Ansys 2024 R1 DEVELOPMENT

Take A Leap Of Certainty™

©2024 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Mechanical User's Guide

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R1
January 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

DRAFT

Table of Contents

Overview xxix
The Ansys Product Improvement Program 31
Application Interface 35
Interface Overview 36
Ribbon 37
File Tab 37
Home Tab 40
Context Tabs 47
Display Tab 78
Selection Tab 88
Automation Tab 94
Learning and Support Tab 95
Graphics Toolbar 97
Clipboard Menu 104
Outline 105
Understanding the Tree Outline 108
Correlating Tree Outline Objects with Model Characteristics 110
Suppressing Objects 113
Filtering the Tree 114
Searching the Tree 119
Details Pane 119
Parameterizing a Variable 128
Geometry Window 129
Status Bar 135
Quick Launch 139
Help Menu 141
Color Theme 141
Ribbon Customization Options 143
Scoping Recovery following Geometry Update 147
Creating User-Defined Buttons 154
Engineering Data Material Window 156
Windows Management 168
Preference Migration 169
Resource Prediction 170
Print Preview 173
Report Preview 174
Publishing the Report 176
Sending the Report 177
Comparing Databases 177
Customizing Report Content 177
Full Screen Mode 178
Contextual Windows 180
Selection Information Window 180
Activating the Selection Information Window 181
Understanding the Selection Modes 181
Using the Selection Information Window 186
Selecting, Exporting, and Sorting Data 190
Worksheet Window 192
Graph and Tabular Data Windows 193

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Messages Window 200
Graphics Annotations Window 202
Section Planes Window 208
Mechanical Wizard Window 208
Group Tree Objects 208
Interface Behavior Based on License Levels 215
Environment Filtering 216
Application Preferences and Default Behaviors 216
Specifying Application Defaults and Preferences 217
Setting Variables 247
Using Macros 248
Data Export 249
Keyframe Animation 255
Graphical Selection and Display 257
Selecting Geometry 258
Selecting Nodes 269
Creating a Coordinate System by Direct Node Selection 274
Specifying Named Selections by Direct Node Selection 275
Selecting Elements and Element Faces 276
Selecting Nodes and Elements by ID 279
Manipulating the Model in the Geometry Window 280
Defining Direction 283
Using Viewports 284
Accelerated Graphics 289
Controlling Graphs and Charts 291
Managing Graphical View Settings 292
Creating a View 292
Applying a View 292
Renaming a View 293
Deleting a View 293
Replacing a Saved View 293
Exporting a Saved View List 293
Importing a Saved View List 293
Copying a View to Mechanical APDL 294
Creating Section Planes 294
Understanding Section Plane Display Differences 303
Working with Section Plane Results 306
Viewing Annotations 306
Specifying Annotation Preferences 310
Controlling Lighting 312
Inserting Comments, Images, and Figures 313
Hotkeys and Keyboard Shortcuts 314
Wizards 320
Mechanical Wizard 321
Steps for Using the Application 323
Create Analysis System 323
Define Materials 328
Attach Geometry/Mesh 330
Importing Geometry or Mesh from Workbench 330
Importing Geometries or Mesh from Mechanical 332
Updating Geometry from Within the Mechanical Application 339

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

CAD Interface Terminology 341
Define Part Behavior 342
Create a Simulation Template 346
Create a Geometry in Mechanical 347
Define Substructures 347
Define Connections 348
Apply Mesh Controls and Preview Mesh 348
Establish Analysis Settings 349
Define Initial Conditions 349
Apply Pre-Stress Effects for Implicit Analysis 352
Apply Loads and Supports 354
Perform Solution 355
Review Results 356
Create Report (optional) 358
Analysis Types 359
Adaptivity Analyses 359
Nonlinear Adaptive Region 360
Geometry Based Adaptivity 369
Coupled Field Analysis Types 376
Coupled Field Harmonic Analysis 380
Prestressed Coupled Field Harmonic Analysis 382
Coupled Field Modal Analysis 386
Prestressed Coupled Field Modal Analysis 388
Coupled Field Static Analysis 391
Coupled Field Transient Analysis 395
Limitations 399
Application Examples and Background 400
Electric Analysis 403
Explicit Dynamics Analysis 406
Fracture Analysis 406
Fracture Analysis Workflows 407
Limitations of Fracture Analysis 413
Fracture Meshing 415
Cracks 424
Crack Overview 424
Defining an Arbitrary Crack 426
Defining a Semi-Elliptical Crack 433
Defining an Elliptical Crack 440
Defining a Ring Crack 445
Defining a Corner Crack 450
Defining an Edge Crack 454
Defining a Through Crack 457
Defining a Cylindrical Crack 461
Defining a Pre-Meshed Crack 465
Special Handling of Named Selections for Crack Objects 466
SMART Crack Growth 468
SMART Crack Growth Application 469
SMART Crack-Growth Assumptions and Limitations 473
Interface Delamination and Contact Debonding 476
Interface Delamination Application 477
Contact Debonding Application 480

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Interface Delamination and Ansys Composite PrepPost (ACP) ... 481
Multi-Point Constraint (MPC) Contact for Fracture ... 482
Solving a Fracture Analysis ... 483
Extracting Fracture Results from a Result File ... 486
Linear Dynamics Analysis Types ... 486
Eigenvalue Buckling Analysis ... 487
Harmonic Response Analysis ... 496
Amplitude Calculation in Harmonic Analysis ... 505
Harmonic Response (Full) Analysis Using Pre-Stressed Structural System ... 506
Harmonic Response Analysis Using Linked Modal Analysis System ... 509
Modal Analysis ... 515
Random Vibration Analysis ... 524
Response Spectrum Analysis ... 533
Substructure Generation Analysis ... 540
Magnetostatic Analysis ... 546
Rigid Dynamics Analysis ... 550
Preparing a Rigid Dynamics Analysis ... 551
Command Reference for Rigid Dynamics Systems ... 561
IronPython References ... 562
The Rigid Dynamics Object Model ... 562
Rigid Dynamics Command Objects Library ... 563
Command Use Examples ... 595
Constraint Equation ... 596
Joint Condition: Initial Velocity ... 598
Joint Condition: Control Using Linear Feedback ... 599
Non-Linear Spring Damper ... 600
Spherical Stop ... 602
Export of Joint Forces ... 603
Breakable Joint ... 605
Debugging RBD Commands with Visual Studio ... 606
Using RBD Commands with Excel ... 609
Using RBD Commands from the IronPython Console ... 610
Using the Variable Load Add-on ... 611
How to Load the Add-on ... 612
Creating Measures ... 612
Body Measures ... 613
Joint Measures ... 614
Derived Measures ... 615
Defining Joint Loads Dependent on one or more Measures ... 616
Defining Force Loads Dependent on one or more Measures ... 621
Known Issues and Limitations ... 622
Using the Motion Load Transfer Add-on ... 622
How to Load the Add-on ... 622
Setting up the Motion Loads Transfer ... 622
Transferring the Motion Loads ... 624
Multibody Dynamics Theory Guide ... 624
Rigid Degrees of Freedom ... 625
Rigid Shape Functions ... 629
Flexible Shape Functions ... 632
Equations of Motion ... 633
Time Integration with Explicit Runge-Kutta ... 638

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Implicit Generalized-α Method 639
Stabilized Implicit Generalized-α Method 641
Moreau-Jean Method 642
Geometric Correction 643
Contact and Stops 645
References 652
Static Structural Analysis 652
Steady-State Thermal Analysis 657
Thermal-Electric Analysis 661
Transient Structural Analysis 665
Transient Structural Analysis Using Linked Modal Analysis System 674
Transient Thermal Analysis 680
Special Analysis Topics 685
Additive Manufacturing Process Simulation 685
Composite Analysis 685
ECAD Analysis using Trace Mapping 686
Trace Mapping in Mechanical 693
ECAD Import Pane (Windows Only) 703
Electromagnetics (EM) - Mechanical Data Transfer 706
Importing Data into Thermal or Structural (Static or Transient) Analyses 707
Importing Data into Harmonic Analyses 710
Importing Forces and Moments 711
Importing Surface Force Density 716
Exporting Results from Thermal or Structural Analyses 718
Electric Machines NVH Analyses (with Waterfall Diagram) 720
External Data 736
Importing Data from Workbench 737
Importing Data from Mechanical 738
Importing Thickness 745
Importing Loads 746
Exporting Data 753
External Study Import 753
Fluid-Structure Interaction (FSI) 756
One-Way Transfer FSI 757
Two-Way Transfer FSI 757
Using Imported Loads for One-Way FSI 758
Face Forces at Fluid-Structure Interface 762
Face Temperatures and Convections at Fluid-Structure Interface 763
Volumetric Temperature Transfer 763
CFD Results Mapping 763
Icepak to Mechanical Data Transfer 764
Inverse Solving for Nonlinear Static Structural Analyses 766
Mechanical-Electronics Interaction (Mechatronics) Data Transfer 772
Overall Workflow for Mechatronics Analysis 772
Set up the Mechanical Application for Export to Twin Builder 773
Polyflow to Mechanical Data Transfer 773
Reinforcement Specification Using Mesh-Independent Method 775
Rotordynamics Analysis 780
Substructure Analysis 781
Condensed Part Overview 784
Condensed Part Application 786

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Condensed Part Worksheet 788
Exporting Condensed Parts 790
Imported Condensed Parts 790
Expansion Pass 800
Limitations 801
Best Practices 805
Static Analysis From Rigid Dynamics Analysis 806
Submodeling 807
Structural Submodeling Workflow 808
Beam-to-Solid/Shell Submodels 813
Thermal Submodeling Workflow 816
Shell-to-Solid Submodels 818
System Coupling 820
Supported Capabilities and Limitations 822
Variables Available for System Coupling 824
System Coupling Related Settings in Mechanical 825
Using Higher-Order Meshes for Coupled Analyses 826
One-Way FSI Transfers Using System Coupling in Workbench 828
Coupled Field Co-Simulation Using System Coupling 830
Coupling Thin Surfaces in CFX 831
Restarting Structural Mechanical Analyses as Part of System Coupling 831
Generating Mechanical Restart Files 832
Specifying a Restart Point in Mechanical 832
Making Setup Changes Before Restarting 833
Recovering the Mechanical Restart Point after a Workbench Crash 834
Restarting a Thermal-Structural Coupled Analysis 835
Running Mechanical as a Coupling Participant in System Coupling's GUI or CLI 835
Troubleshooting Two-Way Coupling Analysis Problems 835
Product Licensing Considerations when using System Coupling 836
Thermal-Stress Analysis 836
Transferring Hydrodynamic Loads to a Structural System 841
Twin Builder/Rigid Dynamics Co-Simulation 842
Co-Simulation Pins 844
Limitations 845
Specifying Geometry 847
Geometry Introduction 847
Parts and Bodies 848
Multibody Behavior and Associativity 850
Geometry Conditions and Requirements 851
Stiffness Behavior 851
Flexible Bodies 852
Rigid Bodies 852
Gasket Bodies 853
Stiff Beam 856
Integration Schemes 857
Common Geometry Display Features 857
Solid Bodies 860
Surface Bodies 860
Preparing Assemblies of Surface Bodies 861
Using Thickness Mode 861
Importing Surface Body Models 862

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Surface Body Thickness 862
Understanding Surface Body Shell Offsets 862
Specifying Surface Body Thickness 864
Specifying Surface Body Layered Sections 868
Defining and Applying a Layered Section 868
Viewing Individual Layers 869
Layered Section Properties 870
Notes on Layered Section Behavior 870
Specifying Surface Body Reinforcements 871
Specifying Faces With Multiple Thicknesses and Layers 871
Line Bodies 872
Simulation without Geometry 878
2D Analyses 878
Using Generalized Plane Strain 881
Construction Geometry 883
Path 883
Surface 890
Solid 892
STL 894
Construction Line 895
Point Mass 901
Distributed Mass 902
Surface Coating 903
Thermal Point Mass 905
Models from External Meshes and Model Assemblies 908
Importing Mesh-Based Geometry 908
Supported External File Types 909
Import Workflow and Interface Options 910
Supported Finite Element Data Types 918
Imported Bolt Pretensions and Premeshed Bolt Pretensions 918
Imported Boundary Conditions 919
Imported Composite Plies 924
Imported Constraint Equations or Coupling 928
Imported Contacts 930
Imported Coordinate Systems 934
Imported Cross Sections 936
Imported Element Orientations 939
Imported Flexible Remote Connectors 942
Imported Named Selections 945
Imported Nodal Orientations 945
Imported Point Mass 947
Imported Rigid Bodies 950
Imported Rigid Remote Connectors 951
Imported Shell Thicknesses 954
Imported Spring Connectors 956
Imported Initial Stresses 961
Automatic Material Assignment 961
Reference Node Naming 962
Importing Mesh-Based Databases in Batch 962
External Model Supported Element Types 969
Supported External Model Commands 973

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

External Model CDB Commands 973
External Model NASTRAN Commands 978
External Model ABAQUS Commands 979
External Model FE Commands Repository 980
Accessing Imported Mesh-Based Databases through ACT 980
Examples of a Synthesized Mesh using Tolerance Angles 986
Assembling External Models and Mechanical Models 989
Assembly Examples 990
Model Assembly Specification 991
Model Alignment 995
Object Renaming 1000
Associativity of Properties 1001
Contact Detection 1002
Mesh Modification 1004
Using Legacy Databases 1005
Limitations and Restrictions for Model Assembly 1005
Element Orientation 1008
Part Transformations 1014
Geometry from Deformation Results 1022
Geometry From Rigid Dynamics Results 1027
Bolt Tools Add-on 1031
Meshing Features 1033
Performing Feature Detection 1033
Specifying Mesh Numbering 1035
Creating Mesh Extrusions 1038
Associating Named Selections To Pull Geometries 1043
Specifying Materials 1049
Material Assignment 1049
Material Plot 1051
Material Combination 1052
Imported Material Fields 1055
Unique Material-Based MAPDL Commands 1056
Working with Named Selections 1057
Create a Named Selection Object 1057
Defining Named Selections 1060
Specifying Named Selections by Geometry Type 1060
Specifying Named Selections using Worksheet Criteria 1061
Specifying Criteria for Geometry-Based Named Selections 1071
Understanding the Named Selections Worksheet 1074
Applying Named Selections via the Ribbon 1075
Promoting Scoped Objects to a Named Selection 1077
Displaying Named Selections 1078
Displaying Interior Mesh Faces 1082
Scoping Analysis Objects to Named Selections 1083
Sending Named Selections to the Solver 1084
Protecting Named Selections 1084
Including Named Selections in Program Controlled Inflation 1084
LS-DYNA Named Selection IDs 1085
Importing Named Selections 1085
Exporting Named Selections 1085
Merging Named Selections 1086

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Converting Named Selection Groups to Mechanical APDL Application Components 1087
Remote Points 1089
Remote Point Application 1090
Geometry Behaviors 1095
Support Specifications 1097
Remote Point Features 1098
Specifying Tables 1103
Table Requirements and Limitations 1103
Create a Table 1105
Import a Table from a Data File 1107
Manually Enter a Table 1109
Enter and Edit Data in a Table 1112
Work with Tables in Free View 1113
Plot Dependent Versus Independent Variables in Free View 1114
Hide Plot in Free View 1115
Sort Data by Column Value or Row Number 1116
Change a Variable for a Column 1117
Add a Column to a Table 1118
Delete a Column from a Table 1119
Add a Row to a Table 1119
Delete a Row from a Table 1119
Work With Tables in Unified View 1120
Enter and Edit Data in Unified View 1121
Plot and Display Variables 1122
Select Active Dependent Variable 1122
Hide Unified View Plot 1123
View Table Details 1123
Rename a Table 1125
Delete a Table 1125
Defining Symmetry 1127
Types of Regions 1128
Symmetry Region Overview 1128
Periodic Region Overview 1131
Electromagnetic Periodic Symmetry 1132
Periodicity Example 1133
Cyclic Region Overview 1134
Pre-Meshed Cyclic Symmetry 1138
Multistage Cyclic Symmetry Analysis 1145
Cyclic Symmetry in a Static Structural or Static Acoustics Analysis 1147
Applying Loads and Supports for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis 1147
Reviewing Results for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis 1149
Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis 1151
Applying Loads and Supports for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis 1151
Non-Cyclic Loading 1153
Reviewing Results for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis 1156
Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis 1159
Applying Loads and Supports for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis 1159

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Settings for Cyclic Symmetry in a Modal Analysis 1160
Analysis Settings for Cyclic Symmetry in a FSI Modal Acoustics Analysis 1161
Reviewing Results for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis 1161
Cyclic Symmetry in a Thermal Analysis 1166
Applying Loads for Cyclic Symmetry in a Thermal Analysis 1166
Reviewing Results for Cyclic Symmetry in a Thermal Analysis 1167
General Axisymmetric Overview 1167
Symmetry Workflow in DesignModeler 1180
Symmetry Workflow in Mechanical 1181
General Axisymmetric Workflow in Mechanical 1188
Setting Up Coordinate Systems 1191
Creating Coordinate Systems 1191
Initial Creation and Definition 1192
Establishing Origin for Associative and Non-Associative Coordinate Systems 1192
Setting Principal Axis and Orientation 1194
Using Transformations 1195
Creating a Coordinate System Based on a Surface Normal 1196
Creating a Coordinate System Based on the Center of Mass 1197
Setting a Coordinate System Origin at the Center of Mass 1197
Importing Coordinate Systems 1198
Applying Coordinate Systems as Reference Locations 1198
Using Coordinate Systems to Specify Joint Locations 1199
Creating Coordinate-Based Section Planes 1199
Transferring Coordinate Systems to the Mechanical APDL Application 1201
Setting Up Coordinate Systems in the Mechanical Configure Tool 1202
Setting Connections 1207
Connections Folder 1207
Connections Worksheet 1210
Connection Group 1212
Connection Features and Operations 1218
Contact 1222
Contact Overview 1222
Contact Formulation Theory 1222
Contact Settings 1226
Scope Settings 1227
Definition Settings 1234
Advanced Settings 1240
Display 1256
Geometric Modification 1257
Supported Contact Types 1265
Setting Contact Conditions Manually 1266
Contact Ease of Use Features 1267
Automatically Generate Objects Scoped to Contact Regions 1268
Controlling Transparency for Contact Regions 1269
Displaying Contact Bodies with Different Colors 1269
Displaying Contact Bodies in Separate Windows 1270
Hiding Bodies Not Scoped to a Contact Region 1271
Renaming Contact Regions Based on Geometry Names 1271
Identifying Contact Regions for a Body 1272
Create Contact Debonding 1272
Flipping Contact and Target Scope Settings 1272

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Setting Default APDL Names 1273
Merging Contact Regions That Share Geometry 1274
Saving or Loading Contact Region Settings 1274
Resetting Contact Regions to Default Settings 1275
Locating Bodies Without Contact 1275
Locating Parts Without Contact 1275
Contact in Rigid Dynamics 1275
Best Practices for Contact in Rigid Body Analyses 1278
Best Practices for Specifying Contact Conditions 1281
Contact Setup and Verification 1281
Solver Preparation 1286
Addressing Non-Convergence 1290
Joints 1292
Joint Characteristics 1293
Joint Types 1298
Fixed Joint 1298
Revolute Joint 1299
Cylindrical Joint 1299
Translational Joint 1300
Slot Joint 1301
Universal Joint 1301
Spherical Joint 1302
Planar Joint 1302
Bushing Joint 1303
Screw Joint 1307
Constant Velocity Joint (Homokinetic Joint) 1308
Distance Joint 1309
General Joint 1310
Point on Curve Joint 1312
Imperfect Joint Types 1314
In-Plane Radial Gap 1314
Spherical Gap 1315
Radial Gap 1316
Joint Properties 1316
Modifying Joint Coordinate Systems 1326
Joint Stiffness 1331
Joint Friction 1335
Joint Friction Definitions 1336
Joint Types 1347
Joint Friction Type 1353
Notes 1354
Manual Joint Creation 1357
Automatic Joint Creation 1359
Joint Stops and Locks 1360
Ease of Use Features 1365
Detecting Overconstrained Conditions 1369
Example: Assembling Joints 1370
Example: Configuring Joints 1380
Example: Configuring Bodies 1391
Springs 1395
Beam Connections 1404

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Spot Welds 1407
End Releases 1409
Bearings 1413
Configuring Analysis Settings 1417
Analysis Settings for Most Analysis Types 1417
Step Controls for Static and Transient Analyses 1418
Step Controls for Harmonic Analysis Types 1426
Solver Controls 1428
Restart Analysis 1437
Restart Controls 1438
Adaptivity Remeshing Controls 1440
Creep Controls 1445
Fracture Controls 1445
Cyclic Controls 1446
Radiosity Controls 1446
Options for Analyses 1447
Coupled Field Harmonic, Harmonic Acoustics, and Harmonic Response Options 1447
Eigenvalue Buckling Analysis Options 1455
Modal Acoustics and Modal Analysis Options 1455
Random Vibration Options 1457
Response Spectrum Options 1458
Substructure Generation Options 1458
Transient Structural (Linked to Modal) Options 1459
Scattering Controls 1461
Advanced 1461
Damping Controls 1462
Nonlinear Controls 1468
Nonlinear Controls for Steady-State, Static, and Transient Analyses 1468
Nonlinear Controls for Transient Thermal Analyses 1472
Nonlinear Controls for Rigid Dynamics Analyses 1472
Output Controls 1473
Analysis Data Management 1484
Rotordynamics Controls 1490
Visibility 1491
Steps and Step Controls for Static and Transient Analyses 1491
Role of Time in Tracking 1491
Steps, Substeps, and Equilibrium Iterations 1492
Automatic Time Stepping 1493
Guidelines for Integration Step Size 1493
Setting Up Boundary Conditions 1497
Boundary Condition Scoping Method 1497
Types of Boundary Conditions 1500
Inertial Type Boundary Conditions 1500
Acceleration 1501
Standard Earth Gravity 1509
Rotational Velocity 1511
Rotational Acceleration 1516
Load Type Boundary Conditions 1520
Pressure 1521
Pipe Pressure 1532
Pipe Temperature 1534

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hydrostatic Pressure 1537
Force 1543
Remote Force 1552
Bearing Load 1561
Bolt Pretension 1566
Moment 1574
Generalized Plane Strain 1582
Line Pressure 1585
PSD Base Excitation 1589
RS Base Excitation 1590
Joint Load 1592
Thermal Condition 1594
Temperature 1598
Convection 1602
Radiation 1609
Heat Flow 1614
Heat Flux 1617
Internal Heat Generation 1620
Mass Flow Rate 1622
Electric Charge 1625
Voltage 1627
Current 1630
Voltage (Ground) 1633
Electromagnetic Boundary Conditions and Excitations 1635
Magnetic Flux Boundary Conditions 1636
Conductor 1637
Solid Source Conductor Body 1638
Voltage Excitation for Solid Source Conductors 1640
Current Excitation for Solid Source Conductors 1641
Stranded Source Conductor Body 1642
Current Excitation for Stranded Source Conductors 1643
Motion Load 1645
Fluid Solid Interface 1648
System Coupling Region 1651
Rotating Force 1655
Imported CFD Pressure 1661
Support Type Boundary Conditions 1663
Fixed Support 1664
Displacement 1666
Remote Displacement 1674
Velocity 1680
Frictionless Support 1682
Compression Only Support 1685
Cylindrical Support 1688
Simply Supported 1690
Fixed Rotation 1692
Elastic Support 1695
Conditions Type Boundary Conditions 1697
Coupling 1698
Voltage Coupling 1700
Constraint Equation 1702

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Pipe Idealization 1704
Nonlinear Adaptive Region 1707
Geometry Based Adaptivity 1707
Element Birth and Death 1707
Contact Step Control 1711
Plastic Heating 1715
Viscoelastic Heating 1717
Direct FE Type Boundary Conditions 1719
Nodal Orientation 1719
Nodal Force 1721
Nodal Pressure 1724
Nodal Displacement 1728
Nodal Rotation 1730
EM (Electro-Mechanical) Transducer 1732
Remote Boundary Conditions 1735
Imported Boundary Conditions 1736
Supported Analysis Types and Systems 1737
Supported Boundary Conditions 1738
Imported Body Force Density 1739
Imported Body Temperature 1740
Imported Boundary Remote Constraint 1744
Imported Convection Coefficient 1744
Imported Cut Boundary Constraint 1745
Imported Cut Boundary Remote Force 1746
Imported Displacement 1746
Imported Force 1747
Imported Heat Flux 1748
Imported Heat Generation 1748
Imported Initial Strain 1749
Imported Initial Stress 1750
Recommendations and Guidelines for Mapping of Initial Stress and Strain Data 1751
Imported Pressure 1752
Imported Remote Loads 1755
Imported Surface Force Density 1755
Imported Temperature 1756
Imported Velocity 1756
Applying Imported Boundary Conditions 1757
Load Mapping Workflow Specification 1762
Spatial Varying Loads and Displacements 1769
Defining Boundary Condition Magnitude 1771
Specifying Boundary Condition Loads With Tables 1781
Specify Pressure Loads with Tables 1781
Specify Thermal Condition Loads with Tables 1783
Specify Temperature Loads with Tables 1785
Matching the Spatial Coordinate System and Table Coordinate System 1786
Specifying Model-Level Boundary Conditions 1787
Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) 1789
Resolving Thermal Boundary Condition Conflicts 1790
Understanding Solving 1791
Understanding Interface Options 1791
Performing the Solution 1793

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using HPC Platform Services to Solve Linked Analyses 1801
Understanding HPC Platform Services (HPS) Limitations 1802
Selecting Solve Modes for an Analysis 1803
Using Solve Process Settings 1804
Memory Tuning the Samcef Solver 1814
Memory Tuning the ABAQUS Solver 1814
Using Solution Restarts 1815
Restart Analysis Extension 1823
Understanding Solve Scenarios 1831
Specifying Solution Information 1833
Postprocessing During Solve 1844
Using Result Trackers 1844
Structural Result Trackers 1847
Thermal Result Trackers 1852
Using Adaptive Convergence 1852
Saving and Managing Results 1858
Writing and Reading Solution Data 1860
Writing and Reading the Mechanical APDL Application Files 1861
Writing and Reading the LS-DYNA Application Files 1864
Writing Ansys Rigid Dynamics Files 1867
Writing NASTRAN Files 1867
NASTRAN Export Supported Features 1870
NASTRAN Export Limitations 1872
Understanding Solving Units 1874
Using Results 1927
Introduction to the Use of Results 1927
Result Outputs 1928
Chart and Table 1929
Line Chart Results (LS-DYNA Only) 1932
Contour Results 1935
Coordinate Systems Results 1935
Nodal Coordinate Systems Results 1935
Elemental Coordinate Systems Results 1936
Rotational Order of Coordinate System Results 1937
Interpolation of Result Values 1937
Line Body Results 1938
Path Results 1940
Probes 1946
Probe Types and Application Overview 1947
Probe Details View 1951
Result Set Listing 1957
Surface Results 1959
Solution Summary Worksheet 1961
Vector Plots 1968
Result Definitions 1968
Applying Results Based on Geometry 1969
Specifying Result Coordinate Systems 1974
Solution Coordinate System 1975
Applying Decibel (dB) Weighting Filters 1977
Defining Result Identifiers 1978
Understanding the Material Properties Used in Postprocessing 1980

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Clearing Result Data 1981
Understanding Averaged and Unaveraged Contour Results 1981
Working with Multiple Result Sets 1990
Displaying Surface Body Results (including Layered Shell Results) 1993
Reviewing Unconverged Results 1995
Handling of Degenerate Elements 1995
Understanding Result Data Display Issues 1996
Result Scoping 1996
Geometry and Mesh 1997
Path Construction Geometry 2002
Surface Construction Geometry 2002
Result File Items 2002
Surface Coatings 2008
Composite Analysis 2012
Imported Condensed Parts 2014
Structural Results 2017
Deformation 2018
Stress and Strain 2026
Equivalent (von Mises) 2027
Maximum, Middle, and Minimum Principal 2028
Maximum Shear 2029
Intensity 2029
Vector Principals 2030
Error (Structural) 2030
Thermal Strain 2032
Equivalent Plastic Strain 2032
Accumulated Equivalent Plastic Strain 2033
Equivalent Creep Strain 2034
Equivalent Total Strain 2035
Membrane Stress 2035
Bending Stress 2036
Stabilization Energy 2036
Strain Energy 2037
Damage Results 2037
Linearized Stress 2041
Contact Results 2043
Frequency Response 2046
Phase Response 2056
Stress Tools 2058
Maximum Equivalent Stress Safety Tool 2059
Maximum Shear Stress Safety Tool 2061
Mohr-Coulomb Stress Safety Tool 2062
Maximum Tensile Stress Safety Tool 2064
Fatigue (Fatigue Tool) 2066
Fracture Results (Fracture Tool) 2066
Composite Failure Tool 2072
Composite Sampling Point Tool 2077
Contact Tool 2079
Contact Tool Initial Information 2084
Bolt Tool 2086
Beam Tool 2087

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Beam Results 2089
Shear-Moment Diagram 2090
Structural Probes 2092
Position 2102
Energy 2105
Reactions: Forces and Moments 2107
Joint Probes 2118
Response PSD Probe 2121
Spring Probes 2123
Bearing Probes 2124
Beam Probes 2125
Bolt Pretension Probes 2125
Generalized Plain Strain Probes 2125
Contact Distance Probes 2126
Fracture Probes (Fracture Tool) 2127
Response PSD Tool 2130
Gasket Results 2131
Campbell Diagram Chart Results 2132
Waterfall Diagrams 2135
Waterfall Diagram Data Filtering Features (RPM Only) 2136
Waterfall Diagram Display Features 2139
Line Pressure Result 2151
Volume Result 2151
Volume Probe 2152
Thermal Results 2153
Temperature 2153
Heat Flux 2153
Heat Reaction 2154
Error (Thermal) 2154
Thermal Flow Results 2155
Thermal Probes 2155
Thermal Contact Results 2156
Magnetostatic Results 2157
Electric Potential 2158
Total Magnetic Flux Density 2158
Directional Magnetic Flux Density 2158
Total Magnetic Field Intensity 2158
Directional Magnetic Field Intensity 2159
Total Force 2159
Directional Force 2159
Current Density 2159
Inductance 2159
Flux Linkage 2160
Error (Magnetic) 2161
Magnetostatic Probes 2161
Electric Results 2162
Electric Probes 2163
Frequency Response for Electric Results 2164
Fatigue Results 2165
Fatigue Material Properties 2165
Fatigue Stress Life versus Strain Life 2167

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Frequency-Based Fatigue 2169
Fatigue Material Properties for Random Vibration (Spectral) Fatigue 2169
Fatigue Result Methods for Random Vibration (Spectral) Fatigue 2173
Fatigue Result Methods for Harmonic Fatigue 2175
Fatigue Analysis Application 2176
Fatigue Results 2181
Fatigue Combination 2188
Mechanical Embedded DesignLife UI 2191
Noise, Vibration, and Harshness 2191
Forced Response 2192
User Defined Results 2192
Overview 2192
Characteristics 2193
Application 2194
Node-Based Scoping 2197
User Defined Result Expressions 2197
Unit Description 2202
User Defined Results for the Mechanical APDL Solver 2203
User Defined Results for the Ansys Rigid Dynamics Solver 2212
Python Result 2212
Insert the Python Result Object 2213
Understanding Data Processing Framework 2216
Using DPF with the Python Result Feature 2218
Limitations of the Python Result Object 2220
Workflow Examples 2220
Display Results for Shared Shell Topologies 2221
Total Deformation 2223
Von Mises Stress 2224
Retrieving Total Deformation from Table/Chart for Time Point 2226
Strain Tensors - Named Selection Scoping 2227
Total Deformation Scoped on Time Step using Property Provider 2228
Total Deformation Scoped on Geometry using Property Provider 2231
Migrating to New Python Result 2242
Animation 2242
User Defined Criteria 2243
Define Primary Criterion for the Measures Object 2244
Define Primary Criterion for a Harmonic Response Analysis 2246
Define Primary Criterion for a Modal Analysis 2247
Define Primary Criterion for a Static Structural Analysis 2248
Combine Primary Criterion Objects 2250
Understanding the Criterion Calculations 2250
Result Utilities 2258
Automatic Result Creation for All Result Sets 2259
Adaptive Convergence 2260
Animation 2260
Animation Options 2260
Animation Behaviors 2264
Animation with Attached Camera 2266
Accelerated Animation 2268
Capped IsoSurfaces 2269
Dynamic Legend 2271

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting Results 2276
Generating Reports 2278
Local Minimum and Maximum Probes 2278
Renaming Results Based on Definition 2283
Results Legend 2284
Results Tab 2288
Solution Combination 2288
Solution Combination Process Requirements and Conditions 2300
Using the Commands Object 2303
Command Entry and Mechanical APDL Entry Options 2304
Commands (APDL) Object Properties 2307
Commands (APDL) Object Post Processing Specifications 2308
Commands (APDL) Objects and the Mechanical APDL Solver 2312
Commands (APDL) Objects and the Rigid Dynamics Solver 2321
Commands (APDL) Objects and the LS-DYNA Solver 2321
Using Python Code 2323
Using Python Code for MAPDL Solver Input 2324
Using Python Code for Event Based Code Execution 2324
Resuming Databases with Python Code Objects 2327
Sharing Scope 2329
Working with the Property Provider 2330
Setting Parameters 2337
Specifying Parameters 2337
CAD Parameters 2340
Productivity Tools 2343
Generating Multiple Objects from a Template Object 2343
Tagging Objects 2349
Creating Tags 2349
Applying Tags to Objects 2349
Deleting a Tag 2349
Renaming a Tag 2350
Highlighting Tagged Tree Objects 2350
CAD System Information 2351
General Information 2352
Troubleshooting 2353
General Product Limitations 2353
Problem Situations 2357
A Linearized Stress Result Cannot Be Solved. 2359
A Load Transfer Error Has Occurred. 2360
A Master Node is Missing from the Condensed Part 2360
Although the Exported File Was Saved to Disk 2360
Although the Solution Failed to Solve Completely at all Time Points. 2360
An Error Occurred Inside the SOLVER Module: Invalid Material Properties 2361
An Error Occurred While Solving Due To Insufficient Disk Space 2362
An Error Occurred While Starting the Solver Module 2362
An Internal Solution Magnitude Limit Was Exceeded. 2363
An Iterative Solver Was Used for this Analysis 2363
At Least One Body Has Been Found to Have Only 1 Element 2364
At Least One Spring Exists with Incorrectly Defined Nonlinear Stiffness 2365
Animation Does not Export Correctly 2365
Application Not Closing as Expected 2365

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Assemblies Missing Parts 2365
Cannot Undo Node Move 2365
CATIA V5 and IGES Surface Bodies 2366
Constraint Equations Were Not Properly Matched 2366
Element *n* Located in *Body* (and maybe other elements) Has Become Highly Distorted 2366
Error Inertia tensor is too large 2366
Equivalent Creep Strain Ratio has Exceeded the Specified Limit Value 2367
Failed to Load Microsoft Office Application 2367
Illogical Reaction Results 2367
Large Deformation Effects are Active 2367
Missing fonts for the Docking Pane Buttons (Linux Platform Only) 2368
MPC equations were not built for one or more contact regions or remote boundary conditions 2368
One or More Contact Regions May Not Be In Initial Contact 2368
One or more MPC or Lagrange Multiplier formulation based contact may have conflicts 2369
One or More Parts May Be Underconstrained 2370
One or More Remote Boundary Conditions is Scoped to a Large Number of Elements 2370
Problems Unique to Background (Asynchronous) Solutions 2371
Problems Using Solution 2372
Proxy Server Environment Variable 2373
Remote Points with Overlapping Geometry Selections are not Recommended within a Condensed Part 2373
Running Norton AntiVirus™ Causes the Mechanical Application to Crash 2374
The Correctly Licensed Product Will Not Run 2374
The Deformation is Large Compared to the Model Bounding Box 2374
The Initial Time Increment May Be Too Large for This Problem 2374
The Joint Probe cannot Evaluate Results 2375
The License Manager Server Is Down 2375
Linux Platform - Localized Operating System 2376
The Low/High Boundaries of Cyclic Symmetry 2377
The Remote Boundary Condition object is defined on the Cyclic Axis of Symmetry 2377
The Solution Combination Folder 2377
The Solver Engine was Unable to Converge 2378
The Solver Has Found Conflicting DOF Constraints 2379
Problem with RSM-Mechanical Connection 2379
Unable to Find Requested Modes 2379
You Must Specify Joint Conditions to all Three Rotational DOFs 2380
Fracture Meshing Problems 2380
Lustre Parallel File Systems on Linux 2383
Recommendations 2383
A. Glossary of General Terms 2385
B. Data Transfer Mapping and Validation 2389
Data Transfer Mesh Mapping 2389
Mapping Validation 2412
C. Workbench Mechanical Wizard Advanced Programming Topics 2417
Overview 2417
URI Address and Path Considerations 2418
Using Strings and Languages 2419
Guidelines for Editing XML Files 2420
About the TaskML Merge Process 2420
Using the Integrated Wizard Development Kit (WDK) 2421
Using IFRAME Elements 2422

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TaskML Reference 2423
Overview Map of TaskML 2423
Document Element 2424
simulation-wizard 2424
External References 2424
Merge 2424
Script 2425
Object Grouping 2425
object-group 2426
object-groups 2426
object-type 2426
Status Definitions 2427
status 2427
statuses 2428
Language and Text 2428
data 2428
language 2429
string 2429
strings 2430
Tasks and Events 2430
activate-event 2430
task 2431
tasks 2432
update-event 2432
Wizard Content 2432
body 2432
group 2433
iframe 2434
taskref 2434
Rules 2435
Statements 2435
and 2435
debug 2436
if then else stop 2436
not 2437
or 2437
update 2438
Conditions 2438
assembly-geometry 2438
changeable-length-unit 2439
geometry-includes-sheets 2439
level 2439
object 2440
zero-thickness-sheet 2441
valid-emag-geometry 2441
enclosure-exists 2441
Actions 2442
click-button 2442
display-details-callout 2443
display-help-topic 2443
display-outline-callout 2444

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

display-status-callout 2445
display-tab-callout 2445
display-task-callout 2446
display-toolbar-callout 2446
open-url 2447
select-all-objects 2447
select-field 2448
select-first-object 2449
select-first-parameter-field 2450
select-first-undefined-field 2451
select-zero-thickness-sheets 2451
select-enclosures 2451
send-mail 2452
set-caption 2452
set-icon 2453
set-status 2453
Scripting 2454
eval 2454
Standard Object Groups Reference 2455
Tutorials 2458
Tutorial: Adding a Link 2459
Tutorial: Creating a Custom Task 2460
Tutorial: Creating a Custom Wizard 2462
Tutorial: Adding a Web Search IFRAME 2463
Completed TaskML Files 2464
Links.xml 2464
Insert100psi.xml 2465
CustomWizard.xml 2466
Search.htm 2467
CustomWizardSearch.xml 2468
Wizard Development Kit (WDK) Groups 2469
WDK: Tools Group 2469
WDK: Commands Group 2470
WDK Tests: Actions 2470
WDK Tests: Flags (Conditions) 2471
Index 2473

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

List of Figures

1. Variable Load Add-on Showing Loaded Status 612
2. Motion Load Transfer Add-on Showing Loaded Status 622
3. Double Pendulum Model 625
4. Absolute Degrees of Freedom 626
5. Relative Degrees of Freedom 627
6. Closed Loop Model 628
7. Generalized Velocities of a Material Point 630
8. Contribution of the Parent Joint to the Generalized Velocities 630
9. Flexible Bodies Kinematics 632
10. Crankshaft Mechanism 636
11. Contact Between Two Convex Bodies 645
12. Stops on a Translational Joint 646
13. One Contact Point 646
14. Two Contact Points 647
15. Cylinder/Cylinder Contact 647
16. Contact Requiring One Single Point 648
17. 3D Solid Submodel Superimposed on Coarse Shell Model 819
18. Node rotations (a) before mapping command, (b) after mapping command 820
19. Two Surfaces in Mechanical with the Correct Offset Parameter for Coupling with a Thin Surface 831
20. Unexpanded One Sector Model Display 1150
21. Expanded Full Symmetry Model Display 1150
22. Initial Geometry 1371
23. Selecting a Face for a Body-Ground Fixed Connection 1372
24. Creating the Reference Mobile System 1373
25. Creating the Reference Coordinate System 1374
26. Creating the Mobile Coordinate System 1375
27. Orienting the Pendulum Axis 1376
28. Oriented Coordinate Systems 1376
29. Scoping the Mobile Coordinate Systems 1377
30. Choose an Edge to Orient the PendulumAxis Geometry 1379
31. Assembled Geometry 1379
32. Equivalent (von-Mises) stress 1752
33. Equivalent (von-Mises) strain (elastic/plastic/equivalent plastic) 1752
34. Interpolating Between Different (but equivalent) Euler Angles 2391
35. Quaternion versus Euler Angle Interpolation 2391
36. Profile Preserving Mapping 2391
37. Conservative Mapping 2392
38. Outside Nodes (Pink) with Mesh Overlay 2403
39. Maximum Distance set to 0.005 (m) 2403
40. Mapped Nodes 2404
41. Imported Data using Maximum Distance for Outside Nodes 2404
42. Interpolating Flipped Orientations 2405
43. Shell-Solid Submodeling with Pinball Factor = 1.0 2406
44. Shell-Solid Submodeling with Pinball Factor = 1.2 2406
45. Shell-Solid Submodeling with Shell Thickness Factor = 0.6 2407
46. Shell-Solid Submodeling with Shell Thickness Factor = 1.2 2407

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

List of Tables

1. Variables On Boundary Wall Regions 824
2. Variables On Body System Coupling Regions 824
3. Comparing Contact Formulations 1225
4. Ansys Workbench Product Adaptivity Methods 1856
5. Acceleration and RS Acceleration 1876
6. Angle 1877
7. Angular Acceleration 1877
8. Angular Velocity 1878
9. Area 1878
10. Capacitance 1879
11. Charge 1880
12. Charge Density 1880
13. Conductivity 1881
14. Current 1881
15. Current Density 1882
16. Decay Constant 1882
17. Density 1883
18. Displacement and RS Displacement 1883
19. Electric Conductance Per Unit Area 1884
20. Electric Conductivity 1885
21. Electric Field 1885
22. Electric Flux Density 1886
23. Electric Resistivity 1886
24. Energy 1887
25. Energy Density by Mass 1887
26. Energy Per Volume 1888
27. Film Coefficient 1889
28. Force 1889
29. Force Intensity 1890
30. Force Per Angular Unit 1891
31. Fracture Energy (Energy Release Rate) 1891
32. Frequency 1892
33. Gasket Stiffness 1892
34. Heat Flux 1893
35. Heat Generation 1893
36. Heat Rate 1894
37. Impulse 1894
38. Impulse Per Angular Unit 1895
39. Inductance 1895
40. Inverse Angle 1896
41. Inverse Length 1896
42. Inverse Stress 1897
43. Length 1897
44. Magnetic Field Intensity 1898
45. Magnetic Flux 1898
46. Magnetic Flux Density 1899
47. Mass 1899
48. Material Impedance 1900
49. Moment 1900

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

50. Moment of Inertia of Area 1901
51. Moment of Inertia of Mass 1902
52. Normalized Value 1902
53. Permeability 1903
54. Permittivity 1903
55. Poisson's Ratio 1904
56. Power 1904
57. Pressure 1905
58. PSD Acceleration 1906
59. PSD Acceleration (G) 1906
60. PSD Displacement 1907
61. PSD Force 1907
62. PSD Moment 1908
63. PSD Pressure 1908
64. PSD Strain 1909
65. PSD Stress 1909
66. PSD Velocity 1910
67. Relative Permeability 1910
68. Relative Permittivity 1911
69. Rotational Damping 1911
70. Rotational Stiffness 1912
71. Seebeck Coefficient 1912
72. Section Modulus 1913
73. Shear Elastic Strain 1913
74. Shock Velocity 1914
75. Specific Heat 1914
76. Specific Weight 1915
77. Square Root of Length 1916
78. Stiffness 1916
79. Strain and RS Strain 1917
80. Strength 1918
81. Stress and RS Stress 1918
82. Stress Intensity Factor 1919
83. Thermal Capacitance 1920
84. Thermal Conductance - 3D Face and 2D Edge 1920
85. Thermal Conductance - 3D Edges and Vertices 1920
86. Thermal Expansion 1921
87. Temperature 1921
88. Temperature Difference 1922
89. Temperature Gradient 1923
90. Time 1923
91. Translational Damping 1924
92. Velocity and RS Velocity 1924
93. Voltage 1925
94. Volume 1925
95. Element Information 2425
96. Element Information 2425
97. Attributes 2431
98. Element Information 2431
99. Element Information 2434

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Overview

Ansys Mechanical is a Workbench application that can perform a variety of engineering simulations, including stress, thermal, vibration, thermo-electric, and magnetostatic simulations.

A typical simulation consists of setting up the model and the loads applied to it, solving for the model's response to the loads, then examining the details of the response with a variety of tools.

Mechanical has "objects" arranged in a tree structure that guide you through the different steps of a simulation. By expanding the objects, you expose the details associated with the object, and you can use the corresponding tools and specification tables to perform that part of the simulation. Objects are used, for example, to define environmental conditions such as contact surfaces and loadings, and to define the types of results you want to have available for review.

The following Help topics describe in detail how to use Mechanical to set up and run a simulation:

- Application Interface (p. 35)
- Steps for Using the Application (p. 323)
- Analysis Types (p. 359)
- Specifying Geometry (p. 847)
- Setting Up Coordinate Systems (p. 1191)
- Setting Connections (p. 1207)
- Configuring Analysis Settings (p. 1417)
- Setting Up Boundary Conditions (p. 1497)
- Using Results (p. 1927)
- Understanding Solving (p. 1791)
- Commands Objects (p. 2303)
- Setting Parameters (p. 2337)

After you become comfortable using Mechanical, you might want to write scripts that automate your routine tasks. Eventually, you might even want to create extensions that customize and automate Mechanical itself. You can accomplish all of this using Ansys ACT and its powerful API (Application Programming Interface).

- For an introduction to writing scripts and information on using the ACT API to access and manipulate objects in the Mechanical tree, see the *Scripting in Mechanical Guide*.
- For descriptions of all ACT API objects, methods, and properties, see the *ACT API Reference Guide*.
- For information on how to use ACT to create apps (extensions) that customize and automate Ansys products, see the *ACT Developer's Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For ACT usage, customization, and automation information specific to Mechanical, see the *ACT Customization Guide for Mechanical*.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables Ansys, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How to Participate

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog appears. Only then will collection of data for this product begin.

How the Program Works

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized Ansys employees. After Ansys receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

Data We Collect

The data we collect under the Ansys Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

Hardware: Information about the hardware on which the product is running, such as the:

- brand and type of CPU
- number of processors available
- amount of memory available
- brand and type of graphics card

System: Configuration information about the system the product is running on, such as the:

- operating system and version
- country code
- time zone
- language used
- values of environment variables used by the product

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Session: Characteristics of the session, such as the:

- interactive or batch setting
- time duration
- total CPU time used
- product license and license settings being used
- product version and build identifiers
- command line options used
- number of processors used
- amount of memory used
- errors and warnings issued

Session Actions: Counts of certain user actions during a session, such as the number of:

- project saves
- restarts
- meshing, solving, postprocessing, etc., actions
- times the Help system is used
- times wizards are used
- toolbar selections

Model: Statistics of the model used in the simulation, such as the:

- number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.
- number of material types, loading types, boundary conditions, species, etc.
- number and types of coordinate systems used
- system of units used
- dimensionality (1-D, 2-D, 3-D)

Analysis: Characteristics of the analysis, such as the:

- physics types used
- linear and nonlinear behaviors
- time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)
- analysis options used

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution: Characteristics of the solution performed, including:

- the choice of solvers and solver options
- the solution controls used, such as convergence criteria, precision settings, and tuning options
- solver statistics such as the number of equations, number of load steps, number of design points, etc.

Specialty: Special options or features used, such as:

- user-provided plug-ins and routines
- coupling of analyses with other Ansys products

Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames
- file names, part names, or other user-supplied labels
- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values
- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

Opting Out of the Program

You may *stop* your participation in the program any time you wish. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

The Ansys, Inc., Privacy Policy

All Ansys products are covered by the Ansys, Inc., Privacy Policy.

Frequently Asked Questions

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   No. You are not enrolled unless you explicitly agree to participate.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by Ansys?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

   Yes, you can stop participating at any time. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

   Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *Ansys Product Improvement Program* description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

   Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

10. *Can I easily opt out of the Product Improvement Program for all clients in my network installation?*

    Yes. Perform the following steps on the file server:

    a. Navigate to the installation directory: `[Drive:]\v241\commonfiles\globalsettings`

    b. Open the file **ANSYSProductImprovementProgram.txt**.

    c. Change the value from "on" to "off" and save the file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical Application Interface

This section describes the elements of the Mechanical interface, their purpose and conditions, as well as the methods for their use.

The following topics are covered in this section:

Interface Overview
Ribbon
Graphics Toolbar
Outline
Details Pane
Geometry Window
Status Bar
Quick Launch
Help Menu
Color Theme
Ribbon Customization Options
Scoping Recovery following Geometry Update
Creating User-Defined Buttons
Engineering Data Material Window
Windows Management
Preference Migration
Resource Prediction
Print Preview
Report Preview
Full Screen Mode
Contextual Windows
Group Tree Objects
Interface Behavior Based on License Levels
Environment Filtering
Application Preferences and Default Behaviors
Using Macros
Data Export
Keyframe Animation
Graphical Selection and Display
Hotkeys and Keyboard Shortcuts

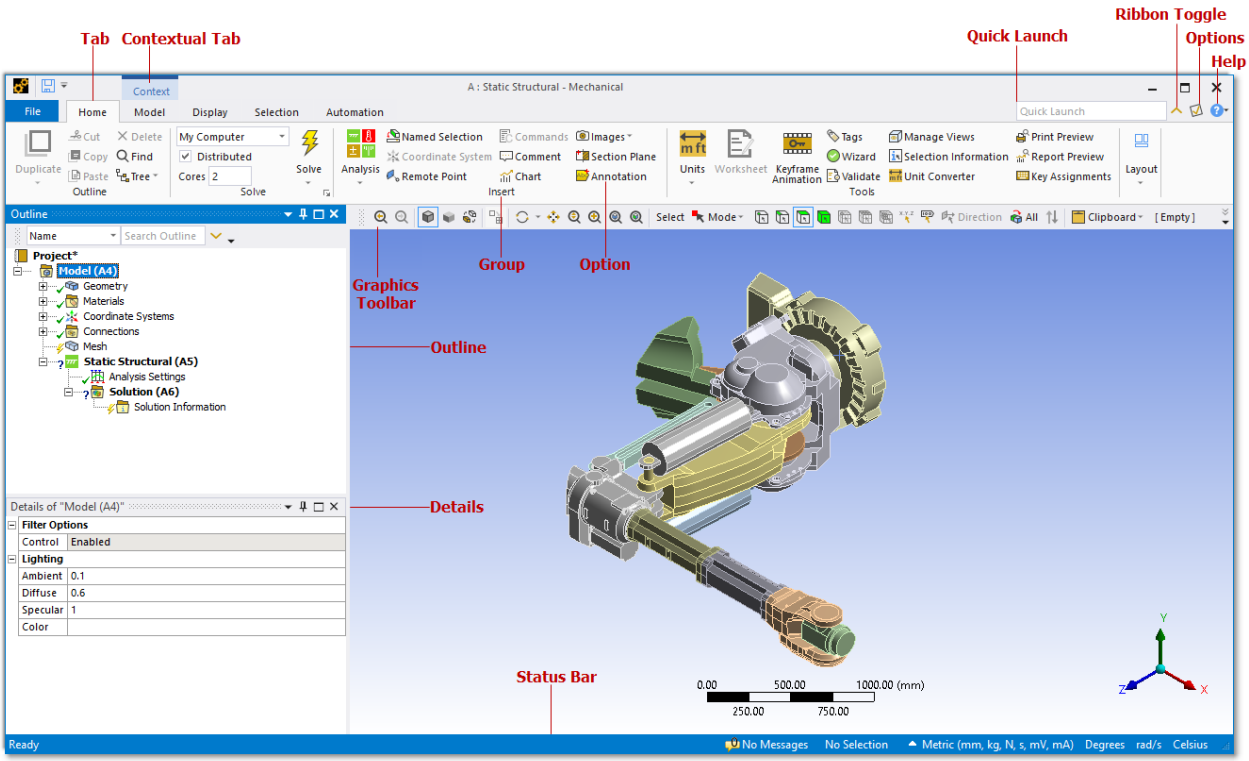
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Wizards

Interface Overview

The Ansys Mechanical application user interface is illustrated below.



The primary interface elements include:

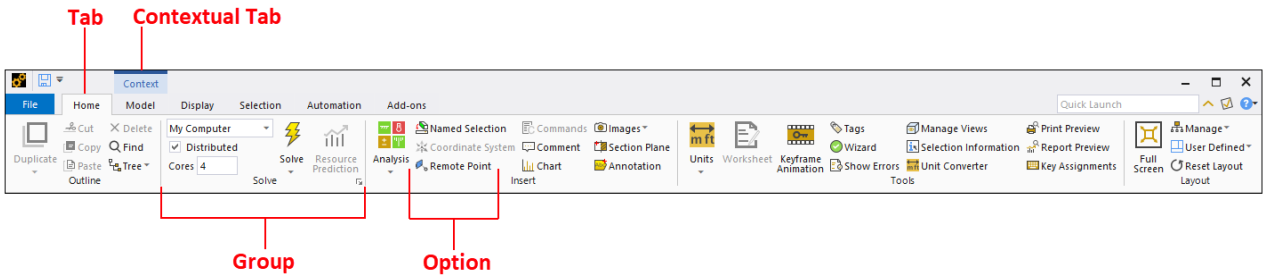
- Ribbon (p. 37)
- Graphics Toolbar (p. 97)
- Outline (p. 105)
- Details View (p. 119)
- Geometry Window (p. 129)
- Status Bar (p. 135)
- Quick Launch (p. 139)
- Help (p. 141)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Ribbon

The ribbon provides easy-to-use option toolbars organized by Tabs. By grouping similar commands together, you will work faster and more efficiently.



Ribbon Structure

The ribbon is organized by Tabs (**Home**, **Display**, **Selection**, **Automation**, etc.). Within each Tab, Options (command buttons) are organized into Groups (**Outline**, **Solve**, etc.) by functionality. This reduces your search time when looking for specific commands. Additionally, a **Context** tab appears based on your currently selected object with options specific to the selected object.

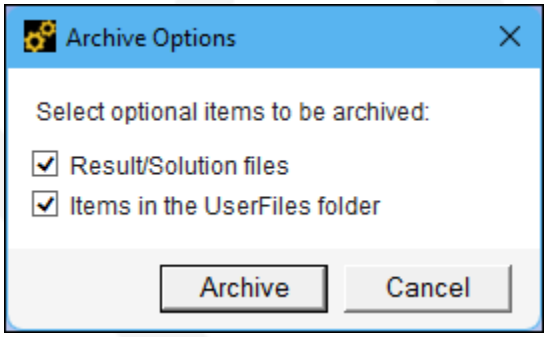
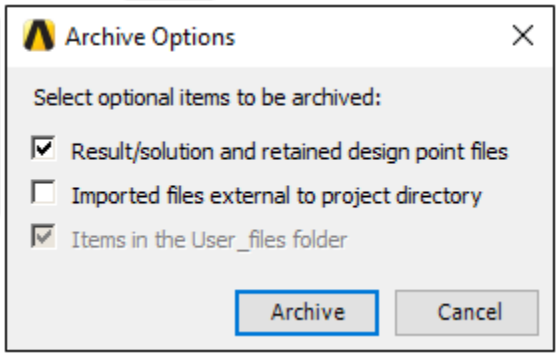
Review the following sections for additional information about each tab:

- File Tab (p. 37)
- Home Tab (p. 40)
- Context Tabs (p. 47)
- Display Tab (p. 78)
- Selection Tab (p. 88)
- Automation Tab (p. 94)
- Add-ons Tab
- Learning and Support Tab (p. 95)

File Tab

The File tab contains a variety of options for managing your project, defining author and project information, saving your project, and launching features that enable you to make changes to default application settings, integrating associated applications, and/or setting up how you want your simulation to operate.

Option	Description
Info	Includes entry fields for project description and ownership (**Project**), a summary of the details of the project (**Model Summary**), as well as a history of when the project was saved (**Save History**). Also see the Project object

Option	Description
	reference section. This information can also be defined in the Details view of the **Project** object. **Note:** When you import a mesh file, the option **Import Summary** becomes available in the **Model Summary** content. When you select this option, the application goes to the **Import Summary** object in the Outline. That object presents upstream source file data.
What's New	Displays an illustrated review of the release's new features and capabilities. This display is also available from the Help (p. 141) drop-down menu option on the title bar.
Home[a]	Go to the Home page. This page displays recently opened projects as well as an option to open a new project.
New[a]	Begin a new session of Mechanical.
Open[a]	Open an existing project.
Save[a]**/Save Project**	Saves your project.
Save Project As[a]**/Save Project As**	Saves the current project under a different name and/or location. You are prompted to specify the name and location for the file.
Archive[a]**/Archive Project**	Generates a single archive file that contains project files. During the archive process, the application prompts you with the following dialog to make optional selections.  

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
	These options enable you to control whether the archive includes certain data. This can be helpful if you have file size concerns. Supported file types include Mechanical (`.mechpz`), Workbench Project Archive (`.wbpz`) or Zip (`.zip`/`.tar.gz`). You can also perform this action in Workbench. See the Archiving Projects section for more information.
Save Database	Enables you to save the current Mechanical session without having to save the entire project. However, you must save the project when you exit the application to properly save your changes.
Refresh All Data	Updates the geometry, materials, and any imported loads that are in the tree.
Clear Generated Data	Clears all results and meshing data from the database depending on the object selected in the tree. This option is available via the right-click context menu on many objects.
Import	Available when you open Mechanical without a geometry or mesh. Selecting **Import** displays two additional options: **Geometry** and **Mesh (External Model)** (p. 908). These options enable you to import a geometry or a mesh file. Select **Geometry** or **Mesh (External Model)** and then select from the **Recent** list or select **Browse** to open a file. Using the **Mesh (External Model)** option automatically inserts and links a corresponding system to the appropriate cells (**Engineering Data** and **Model**) of the existing system. **Note:** Linux platform does not support the: • **Import** option. • Link between the **External Model** system and the **Engineering Data** cell.
Export	Exports your project. You can export a `.mechdat` file (when running the Mechanical application) that later can be imported into a new Workbench project. Note that only the data native to the Mechanical application is saved to the `.mechdat` file. External files (such as solver files) will not be exported. You can also export the mesh for input to any of the following: Fluent (`.msh`), Polyflow (`.poly`), CGNS (`.cgns`), and ICEM CFD (`.prj`).
Addins	Launches the **Addins** dialog that enables you to load/unload third-party add-ins that are specifically designed for integration within the Workbench environment.
Options	Opens the **Options** (p. 217) dialog. This dialog enables you to customize the application and to control the behavior of Mechanical application functions. This option is also available on the title bar of the application, beside the **Quick Launch** feature.
Solve Process Settings	Displays the Solve Process Settings (p. 1804) dialog to configure your solution process.

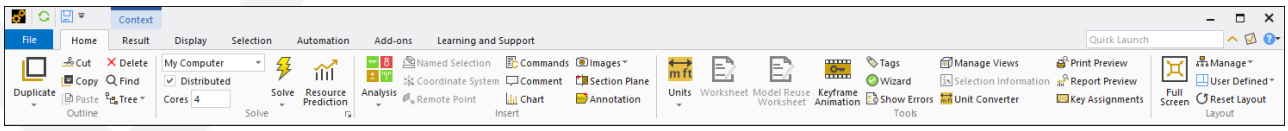
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Variable Manager	Opens the **Variable Manager** (p. 247) dialog. This dialog enables you to enter an application variables that can override default settings.
Licensing	Displays the **License Options** pane. This pane displays a list of all the licenses available to you as a user. Mechanical uses the first relevant license in the list. You can change the order using the **Up**/**Down** and **Save** options. You can also use the **Disable** option to exclude a potential license from your current and future Mechanical sessions. The application checks out a license for a session based on these preferences. For any subsequent license requests, the application refers to the preferences to fulfill the request. If any other license is available, individual or shared, the application uses that license. As indicated by a displayed message, it is necessary to close and reopen Mechanical for licensing changes to take effect. **Shared Licenses** Workbench controls shared licenses. Using shared licensing, the active application holds the license, preventing other applications that are sharing that license from using it during that time. The application or operation requiring use of the license is called a concurrency event. For example, meshing and solving would each be a concurrency event. Single license sharing applies only to licenses of the same type (for example, Mechanical Enterprise). If a license is not available/checked out when you open Mechanical, the application automatically opens in **Read-Only Configuration** mode (as displayed on the Status (p. 135) bar). Under the **License Options**, the **Activate License** button displays. This option is only available when no license is checked out and the application is in **Read-Only Configuration** mode. From the **License Options** pane select a license and click the activate button. This removes the application from read-only mode. Review the material in the Ansys Workbench Licensing Methods section of the Workbench User's Guide for additional information.
Close Mechanical	Exits your current Mechanical session.

[a] Mechanical opened independently.

Home Tab

The **Home** tab displays by default when you open the application.



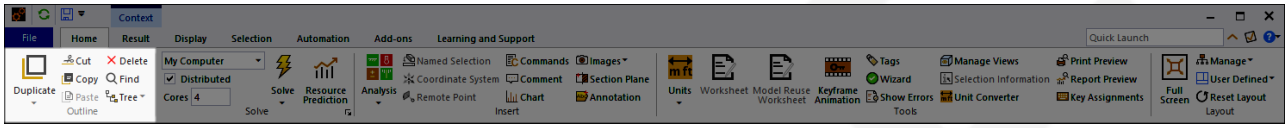
This tab contains the following **Groups**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Outline** (p. 41)
- **Solve** (p. 41)
- **Insert** (p. 42)
- **Tools** (p. 45)
- **Layout** (p. 47)

Outline

The **Outline** group provides options that enable you to make basic changes to Outline pane objects.



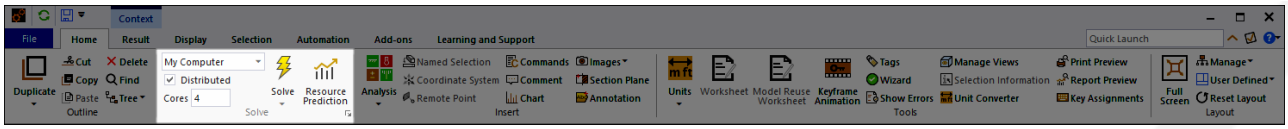
Options for this group include:

Option	Description
Duplicate and **Duplicate Without Results**	Duplicates a selected **Outline** object. This option is only available if an object supports being duplicated. A drop-down menu is also available from this option. Once you have solved your analysis, the additional option **Duplicate Without Results** becomes available in the drop-down. This option is only available when you select a result object. It duplicates your selected result object, including all subordinate objects. This is a faster option than duplicating a result that includes result data.
Cut/**Copy**/**Paste**	Cuts, copies, or pastes **Outline** objects.
Delete	Deletes a selected **Outline** object.
Find	Displays the **Find in Tree** (p. 119) dialog that enables you to search Outline objects, such as the name of an object or objects or a string of characters that are included in the name of objects.
Tree	Displays a drop-down menu of the following options: • **Expand All**: Expands all **Outline** objects • **Collapse All**: Collapses all **Outline** objects • **Collapse Environments**: Collapses only the Environment objects.

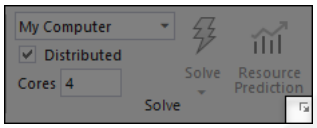
Solve

The **Solve** group provides options that enable you to specify some basic solution configurations and to solve your analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Options for this group include:

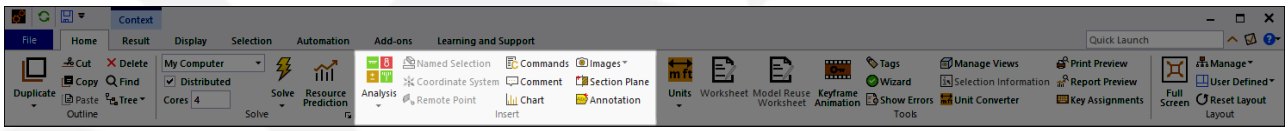
Option	Description
Solve	Displays a drop-down menu for **My Computer** and **My Computer, Background** solution selections. Initiates the solution when selected.
My Computer / **My Computer, Background**	Provides a drop-down menu to specify your desired selection only.
Resource Prediction (p. 170)	Opens a window that enables you to select an available analysis and then generate an estimate of the computing resources needed to perform a solution for the environment.
Solve Process Settings (p. 1804)	 Launches the **Solve Process Settings** (p. 1804) dialog. This dialog enables you to configure solution settings.

Note:

The **Solve** drop-down menu and dialog options are also available on a number of Context tabs (Environment, Solution, etc.).

Insert

The **Insert** group provides a variety of regularly used options.



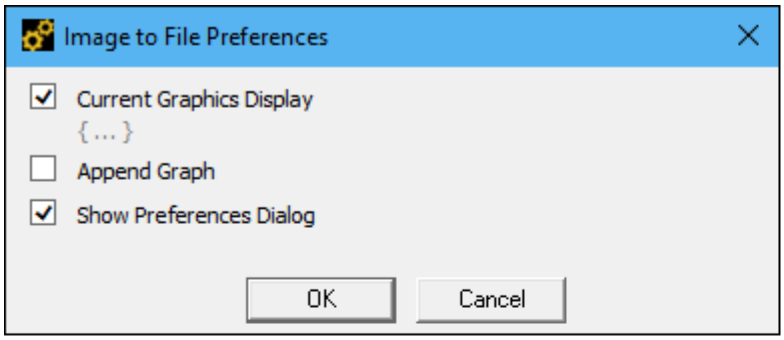
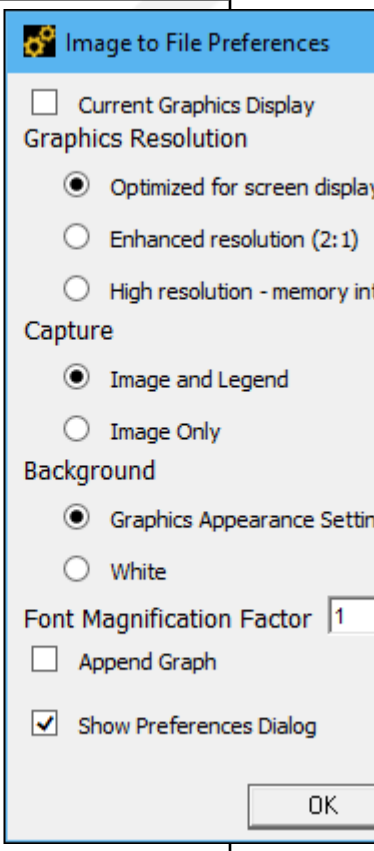
Options for this group include:

Option	Description
Analysis	Displays a drop-down menu of standalone analysis selections to add to your existing model. A corresponding analysis system, with the appropriate connections, is also included in the **Project Schematic**. The new analysis shares the **Engineering Data**, **Geometry**, and **Model** cells with the other analysis systems under the model.
Named Selection	For a supported parent object, inserts a Named Selection, and parent folder as needed, into the Outline.
Coordinate System	This option is available when the **Coordinate Systems** object is selected. It inserts a new **Coordinate System** object.

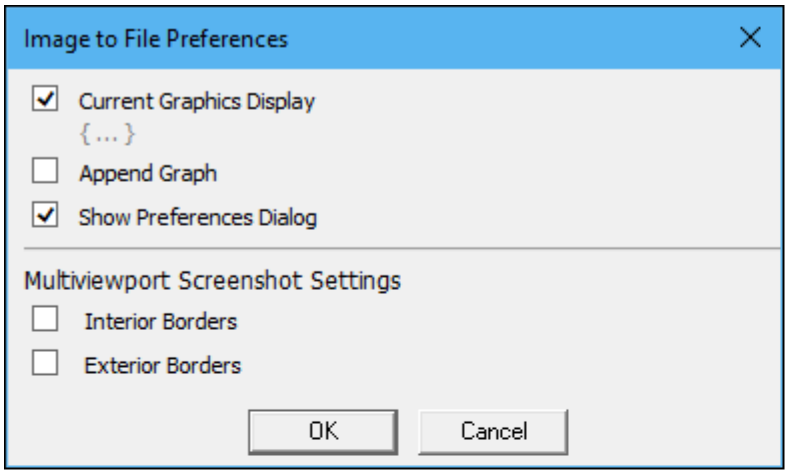
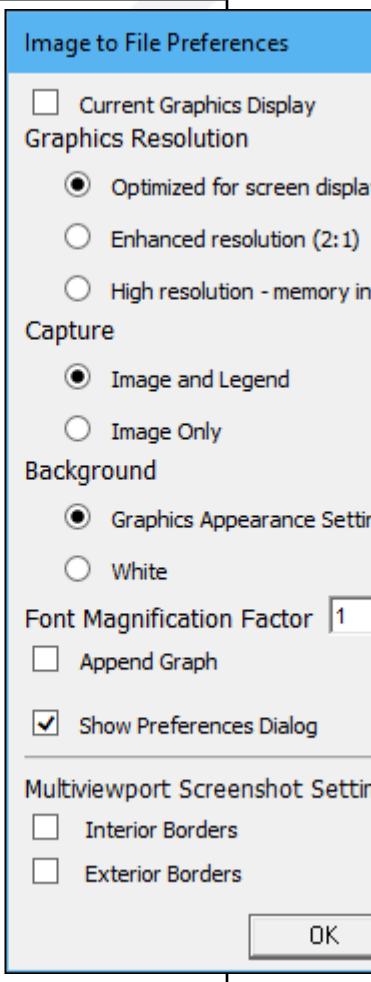
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Remote Point	This option is available when the **Model** object is selected. It inserts a new Remote Point object and parent folder as needed.
Commands	For a supported parent object, inserts and specifies a new Commands object.
Comment	For a supported parent object, inserts and specifies a new Comment object.
Chart	Inserts and specifies a new Chart object.
Images	Displays a drop-down menu of the following options: • **Figure**: Captures the current **Geometry** window content and places it under the currently selected object. You can manipulate **Figure** objects in the **Geometry** window as well as use other options on the object, such as adding an Annotation. • **Image**: Captures a two-dimensional screen shot of the **Geometry** window content and places it under the currently selected object. • **Image from File**: Imports an existing image and places it under the currently selected object. • **Image Plane from File**: Imports an image into Mechanical and places it on or around your model based on the XY-plane of the selected coordinate system. This option is used primarily with the Construction Line (p. 895) feature to overlay and accurately sketch one or more line segments. However, it has many applications. For example, you can place logos on your model, specify a background, set up physical rulers to help measure deformations, incorporate image planes into custom ACT extensions, etc. • **Image to File**: Saves an image of the active viewport of the **Geometry** window. When you select this option a dialog displays, as illustrated below. By default, the option **Current Graphics Display** is active. With this option selected, the application captures the content of the **Geometry** window using the application default settings. In order to make any changes on the dialog, you must first deselect the **Current Graphics Display** option. The expanded dialog, illustrated below, provides graphical resolution and image capture options that you can modify. If you change the settings, the application saves your selections for future use of the feature. In addition, the dialog includes the **Append Graph** option that enables you to add a graph of the **Graph** window data in the lower left-hand corner of the image.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

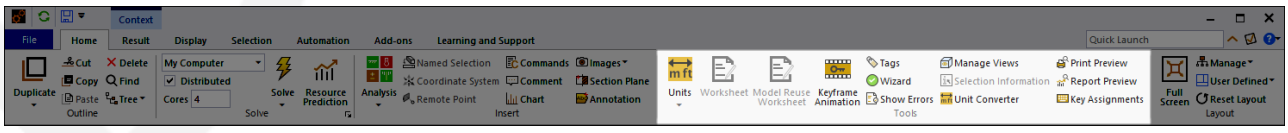
Option	Description
	  Deselecting the **Show Preference Dialog** instructs the application to not show the above dialog and immediately display the **Save As** window. Supported file formats for this feature include: PNG (.png), JPEG (.jpg), and BMP (.bmp). **Note:** You can change the default settings for these options using the properties of the Image Export (p. 232) group of the **Graphics** category of the **Options** (p. 217) dialog. • **Composite Image to File**: This option is available when you have specified more than one viewport (p. 284). This option acts the same as the **Image to File** option. It includes the same options as the above dialog, with additional options that are specific to saving viewport images, including the options **Interior Borders** and **Exterior Borders** under the **Multiviewport Screenshot Settings** title, as illustrated below. These options enable you to add borders around and in-between the viewports you capture.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
	  • **Image to Clipboard**: Copies **Geometry** window content to the clipboard. The image may then be pasted into different applications. This feature is for the Windows platform only.
Section Plane	Displays the **Section Planes** window to specify a section cut-through on your model in order to view a cross section of your geometry, mesh, or of a result. See the Creating Section Planes (p. 294) section for additional information about this feature.
Annotation	Adds a text comment to a particular spot of your model. See the Graphics Annotations Window (p. 202) section for additional information about this feature.

Tools

The **Tools** group provides a variety of display-based options.



Options for this group include:

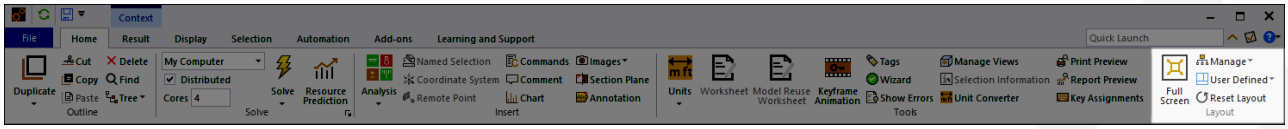
Option	Description
Units	Displays the Unit Systems drop-down menu. Modify unit system as desired. **Note:** The **Celsius** and **Kelvin** settings are not available if you select either of the U.S. Customary settings.
	For a supported objects, displays (or hides) the **Worksheet** (p. 192) window.
Model Reuse Worksheet	Displays (or hides) the **Model Reuse Worksheet** pane. This pane enables you to specify the use of boundary conditions across multiple analysis environments (p. 1787).
Keyframe Animation	Displays the **Keyframe Animation** (p. 255) window.
Tags	Displays the **Tags** window to apply meaningful labels to objects that can then be filtered. See the Tagging Objects (p. 2349) section for additional information about this feature.
Wizard	Activates the Mechanical Wizard (p. 321). This feature helps you construct your simulation.
Show Errors	Displays error messages, in the **Messages** window, for invalid **Outline** pane objects. Once you correct an invalid object, selecting this option removes the error message from the window. **Note:** Even though the application does not check for solution and mesh related errors with this feature, selecting the option may also clear those errors from the **Messages** window.
Manage Views	Displays the **Manage Views** (p. 292) window. This feature enables you to save a graphical view of your model.
Selection Information	Displays the **Selection Information** (p. 180) window.
Unit Converter	Displays a Unit Conversion tool. It is a built-in conversion calculator that enables you to perform conversions between consistent unit systems (p. 1874). The Units menu sets the active unit system. The status bar shows the current unit system. The units listed in the tool and in the Details pane are in the proper form (include no parenthesis).
Print Preview	Displays a printable image of the currently selected object. See the **Print Preview** (p. 173) section for more information about this feature.
Report Preview	Displays your analysis in the Report Preview view. See the **Report Preview** (p. 174) section for more information about this feature.
Key Assignments	Displays a dialog that lists all available hotkey and hotkey combinations that enable you to quickly perform certain actions. See the Key Assignments (p. 314) section for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Layout

The **Layout** group provides options to manage the display of the interface.



Options for this group include:

Option	Description
Full Screen	Activates a full screen display (p. 178). This display can also be turned on and off using the **F11** key.
Manage	Provides a drop-down menu of interface display selections.
User Defined	Displays a drop-down menu of the following options: • **Store Layout**: Saves an interface layout that you have created. For example, you may like to size the interface windows in a specific way or you like to display certain interface windows, such as Section Planes, or you may wish to hide certain interface windows. Once you have designed/configured an interface layout, you select the **Store Layout** option and then enter a name for the layout. This name then displays in the drop-down menu enabling you to select it and any time. • **Remove Layout**: Becomes available once you have saved a layout. Selecting this option displays a small dialog that you use to delete existing layouts.
Reset Layout	Restores the interface layout to the default setting.

Context Tabs

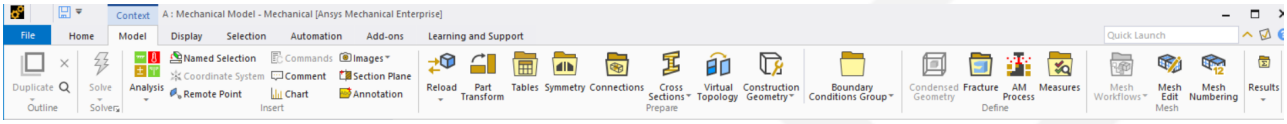
The ribbon contains a **Context** tab for most objects. The Context tabs provides relevant options based on the selected object. Primary Context tabs include:

- Model Context Tab (p. 48)
- Geometry Imports (p. 55)
- Geometry Context Tab (p. 56)
- Materials Context Tab (p. 57)
- Tables Context Tab (p. 57)
- Cross Section Context Tab (p. 58)
- Coordinate Systems Context Tab (p. 58)
- Connections Context Tab (p. 50)
- Mesh Context Tab (p. 58)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Environment Context Tab (p. 61)
- Environment Context Tab Display Group for Variable Data (p. 61)
- Solution Context Tab (p. 62)
- Solution Information Tab (p. 62)
- Result Context Tab (p. 62)

Model Context Tab



The **Model** Context tab becomes active when the **Model** object is selected in the **Outline**. The **Model** Context tab contains options for creating objects related to the model, as described below.

Option	Description
Import Geometry/Reload	The import option for geometry is labeled **Import Geometry** if no primary geometry has been imported and **Reload** if a geometry has been imported. Therefore, the **Reload** option imports secondary geometries into your analysis. See the Geometry Imports (p. 55) description below.
Tables	Inserts a Tables object that houses all of the Table objects that you create.
Part Transform	Inserts a Geometry Transforms folder object that houses all of the part transformations (p. 1014) (via Part Transform objects) that you create.
Symmetry	Inserts a **Symmetry** object. For symmetric (p. 1127) bodies, you can remove the redundant portions based on the inherent symmetry, and replace them with symmetry planes. Boundary conditions are automatically included based on the type of analyses. Also see the Symmetry Context Tab (p. 50) topic below.
Connections	Becomes available only if a Connections object is **not** already included in the Outline (such as a model that is not an assembly), and you wish to create a connections object. See the Connections Context Tab (p. 50) topic below. Connection objects include contact regions, joints, and springs. You can transfer structural loads and heat flows across the contact boundaries and "connect" the various parts. See the Contact (p. 1207) section for details. A joint typically serves as a junction where bodies are joined together. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free. See the Joints (p. 1292) section for details. You can define a spring (longitudinal or torsional) to connect two bodies together or to connect a body to ground. See the Springs (p. 1395) section for details.
Cross Sections	Inserts a desired cross section type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

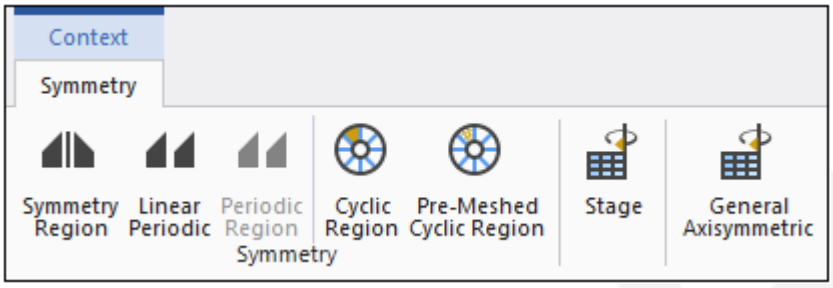
Published: 2023-10-23

Option	Description
Virtual Topology	You can use the **Virtual Topology** option to reduce the number of elements in a model by merging faces and lines. This is particularly helpful when small faces and lines are involved. The merging will affect meshing and selection for loads and supports. See Virtual Topology Context Tab (p. 51) below as well as the Virtual Topology Overview (p. 349) section for additional details.
Construction Geometry	See the Specifying Construction Geometry (p. 883) section for additional details.
Boundary Conditions Group	Define boundary conditions to be used across multiple analysis systems included in your simulation. See the Specifying Model-Level Boundary Conditions (p. 1787) section for more information.
Condensed Geometry	Inserts a **Condensed Geometry** object. See the Condensed Geometry Context Tab (p. 52) topic below as well as the Substructure Analysis (p. 781) section for additional information.
Fracture	Inserts a **Fracture** object. See the **Fracture Context Tab** (p. 52) topic below as well as the Performing a Fracture Analysis (p. 406) section for additional information.
AM Process	Inserts an AM Process object. By default, it is inserted along with the child object Build Settings. You use this object when you are performing an additive manufacturing simulation. Also see the AM Process Context Tab (p. 53) topic below.
Measures	Inserts a Measures object. See the Measures Context Tab (p. 53) topic below.
Mesh Workflows	Inserts a **Mesh Workflows** object. Also, see the Mesh Workflows Context (p. 53) topic below.
Mesh Edit	Inserts a **Mesh Edit** object. Also see the Mesh Edit Context (p. 54) topic below.
Mesh Numbering	Enables you to renumber the node and element numbers of a generated meshed model consisting of flexible parts. See the Specifying Mesh Numbering (p. 1035) section for details.
Solution Combination	Combines multiple environments and solutions to form a new solution. A solution combination folder can be used to linearly combine the results from an arbitrary number of load cases (environments). Note that the analysis environments must be static structural with no solution convergence. Results such as stress, elastic strain, displacement, contact, and fatigue may be requested. To add a load case to the solution combination folder, right-click the worksheet view of the solution combination folder, choose add, and then select the scale factor and the environment name. An environment may be added more than once and its effects will be cumulative. You may suppress the effect of a load case by using the check box in the worksheet view or by deleting it through a right-click. For more information, see Solution Combination (p. 2288).
Fatigue Combination	Inserts a Fatigue Combination object. When you are running an analysis that includes multiple systems that each include a Fatigue Tool object, the **Fatigue Combination** feature enables you to sum (generate a sum total of) the Damage results for all of the linked systems. This option only supports all analysis types that support the use of the **Fatigue Tool**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

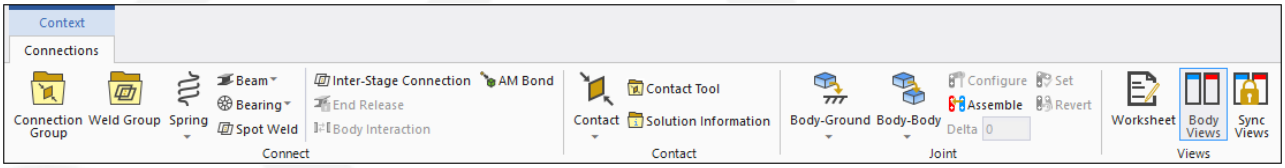
Option	Description
Ply	When you select a ply object, the **Ply** group displays and contains the **Direction** drop-down menu. The options of the menu enable you to graphically display ply and element directions for imported ply structures. • **Fiber**: Ply Fiber Direction. • **Transverse**: Ply Transverse Direction. • **Normal**: Ply Normal Direction. • **Element Reference**: Element Reference Direction. • **Element Normal**: Element Normal Direction.

Symmetry Context Tab



Based on your analysis type, the **Symmetry** (p. 1127) Context tab includes options to insert **Symmetry Region** (including **Linear Periodic**), **Periodic Region**, **Cyclic Region**, **Pre-Meshed Cyclic Region**, **Stage**, and **General Axisymmetric** objects.

Connections Context Tab



The **Connections** Context tab includes the following options and functions:

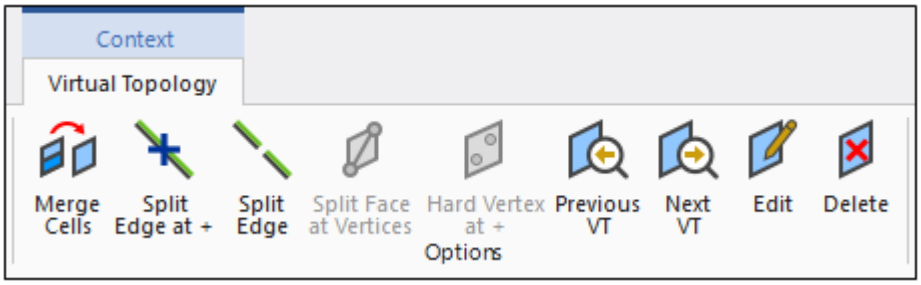
Option	Description
Connection Group	Inserts a **Connection Group** (p. 1212) object.
Spring	Inserts a **Spring** (p. 1395) object, either **Body-Ground** or **Body-Body**
Beam	Inserts a Beam object, either **Body-Ground** or **Body-Body**.
Bearing	Inserts a **Bearing** (p. 1413) object, either **Body-Ground** or **Body-Body**.
Spot Weld	Inserts a **Spot Weld** (p. 1407) object.
Interstage	Inserts an Interstage object..
End Release	Inserts an **End Release** (p. 1409) object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	Description
Body Interactions	See the Body Interactions in Explicit Dynamics Analyses section for additional information.
AM Bond	Inserts an AM Bond object.
Contact	Inserts a specific type of Contact Region (p. 1234).
Contact Tool	Inserts a **Contact Tool** (p. 2079) object.
Solution Information	Insert a **Solution Information** (p. 1833) object.
Body-Ground	Enables you to insert and specify a certain type of Body-to-Ground Joint (p. 1298) object.
Body-Body	Enables you to insert and specify a certain type of Body-to-Body Joint (p. 1298) object.
Configure, **Set**, and **Revert** options and **Delta** field	Graphically configure the initial positioning of a joint. See the Example: Configuring Joints (p. 1380) example. The **Assemble** option performs the assembly of the model, finding the closest part configuration that satisfies all the joints. **Important:** When a model contains a **Point On Curve** (p. 1312) joint, the **Configure** and **Assemble** options (p. 51) are disabled for all the joints. This is also the case for a redundancy analysis that includes a **Point On Curve** joint.
Body Views	Toggles the display of parts and connections in separate auxiliary windows for contact regions, beams, bearings, joints, and spring connections.
Sync Views	When the **Body Views** option is selected, you can select this option synchronize the movements of your model in the **Geometry** window with the views of the auxiliary windows. and vice versa.

Virtual Topology Context Tab



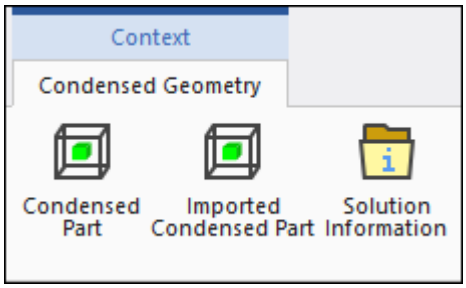
The Virtual Topology Context tab includes the following options:

Option	Description
Merge Cells	Creates Virtual Cell objects you can use to group faces or edges.
Split Edge at + and **Split Edge**	Create Virtual Split Edge objects that enable you to split an edge to create two virtual edges.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

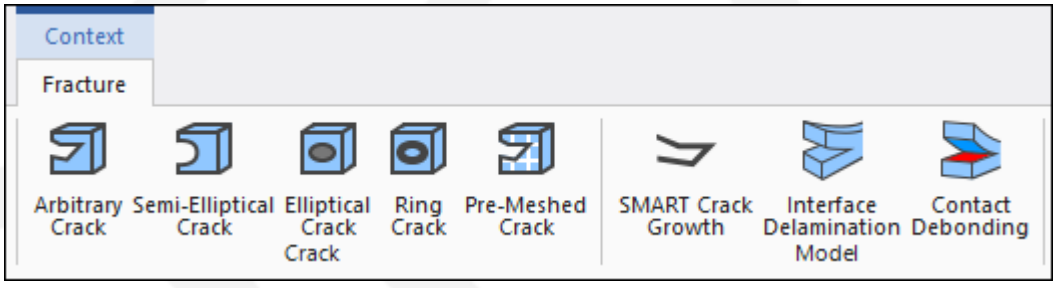
Option	Description
Split Face at Vertices	Creates Virtual Split Face objects to split a face along two vertices to create 1 to N virtual faces. The selected vertices must be located on the face that you want to split.
Hard Vertex at +	Creates Virtual Hard Vertex objects to define a hard point according to your cursor location on a face, and then use that hard point in a split face operation.
Previous VT/Next VT	Enable you to cycle through virtual topology entities in the sequence in which they were created. If any virtual topologies are deleted or merged, the sequence is adjusted automatically. See Cycling Through Virtual Entities in the Geometry Window.
Edit	Edits virtual topology entities.
Delete	Deletes selected virtual topology entities, along with any dependents if applicable.

Condensed Geometry Context Tab



The **Condensed Geometry** Context tab enables you to apply the objects associated with substructuring (p. 781), including the Condensed Part object, Imported Condensed Part, as well as a Solution Information (p. 1833) object.

Fracture Context Tab

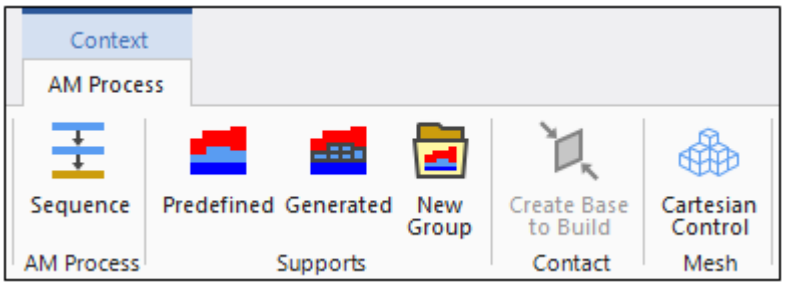


The **Fracture** Context tab enables you to insert the objects associated with a Fracture Analysis (p. 406), including different **Crack** (p. 424) types, SMART Crack Growth (p. 468), as well as progressive failure features (p. 476) in the form of Interface Delamination and Contact Debonding objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

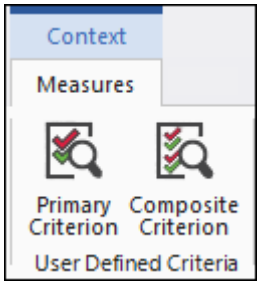
Published: 2023-10-23

AM Process Context Tab



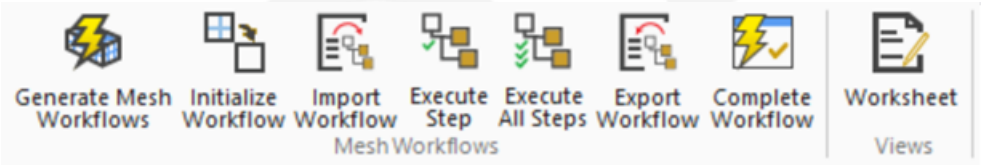
This tab displays when you insert an AM Process object into the **Outline**.

Measures Context Tab



This tab displays when you insert a **Measures** object into the **Outline**. You use the this object to insert User Defined Criteria (p. 2243).

Mesh Workflows Context Tab



The **Mesh Workflows Context** tab helps you to:

- Generate workflows from predefined steps that can be customized
- Initialize, import, execute, and export the workflows
- Visualize the workflow properties using the worksheet

The **Mesh Workflows** Context tab includes the following options:

Option	Description
Generate Mesh Workflows	Generate mesh workflows.
Initialize Workflow	Initialize mesh workflow, transferring selected input geometry to the mesh workflow operations. **Named Selections** scoped to the input geometry translate into corresponding **Labels** after initializing the workflow.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Import Workflow	Import workflow steps from a .emx file. **Import workflow** is available after deleting steps and clearing data from the current workflow.
Execute Step	Initialize the workflow and execute the next step of the specified mesh workflow.
Execute All Steps	Initialize the workflow and execute all steps of the specified mesh workflow.
Export Workflows	Export the workflow steps as a template to a .emx file.
Complete Workflow	Complete the workflow by transferring generated data back to new mechanical geometry parts along with the corresponding part meshes. You cannot edit the specified mesh workflow type after completing the workflow.
Worksheet	Provides a view of the published mesh workflow operations, controls, and its properties.

Mesh Edit Context Tab

The **Mesh Edit** Context tab enables you to modify and create Mesh Connection objects that enable you to join the meshes of topologically disconnected surface bodies and also move individual nodes on the mesh. The **Mesh edit** Context tab includes the following options:

Option	Description
Mesh Connection Group	Inserts a Mesh Connection Group folder object.
Manual Mesh Connection	Inserts a **Mesh Connection Group** folder that includes a Mesh Connection object.
Contact Match Group	Inserts a Contact Match Group folder object.
Contact Match	Inserts a Contact Match folder object.
Node Merge Group	Inserts a Node Merge Group folder object.
Node Merge	Selects geometries and merge coincident mesh nodes.
Node Move	Selects and move individual nodes on the mesh. Requires mesh generation.
Pull	Creates a mesh volume extrusion.
Body Views	Only visible when Mesh Connection object selected. Toggle button to display parts in separate auxiliary windows.
Sync Views	Only visible when Mesh Connection object selected. Toggle button that you can use when the **Body Views** button is engaged. Any change to the model in the **Geometry** window is reflected in both auxiliary windows.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

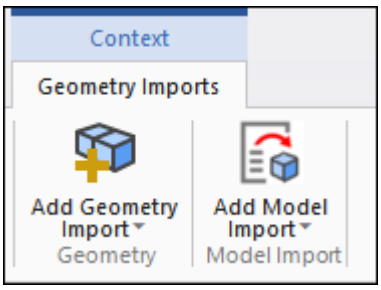
Published: 2023-10-23

Geometry Imports

The options of the **Geometry Imports** Context tab depend upon how you have open Mechanical as well as whether you have imported geometry.

Mechanical Opened Independently

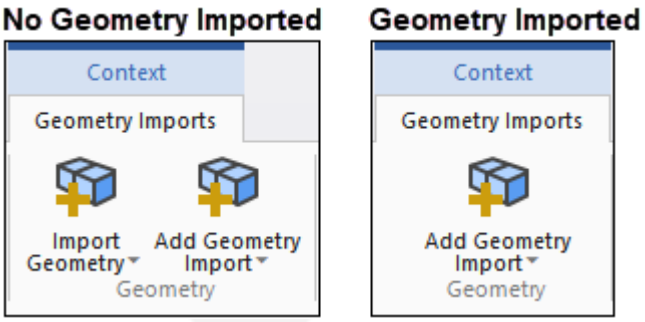
The following Context tab displays when you open Mechanical independently enabling you to open geometry or an external model file.



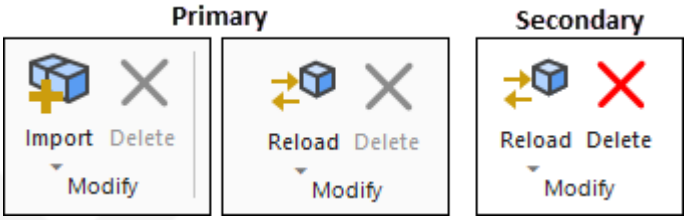
Both options include drop-down menus that provide a **From File** option to import files from a file location as well as a list of recently imported files that you can choose from.

Mechanical Opened from Workbench

As illustrated, if you have not imported a geometry, the option **Import Geometry** enables you to do so. The **Add Geometry Import** option imports secondary geometries into your analysis. All options include drop-down menus that provide a **From File** option to import models from a file location as well as a list of recently imported geometries that you can choose from.

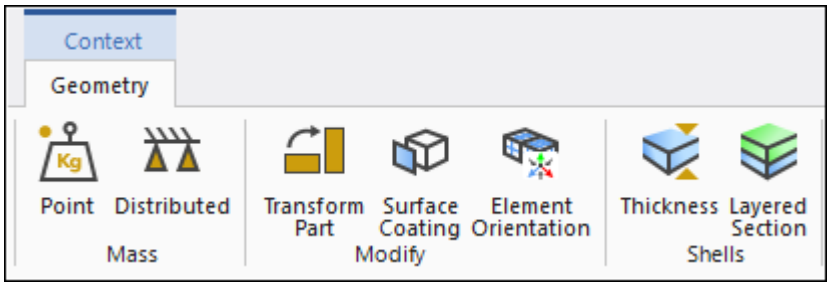


Similarly, when you select a child object (**Geometry Import** primary/secondary), the options of the primary object depend upon whether you have imported a geometry. For the secondary object, you use the **Reload** option to change your geometry. Or you can use the **Delete** option to remove the geometry from your analysis.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Geometry Context Tab



The **Geometry** context tab is active when you select the **Geometry** object in the Outline or any child objects (part/body) of the **Geometry** object. The tab includes the following options:

Option	Description
Modify Geometry (not illustrated)	For electronic computer-aided design (ECAD) models, displays the ECAD Import (p. 703) pane.
Point	Specifies a **Point Mass** (p. 901).
Distributed	Specifies a **Distributed Mass** (p. 902).
Transform Part	Becomes visible when you have a Part or Body selected. For the selected Part or Body, this option automatically scopes the geometry to a Part Transformation object.
Delete Part(s) (not illustrated)	Becomes visible when you have a Part or Body selected, enabling you to delete the selected part or body. Multiple part or body selection is supported.
Surface Coating	Specifies a **Surface Coating** (p. 2008).
Element Orientation	Specifies **Element Orientations** (p. 1008).
Imported Element Orientation	Available when you open Mechanical independently. Inserts an **Imported Element Orientation** folder object, an associated **Element Orientation** object, and displays the **External Data** dialog. You use this dialog to specify the desired external file(s) you wish to import.
Thickness	For surface bodies, enables you to add a Thickness object.
Imported Thickness	Available when you open Mechanical independently. Inserts an **Imported Thickness** folder object, an associated **Imported Thickness** object, and displays the **External Data** dialog. You use this dialog to specify the desired external file(s) you wish to import.
Layered Section	For surface bodies, enables you to add a **Layered Section** (p. 868) object to define layers applied to surfaces.

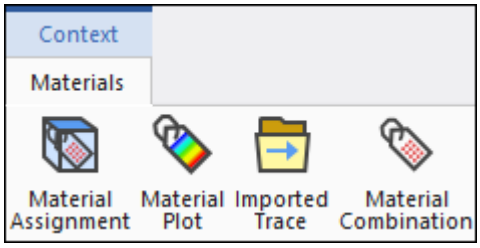
Imported Fields Context Tab

If the **Geometry** object includes an imported object, such as Imported Thickness or Imported Element Orientation, an **Imported Fields** Context menu displays when you select the imported object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Materials Context Tab



The **Materials** context tab is active when you select the Materials, **Material Assignment** (p. 1049), **Material Plot** (p. 1051), or **Material Combination** objects. Use this tab to employ the features related to the **Materials** object. Options include:

- Material Assignment (p. 1049)
- Material Plot (p. 1051)
- Material Combination (p. 1052)

Imported Fields Context Tab

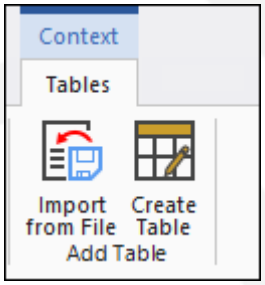
If you import Trace Mapping (p. 686) from an ECAD file, an Imported Trace group folder is placed under the Materials folder. This group folder displays the **Imported Fields** Context tab that includes the option **Trace**.

Imported Material Fields

If you import initial user-defined Field Variable values using the **External Data** (p. 736) system, an Imported Material Fields (p. 1055) group folder is placed under the **Materials** folder.

As a result of your data import, the folder contains an Imported Material Field object. You can specify additional **Imported Material Field** objects using the option of this tab. In addition, the **Variable Data** (p. 61) tab displays when **Imported Material Field** objects are selected.

Tables Context Tab

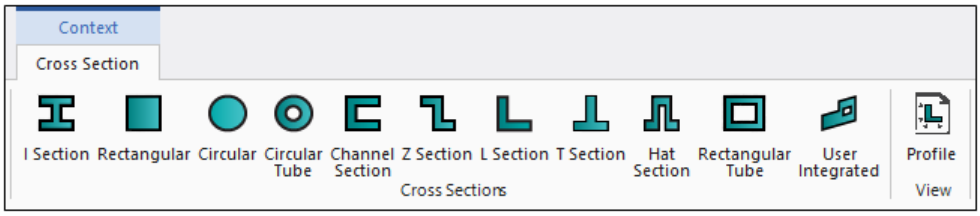


The **Tables** context tab is active when you select the Tables object. Use this tab to add tables for defining pressure, temperature, and convection loads. Options include:

- Import from File (p. 1107)
- Create Table (p. 1109)

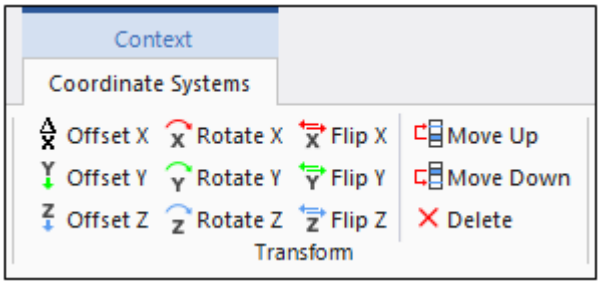
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Cross Section Context Tab



The **Cross Section** Context tab provides cross section type options that enable you to manually define a cross section for your line body model. There is also a **Profile** option that displays a window that enables you to view the cross section dimensions, during construction as well as when you are complete.

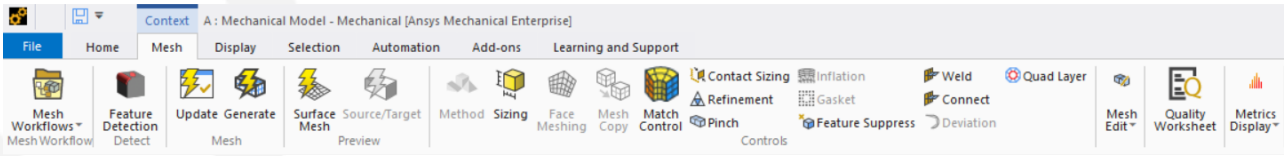
Coordinate Systems Context Tab



The **Coordinate Systems** Context tab is available when you have a user-defined Coordinate System object selected. It includes the following transformation options:

Option	Description
Offset X/Y/Z	Create an **Offset** in the **Transformations** category of the Details view. These options require to enter a value.
Rotate X/Y/Z	Create a **Rotate** transformation in the **Transformations** category of the Details view. These options require to enter a value.
Flip X/Y/Z	Create a **Flip** transformation in the **Transformations** category of the Details view. These options flip the coordinate system about a desired axis.
Move Up/Move Down	Scroll up or down through the **Transformations** category properties/transformations that you have created.
Delete	Deletes a property/transformation from the **Transformations** category.

Mesh Context Tab



The **Mesh Context Tab** includes the following options:

In the **Mesh Workflow** group, the following option is available:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	Description
Mesh Workflows	Generate **Mesh Workflows** based on predefined workflows that can be customized.

In the **Detect** group, the following option is available:

Option	Description
Feature Detection	Detects feature based on Feature Detection (p. 1033).

In the **Mesh** group, the following options are available:

Option	Description
Update	Updates a cell that references the current mesh. This includes mesh generation as well as generating any required outputs.
Generate	Generates a mesh.

In the **Preview** group, the following options are available:

Option	Description
Surface Mesh	Enables you to preview the Surface Mesh.
Source/Target	Enables you to preview the source and target meshes for scoped bodies.

In the **Controls** group, the following options are available:

Option	Description
Method	Selects Method Control.
Sizing	Selects Sizing Control.
Face Meshing	Selects Face Meshing Control.
Mapped Meshing	Selects Mapped Meshing Control.
Mesh Copy	Selects Mesh Copy Control.
Match Control	Selects Match Control.
Contact Sizing	Selects Contact Sizing Control.
Refinement	Selects Refinement Control.
Pinch	Selects Pinch Control.
Inflation	Selects Inflation Group.
Gasket	Selects Gasket Mesh Control.
Feature Suppress	Selects Feature Suppress Control.
Weld	Selects Weld Control.
Connect	Selects Connect Control.
Deviation	Selects Deviation Control.
Quad Layer	Selects Quad Layer Control.

In the **Mesh Edit** group, the following options are available:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Mesh Connection Group	Selects Mesh Connection Group.
Contact Match Group	Selects Contact Match Group.
Node Merge Group	Selects Node Merge Group.
Mesh Edit	Selects Mesh Edit.
Mesh Numbering	Selects Mesh Numbering (p. 1035).
Manual Mesh Connection	Creates manual Mesh Connections.
Contact Match	Selects Contact Match.
Node Merge	Selects geometries and merges coincident mesh nodes.
Node Move	Selects Node Move.
Pull	Creates a mesh volume extrusion.

In the **Quality Worksheet** group, the following option is available:

Option	Description
Quality Worksheet	Displays the Mesh Quality Worksheet.

In the **Metrics Display** group, the following options are available:

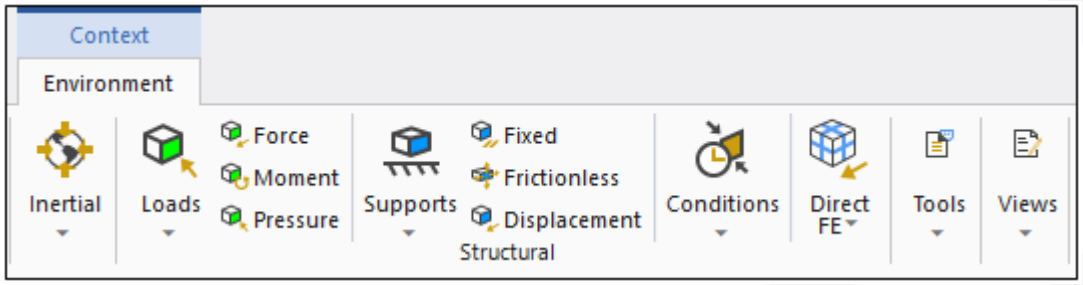
Option	Description
Metric Graph	Shows and/or hides the Mesh Metrics bar graph.
Edges	A drop-down menu options to change the display of your model, including: • **No Wireframe**: Displays a basic picture of the body. • **Show Elements**: Displays element outlines. These options are the same options that are available on the Meshing Edit Context Toolbar (p. 54).
Probe, **Max**, and **Min** (**Snap** and **Advanced**)	These are annotation display options. These options are not visible if the **Mesh** object **Display Style** property is set to the default setting, **Use Geometry Setting**. Selecting the **Max** and/or **Min** buttons displays the maximum and minimum values for mesh criteria (Element Quality, Jacobian Ratio, etc.) that you have selected. The **Probe** feature is also criteria-based. You place a **Probe** on a point on the model to display an annotation on that point. **Probe** annotations show the mesh criterion-based value at the location of the cursor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	Description
	See the Probe, Maximum, and Minimum (p. 71) topic below for more information.

Environment Context Tab



The **Environment** Context tab enables you to apply loads to your model. Tab groups and options vary depending on the type of analysis you are performing. For example, the groups and options for a **Static Structural** analysis is shown above.

Environment Context Tab Display Group for Variable Data

When your analysis includes imported boundary conditions (p. 1736) or imported thicknesses or you have specified spatial varying loads and displacements (p. 1769), the application displays contours or isoline representations of the associated variable data once you have generated the mesh on the model.

The **Display** group (shown below) becomes visible on the Environment Context Tab (p. 61) when variable data is available. The **Variable Data** drop-down menu provides the display options: **Smooth Contours**, **Contour Bands**, and **Isolines**. When you select the **Isolines** display option, the **Isoline Thickness** drop-down menu enables you to change the thickness of the displayed lines. Options include **Single** (default), **Double**, or **Triple**. The toolbar also contains options to display the **Maximum** and **Minimum** values of the imported data or spatial varying loading. You can toggle these min/max options on (default) and off.

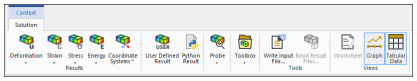
Note:

- The **Isolines** option is drawn based on nodal values. When drawing isolines for imported loads that store element values (**Imported Body Force Density**, **Imported Convection**, **Imported Heat Generation**, **Imported Heat Flux**, **Imported Pressure**, and **Imported Surface Force Density**), the program automatically calculates nodal values by averaging values of the elements to which a node is attached.
- This feature is not available for **Imported Loads** that are scoped to nodal-based **Named Selections**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you select multiple **Convection** load objects that include variable data, the application displays only one solid color for the scoped entities.

Solution Context Tab



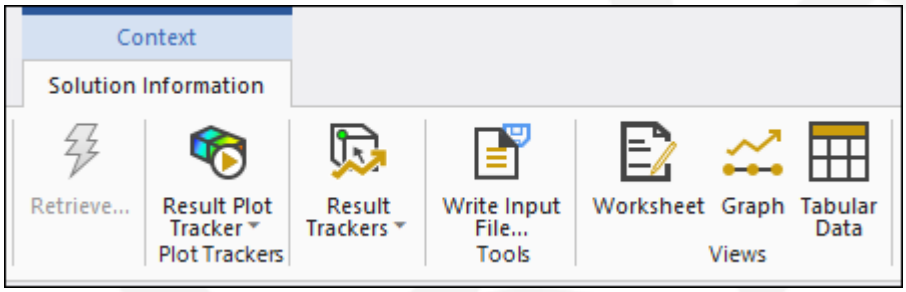
The **Solution** tab applies to Solution-level objects that either:

- Never display contoured results (such as the **Solution** object), or
- Have not yet been solved (no contours to display).

The options displayed on this tab are based on the type of analysis that is selected. The example shown above displays the solution options for a Static Structural analysis. Objects inserted using the **Solution** tab are automatically selected in the **Outline**.

The Applying Results Based on Geometry (p. 1969) section outlines which bodies can be represented by the various choices available in the drop-down menus of the Solution tab.

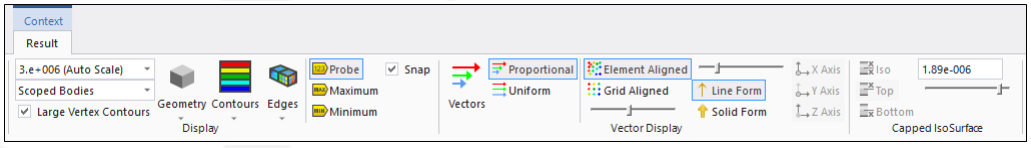
Solution Information Tab



Selecting the **Solution Information** (p. 1833) object displays a corresponding tab. The tab includes the **Retrieve** (p. 1795) option that you use to track background solution processes as well as the **Result Tracker** (p. 1844) and Result Plot Tracker options. The **Write Input File** option as well as some additional display options, **Worksheet**, **Graph**, **Tabular Data**, are also included on the tab.

Result Context Tab

The **Result** tab provides display options for your solved result objects.



The following subsections describe the options available on this tab.

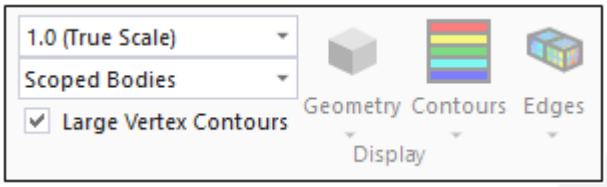
- Scaling Menus for Deformed Shapes (p. 63)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Geometry (p. 67)
- Contours (p. 67)
- Edges (p. 68)
- Probe, Maximum, and Minimum (p. 71)
- Vector Display (p. 74)
- Capped Isosurface (p. 2269)

Scaling Menus for Deformed Shapes

The **Display** group contains the following scaling options for deformations.

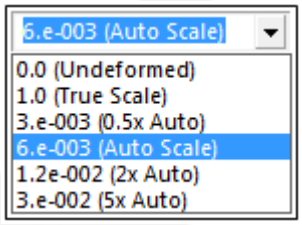


Review the following topics for additional information:

- Deformation Scale Menu (p. 63) and Relative Scaling (p. 65)
- Display Menu (p. 65)
- Large Vertex Contours (p. 67)

Deformation Scale Menu

For results with an associated deformed shape, the scaling menu provides display selections.



Scale factors precede the descriptions in parentheses in the list. The scale factors shown above apply to a particular model's deformation and are intended only as an example. Scale factors vary depending on the amount of deformation in the model. You can choose a preset option from the list, or you can enter a customized scale factor relative to the scale factors in the list.

For customized entries, the application follows a scheme where positive numbers affect the corresponding scaling differently than negative numbers. Positive numbers affect the **True Scale** factor (actual deformation) and negative numbers the **Auto Scale** factor (deformation is visible but not distorted). For example:

- A scale factor of +2.0 equates to 2 times the True Scale factor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- A scale factor of -2.0 equates to 2 times the Auto Scale factor

Available scale factors include:

- `Undeformed` does not change the shape of the part or assembly.
- `True Scale` is the actual scale.
- `Auto Scale` scales the deformation so that it's visible but not distorting.
- The remaining options provide a wide range of scaling.

The system maintains the selected option as a global setting like other options in the Result tab. As with other presentation settings, figures override the selection. For results that are not scaled, the menu selection has no effect.

Note:

Most of the time, a scale factor selected by the application to create a deformed shape that will show a visible deflection to allow you to better observe the nature of the results. However, under certain conditions, the `True Scale` displaced shape (scale factor = 1) is more appropriate and is therefore the default if any of the following conditions are true:

- Rigid bodies exist.
- A user-defined spring exists in the model.
- Large deflection is on.

This applies to all analyses except for Modal and Eigenvalue Buckling analyses (in which case `True Scale` has no meaning).

Important:

Scaling of Rigid Body Part Displacements in Modal or Eigenvalue Buckling Analyses

Note the following restrictions that apply when scaling rigid part displacements during Modal or Eigenvalue Buckling analyses.

- (Currently) If you are performing a Modal or Eigenvalue Buckling analysis that includes rigid body parts, the application experiences a limitation while scaling and/or animating results.
- The motion of rigid parts in Mechanical is characterized by the changes in the:
  - Position of the center of mass, referred to as linear displacement.
  - Euler angles of the element coordinate system, referred to as angular displacement.

  Because of the difference in the nature of these concepts, a unified scaling algorithm that satisfies both scenarios has not yet been implemented for auto

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

scaling. With the `Auto Scale` option, Mechanical displays rigid parts as white asterisks at the centroid of the part. The application maintains the correct position of the rigid parts with respect to the flexible parts, however, the displayed asterisks do not indicate angular displacement or rotation.

- `True Scale` will not properly display the shapes in Modal or Buckling analysis and should not be used.
- For the best scaling results when working on a Modal analysis (where displacements are not true), use the `Auto Scale` option. If a given body's optimal scaling is `True` and another body's optimal scaling is `Auto Scale`, the graphical display of the motion of the bodies may not be optimal.

Important:

For the following analyses and/or configuration conditions, Mechanical sets the scale factor to zero so that the image of the finite element model does not deform.

- Random Vibration (PSD).
- Response Spectrum.
- Amplitude results for Harmonic Response analyses.
- When the By property of a result is set to:
  - **Maximum Over Time** or **Minimum Over Time**
  - **Time of Maximum** or **Time of Minimum**

Relative Scaling

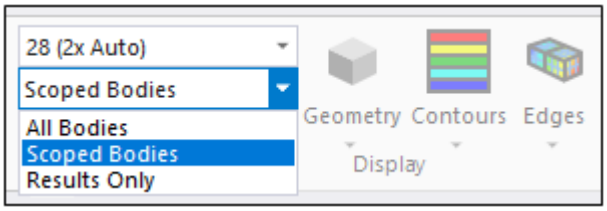
The menu provides the following "relative" scaling options. These options automatically scale deformations relative to preset criteria.

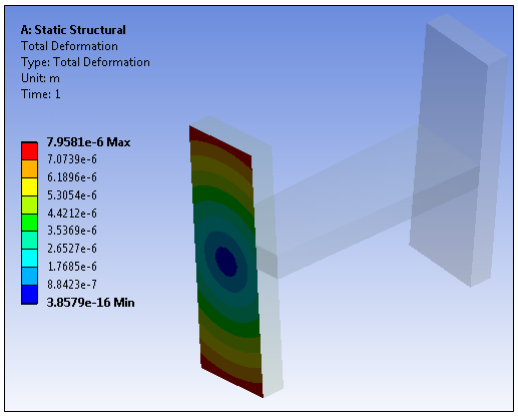
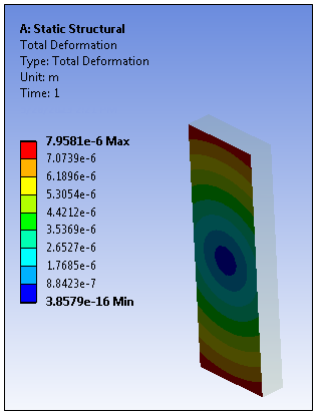
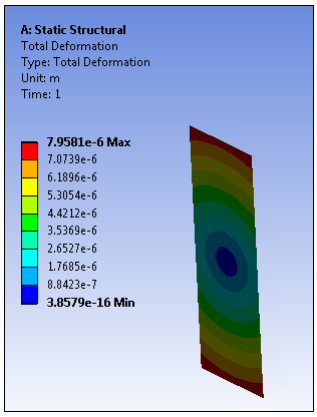
- **Undeformed**
- **True Scale**
- **0.5x Auto**
- **Auto Scale**
- **2x Auto**
- **5x Auto**

Display Menu

The **Display** drop-down menu enables you to view:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Option	Description	Example
All Bodies	Regions of the model not being drawn as a contour are plotted as translucent even for unscoped bodies as long as the bodies are *visible* (not hidden (p. 857))	
Scoped Bodies	Default setting. Regions of the model not being drawn as a contour are plotted as translucent for scoped bodies only. Unscoped bodies are not drawn.	
Results Only	Only the resultant contour or vector is displayed.	

Limitations

Note the following limitations for the display selections:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The **Scoped Bodies** and **Results Only** options support geometry-based scoping (**Geometry Selection** property = **Geometry**) and **Named Selections** that are based on geometry selections or worksheet criteria.
- The **Scoped Bodies** and **Results Only** options do not support Construction Geometry features Path (p. 883) and Surface (p. 890).
- The **Results Only** option does not support the Explicit Dynamics Solver.
- For the **Scoped Bodies** option for results that are scoped across multiple entities (vertices, edges, faces, or volumes), all of these entities may not display because there are times when only the nodes of one of the shared entities are used in the calculation.

Large Vertex Contours

The **Large Vertex Contours** check box option is used for node-based result scoping. It toggles the size of the displayed dots that represent the results at the underlying mesh nodes on and off.

Geometry

You can observe different views from the **Geometry** drop-down menu, including:

Option	Description
Exterior	Displays the exterior results of the selected geometry.
IsoSurfaces	For contour results, displays a collection of surfaces of equal value of the chosen result, between its minimum and a maximum as defined by the legend settings. The application displays the interior of the model only.
Capped IsoSurfaces	Represents mainly a set of all points that equal a specified result value within the range of values for the result with additional features. This option provides three display selections. A display based on all points of a specified result, all points equal to and less than the specified result, and all points equal to and greater than the specified result value. Refer to Capped Isosurfaces (p. 2269) section for a description of the controls with this option. This view displays contours on the interior and exterior.
Section Planes	Displays planes cutting through the result geometry; only previously drawn Section Planes (p. 294) are visible.

Contours

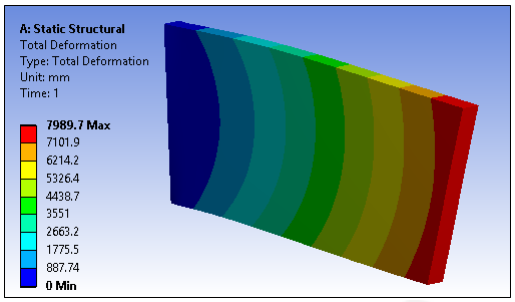
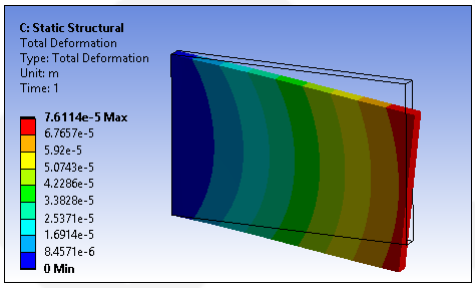
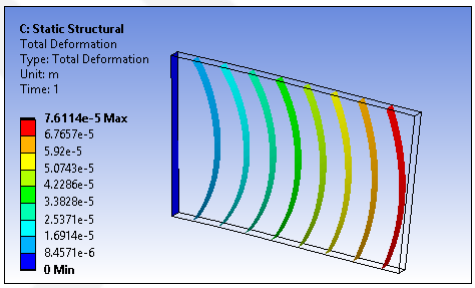
The **Contours** drop-down menu enables you to change the way you view your results. Options include:

Option	Description
Smooth Contours	Displays gradual distinction of colors.
Contour Bands	Displays the distinct differentiation of colors.
Isolines	Displays a line at the transition between values.
Solid Fill	Displays the model only with no contour markings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

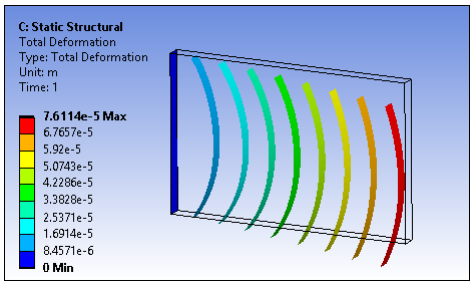
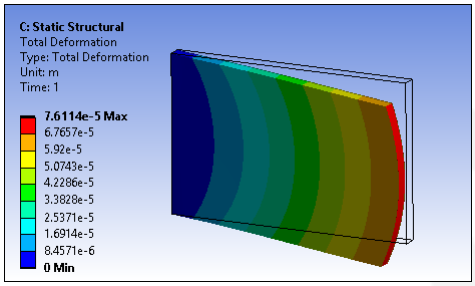
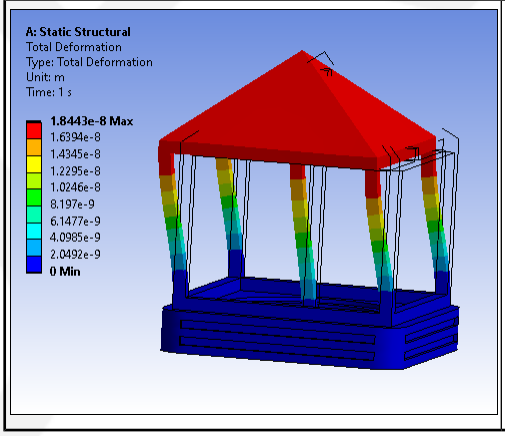
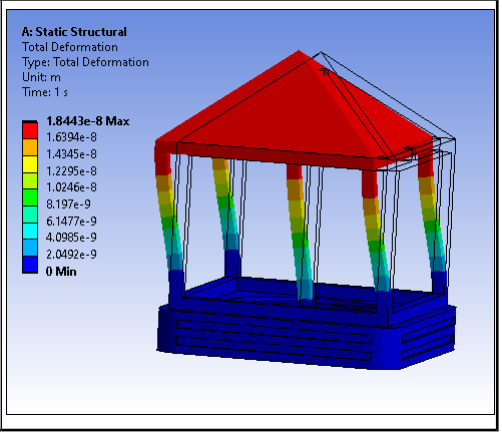
Edges

What is displayed by the options of the **Edges** drop-down menu, depends upon the selections you make in the other result display menus.

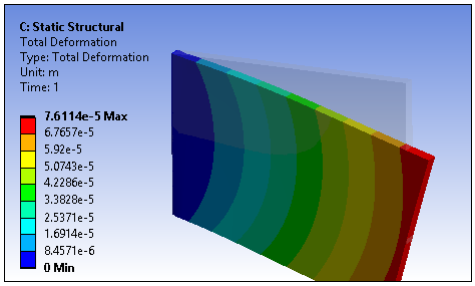
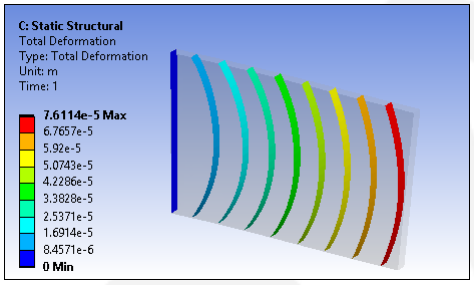
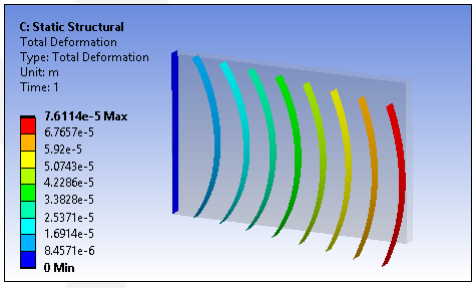
Option	Description
No Wireframe	Displays the result in its deformed state. The result's display is based on your selections in the **Geometry** and **Contours** menus (see above). **Example** 
Show Undeformed Wireframe	Displays the result with an undeformed wireframe overlay, as illustrated by the first image below. By default, this undeformed display is only supported for the **Exterior** option of the **Geometry** menu. The **IsoSurfaces**, **Capped IsoSurfaces**, and **Section Planes** options of the **Geometry** menu display the result with the wireframe overlay in a deformed state, as illustrated. **Geometry Menu Exterior** Option: Wireframe Not Deformed with Result  **Geometry Menu IsoSurfaces** Option: Wireframe Deformed with Result  You can change this default setting for the deformation display using the preferences of the **Options** dialog. Under the **Graphics** (p. 228) category, set the **Use Deformed Edge for Slice ISO Option** to **No**. For the **IsoSurfaces**, **Capped IsoSurfaces**, and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

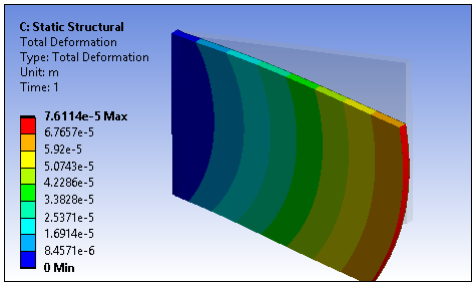
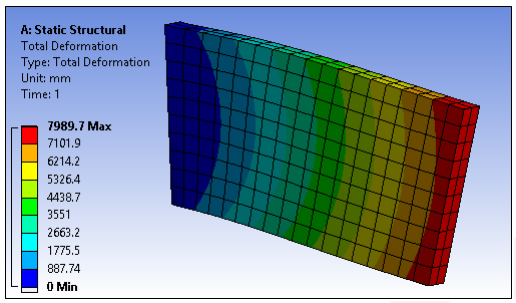
Published: 2023-10-23

Option	Description
	Section Planes options, you can display the result in a deformed state and the wireframe overlay in an undeformed state, as illustrated below. **Geometry Menu IsoSurfaces** Option: Wireframe Not Deformed with Result  **Geometry Menu Capped IsoSurfaces** Option: Wireframe Not Deformed with Result  **Correcting Edge Display for Large Face Angles** You may see wireframe display issues for edges when the tolerance angle between two faces is insufficient. Using the Graphics (p. 228) category preference, **Maximum Face Angle to Preserve Edges**, you can increase this value to properly render the edge display. As illustrated below, note the display change when the angle is increased. **Face Angle 120°**  **Face Angle 130°** 

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Show Undeformed Model	Displays the result with a translucent overlay of the undeformed model, as illustrated below. By default, this undeformed display is only supported for the **Exterior** option in the **Geometry** menu. The **IsoSurface**, **Capped IsoSurface**, and **Section Planes** options display the result with the translucent overlay of the model in a deformed state, as illustrated. **Geometry Menu Exterior** Option - Translucent Overlay Not Deformed  **Geometry Menu IsoSurfaces** Option - Translucent Overlay Deformed with Result  As stated above, you can change this default setting for the deformation display using the preferences of the **Options** dialog. Under the **Graphics** (p. 228) category, set the **Use Deformed Edge for Slice ISO Option** to **No**. For the IsoSurface, Capped IsoSurface, aand Section Planes options, you can display the result in a deformed state and the translucent overlay of the model in an undeformed state, as illustrated. **Geometry Menu IsoSurfaces** Option: Translucent Overlay Not Deformed with Result  **Geometry Menu Capped IsoSurfaces** Option: Translucent Overlay Not Deformed with Result

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

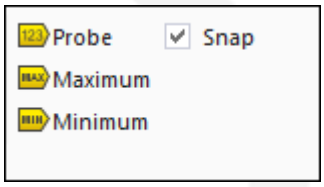
Option	Description
	
Show Elements	Displays the result in its deformed state and includes mesh elements. The result's display is based on your selections in the **Geometry** and **Contours** menus. **Example** 

Probe, Maximum, and Minimum

You use the **Probe** option to interactively place probe annotations on your model. If you display the Graphics Annotations (p. 202) window, you can view the result value at the location of your probe annotation, the unit of the result, as well as the coordinate values for the probe. You use the **Maximum** and **Minimum** options to display (or hide) annotations of the maximum and/or minimum values for the selected result.

Note:

The **Maximum** and **Minimum** options consider only visible bodies. If you hide a body that includes one of these annotations, you could see a discrepancy between the minimum and maximum values displayed in the Details properties for the result versus the (new) location of the annotations.



The **Snap** option (active by default), automatically places probes on the nearest mesh node (the probe "snaps" to the node). For high order elements, this includes midside nodes as well as the centroids of element faces. Additionally, using the **Show Mesh Entity ID in Snap-Probe Labels** preference (also active by default) of the Graphics (p. 228) category in the **Options** dialog, the application automatically includes either the Node ID or the Element ID in the probe annotation label (see example

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

below). Furthermore, the **Graphics** category (under the Interactive and Min/Max Probes category (p. 231)) includes a property, **Show Detailed Information in Min/Max Probe Labels**, that enables you to also display additional information on the Min/Max probe, such as Node/Element ID and actual Min/Max values.

Tip:

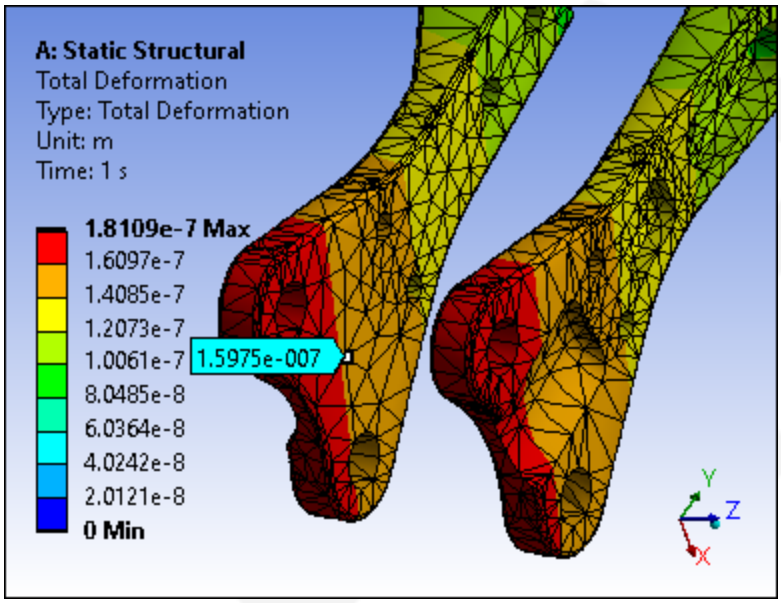
The application does not automatically save newly created probe annotations. You need to save the project manually.

Note:

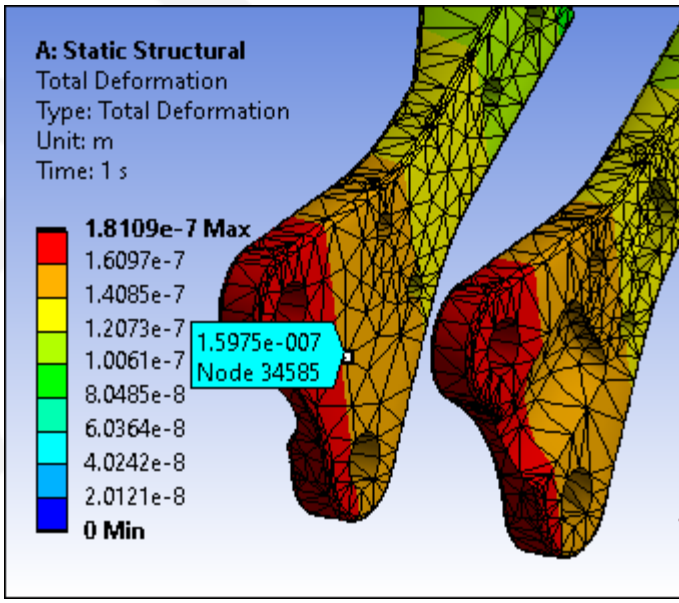
The **Snap** feature does not support the Geometry menu display options **IsoSurfaces**, **Capped IsoSurfaces**, or **Section Planes**.

Probe Labelling

Probe Label



Probe Label with Node ID

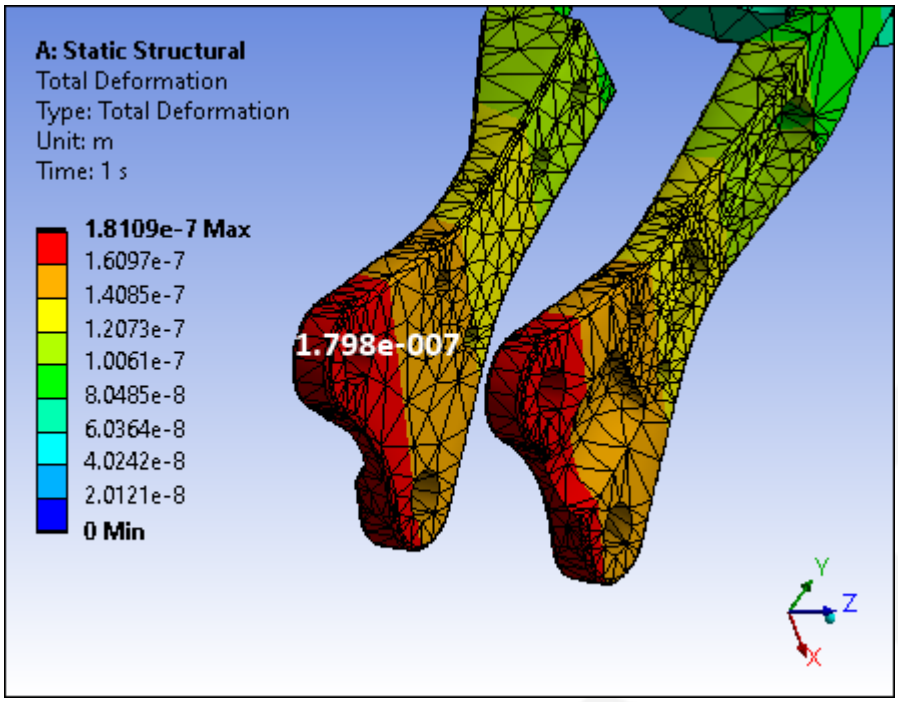


Probe Display Font

When the **Probe** option is selected and you move the cursor over your model, result values display. You can change the default display settings for the font using the **Probe Tool and Hit Point Coordinate (Windows Only)** preference of the **Graphics** (p. 228) category in the **Options** dialog. This preference enables you to change the attributes (font, size, style, color) of the font displayed (illustrated below is Bold Calibri Size 18).

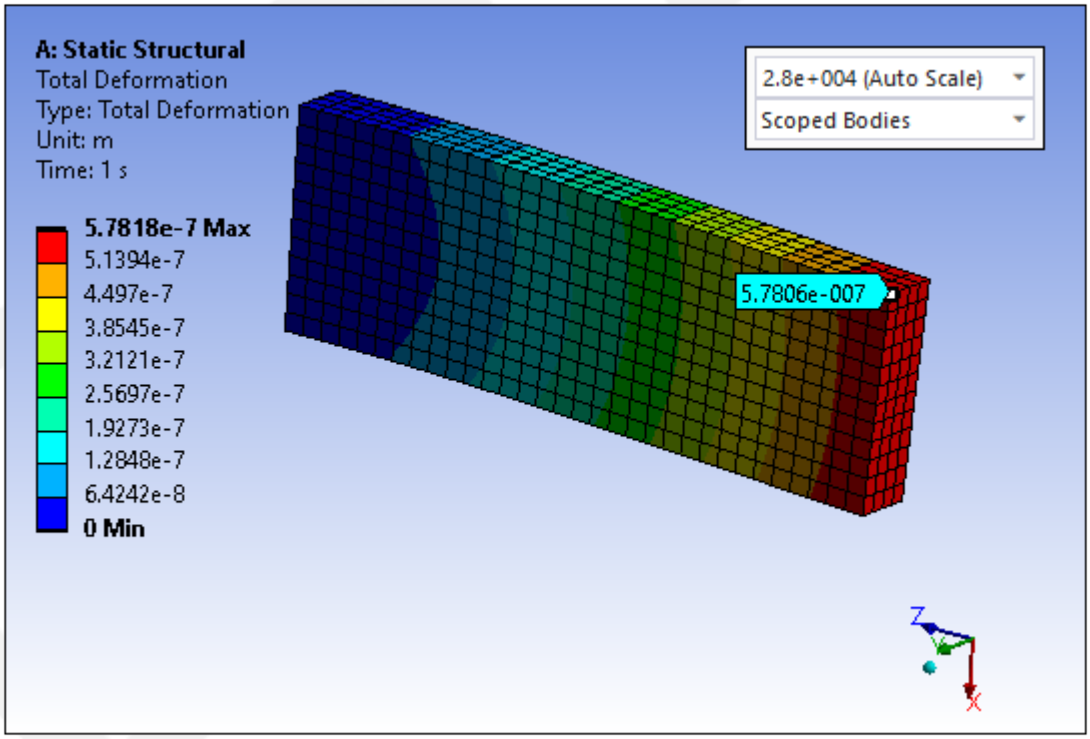
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

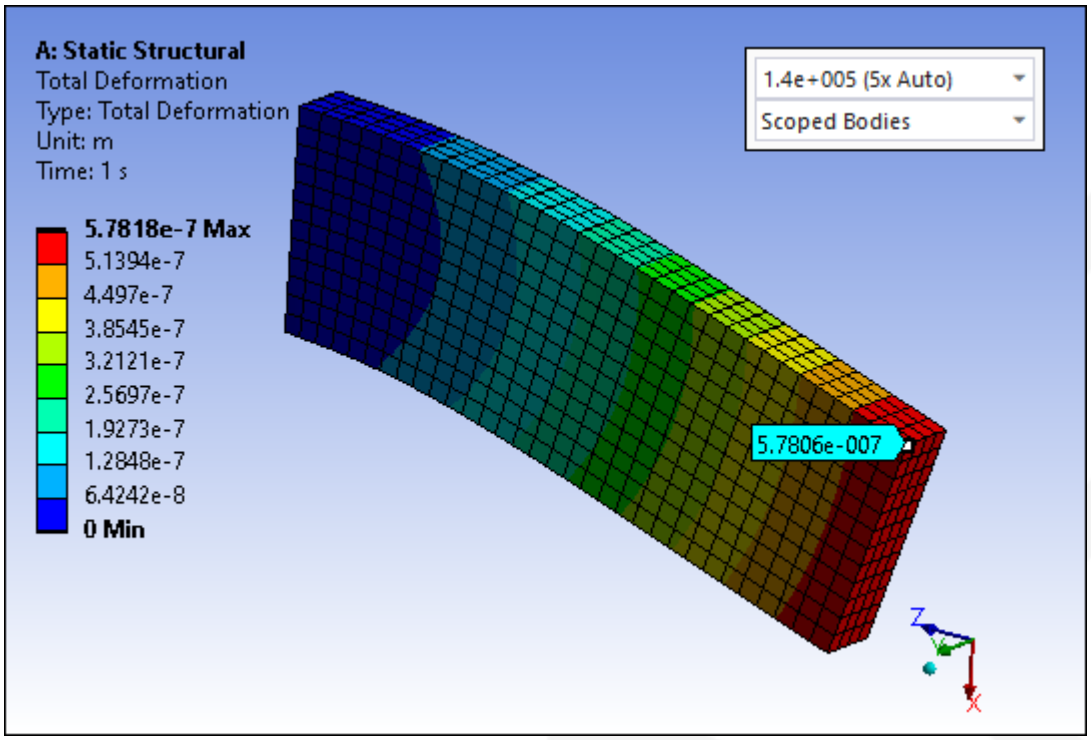


Probe Label and Scaling

Another **Graphics** (p. 228) category preference is the **Move Probe Labels with Deformation Scale Factor** setting. This preference is the default setting. It enables the probe to be "flexible" and move with the deformed mesh whenever you make a change to the Deformation Scale Factor, as illustrated below. If turned off, the application deletes probe labels whenever you make a change to the scale factor.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

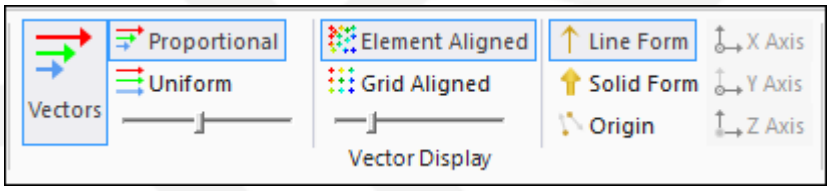


Note:

The application does not support the ability to flex a probe's location using the **Geometry** menu display options **IsoSurfaces**, **Capped IsoSurfaces**, or **Section Planes**. If you place a probe when any of these options are active and then change the scaling, the application automatically removes/deletes the probe labels.

Vector Display

Using the **Vector Display** group, you can display results as vectors. This group includes various options for controlling the display of the vectors.



Note:

These vector display options are also available for Element Orientations (p. 1008) and certain vector based-loads.

When you select the **Vectors** option, the following associated options may be used.

Option	Description
Proportional	Displays vector length proportional to the magnitude of the result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

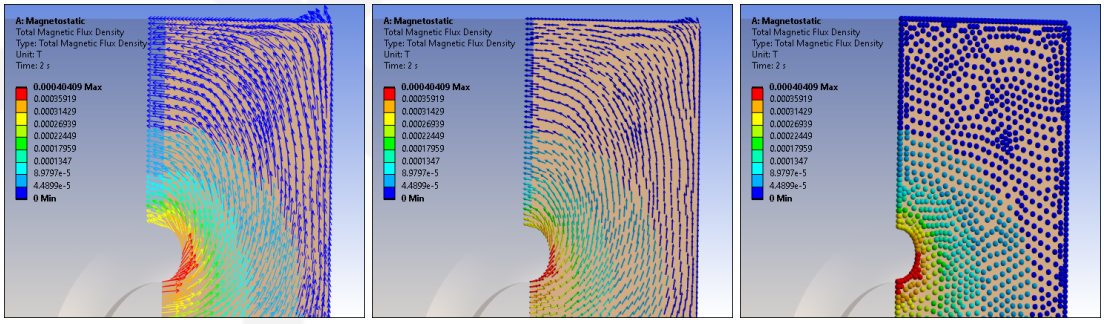
Published: 2023-10-23

Option	Description
Uniform	Displays a uniform vector length, useful for identifying vector paths.
Element Aligned	Displays all vectors, aligned with each element.
Grid Aligned	Displays vectors, aligned on an approximate grid.
Length Slider	Controls the relative length of the vectors in incremental steps from 1 to 10 (default = 5), as displayed in the tool tip when you drag the mouse cursor on the slider handle.
Grid Slider	Controls the relative size of the grid, which determines the quantity (density) of the vectors. The control is in uniform steps from 0 [coarse] to 100 [fine] (default = 20), as displayed in the tool tip when you drag the mouse cursor on the slider handle. **Note:** This slider control is active only when the adjacent button is chosen for displaying vectors that are aligned with a grid.
Line Form	Displays vector arrows in line form.
Solid Form	Displays vector arrows in solid form.
Origin	Displays origin of the nodes included in the load and/or result.
X Axis/Y Axis/Z Axis	When solving principle stresses or principle strain, these buttons enable you to display (or hide) the vectors for Maximum Principal, Middle Principal, and Minimum Principal at each node. When solving Nodal Triads or Elemental Triads, these buttons enable you to display (or hide) the vectors for X Axis, Y Axis, and Z Axis at each node or element.

Vector Display Examples

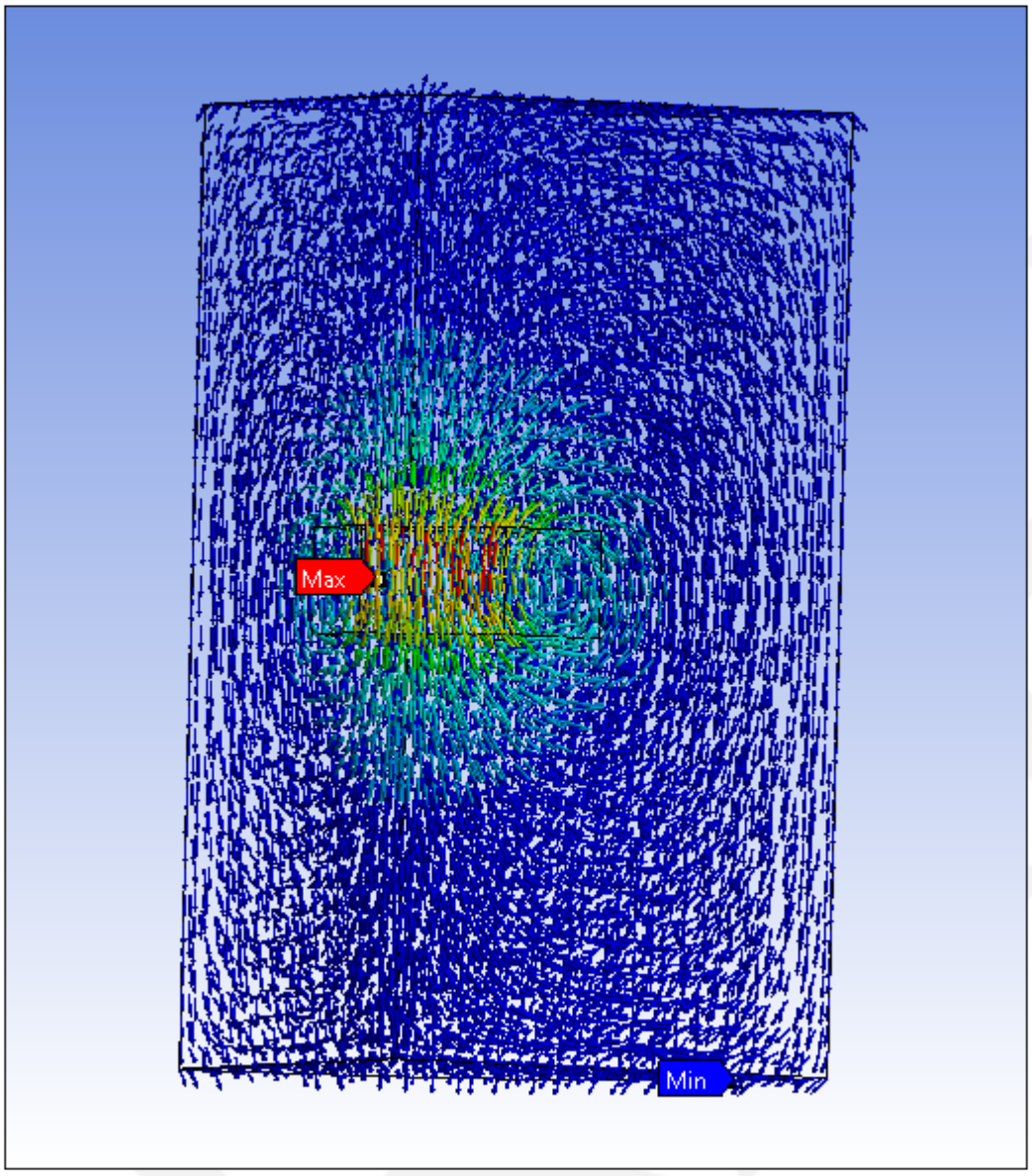
Here are examples of uniform vectors identifying paths in line, solid, and origin form.

Line Form **Solid Form** **Origin Form**



Here are vector arrows in solid form using the wireframe option.

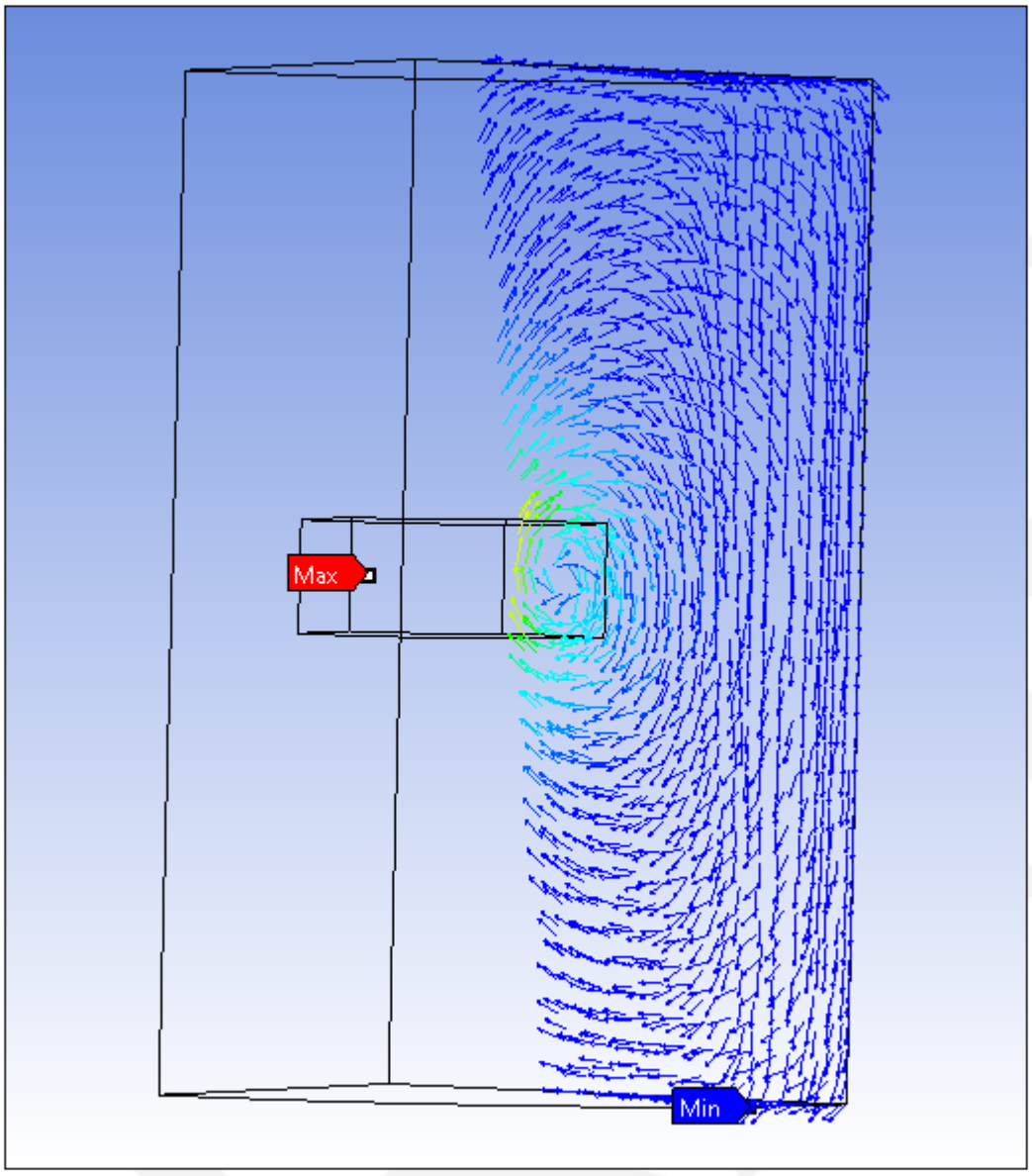
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



This is an example of uniform vector in solid form that have a Section Plane (p. 294) inserted.

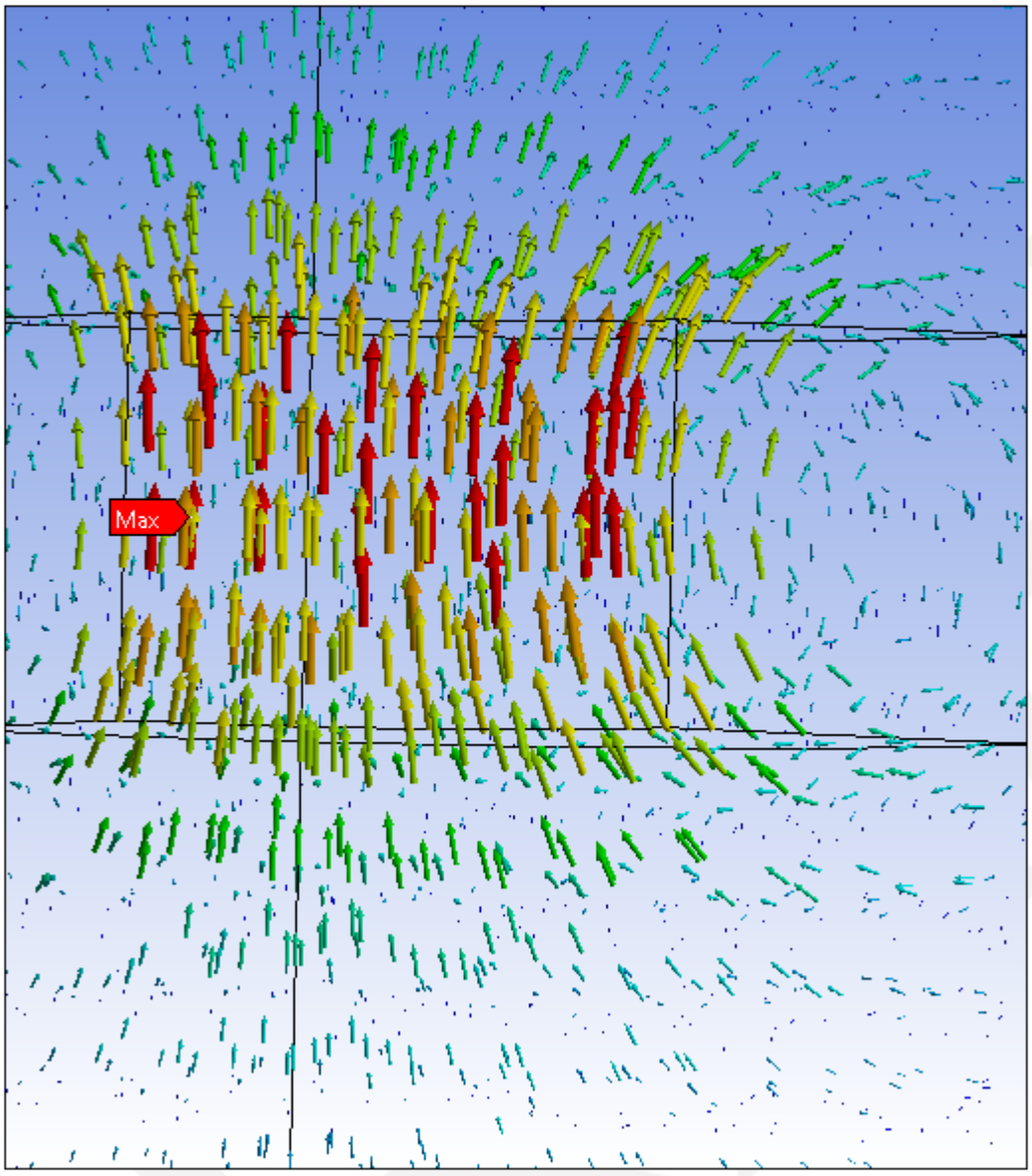
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



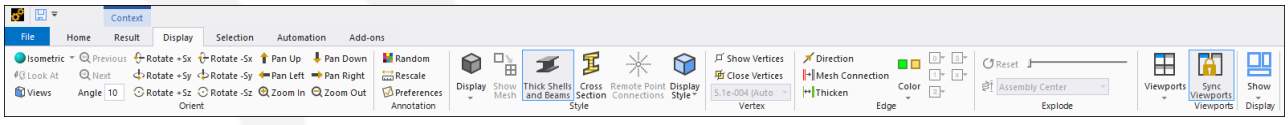
Here is a zoomed-in example of uniform vectors with arrow scaling in solid form.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Display Tab

The **Display** tab contains options for moving your model within the **Geometry** window as well as a variety of display-based options such as wireframe, edge thickness, ply directions, etc.



This tab contains the following **Groups**.

- **Orient** (p. 79)
- **Annotation** (p. 79)
- **Style** (p. 80)

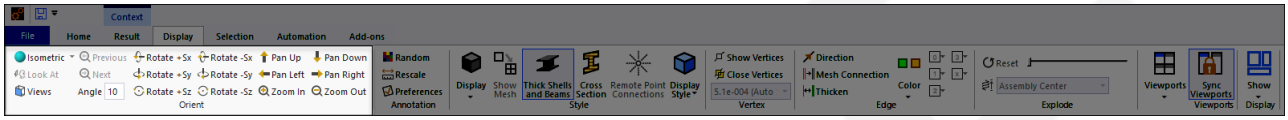
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Vertex** (p. 83)
- **Edge** (p. 84)
- **Explode** (p. 86)
- **Viewports** (p. 87)
- **Display** (p. 87)

Orient

The **Orient** group provides model orientation options.

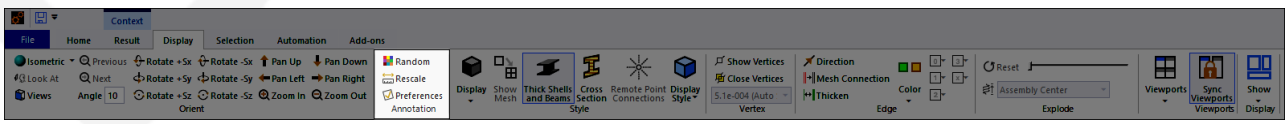


Options for this group include:

Option	Description
Isometric	Reorients your model into the isometric view. This option also includes the following drop-down menu options: • **Set**: Orient your model to a desired view and select this option to define a new default view for the **Isometric** option. • **Restore Default**: Resets the view of the **Isometric** option to the application default.
Look At	Centers your model in the **Geometry** window based in the currently selected face or plane.
Views	Provides a drop-down menu of options that enable you to change the viewpoint (front, back, right, etc.) of your model as well as an option to orient your model in the isometric view.
Previous/Next	Scrolls forward or backward from the last view displayed in the **Geometry** window.
Rotate ± X/Y/Z	Rotates your model in the **Geometry** window about the axis.
Pan Up/Down/Right/Left	Pans your model in the **Geometry** window.
Zoom In/Out	Zooms in or out of your model.

Annotation

The **Annotation** group enables you to make changes to how Annotations are displayed in the **Geometry** window as well as specify preferences.



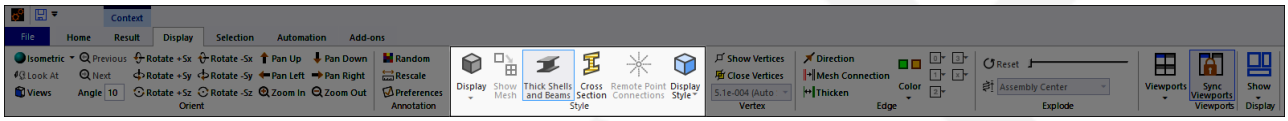
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options for this group include:

Option	Description
Random	By default, the annotations for objects types (loads, supports, named selections, etc.) are shown by a unique color. That is, all loads are red and all supports are blue etc. Selecting the **Random** option, you change the colors used for annotations.
Rescale	Changes the size of annotation symbols, such as load direction arrows.
Preferences	Displays the **Preferences** (p. 310) dialog that you use to set preferences for the display of annotations.

Style

The **Style** group provides model display options such as wireframe, showing the mesh, etc.



Options for this group include:

Option	Description
Display	Provides a drop-down menu of the following model display options: • **Shaded Exterior and Edges**: Displays the model in the **Geometry** window with shaded exteriors and distinct edges. This option is mutually exclusive with Shaded Exterior and Wireframe. • **Shaded Exterior**: Displays the model in the **Geometry** window with shaded exteriors only. This option is mutually exclusive with Shaded Exterior and Edges and Wireframe. • **Wireframe**: Displays the model in the **Geometry** window with a wireframe display rather than a shaded one (recommended for seeing gaps in surface bodies). This option is mutually exclusive from the above two options. The Wireframe option not only applies to geometry, mesh, or named selections displayed as a mesh, but extends to probes, results, and variable loads to enable a better understanding of regions of interest. When Wireframe mode is set, just the exterior faces of the meshed models are shown, not the interior elements. **Note:** – When this option is on, green scoping is not drawn on probes. Also, elements are shown on probes and results, whereas the outline of the mesh is shown on isoline contour results.

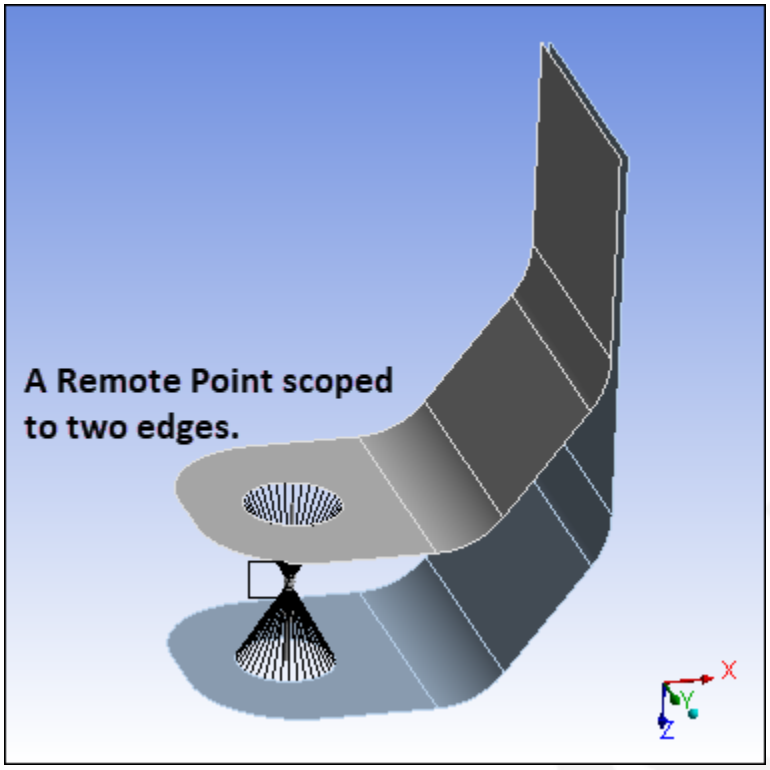
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

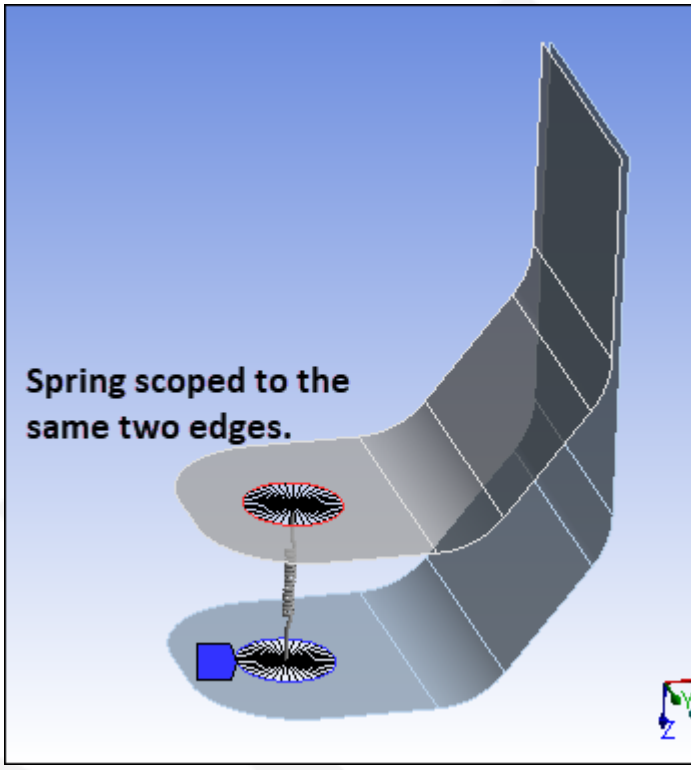
Published: 2023-10-23

Option	Description
	– Selecting any of the edges options (p. 68) on contour results automatically closes Wireframe mode.
Show Mesh	Displays your model's mesh regardless of the selected Outline object. When enabled, to make sure that Annotations display properly, also turn on Wireframe mode. See the related Note (p. 82) below.
Thick Shells and Beams	Displays the thickness of shells, beams, and particles, when viewing mesh or results. By default, this option is inactive. Review the Note (p. 82) below for additional display characteristics associated with this feature.
Cross Section	Displays line body cross sections as 3D geometry. See Viewing Line Body Cross Sections (p. 877) for details.
Remote Point Connections	Displays connection lines between geometry and Remote Points (p. 1089) or between supported remote boundary conditions (p. 1735) and Remote Points. See the example below. Once you have generated the mesh, the connection lines are drawn between the Remote Point and the nodes on the corresponding mesh of the underlying geometry or boundary condition. **Important:** Only the nodes that are contained inside of the Pinball Radius property setting are displayed.
Display Style	Provides a drop-down menu with options to display the parts and bodies of your model based on the available styles. For example, if an assembly is made of parts of different materials, you can color the parts based on the material; that is, all structural steel parts have the same color, all aluminum parts have the same color and so on. See the Color Coding of Parts (p. 858) topic.

Remote Point Connections Display Example

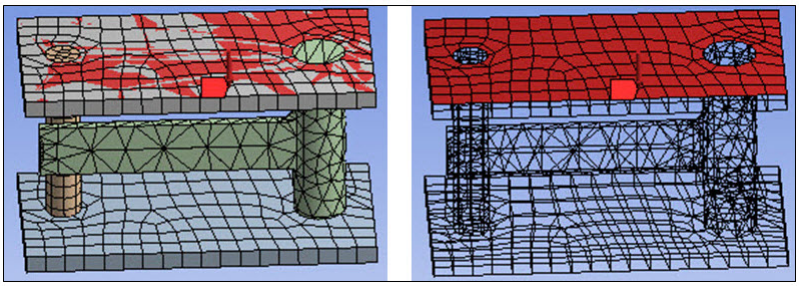
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Note:

As illustrated below, annotations may not always display properly when the **Show Mesh** option is activated. Turning on Wireframe mode accurately displays Annotations when **Show Mesh** is selected.



Note:

- **Displaying Shells for Large Deflections**: The display of shells may become distorted for large deformations such as in large deflection or during an Explicit Dynamics analysis. A workaround for this is to disable Shell Thickness by toggling the **Thick Shells and Beams** option. Or, set a variable, **UsePseudoShellDisp = 1**, using the **Variable Manager** option from the **File** tab. It may be necessary to toggle the deformation scaling from True Scale to Undeformed to True Scale again (see **Scaling Deformed Shape** in the Context Tabs (p. 47) section). Note that this option requires **True Scaling** to work properly.

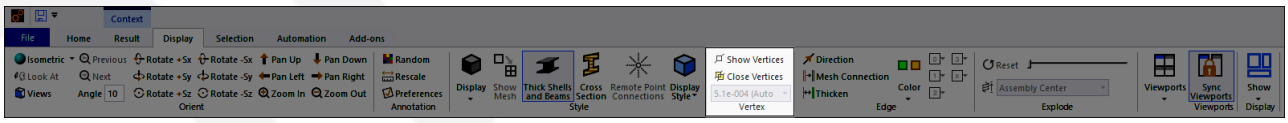
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Displaying Shells with Thickness on Geometry that Spans Large Angles**: The graphical representation of your meshed shell model may appear distorted when the shell spans a large angle, such as a 90° angle. Ordinarily, the application calculates an average of the normals between elements (based on a default setting of 180°). Given too large of an angle, a graphical abnormality may occur. Modify the default setting using the Graphics (p. 228) option in the **Options** (p. 217) dialog box. **60°** is the recommended setting to avoid the display of any graphical abnormalities.
- **Displaying Results on Very Thin Shell Bodies**: If you are viewing result contours of a very thin geometry, you could observe a graphical distortion as a result of colors from the back face of the geometry bleeding onto the front face of the geometry. This is a graphics-based limitation. In addition, turning off the **Thick Shells and Beams** option can cause the distortion to worsen.
- **Displaying Shells on Shared Entities**: The display of shells is done on a nodal basis. Therefore, graphics plot only 1 thickness per node, although node thickness can be prescribed and solved on a per elemental basis. When viewing shell thickness at sharp face intersections or a shared body boundary, the graphics display may become distorted.
- **Shell Element Display from Mesh Changes**. If you employ a feature that changes the model's mesh, such as the **Nonlinear Adaptive Region** or **Fracture**, you may see display errors for expanded shell elements as a result of the changing mesh. Disable the **Thick Shells and Beams** option to properly display the elements.
- **Displaying Contours and Displaced Shapes on Line Bodies**: The contour result on a line body are expanded to be viewed on the cross section shape, but only one actual result exists at any given node and as a result no contour variations across a beam section occur.
- **Display Pipes using Pipe Idealizations**: Although the solution will account for cross section distortions, the graphics rendering for the results display the cross sections in their original shape.

Vertex

The **Vertex** group provides vertex display options.



Options for this group include:

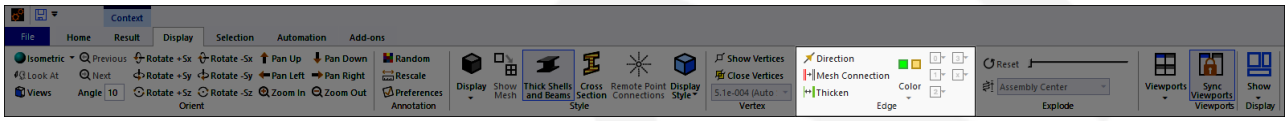
Option	Description
Show Vertices	Highlights all vertices on the model. This feature is especially useful when examining complex assemblies where vertices might normally be hidden from view. It can also be used to ensure that edges are complete and not segmented unintentionally.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Close Vertices	Displays tightly clustered vertices on your model. This vertex display feature has an accompanying scale menu. When you select this option, a corresponding scale menu also activates and includes application generated tolerances as well as an option that enables you to enter a tolerance value. The application calculates the default tolerance (**Auto Scale**). This value is 0.1% of the diagonal measurement of your model's **Bounding Box** dimensions. Additional system options are factors of this base measurement and you can manually enter a tolerance using the **Custom Value** option. The **Custom Value** cannot exceed 5% of the model's **Bounding Box** dimensions. Based on the selected tolerance, the application highlights pairs of vertices that are closer to one another than the specified tolerance and draws segments between the vertices to further illustrate proximity.

Edge

The **Edge** group provides display options used to display the edges on your model, their connectivity, and how they are shared by faces. Also see the Preparing Assemblies of Surface Bodies (p. 861) section for more information.



Options for this group include:

Option	Description
Direction	Displays model edge directions. The direction arrow appears at the midpoint of the edge. The size of the arrow is proportional to the edge length.
Mesh Connection	Displays the edges using coloring schema, by taking into account the mesh connection information.
Thicken	For annotations scoped to lines (for example, annotations representing loads, named selections, point masses, and so on), enabling this option thickens these lines so they are more easily identifiable on the screen.
Color	The options of this drop-down menu highlight the edges of your model using specific colors. Options include: • **By Body Color**: Highlight the edges of your model using the colors assigned to the bodies. • **By Connection**: Highlight the edges of your model based on the number of element faces shared by the edge. Colors include blue (not shared, labeled “Free”), red (shared by one face, labeled “Single”), black (shared by two faces, labeled “Double”), pink (shared by three faces, labeled “Triple”) and yellow (edge shared by more than three faces, labeled “Multiple”). The legend for this option is titled “Edge/Face Connectivity.”

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	Description
	• **By Body Connection**: Highlight the edges of your model based on edge-body connections. Colors include black (edge shared by one body, labeled Single), pink (edge shared by two bodies, labeled Double), and yellow (edge shared by more than three bodies, labeled "Multiple"). The legend for this option is titled "Edge/Body Connectivity." • **Black**: Highlight the edges of your model with black lines. This option turns off the edge-face connectivity display.
Free (edge display)	Displays drop-down menu with the following options: • **Hide Free**: Hides only edges not shared by any faces. • **Show Free**: Displays only edges not shared by any faces. • **Thick Free**: Displays only edges not shared by any faces at a different edge thickness compared to the rest of the model.
Single (edge display)	Displays drop-down menu with the following options: • **Hide Single**: Hides only edges that are shared by one face. • **Show Single**: Displays only edges that are shared by one face. • **Thick Single**: Displays only edges that are shared by one face at a different edge thickness compared to the rest of the model.
Double (edge display)	Displays drop-down menu with the following options: • **Hide Double**: Hides only edges that are shared by two faces. • **Show Double**: Displays only that are shared by two faces. • **Thick Double**: Displays only edges that are shared by two faces at a different edge thickness compared to the rest of the model.
Triple (edge display)	Displays drop-down menu with the following options: • **Hide Triple**: Hides only edges that are shared by three faces. • **Show Triple**: Displays only that are shared by three faces. • **Thick Triple**: Displays only edges that are shared by three faces at a different edge thickness compared to the rest of the model.
Multiple (edge display)	Displays drop-down menu with the following options: • **Hide Multiple**: Hides only edges that are shared by more than three faces. • **Show Multiple**: Displays only that are shared by more than three faces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
	• **Thick Multiple**: Displays only edges that are shared by more than three faces at a different edge thickness compared to the rest of the model.

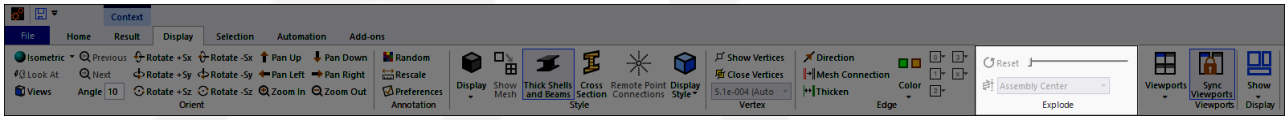
Note:

Note the following restrictions when you are using the **Edge** options on the mesh, as compared to their use on geometry:

- When you are using the **Edge Coloring** options when viewing the mesh, the application only draws the corner nodes to display the outline of the elements (mid-side nodes are ignored if available). You can use the Wireframe tool and also hide bodies to properly display the colored edges. And in doing so, you can see where mid-side nodes are located, if available.
- Not all of the buttons/options are functional, for example, **Double** always displays thin black lines. The width of the colored lines cannot be changed. They are always thick.
- During slicing, the colors of shared element edges are not drawn. They display as black and appear only when the selected section plane is losing focus in the slice tool pane.

Explode

The **Explode** group is a graphical display feature used to create imaginary distance between geometry bodies (only) of your model for viewing purposes.



Once the mesh is generated, this feature is not supported when you have the **Mesh** object selected or when the **Show Mesh** feature is turned on. In addition, when viewing the mesh, exploded geometry bodies, although not visible in the **Geometry** window, are still in an exploded state and passing the cursor over an exploded body will highlight the (otherwise invisible) body and it is also selectable at this time.

Reset Button

This button reassembles the parts of your model to their original position.

Explode View Factor Slider

This slider tool enables you to change the exploded distance between the parts from their original position.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Move Springs/Beams with Parts

The button for this option enables you to see an accurate representation of connections on your model, such as Springs and Beams, by showing the connections stretched from the assigned locations on the moving parts. Because the display is graphically accurate, the processing requirements are intensive. Use the default position (not active/depressed) when moving the slider for large models and when connection representations are not critical.

Assembly Center Drop-Down List

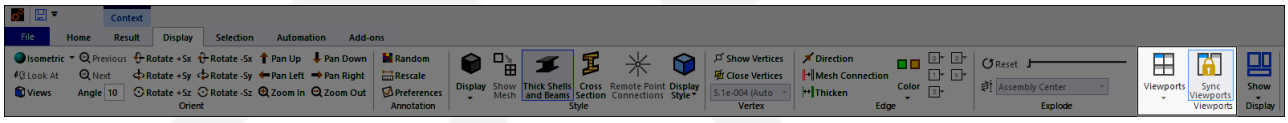
This drop-down list provides the available coordinate system options as well as the **Assembly Center** option (default setting) that defines the position in space from which the exploded view originates and the **Assembly Center (Visible)** option that accounts for the visible parts only. The **Global Coordinate System** is always an available option as well as any user-defined coordinate systems (p. 1192).

Note:

The explode view feature does not support the **Body Views** display, such as when you are displaying contact bodies in separate windows (p. 1270).

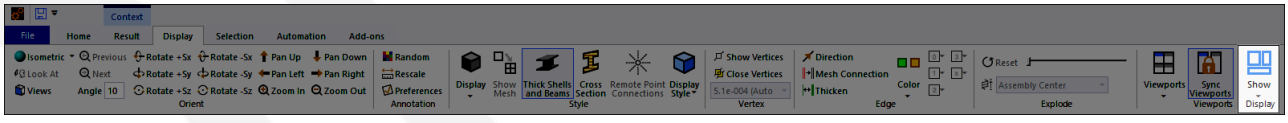
Viewports

The **Viewports** group enables you to split the **Geometry** window into multiple windows, synchronize the windows as desired, and/or capture an image of the currently displayed viewports. See the Using Viewports (p. 284) section for more information.



Display

The **Display** group contains the **Show** drop-down menu that provides several general display options, such as the ruler and legend.



Options of the menu include:

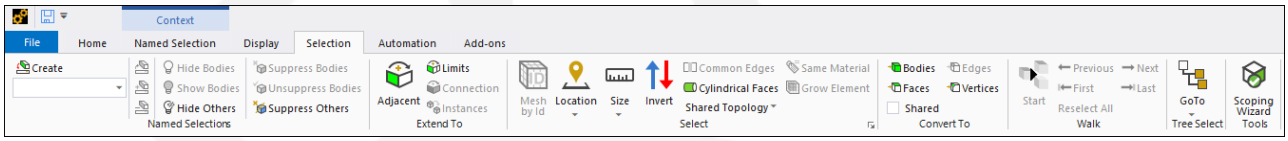
Option	Description
Ruler	Turns the **Geometry** window ruler on and off.
Legend	Turns the **Geometry** window legend on and off.
Triad	Turns the **Geometry** window triad on and off.
Show Mesh	Displays the model's mesh.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
All Coordinate Systems	Displays all the available coordinate systems defined on the model – default as well as user defined.
Visual Expansion	Toggles the visibility of either a single cyclic sector mesh or the full symmetry mesh in a cyclic symmetry analysis (p. 1147). Toggling this option can help preview before solving the density of nodes on the sector boundaries, or it can help confirm the expanded mesh in each case.
Erodes Nodes	Turns the visibility of eroded nodes for explicit dynamics analyses on or off.
Draw Face Mode	Displays a drop-down menu with options to change how faces are displayed as a function of back-face culling. Options include: • **Auto Face Draw**: Turning back-face culling on or off is program controlled. Using Section Planes is an example of when the application would turn this feature off. • **Draw Front Faces**: Forces face culling to stay on. Back-facing faces will not be drawn in any case, even if using Section Planes. • **Draw Both Faces**: Turns off back-face culling. Both front-facing and back-facing faces are drawn.

Selection Tab

The **Selection** tab facilitates the selection of geometric and/or mesh entities either through graphical picking or through some criterion-based selection feature, such as size or location.



Note:

The tab's functionality uses Ansys ACT. The relevant python modules (`selection.py` and `toolbar.py`) are available for review in the install folder: **aisol/DesignSpace/DSPages/Python**.

This tab contains the following **Groups**.

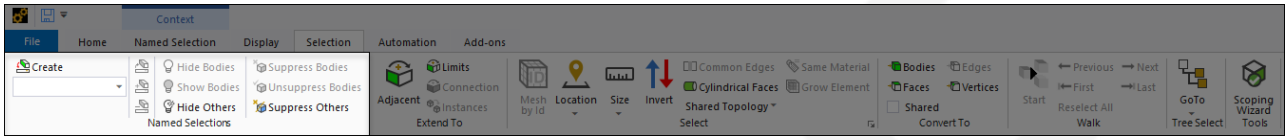
- **Named Selections** (p. 89)
- **Extend To** (p. 89)
- **Select** (p. 89)
- **Convert To** (p. 92)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Walk** (p. 93)
- **Tree Select** (p. 93)
- **Tools** (p. 93)

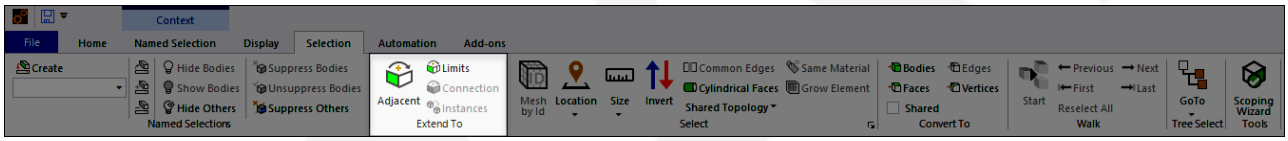
Named Selections

The **Named Selections** group enables you to select, add to, and remove items from existing user-defined named selections as well as modify visibility and suppression states. See the Applying Named Selections via the Ribbon (p. 1075) section for detailed description of the options.



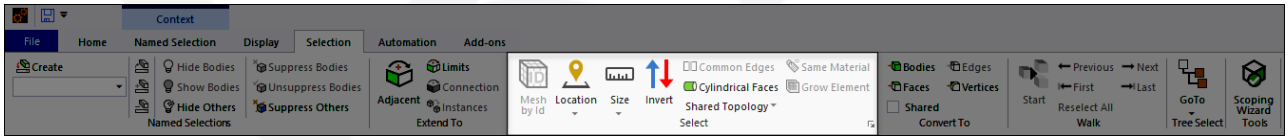
Extend To

The **Extend To** group enables you to add adjacent faces or edges, within angle tolerance, to the currently selected face or edge set, or adds tangent faces or edges within angle tolerance, to the currently selected face or edge set. See the Extend To (p. 264) topic for additional information about these options.



Select

The **Select** group provides options for making and/or manipulating geometry selections.



Note:

Many of these options are also available from the **Select By** menu on the Graphics Toolbar (p. 97).

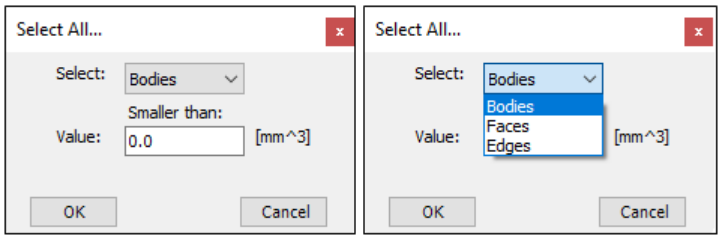
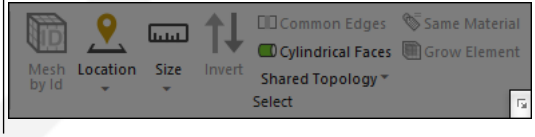
Options for this group include:

Option	Description
Mesh by Id	Opens a dialog that enables you to select mesh nodes and mesh elements using their IDs once you have generated the mesh for your model. This feature is *modeless* and therefore enables you to work with the user interface while the dialog box is displayed. This feature is also available from the context (right-click) menu, **Select Mesh by ID (M)**, in the **Geometry**

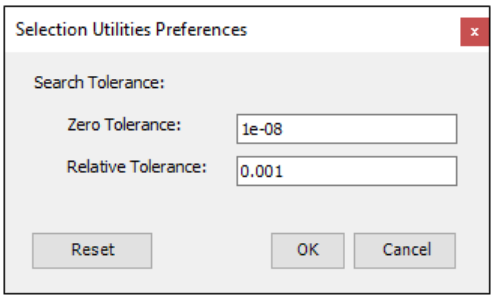
Option	Description
	window. You can also activate the feature using the **M** key, when the **Geometry** window has focus. See the Selecting Nodes and Elements by ID (p. 279) section for more information.
Location **Note:** For a line body geometry, the location is estimated as the weighted arithmetic mean of the centroids of its edges. The weight is based on the edge lengths.	Displays a drop-down menu of the following options: • **Select All Entities With the Same X Location**: Adds all geometry entities in the model with the same X location in the Global Coordinate System as the current selection, to the current selection. • **Select All Entities With the Same Y Location**: Adds all geometry entities in the model with the same Y location in the Global Coordinate System as the current selection, to the current selection. • **Select All Entities With the Same Z Location**: Adds all geometry entities in the model with the same Z location in the Global Coordinate System as the current selection, to the current selection. • **Select All Coplanar Entities With the Same X Location**: Adds all geometry entities of the model that are in the same plane with the same X location, in the Global Coordinate System as the current selection, to the current selection. • **Select All Coplanar Entities With the Same Y Location**: Adds all geometry entities of the model that are in the same plane with the same Y location in the, Global Coordinate System as the current selection, to the current selection. • **Select All Coplanar Entities With the Same Z Location**: Adds all geometry entities of the model that are in the same plane with the same Z location, in the Global Coordinate System as the current selection, to the current selection.
Size	Displays a drop-down menu of the following options: • **Select All Entities With the Same Size**: Adds all geometry entities in the model with the same size as the current selection to the current selection. • **Select All Entities Smaller than Selection**: Adds all geometry entities in the model that are smaller than the current selection to the current selection. • **Select All Entities Smaller than**: Displays a dialog box that enables you to specify the type of geometric entity as well as a reference value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

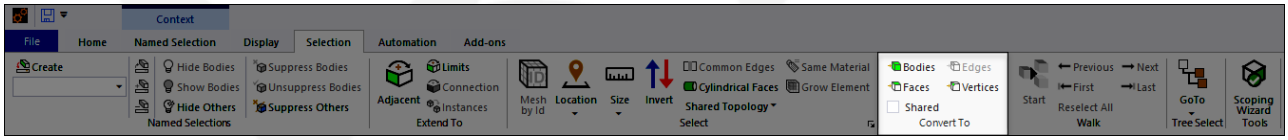
Option	Description
	 The **Select** drop-down menu default option is **Bodies**. The application overrides this default if you have actively selected edges or faces. The default of the **Value** field equals the size of the current selection. The application adds all entities in the model for the given type that are smaller than the reference value to the current selection. Note that the units are based on the active unit system when the dialog was first launched and the type of entity. • **Select All Entities Larger than Selection**: Adds all geometry entities in the model that are larger than the current selection to the current selection. • **Select All Entities Larger than**: Displays a dialog box (shown above) that enables you to specify the type of geometric entity as well as a reference value. The application adds all entities in the model for the given type that are larger than the reference value to the current selection. The units are based on the active unit system . **Note:** These options are also available from the **Select By** > **Size** menu on the Graphics Toolbar (p. 97).
Invert	Selects all entities (face, edge, etc.) that are **not** currently selected. The option only selects entities of the same type. For example, if you have a face selected and select Invert, the application selects all the faces on your model *except* the face that you had selected.
Common Edges	Selects common edges of selected faces.
Cylindrical Faces	Selects all faces on the model that are cylindrical (they do not need to be full cylinders).
Shared Topology	Displays a drop-down menu of options, including, **All Edges** and **All Faces**. These options select any edge or face on the interior of a multi-body part.
Same Material	Selects all bodies with the same Material Assignment as the currently selected body.
Grow Element	Selects all elements adjacent to your current element selection. This option effectively grows the element selection by one layer of elements.
Set Tolerances	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Description
	Opens a dialog box that enables you to specify a search tolerance for your geometric entity selections. The dialog fields include **Zero Tolerance** and **Relative Tolerance**. **Note:** Tolerance settings are only applicable when using the **Select All Entities With Same Size** option or a **Location** option (see above).  By default, the **Zero Tolerance** property is set to **`1.e-08`** and the Relative Tolerance value is **`0.001`**. Relative tolerance is a multiplying factor applied to comparisons. For example, if you want a tolerance of 1%, enter **`.01`** in the **Relative Tolerance** field. Tolerance values are dimensionless. All comparisons are done in the CAD unit system. Review the Adjusting Tolerance Settings for Named Selections by Worksheet Criteria (p. 1061) topic in the *Specifying Named Selections using Worksheet Criteria* section for additional information.

Convert To

The **Covert To** group enables you to change (convert) your currently selected geometric entity or mesh item to a different geometric entity or mesh item.



This group includes the following options:

Option	Description
Shared	Selects only geometric entities that are shared by all currently selected entities.
Bodies	Selects all bodies associated with your current selection of either faces, edges, vertices, elements, or nodes. The selection mode automatically changes to Body selection.
Faces	Selects all Faces associated with your current selection of either bodies, edges, vertices, elements, or nodes. For example, if your selection is a body or bodies, all faces on that body will be selected. The selection mode automatically changes to Face selection.
Edges	Selects all Edges associated with your current selection of bodies, faces, vertices, elements, or nodes. For example, if vertices are selected, any edges associated with the vertices will be selected. The selection mode automatically changes to Edge selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

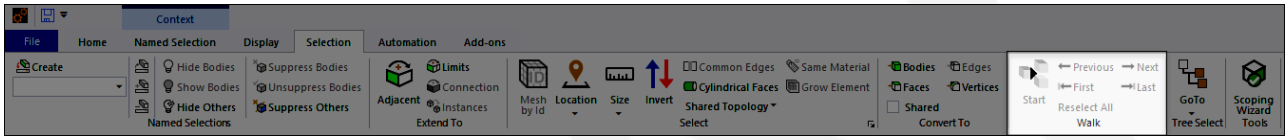
Option	Description
Vertices	Selects all Vertices associated with your current selection of either bodies, faces, edges, elements, or nodes. The selection mode automatically changes to Vertex selection.

Note:

These options, except for the **Shared** option, are also available from the **Convert** menu on the Graphics Toolbar (p. 97).

Walk

The **Walk** group enables you to highlight and zoom in on entities of your model. Once you select more than one geometric or mesh entity (using Graphics Toolbar (p. 97) options) from the **Geometry** window and select the **Start** option, the application automatically highlights and zooms in on each entity in turn as you use the navigation options. The feature supports all geometric and mesh (mesh generation required) selections, including Body, Face, Edge, Vertex, Node, Element Face, and Element.



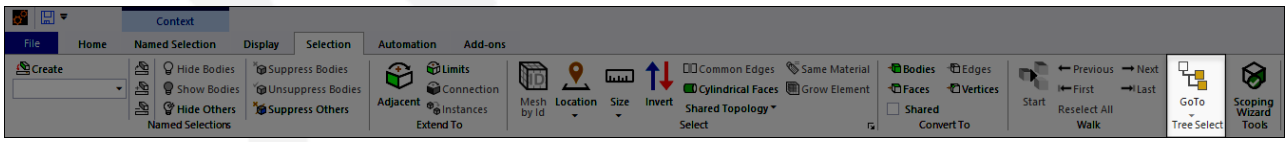
Once you have made selections and selected the **Start** option, the application caches each selected entity into memory. You then use the navigation options (**Previous**/**Next**/**First**/**Last**) to step through your selections. The cached selections are maintained until you make new selections and click the **Start** option.

Tip:

When the **Geometry** window has focus, the **Select All** (or **[Ctrl]**+**[A]**) context (right-click) menu option can be useful when using this feature.

Tree Select

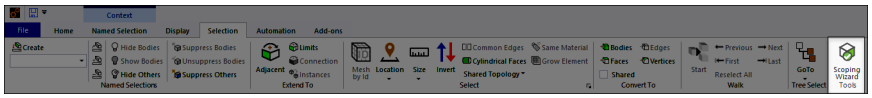
This group provides a drop-down menu of **Go To** options. These options correlate model selections to tree objects. Selecting an option highlights a tree object. See the Correlating Tree Outline Objects with Model Characteristics (p. 110) section for more information. The **Go To** options are also available from the context (right-click) menu of the **Geometry** (p. 129) window.



Tools

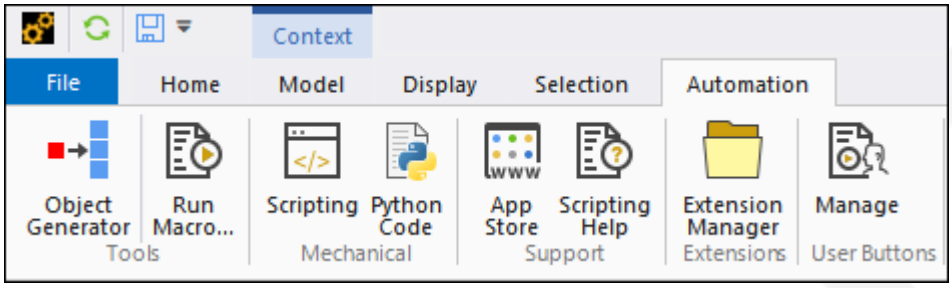
The **Tools** group contains the **Scoping Wizard** option. This option displays the **Scoping Wizard** (p. 147) pane. This feature enables you to reidentify an object's scoping lost because of a geometry update. Once identified, you can use the options of the feature to reestablish the original scoping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



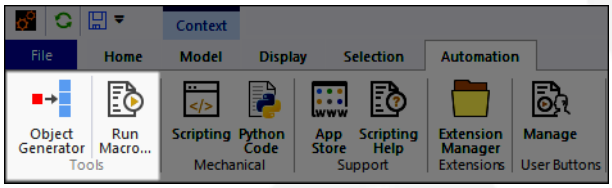
Automation Tab

The **Automation** tab provides productivity and customization features.



Tools

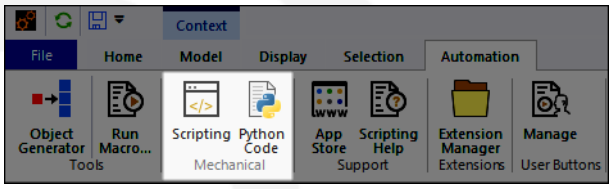
The **Tools** group contains the **Object Generator** (p. 2343) option as well as the **Run Macro** option that open a dialog box to locate a desired script file. Macros can be written in the Python (.py) programming language. For additional information, refer to the Scripting in Mechanical Quick Start Guide.



Mechanical

The **Mechanical** group includes the following options:

- **Scripting**: Launch the **Mechanical Scripting** pane.
- **Python Code**: Insert a Python Code object into the Outline.



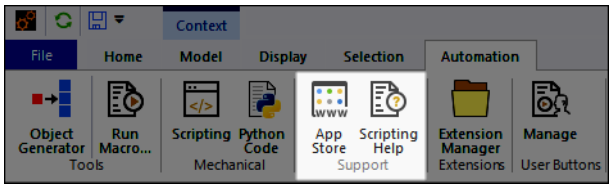
Support

The **Support** group includes the following options:

- **App Store**: Open the Ansys Application Store website.
- **Scripting Help**: Open the Scripting Introduction Help page.

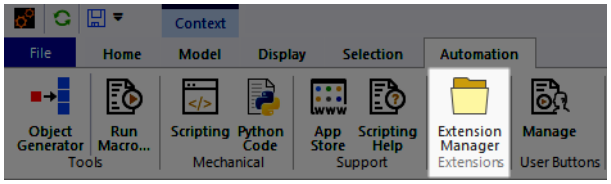
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



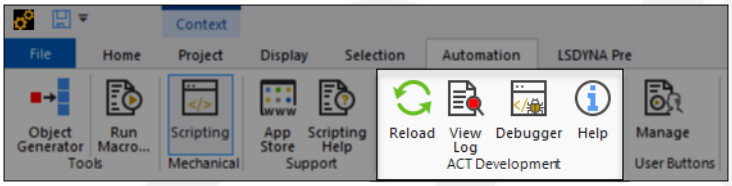
Extension Manager

When you open Mechanical independently (p. 323), the **Automation** tab contains an **Extension Manager** option. Use this feature to load/unload ACT extensions just as you would from the Manage Extensions in Workbench.



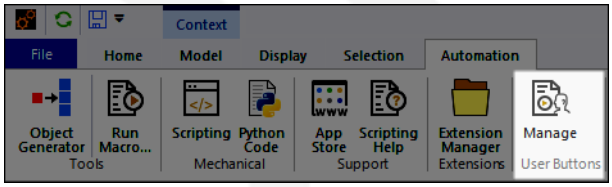
ACT Development

The **ACT Development** group displays when you have activated the debug mode. From Workbench, active **Debug Mode** using the options **Tools** > **Options** > **Extensions**. When you launch Mechanical independently, activate debug mode by selecting the **Extension Manager** option. This opens the Extension **Manager** pane. From this pane, select the gear icon and then select the **Debug Mode** checkbox.



User Buttons

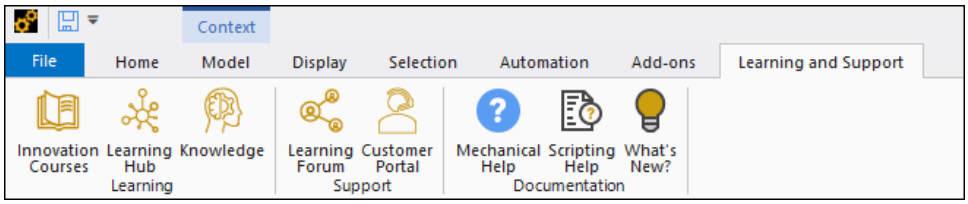
The **User Buttons** group enables you to create, edit, and manage custom options. Custom options are added to this group when created. See the Creating User Defined Buttons (p. 154) section for more information.



Learning and Support Tab

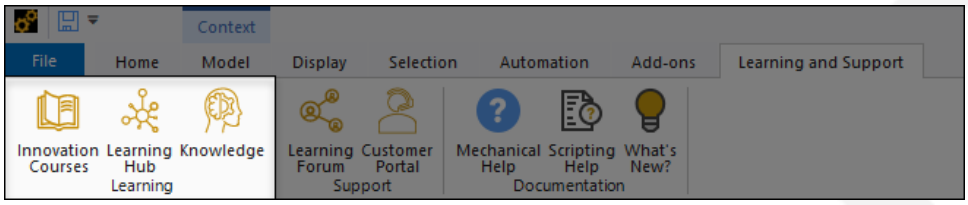
The **Learning and Support** tab contains links to various learning and support resources for Mechanical.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Learning

The **Learning** group provides access to learning resources.

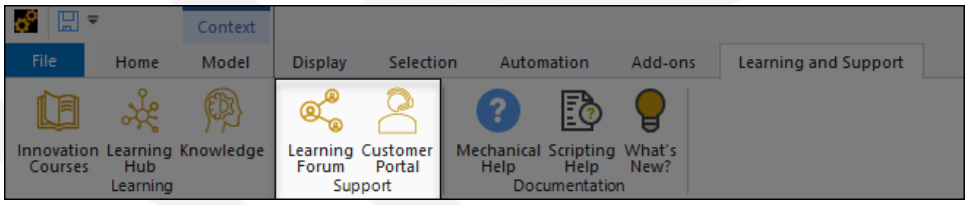


Options for this group include:

- **Innovation Courses**: This option links to free structural engineering courses in Ansys Innovation Space. These self-paced courses include videos, handouts, and quizzes to help reinforce physics and engineering fundamentals.
- **Learning Hub**: This option links to subscription-based virtual courses in Ansys Learning Hub. Within the hub, you can attend live or self-paced classes with dedicated learning rooms for discussions and continuous learning support from Ansys experts.
- **Knowledge**: This option links to free general learning materials for structural engineering in Ansys Innovation Space. The materials include expert-curated FAQs and short video tutorials to help answer simulation-related questions.

Support

The **Support** group provides access to support options.



Options for this group include:

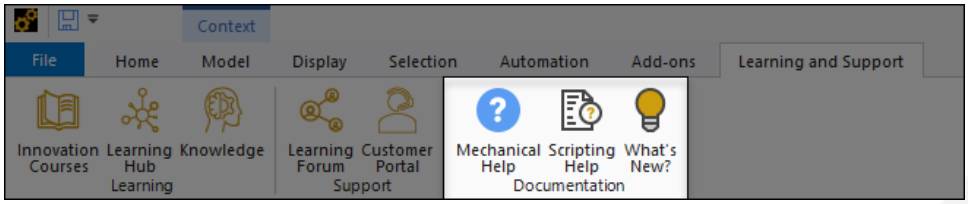
- **Learning Forum**: This option links to a public user forum for mechanical engineering discussions in Ansys Innovation Space. You can find answers to your questions and learn from the global Ansys user community.
- **Customer Portal**: This option links to a private support portal for commercial customers. You can receive technical support from Ansys during business hours. You can also log support requests, download product software, and manage licensing and subscriptions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Documentation

The **Documentation** group provides access to our product documentation for Mechanical.

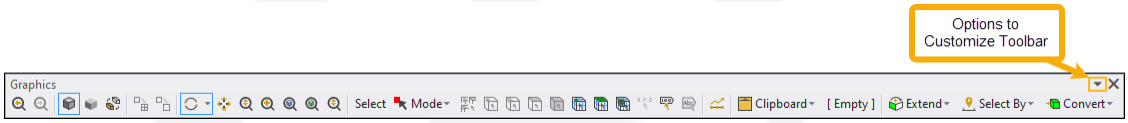


Options for this group include:

- **Mechanical Help**: This option links to the *Mechanical User's Guide (p. 1)* with detailed information about Mechanical's features.
- **Scripting Help**: This option links to the *Scripting in Mechanical Guide*. The guide provides an introduction to writing scripts that automate your routine tasks and information on using the ACT API to access and manipulate objects in the Mechanical tree.
- **What's New?**: This option provides information about the new Mechanical features at the current release.

Graphics Toolbar

The **Graphics** toolbar sets the selection/manipulation mode for the cursor in the **Geometry** window. The toolbar also provides commands for modifying a selection or for modifying the viewpoint. You can turn this toolbar on and off using the **Graphics Toolbar** option in the **Manage** option's drop-down menu located in the **Layout** (p. 47) group on the **Home** tab.



You can add or remove options from this toolbar using the Customization Menu shown below. You access this menu using the down-arrow drop-down menu at the far end of the toolbar.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Customization Menu	Option	Description
	Previous/Next	To return to the last view displayed in the Geometry window, click the Previous View button on the toolbar. By continuously clicking you can see the previous views in consecutive order. After displaying previous views in the Geometry window, click the Next View button on the toolbar to scroll forward to the original view.
	Shaded Exteriors and Edges	Displays the model in the Geometry window with shaded exteriors and distinct edges. This option cannot be used in combination with either the Shaded Exterior or Wireframe views.
	Shaded Exteriors	Displays the model in the Geometry window with shaded exteriors only. This option cannot be used in combination with either the Shaded Exterior and Edges or Wireframe views.
	Wireframe	Enable the **Wireframe** display mode. The model displays in the Geometry window with a wireframe display rather than a shaded (recommended for seeing gaps in surface bodies) display. The Wireframe option not only applies to geometry, mesh, or named selections displayed as a mesh, but extends to probes, results, and variable loads to enable a better understanding of regions of interest. When set, just the exterior faces of the meshed models are shown, not the interior elements. Selecting any of the edges options (p. 68) on contour results automatically closes Wireframe mode. **Note:** When active, green scoping is not drawn on probes. Also, elements are shown on probes and results, whereas the outline of the mesh is shown on isoline contour results.
	Show Mesh	Display the model's mesh. Enabling this option displays the model's mesh regardless of the selected tree object. When enabled, to make sure that Annotations display properly, also turn on Wireframe mode. See Note below.
	Rotate	Activates the model Rotate feature. Free

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Customization Menu	Option	Description
		rotation is the default behavior. See the rotation topic in the Manipulating the Model in the Geometry Window section for additional options associated with this selection.
	Pan	Reposition your model laterally in the Geometry window.
	Zoom	Displays a closer view of the body by dragging the mouse cursor vertically toward the top of the Geometry window, or displays a more distant view of the body by dragging the mouse cursor vertically toward the bottom of the Geometry window. **Note:** If you zoom in on a model at an extreme factor (over 10^5), you may see the graphical display of geometry edges or mesh element edges appear distorted. These display distortions are due to the limitations of the computer's architecture (precision of storing data) and the display screen resolution.
	Zoom To Fit	Fit the entire model in the Geometry window.
	Zoom To Selection	Zoom in on the currently selected item in the Geometry window.
	Toggle Magnifier Window On/Off	Displays a Magnifier Window, which is a shaded box that functions as a magnifying glass, enabling you to zoom in on portions of the model. When you toggle the Magnifier Window on, you can: • *Pan* the Magnifier Window across the model by holding down the *left* mouse button and dragging the mouse. • *Increase the zoom* of the Magnifier Window by adjusting the *mouse wheel*, or by holding down the *middle* mouse button and dragging the mouse upward. • *Recenter* or *resize* the Magnifier Window using a *right* mouse button click and choosing an option from the context

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Customization Menu	Option	Description
		menu. Recenter the window by choosing **Reset Magnifier**. Resizing options include **Small Magnifier**, **Medium Magnifier**, and **Large Magnifier** for preset sizes, and **Dynamic Magnifier Size On/Off** for gradual size control accomplished by adjusting the *mouse wheel*. Standard model zooming, rotating, and picking are disabled when you use the Magnifier Window.
	Select	Label only.
	Mode	Display a drop-down menu of options that define how geometry, node, or element selections are made: • **Single Select** • **Box Select** • **Box Volume Select** • **Lasso Select** • **Lasso Volume Select** These options are used in conjunction with the selection filters (p. 261) (**Vertex**, **Edge**, **Face**, **Body**, **Node**, **Element**) **Note:** Selection shortcuts: • When you place your cursor within the **Geometry** window, you can change your selection mode from **Single Select** to **Box Select** by holding the right mouse button and then clicking the left mouse button. • When you place your cursor within the **Geometry** window, and given a generated mesh, and the **Node** or **Element** selection

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Customization Menu	Option	Description
		option is active, holding the right mouse button and then clicking the left mouse button scrolls through the available selection options (single section, box selection, box volume, lasso, lasso volume).
	Smart Select	For **Single Select** mode, use this option to select or highlight vertices, edges, and faces on the model without needing to specify a selection filter (Vertex, Edge, or Face). Use the **[Ctrl]** key or hold down the left-mouse button to select multiple entities. When you make a selection, an icon displays that enables you to select the parent body for the current selection(s). This option supports the **Select All**, **Zoom To Selection**, and Mini Selection Toolbar (p. 264) features. This option does not currently support depth picking (p. 261).
	Vertex	Activate Vertex geometry selection option.
	Edge	Activate Edge geometry selection option.
	Face	Activate Face geometry selection option.
	Body	Activate Body geometry selection option.
	Node	Activate Node mesh selection option.
	Element Face	Activate Element Face mesh selection option.
	Element	Activate Element mesh selection option.
	Hit Point Coordinate	Available only if you are setting a location, for example, a local coordinate system. This option enables the exterior coordinates of the model to display adjacent to the cursor and updates the coordinate display as the cursor is moved across the model. If you click with the cursor on the model, a label displays the coordinates of that location. This feature is functional on faces only. It is not functional on edges or line bodies. **Note:** You can change the default display settings for the font (size, style, color) used with this

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Customization Menu	Option	Description
		feature by changing the settings of the **Probe Tool and Hit Point Coordinate (Windows Only)** preference in the Graphics (p. 228) category of the **Options** dialog.
	Label	Select and drag and drop an annotation label anywhere on the corresponding scoping. Not all objects support this option.
	Highlight Imported Data	For certain data imported through the External Model system, this option enables you to highlight the graphical representation of the data in the **Geometry** window. This option behaves just like you were selecting a row in the Worksheet for the imported data. See the User Interface Options (p. 910) topic in the *Importing Mesh-Based Databases* section for more information.
	Direction	Chooses a direction by selecting a single face, two vertices, or a single edge (enabled only when Direction field in the Details view has focus). See Pointer Modes (p. 258).
	All	Select all geometric entities of your model based on the current selection filter type (vertex, edge, face, or body). **Ctrl**+**A** also performs this action.
	Invert	Automatically select entities of the same type (face, edge, etc.) that are not currently selected. Any selection made before selecting this option is removed.
	Display Graph (p. 197)	Display the content of the **Graph** window in the **Geometry** window.
	Clipboard Information	To help you keep track of what is contained in the Clipboard, once you make selections (or add or change) the Clipboard menu displays the current number of entities contained in the Clipboard, such as 1 Body, as illustrated above. When no selections are contained in the Clipboard, this field contains the text string "Empty." Also note that the status bar displays active Geometry window selections.
	Extend	Adds adjacent faces (or edges) within angle tolerance, to the currently selected face (or edge) set, or adds tangent faces (or edges)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

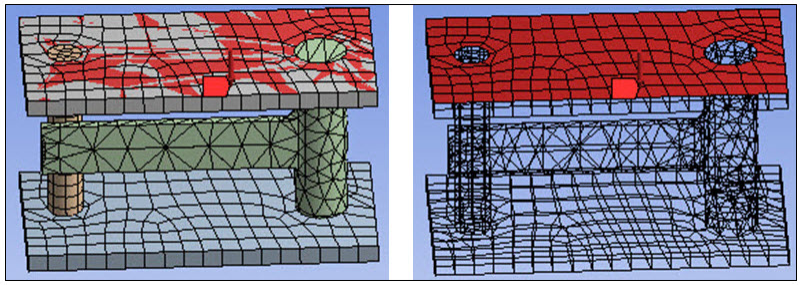
Customization Menu	Option	Description
		within angle tolerance, to the currently selected face (or edge) set. See the Extend Selection Menu (p. 264) topic for additional information about these options.
	Select By	Display a drop-down menu of selection options. See the **Select** group topic (p. 89) in the Selection Tab (p. 88) section for more information.
	Convert	Display a drop-down menu of options to change your currently selected geometric entity or mesh item to a different geometric entity or mesh item. See the **Convert To** group topic (p. 92) in the Selection Tab (p. 88) section for more information.
	Viewports	Split the Geometry window into multiple windows (p. 284).
	Sync Viewports	Synchronize the Viewports display in each window to reorient/move (pan, zoom, rotate) your model in each window simultaneously.
	Front/Back/Right/Left/Top/Bottom	These options reorient the display of your model.
	Look At	
	Accelerated Display (p. 289)	Improve the display time for mesh and result objects.
	Reset Toolbar	Reset the Graphics Toolbar to the default display configuration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Customization Menu	Option	Description

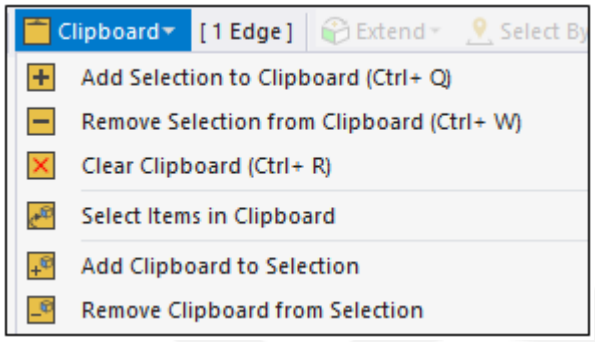
Note:

As illustrated below, annotations may not always display properly when the **Show Mesh** option is activated. Turning on Wireframe mode accurately displays Annotations when **Show Mesh** is selected.



Clipboard Menu

The **Clipboard** menu is a selection feature that enables you to make geometry and mesh selections in the **Geometry** window.



Using the options of this menu, you can create, change, add to, and overwrite the selections of the **Clipboard**. This feature enables you to select only one type of geometric (face/edge/body/vertex) or mesh (node/element) entity at a time. Once you have made your desired selections and included them in the **Clipboard**, these selections are available for use during your analysis. When desired, use the menu option **Select Items in Clipboard** in order to activate your clipboard selections. You can also change active selections using the menu options **Add Clipboard to Selection** or **Remove Clipboard from Selection**.

To help you keep track of what is contained in the **Clipboard**, once you make selections (or add or change) the **Clipboard** menu displays the current number of entities contained in the **Clipboard**, such as [1 Edge], as illustrated above. When no selections are contained in the **Clipboard**, this field

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

contains the text string "Empty." Also note that the status bar (p. 135) displays active **Geometry** window selections.

Note:

This feature does not currently support **Element Face** selection.

The **Clipboard** menu contains the following options:

- **Add Selection to Clipboard**: This option adds your current selection to the existing selection(s) contained in the clipboard. Default hotkey: **Ctrl**+**Q**.
- **Remove Selection from Clipboard**: This option removes your current selection from the existing selection(s) contained in the clipboard. Default hotkey: **Ctrl**+**W**.
- **Clear Clipboard**: This option clears clipboard selections. Default hotkey: **Ctrl**+**R**
- **Select Items in Clipboard**: This option replaces your current selection with the selection contained in the clipboard.
- **Add Clipboard to Selection**: This option adds the selection contained in the clipboard to your currently selected geometry or mesh.
- **Remove Clipboard from Selection**: This option removes the selection contained in the clipboard from your currently selected geometry or mesh.

Outline

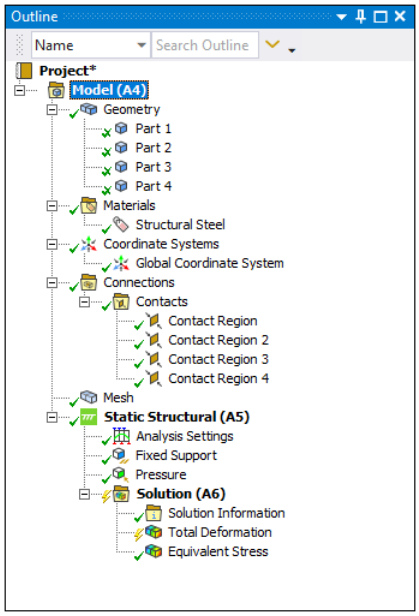
Go to a section topic:

- Outline Pane Overview (p. 105)
- Context Menu Options (p. 106)
- Object Details (p. 108)
- Additional Capabilities (p. 108)

Overview

You use the **Outline** pane, as illustrated below, to set up your model using the "objects" arranged in the tree structure. These objects guide you through the different steps (p. 323) of a simulation. By expanding the objects, you expose the details associated with the object, and you can use the corresponding tools and specification tables to further define your simulation. Refer to the Objects Reference section for a listing and description of all of the objects available in the application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Important:

If your analysis includes an exceptionally large number of objects, the **Outline** content may appear incomplete. The application has a display limitation of 65,536 objects. If the number of objects exceeds this threshold, any additional objects are not displayed.

Note:

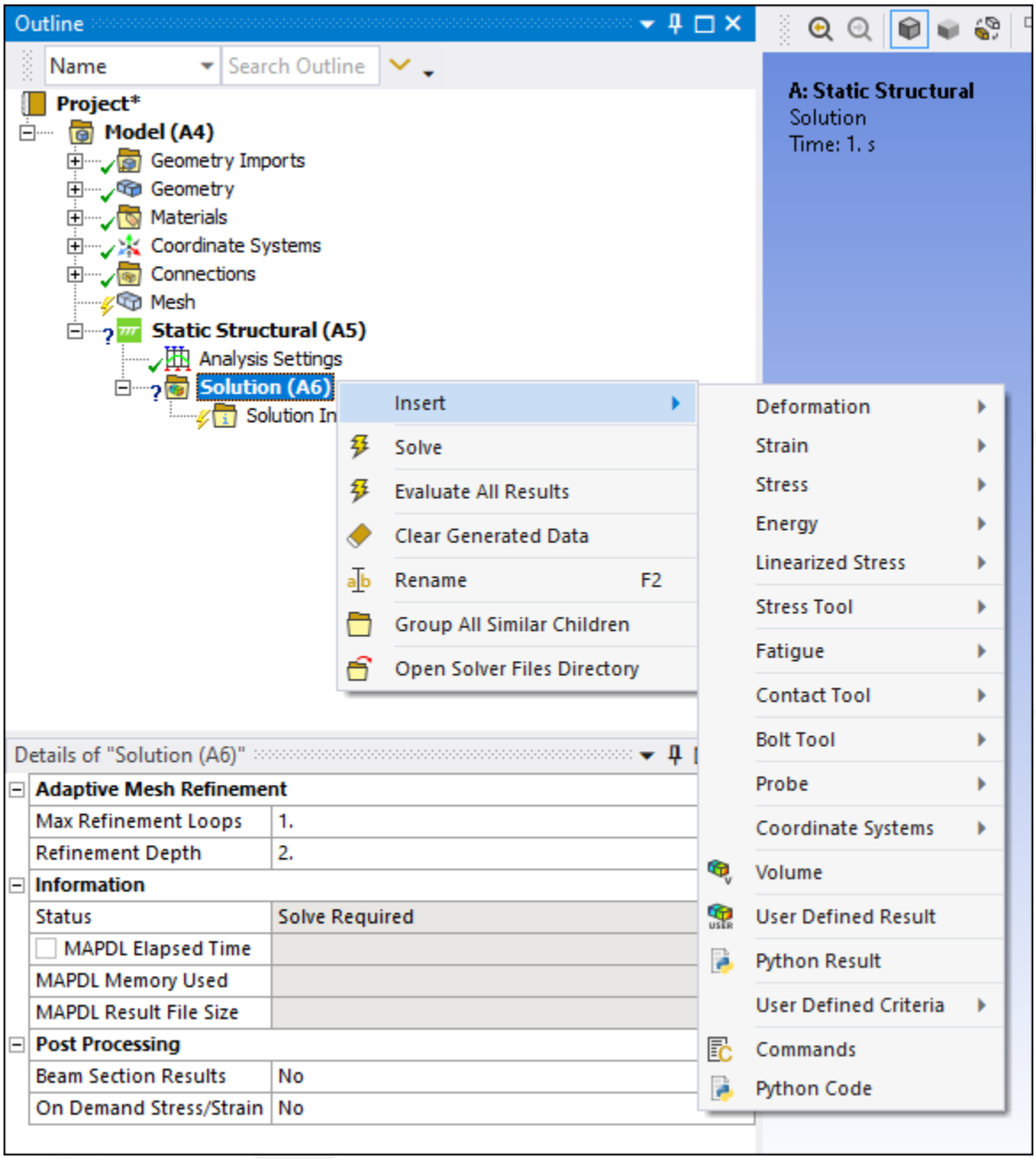
Numbers preceded by a space at the end of an object's name are ignored. This is especially critical when you copy objects or duplicate object branches. For example, if you name two force loads as **Force 1** and **Force 2**, then copy the loads to another analysis environment, the copied loads are automatically renamed **Force** and **Force 2**. However, if you rename the loads as **Force_1** and **Force_2**, the copied loads retain the same names as the two original loads.

Contextual Options

Each **Outline** object provides contextual (right-click) menus, as illustrated below. These menu options are generally based on the selected object. For example, if you select the **Solution** object, most of the menu options are specific to obtaining solution data or further defining the results of the simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Common Context Menu Options

Most of the context menus also provide a variety of common options, including:

- **Insert**: The **Insert** menu is specific to the object. You use the insert menu to add contacts, loads, results, etc.
- **Duplicate**: Duplicate the selected object, including scoping and Details definition.
- **Copy**/**Cut**: Copy or Cut the selected object.
- **Paste**: Paste supported objects (that you copied) within the active Mechanical session.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Copy To Clipboard**/**Paste From Clipboard**: Any object that is scoped to a geometric entity (mesh-based scoping is not supported), and that supports Copy/Cut, also supports copying and pasting to and from the clipboard. These options enable you to paste objects copied to the clipboard from one Mechanical session to another. These clipboard actions can only be performed once. The clipboard is cleared once pasted.
- **Delete**: Delete the selected object.
- **Rename**: Rename the selected object.
- Group (p. 208)/**Ungroup**: Group objects in the Outline as desired.
- Rename Based on Definition (p. 2283): The application automatically renames the object based on its type and scoping.

Tip:

The sections of the Objects Reference provide more information about the context menu options specific to the various objects.

Object Details

Selecting an object in the **Outline** displays attributes and controls for the selected object in the **Details** (p. 119) pane. The **Geometry** window displays your CAD model and, based on the object selected, displays pertinent information about object specifications and how they relate to the displayed geometry.

Additional Capabilities

The following topics present further details related to the **Outline** features.

Understanding the Tree Outline

Correlating Tree Outline Objects with Model Characteristics

Suppressing Objects

Filtering the Tree

Searching the Tree

Understanding the Tree Outline

The tree Outline uses the following conventions:

- Icons appear to the left of objects in the tree. Their intent is to provide a quick visual reference to the identity of the object. For example, icons for part and body objects (within the **Geometry** object folder) can help distinguish solid, surface and line bodies.
- A ⊞ symbol to the left of an item's icon indicates that it contains associated subitems. Click to expand the item and display its contents.
- To collapse all expanded items at once, double-click the Project name at the top of the tree.
- Drag-and-drop function to move and copy objects.

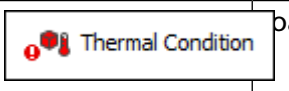
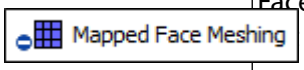
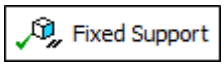
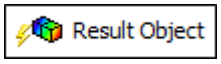
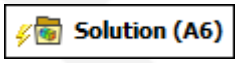
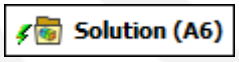
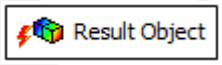
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- To delete a tree object from the Outline (p. 105), right-click the object and select **Delete**. A confirmation dialog asks if you want to delete the object.
- Filter tree contents and expand the tree by setting a filter (p. 114) and then clicking the **Expand on Refresh** button.

Status Symbols

As described below, a small status icon displays to the left of the object icon in the Outline (p. 105).

Status Symbol Name	Symbol	Example
Underdefined		A load requires a nonzero magnitude.
Error		bad attachments may break during an Update.
Mapped Face or Match Control Failure		Face could not be mapped meshed, or mesh of face pair could be matched.
Ok		The object is defined properly and/or any specific action on the object is successful.
Needs to be Updated		Equivalent to "Ready to Answer!"
Hidden		A body or part is hidden.
Meshed		The symbol appears for a meshed body within the **Geometry** folder, or for a multibody part whose child bodies are all meshed.
Suppress		An object is suppressed.
Solve	  	*Yellow lightning bolt*: Item has not yet been solved. *Green lightning bolt*: Solve in progress. *Green check mark*: Successful solution. *Red lightning bolt*: Failed solution. An overlaid pause icon indicates the solution could resume with the use of restart points. *Green down arrow*: Successful background solution (p. 1803) ready for download. *Red down arrow*: Failed background solution (p. 1803) ready for download.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

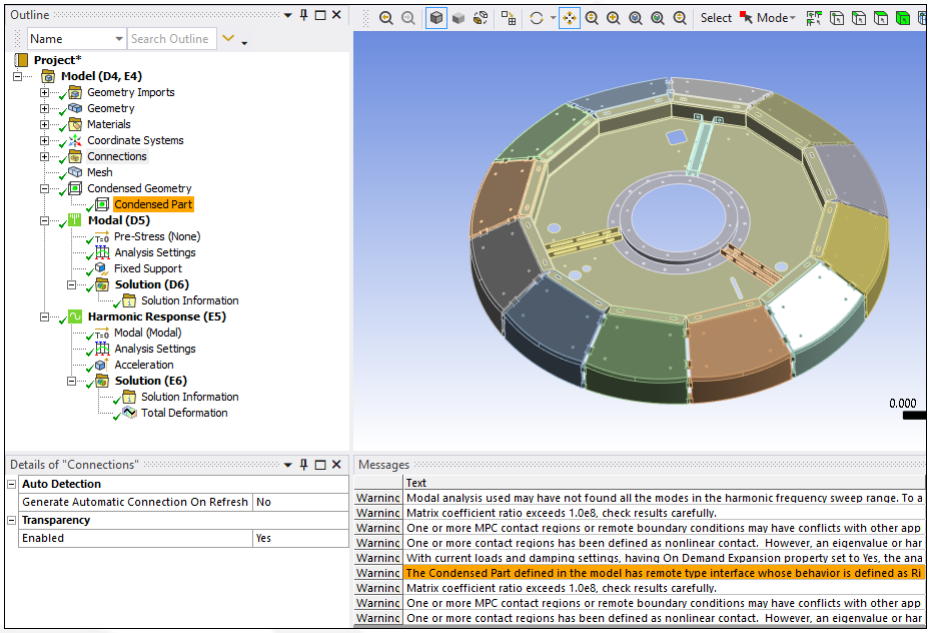
Status Symbol Name	Symbol	Example
		Yellow down arrow: You can download obsolete results from a background solution.

Note:

The state of an environment folder can be similar to the state of a **Solution** folder. The solution state can indicate either solved (check mark) or not solved (lightning bolt) depending on whether or not an input file has been generated.

Status Coloring

In addition to the status icons, you may see objects highlighted in orange to indicate that there is a potential problem related to the object or to a child-object. Objects highlighted in orange have a corresponding message in the Messages (p. 200) window. You can turn this feature off using the **Options** (p. 217) dialog box (see the Messages (p. 236) category).



Also see the Outline (p. 105) section.

Correlating Tree Outline Objects with Model Characteristics

The **Go To** feature provides you with instant visual correlation of objects in the tree as they relate to various characteristics of the model displayed in the **Geometry** window.

To activate this feature, right-click anywhere in the **Geometry** window, choose **Go To**, then choose an option in the context menu. In some cases (see table below), you must select geometry prior to choosing the **Go To** feature. The resulting objects that match the correlation are highlighted in the tree and the corresponding geometry is highlighted on the model. **Go To** options are also available from the Selection (p. 88) tab. For example, you can identify contact regions in the tree that are associated with a particular body by selecting the geometry of interest and choosing the **Contacts for**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Selected Bodies option. The contact region objects associated with the body of the selected items will be highlighted in the tree and the contact region geometry will be displayed on the model.

Several options are filtered and display only if specific conditions exist within your analysis. The **Go To** options are presented in the following table along with their description and the conditions under which they appear in the context menu.

Go To Option	Description / Application	Required Conditions for Option to Appear
Corresponding Bodies in Tree	Identifies body objects in the tree that correspond to selections in the **Geometry** window.	At least one vertex, edge, face, or body is selected.
Hidden Bodies in Tree	Identifies body objects in the tree that correspond to hidden bodies in the **Geometry** window.	At least one body is hidden.
Suppressed Bodies in Tree	Identifies body objects in the tree that correspond to suppressed bodies in the **Geometry** window.	At least one body is suppressed.
Bodies Without Contacts in Tree	Identifies bodies that are not in contact with any other bodies. When you are working with complex assemblies of more than one body, it is helpful to find bodies that are not designated to be in contact with any other bodies, as they generally cause problems for a solution because they are prone to rigid body movements.	More than one body in an assembly.
Parts Without Contacts in Tree	Identifies parts that are not in contact with any other parts. When you are working with complex assemblies of more than one multibody part, it is helpful to find parts that are not designated to be in contact with any other parts. For example, this is useful when dealing with shell models which can have parts that include many bodies each. Using this feature is preferred over using the **Bodies Without Contact in Tree** option when working	More than one part in an assembly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Go To Option	Description / Application	Required Conditions for Option to Appear
	with multibody parts mainly because contact is not a typical requirement for bodies within a part. Such bodies are usually connected by shared nodes at the time of meshing.	
Contact Sizing Common to Selected Bodies	Identifies Contact Sizing controls that exist between the selected bodies. This option may be useful when you want to delete common contact sizing controls.	Two bodies are selected.
Contacts for Selected Bodies	Identifies contact region objects in the tree that are associated with selected bodies.	At least one vertex, edge, face, or body is selected.
Contacts Common to Selected Bodies	Identifies contact region objects in the tree that are shared among selected bodies.	
Joints for Selected Bodies	Identifies joint objects in the tree that are associated with selected bodies.	
Joints Common to Selected Bodies	Identifies joint objects in the tree that are shared among selected bodies.	
Springs for Selected Bodies	Identifies spring objects in the tree that are associated with selected bodies.	
Mesh Controls for Selected Bodies	Identifies mesh control objects in the tree that are associated with selected bodies.	
Mesh Connections for Selected Bodies	Highlights **Mesh Connection** objects in the tree that are associated with the selection.	At least one vertex, edge, face, or body is selected and at least one mesh connection exists.
Mesh Connections Common to Selected Bodies	Highlights **Mesh Connection** objects in the tree that are shared among selected bodies.	At least one vertex, edge, face, or body is selected.
Field Bodies in Tree	Identifies enclosure objects in the tree that are associated with selected bodies.	At least one body is an enclosure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Go To Option	Description / Application	Required Conditions for Option to Appear
Bodies With One Element Through the Thickness	Identifies bodies in the tree with one element in at least two directions (through the thickness). This situation can produce invalid results when used with reduced integration. See At Least One Body Has Been Found to Have Only 1 Element (p. 2364) in the troubleshooting section for details.	At least one body with one element in at least two directions (through the thickness).
Thicknesses for Selected Faces	Identifies objects with defined thicknesses in the tree that are associated with selected faces.	At least one face with defined thickness is selected.
Body Interactions for Selected Bodies	Identifies body interaction objects in the tree that are associated with selected bodies.	At least one body interaction is defined and at least on body is selected.
Body Interactions Common to Selected Bodies	Identifies body interaction objects in the tree that are shared with selected bodies.	At least one body interaction is defined and at least on body is selected.
Named Selections Containing to Selection	Highlight all Named Selection objects in the Outline that contain any of the currently selected geometric or mesh entities.	The selection filter (face, edge, element, etc.) must match the definition of the Named Selection. That is, Face selections are only applicable to face-based Named Selections and so on.
Named Selections Common to Selection	Highlight all Named Selection objects in the Outline that include all of the currently selected geometric or mesh entities.	The selection filter (face, edge, element, etc.) must match the definition of the Named Selection. That is, Face selections are only applicable to face-based Named Selections and so on.

Suppressing Objects

Suppression Behaviors

Mechanical provides an option to suppress tree Outline (p. 105) objects. This feature enables you to remove an individual object or multiple objects from the analysis. Any corresponding (scoped) objects are also affected. For example, when you suppress a part, the application automatically removes the part from the display, under-defines any object that is scoped to the part, and clears data from all solution objects.

This can be useful when you are applying different types of loading conditions. You can quickly include and/or remove conditions through suppression. Not all tree objects provide the suppression capability.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For child objects of the **Geometry** and the **Environment** objects: the application removes suppressed objects from the solution process. You can also use the Grouping feature (p. 208) on the **Geometry** object to select and suppress (and unsuppress) one or more objects.

For the **Solution** object: the application clears result data for suppressed objects and the object is not included during any subsequent solution processing. You can use this feature to remove under-defined result objects and/or perform comparisons for different result types.

How to Suppress or Unsuppress Objects

If available, set the **Suppressed** option in the Details view (p. 119) to **Yes**. Conversely, you can unsuppress items by setting the **Suppressed** option to **No**.

You can also suppress/unsuppress these items through context menu options available via a right mouse button click. Included is the context menu option **Invert Suppressed Body Set**, which enables you to reverse the suppression state of all bodies (unsuppressed bodies become suppressed and suppressed bodies become unsuppressed). You can suppress the bodies in a named selection using either the context menu options mentioned above, or through the Named Selection (p. 89) group.

Another way to suppress a body is by selecting it in the Geometry window, then using a right mouse button click in the Geometry window and choosing **Suppress Body** in the context menu. Conversely, the **Unsuppress All Bodies** option is available for unsuppressing bodies. Options are also available in this menu for hiding or showing bodies. Hiding a body only removes the body from the display. A hidden body is still active in the analysis.

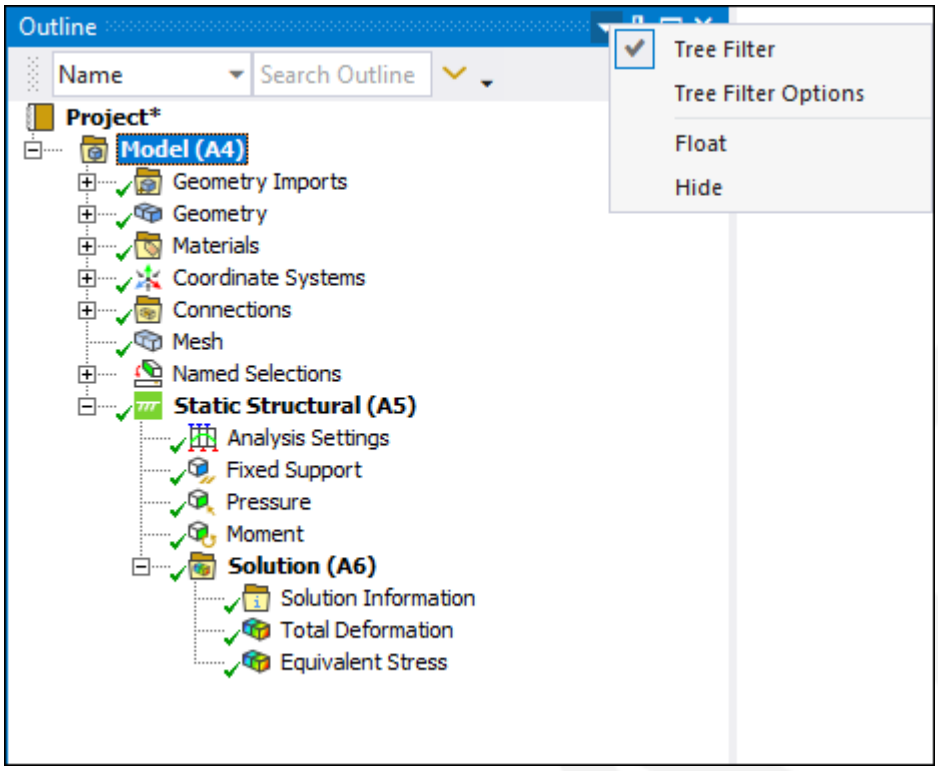
Filtering the Tree

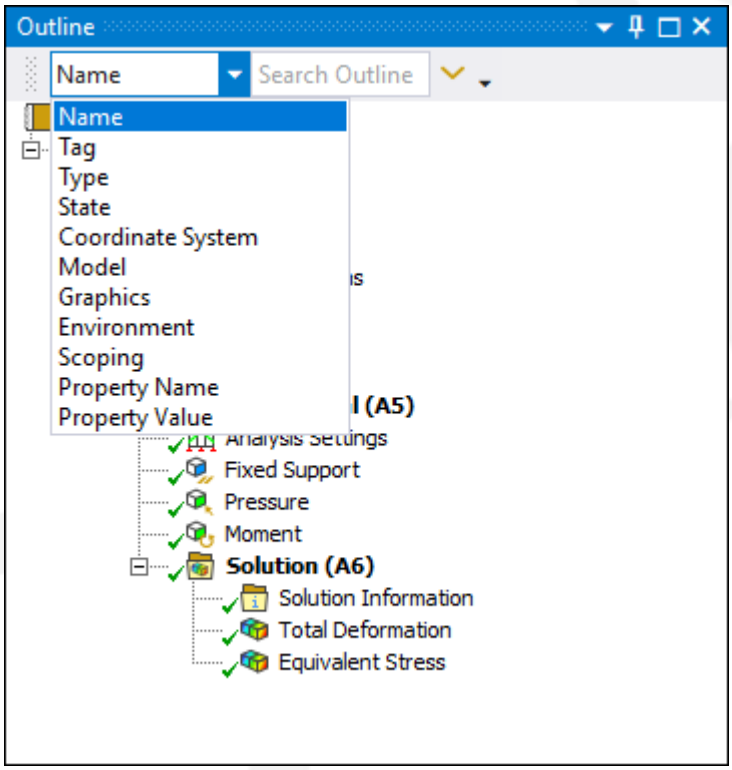
At the top of the **Outline** pane is the Tree Filter option. As illustrated below, this option has a search feature as well as associated filter options. The search feature filters objects based on criteria. The filter options can be hidden by right-clicking on a non-interactive region of the **Outline** pane, on the pane's title bar, or using the drop-down (arrow) menu on the pane's title bar.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Filtering options include the following:

Filter Type	Description
Name	Filters the Outline for one or more specified search terms.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Filter Type	Description
Tag	Filters for Outline objects marked with one or more specified tag names. See the Tagging Objects (p. 2349) section.
Type	Provides a drop-down list of objects for which you can filter. The options include: • **All**: This default option displays all Outline objects and requires you to make a selection to initiate the filter process. • **Boundary Conditions** • **Commands** • **Comments** • **Connections** • **Figures** • **Images** • **Results**
State	Provides a drop-down list of filters for a selected state. State options include: • **All** • **Suppressed** • **Not Licensed** • **Underdefined** • **Ignored**
Coordinate System	Provides a drop-down list of all coordinate systems in the Outline. You can choose to filter **All** coordinate system objects or you can select an individual coordinate system object. The filter displays all objects within the Outline that employ the individually selected coordinate system. **Note:** *All* coordinate systems display in the filter. There are cases where an object does not have a coordinate system property in its Details view, but it does have an associated coordinate system as a requirement. As a result, it may appear as though an unaccounted for coordinate system is present. This is especially true for the Global Coordinate System.
Model	Provides a drop-down list of all source models (External Model, Mechanical Model, etc., including the source model's cell ID) that create an assembly. You can choose to filter **All** source models or you can select a specific model. The current system is

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Filter Type	Description
	the first item. This feature is only supported for models assembled in Ansys Mechanical 2024 R1 or greater.
Graphics	The default option, **All**, displays all Outline objects. The **Visible Bodies** option filters the Outline so that only visible bodies and objects *associated* with any visible body display. Objects independent of geometry, that is, those that do not require scoping, are always shown, such as Analysis Settings. The default setting for this selection can be modified using the **Options** dialog box. See the Specifying Options (p. 217) section of the Help under **Visibility** (p. 243).
Environment	For an analysis with multiple environments, this selection provides a drop-down list of all of the system's environments. You can choose to filter **All** (default) environments or you can select a specific environment. Once selected, all objects specific to the environment are displayed in the tree.
Scoping	The default option, **All**, displays all Outline objects. The **Partial** option filters the tree to only show objects that have partial scoping. These objects require your attention because scoping was lost due to upstream geometry updates.
Property Name	Filters Outline objects based on the name of a property, such as Magnitude, and displays objects containing the property name.
Property Value	Filters Outline objects based on the specified property value searched and displays objects containing the property value. You can also perform special query such as, "Property Name = Property Value." For example: "Formulation = MPC," or "Pinball Radius = 2," or "Solver Type = Direct."

Note:

Performing a search for an object that does not exist in the tree results in all objects being displayed.

Tree Filter Options

The filter options perform the following actions.

Refresh Search

Refreshes the search criteria that you have specified following changes to the environment.

Clear Search

Clear the filter and returns the tree to the full view.

Remove

Turned off by default. Depressing this button turns the feature on and off. When active, it removes the objects in question from the tree display.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Expand on Refresh

Selecting this option enables filtering updates to automatically display. The default setting is off. Select the button to turn the feature on and off. This option can be configured so that the filter will be automatically applied when bodies are hidden or shown. See the Specifying Options (p. 217) section of the Help under **Visibility** (p. 243).

Hide Folders

Selecting this option hides all grouping objects present in the tree. The default setting is off. Select the button to turn the feature on and off. If active, the grouping feature (p. 208) is unavailable and the tree displays in the default view, that is, no grouping.

Sort Ascending

Selecting this option sorts tree objects in alphanumeric order. This excludes most parent objects such as **Geometry**, **Coordinate Systems**, **Connections**, and **Named Selections** objects, however, child objects are sorted. For example, selecting this option would sort all contact regions, user- or system defined named selections, loading conditions, results, etc., in alphanumeric precedence.

Using the Filter Feature

To filter the Outline:

1. Select a filter type:
   - **Name**
   - **Tag**
   - **Type**
   - **State**
   - **Coordinate System**
   - **Model**
   - **Graphics**
   - **Environment**
   - **Scoping**
   - **Property Name**
   - **Property Value**
2. For the **Name**, **Tag**, **Property Name**, or **Property Value** options, you enter one or more search terms. For all other filters, select an option from the drop-down list to further specify your inquiry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

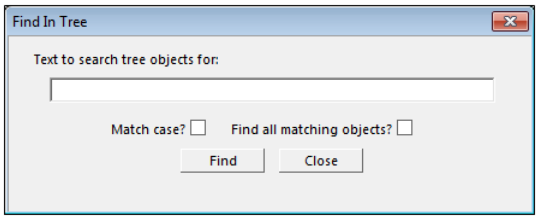
Published: 2023-10-23

3. Click the **Refresh Search** button (or press **Enter**) to execute your search. If you want to eliminate content from the tree, click the **Remove** button and then click **Refresh Search** to remove the requested objects.

4. When searching, the tree displays only objects matching your search criteria. If you enter multiple search terms, the tree shows only objects matching all of the specified terms. When removing objects, the requested objects do not display.

Searching the Tree

The **Find In Tree** option provided through the **Edit Menu** (or **F3** key) enables you to search tree objects whose names match your search criteria. The search tool is illustrated below.



Once you make an entry and click the **Find** button, the application highlights the first instance of the search string. The application will cycle through (highlight) each instance of the string as you continue to press the **Find** button. Furthermore, this cycle is sensitive to the order in which objects were generated, created, or renamed.

Search options may be case sensitive and you can search tree objects for all instances of a name/text-based string. The application highlights all objects in your specified string when you select the **Find all matching objects** option.

Details Pane

You use the **Details** pane to specify the attributes of an object selected in the Outline (p. 105). The pane provides categorized groups of properties for the selected object. You define the various properties in different ways. Some require you to make a selection in the **Geometry** window, others require a value, and so on. This is the primary entry point to properly define the environmental conditions of your simulation.

Here is an example of the **Details** pane for the Geometry object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Geometry"

Definition	
Source	C:\Users\
Type	Parasolid
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	7.e-002 m
Length Y	7.e-002 m
Length Z	6.4e-002 m
Properties	
Volume	8.285e-005 m^3
Mass	0.65037 kg
Scale Factor Value	1.
Statistics	
Bodies	4
Active Bodies	4
Nodes	0
Elements	0
Mesh Metric	None
Update Options	
Assign Default Material	No
Basic Geometry Options	
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No
Parameters	Independent
Parameter Key	ANS;DS
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Use Associativity	Yes
Coordinate Systems	No
Reader Mode Saves Updated File	No
Use Instances	Yes
Smart CAD Update	Yes
Compare Parts On Update	No
Analysis Type	3-D
Mixed Import Resolution	None
Clean Bodies On Import	No
Stitch Surfaces On Import	None
Mixed Import Resolution	None
Decompose Disjoint Geometry	Yes
Enclosure and Symmetry Processing	Yes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For more information, see the topic below:

Features (p. 121)
Pausing Updates (p. 121)
Header (p. 122)
Categories (p. 122)
Undefined or Invalid Fields (p. 122)
Decision-Based (p. 123)
Text Entry (p. 124)
Numeric Values (p. 126)
Ranges (p. 126)
Increments (p. 127)
Scoping (p. 128)
Parameters (p. 128)
Options (p. 128)

Features

The **Details view** enables you to enter information that is specific to each section of the Outline. It automatically displays details for branches such as Geometry, Model, Connections, etc. Features of the Details pane include:

- Collapsible bold headings.
- Dynamic cell background color change.
- Row selection/activation.
- Auto-sizing/scrolling.
- Sliders for range selection.
- Combo boxes for boolean or list selection.
- Buttons to display dialog box (browse, color picker, etc.).
- Apply / Cancel buttons for geometry selection.
- Obsolete items are highlighted in red.

Pausing Updates

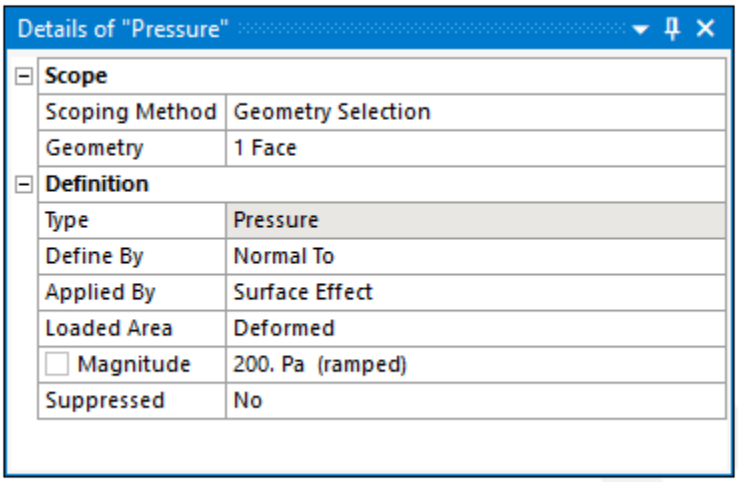
Certain actions instruct the application to update the content of the Details view pane. Depending upon the action, this can take an undesirable amount of time. The **Options** (p. 217) dialog preference **Pause View Update** enables you to halt Details view updates. This may be desirable until you have completed all desired actions when configuring an analysis. The options of this preference are **Yes** and **No** (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When active, the title of the pane displays the message “Details view Update Paused.” You must change the setting in the **Options** dialog to deactivate the setting. The setting for this preference carries over to future Mechanical session if not changed.

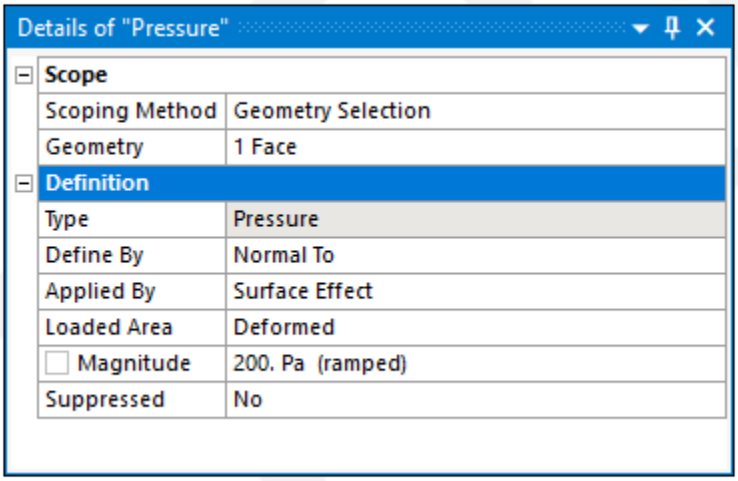
Header

The heading of the pane identifies the name of the selected object (contained in parenthesis), such as the "Pressure" load shown here. For certain objects, the heading may also display the type of object currently selected in addition to the object name.



Categories

The category label in the pane organizes associated properties. The **Definition** category, a common object category, is highlighted in the following example. Double-click a category's name to expand or collapse the category.

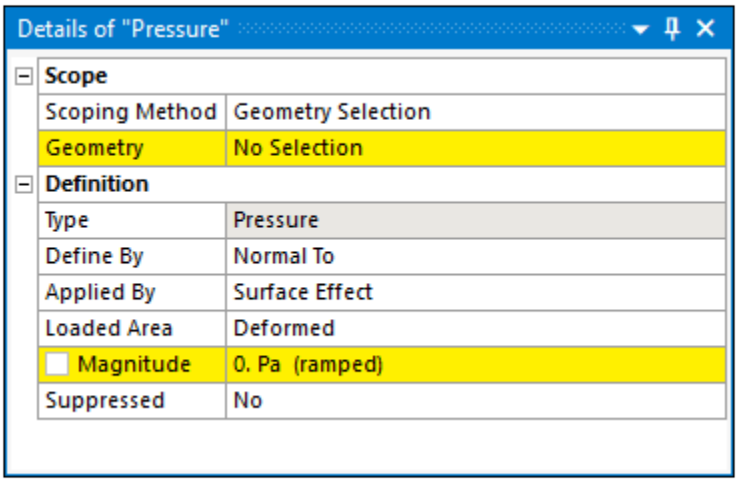


Undefined or Invalid Fields

Fields whose value is undefined or invalid are highlighted in yellow:

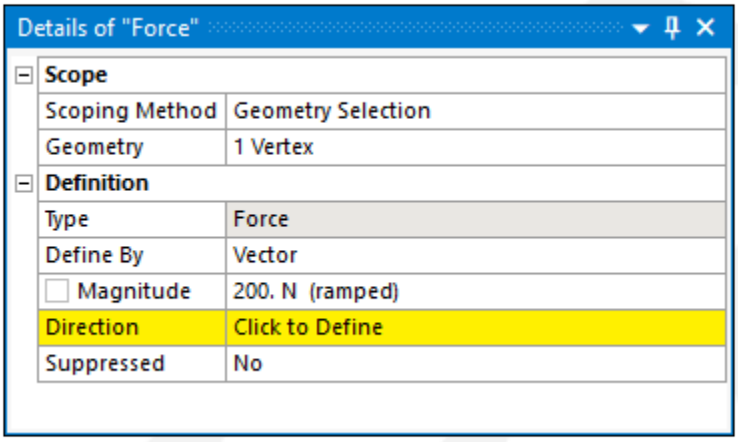
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

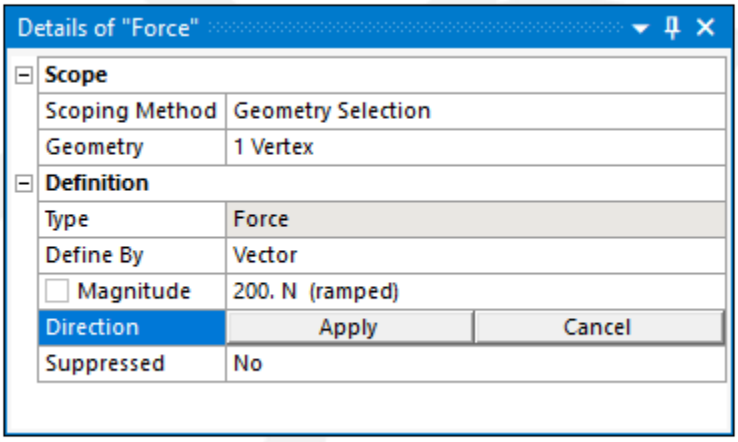
Published: 2023-10-23



Decision-Based

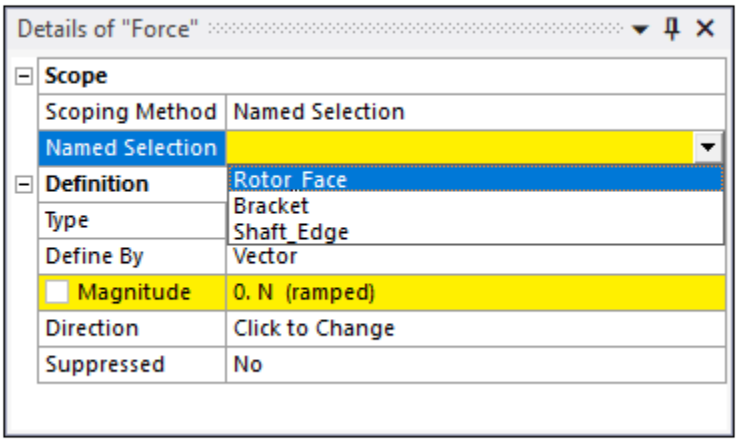
Some properties require a selection in order to specify an attribute, such as the **Direction** property shown here. This property requires additional specification actions that you then "Apply."



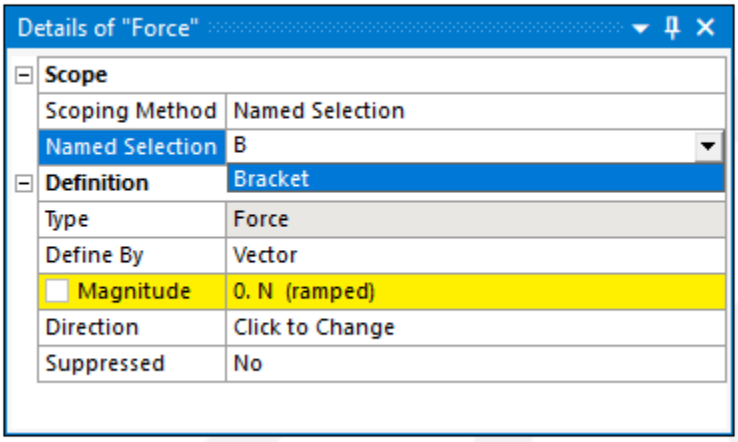


The properties associated with decision-based fields often provide a drop-down list of options, such as the list of Named Selections shown here.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



You can search these option-based fields. For example and as illustrated below, an "B" was entered in the field and the application filtered all of the options that included that letter. This search feature is not case sensitive. And, you can change disable this capability (turned on by default) under the **UI Controls** category of the Miscellaneous Options (p. 235).



Note:

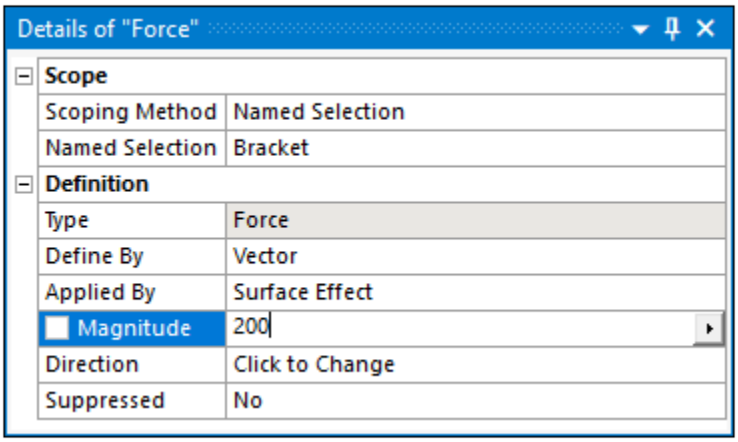
The left column always adjusts to fit the widest visible label. This provides maximum space for editable fields in the right column. You can adjust the width of the columns by dragging the separator between them.

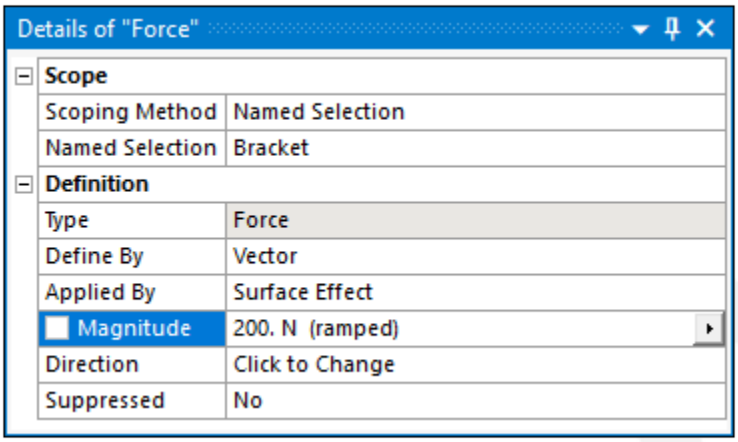
Text Entry

Text entry fields may be qualified as strings, numbers, or integers. Units are automatically removed and replaced to facilitate editing:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Inappropriate characters are discarded (for example, typing a Z in an integer field). A numeric field cannot be entered if it contains an invalid value. It is returned to its previous value.

Separator Clarification

Some languages use "separators" within numerical values whose meanings may vary across different languages. For example, in English the comma separator [,] indicates "thousand" ("2,300" implies "two thousand three hundred"), but in German the comma separator indicates "decimal" ("2,300" implies "two and three tenths", equivalent to "2.300" in English). To avoid misinterpretation of numerical values you enter that include separators, you are asked to confirm such entries before they are accepted.

For example, in English, if you enter "2,300", you receive a message stating the following:

"Entered value is 2,300. Do you want to accept the correction proposed below?

2300

To accept the correction, click **Yes**.

To close this message and correct the number yourself, click **No**.

Note:

If an invalid entry is detected, an attempt is made to interpret the entry as numerical and you receive the message mentioned above if an alternate value is found. If an invalid value

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

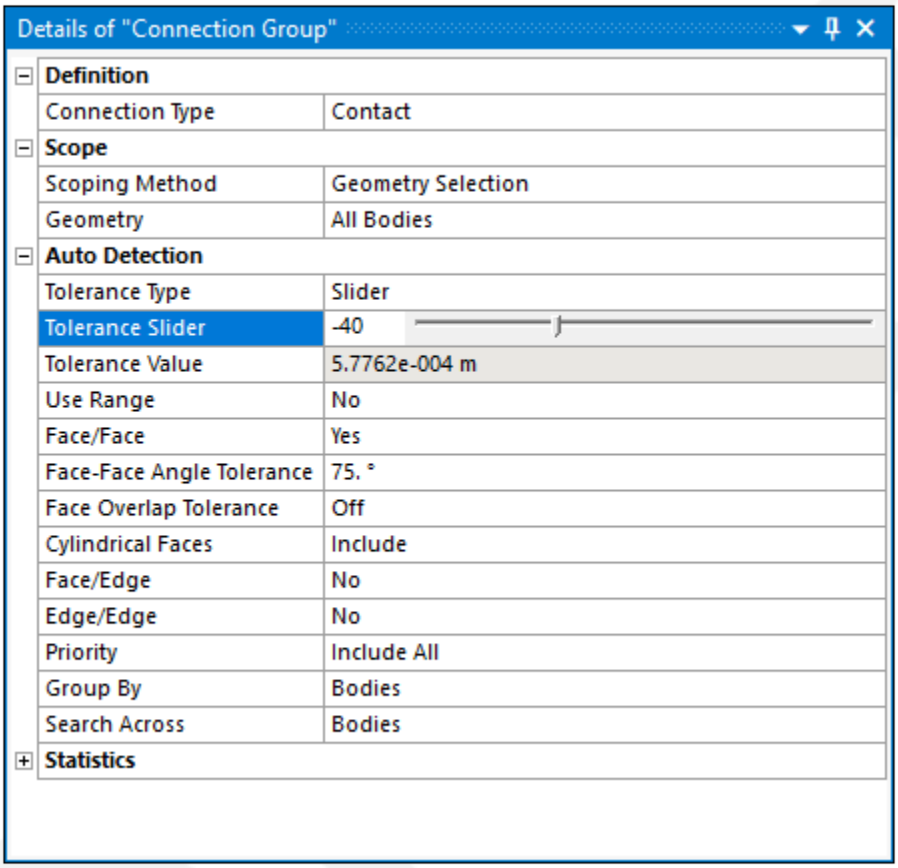
is entered, for example "a1.3.4", and no numerical alternative is found, the entry is rejected and the previous value is re-displayed.

Numeric Values

You can enter numeric expressions in the form of a constant value or expression, tabular data, or a function. See Defining Boundary Condition Magnitude (p. 1771) for further information.

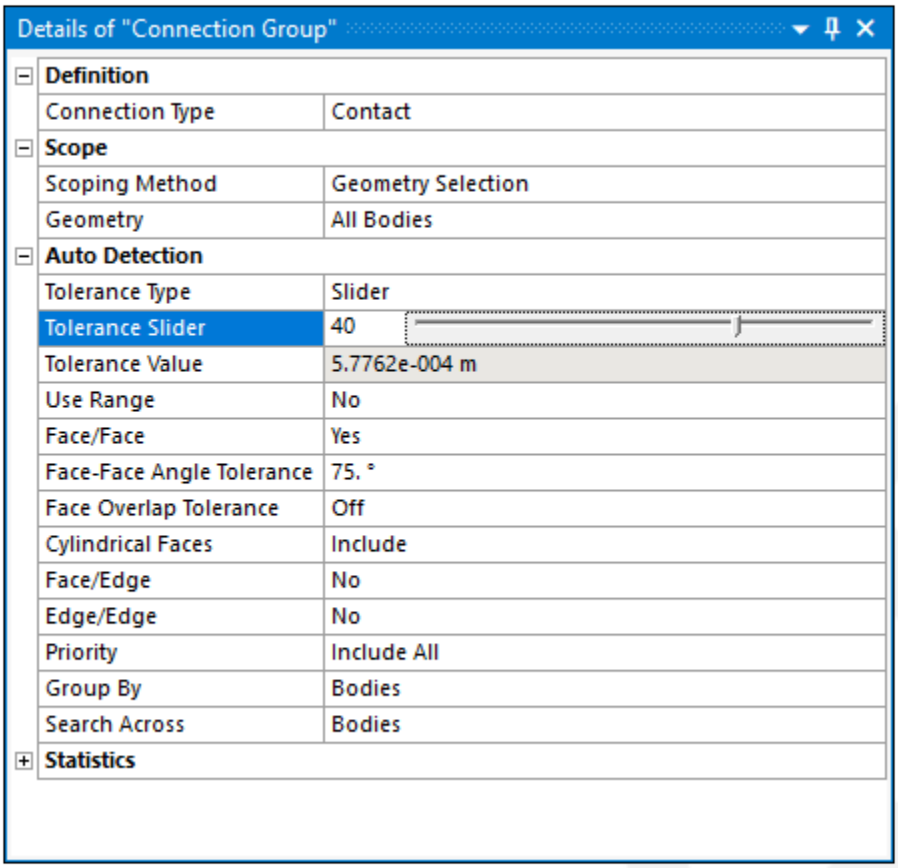
Ranges

If a numeric field has a range, a slider appears to the right of the current value. If the value changes, the slider moves. And when the slider moves, the value updates.



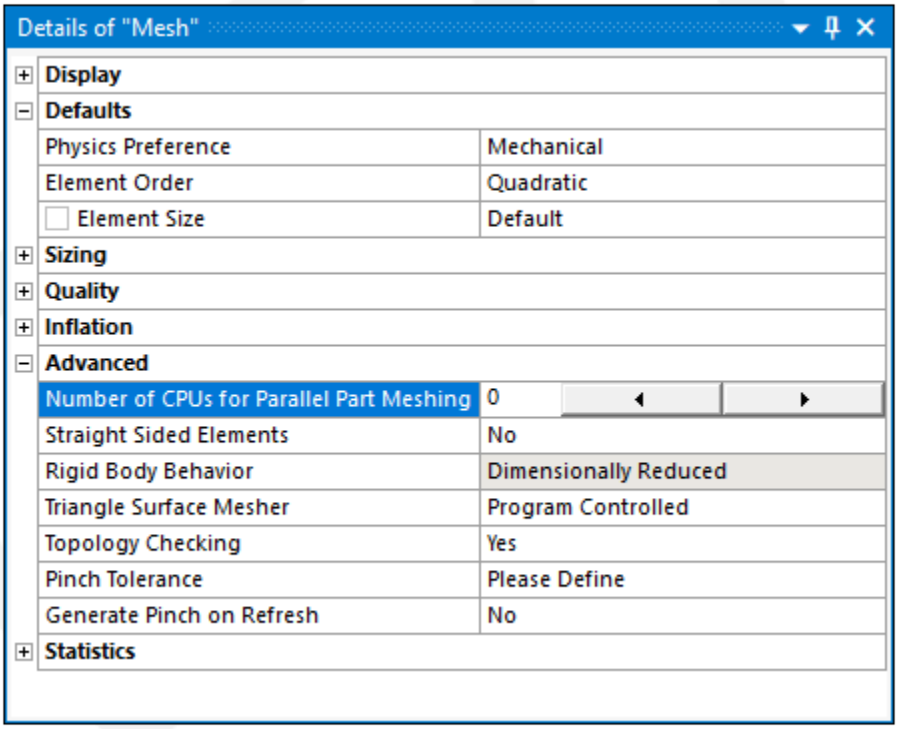
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Increments

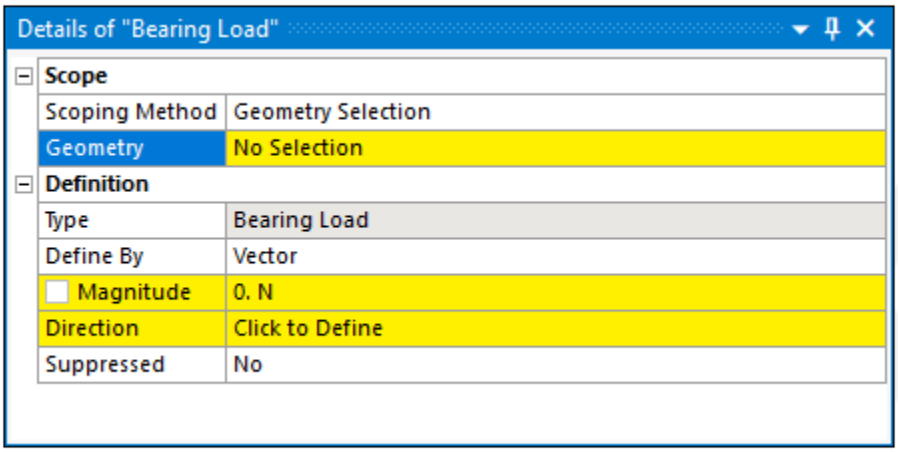
If a numeric field has an increment, a horizontal up/down control appears to the right of the current value. The arrow buttons enable you to increase/decrease the property's value.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Scoping

The **Scoping Method** and its associated properties, usually **Geometry** and **Named Selection**, may have selection requirements. For example, the application only supports face scoping for a Bearing Load (p. 1561). If you try to scope the load to any other geometric entity, or a Named Selection that is **not** face-based, the application presents an invalid state.



Parameters

Variables that you can parameterize display in the interface with a check box. Clicking the box activates the property as a parameter. For more information, see Parameterizing a Variable (p. 128).

Options

Option fields allow you to select one item from a short list. Options work the same way as Decision-Based (p. 123), but don't affect subsequent fields. Options are also used for boolean choices (true/false, yes/no, enabled/disabled, fixed/free, etc.) Double-clicking an option automatically selects the next item down the list.

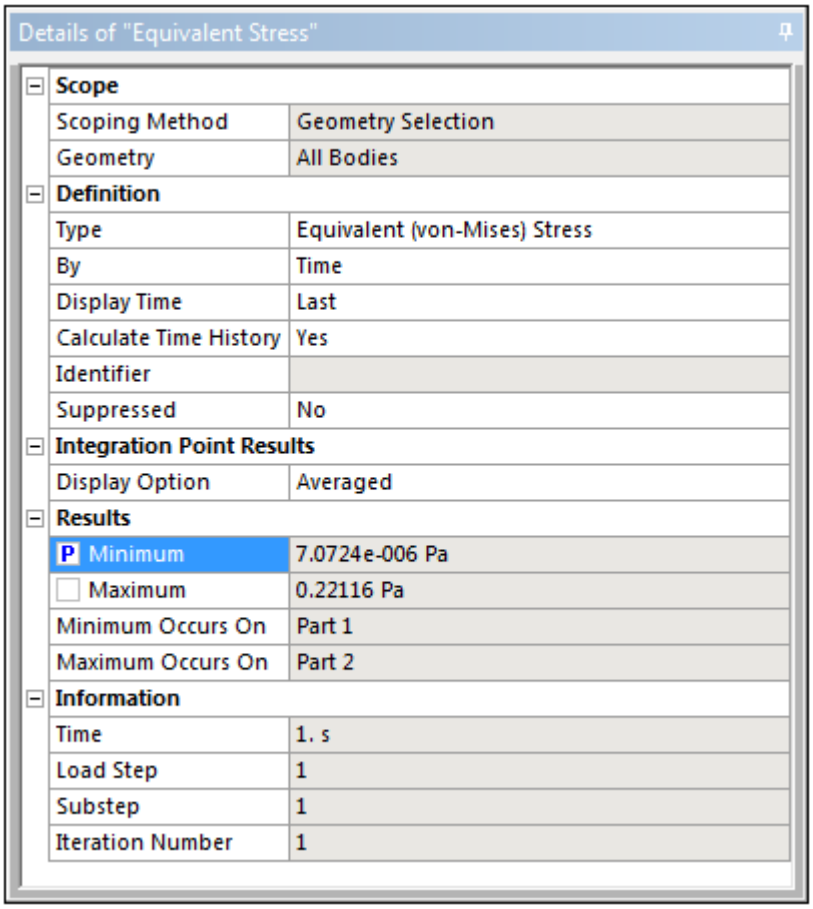
Selecting an option followed by an ellipsis causes an immediate action.

Parameterizing a Variable

Variables that you can parameterize display in the interface with a check box. Clicking the check box displays a blue capital "**P**", as illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



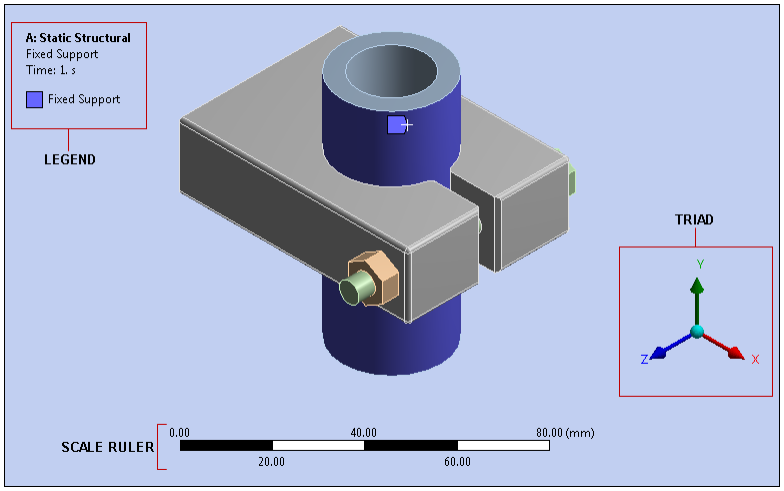
The boxes that appear in the Mechanical application apply only to the Parameter Workspace. Checking or clearing these boxes will have no effect on which CAD parameters are transferred to Design Exploration.

For more information, see Setting Parameters (p. 2337).

Geometry Window

The **Geometry** window/tab displays a 3D graphical representation of your model. All view manipulation, geometry selection, and graphics display of a model occurs in this window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The default components of the window, as illustrated above, include:

Legend

The information presented by the Legend varies, but in general, it provides information about the currently selected object or objects as well as the analysis type, and includes annotations. For the above example, a **Fixed Support** object is selected. The face to which the support is applied is shown via color coding.

You can reposition the legend by dragging and dropping it to a location in the window.

Scale Ruler

Based on the selected unit of measure, the ruler provides a reference for your geometry.

Triad

Shows the global XYZ coordinate triad. The axes are color-coded as follows:

- Red: X
- Green: Y
- Blue: Z

The Triad enables you to reorient the position of your model based on a desired axis as well as reset the isometric view (light blue ball). If you move your cursor around the triad, you will see an arrow appear that shows the direction that corresponds to the position of your cursor (+x, -x, +y, -y, +z, -z). If you click the arrow, it changes your view so that the axis indicated by the arrow is facing outward.

You can turn these options on and off using selections **Show** drop-down menu on the **Display** (p. 87) group on the **Display** tab.

Full Screen Mode

Mechanical offers a full screen mode so that you can view and/or present the results of your analysis by maximizing the **Geometry** window and hiding all other interface elements (by default). You activate

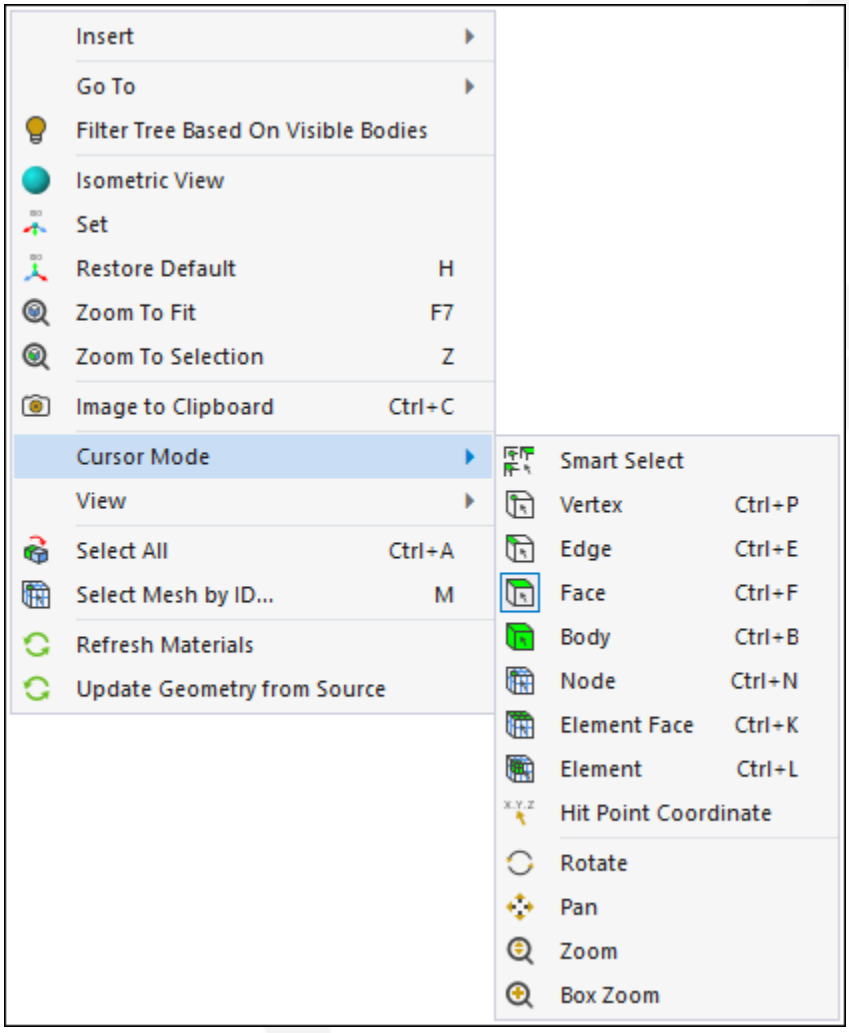
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

full screen mode by selecting the **Full Screen** option from the **Layout** (p. 47) group on the **Home** tab. Or, you can use the **F11** key when the **Geometry** window, the tree **Outline**, or the **Details** view have focus. The tab option and the **F11** key toggle the display on and/or off. Also note that you can use Key Assignments (p. 314) from when the Full Screen active to displays other interface panes, such as the **Outline**, if desired.

Contextual Menu Options

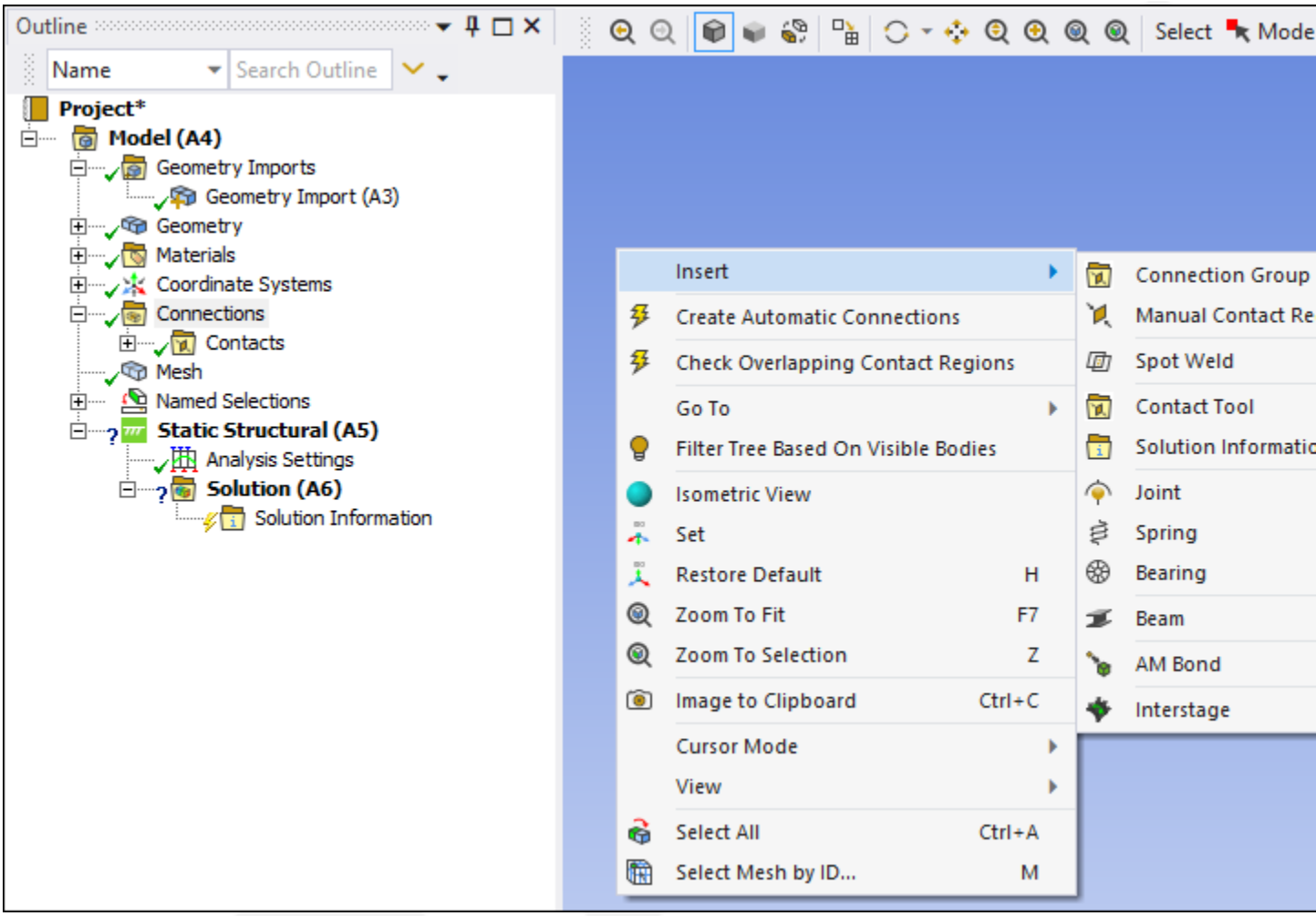
Right-clicking the mouse in the Geometry window provides a context menu with a variety of options. The common menu options and specifically, the **Cursor Mode** menu, are illustrated below. Furthermore, the menu displays additional options that are based on the object you have selected. For example, the **Insert** and **Go To** options shown below are often available and their menu selections depend upon the tree object that is selected.



Insert

The options of the **Insert** menu depend upon the object selected in the tree. For example, with the Connections object selected, right-click in the geometry window and then select **Insert** to display the menu of connection options illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Tip:

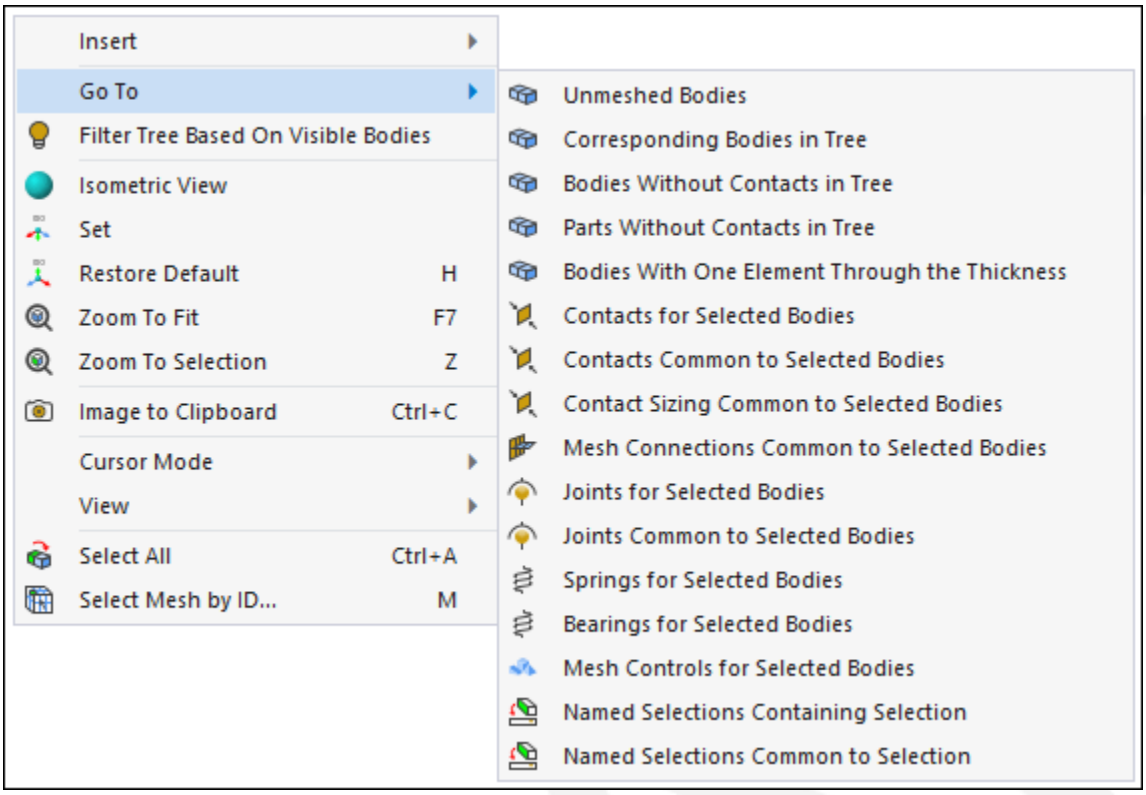
The Objects Reference section provides descriptions for all objects and their context menu options.

Go To

The options of the **Go To** menu depend upon the object selected in the tree. For example, with a geometric entity selected on the model, right-click in the geometry window and then select **Go To** to display the menu of options illustrated below. See the Correlating Tree Outline Objects with Model Characteristics (p. 110) section for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

Got To options are also available from the Selection (p. 88) tab.

Filter Tree Based on Visible Bodies

Filters the tree so that only visible bodies and objects associated with any visible body display.

Isometric View

Displays your model in the default isometric view.

Set

This option enables you can define a custom isometric viewpoint based on the current viewpoint. That is, you position your model where you would like it, using the other view options as desired, and then selecting this option establishes a new **Isometric View**.

Restore Default

This option resets the **Isometric View** to the application default.

Zoom To Fit

Fits the entire model in the Geometry window.

Zoom To Selection

Zoom in on the currently selected item (geometry/mesh/etc.) in the Geometry window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Image to Clipboard (Ctrl+C)

For the Windows platform, this option performs a snapshot of whatever is currently displayed in the **Geometry** window and copies it to the clipboard so that you can paste it into compatible applications.

Cursor Mode

This option provides a different method for selecting the cursor mode. See the Graphics Toolbar (p. 97) section of the Help for a description of each selection.

View

This option changes the viewpoint of your model. It operates much like the Triad.

Select All

Selects all items in the Model of the current selection filter type (vertex, edge, face, or body).

Select Mesh by ID (M)

This option enables you to select mesh nodes or elements using their ID (p. 279). This feature can be activated with the **M** key when the Geometry window is active. And, it also provides an option to create Named Selections for your selections.

Reset Body Colors

Reset the body colors back to the default color scheme.

Note:

As applicable and based on the object that you have selected in the tree, the contextual menu also provides options specific to that object.

Discrete Legends in the Mechanical Application

The following additional legends are available based on the object you have selected in the application.

- **Geometry Legend (p. 858)** : Content is driven by **Display Style** selection in the Details view panel.
- **Joint Legend (p. 1366)** : Depicts the free degrees of freedom characteristic of the type of joint.
- **Results Legend (p. 2284)** : Content is accessible via the right mouse when the legend for a solved object in the **Solution** folder is selected.

Displaying Shells for Large Deflections

The display of shells may become distorted for large deformations such as in large deflection, explicit dynamics analyses, etc. A workaround is to disable shell thickness by toggling **Thick Shells and Beams** option in the **Style** group (p. 80) of the **Display** tab. Or, set a Workbench variable, **`UsePseudoShellDisp`** = **`1`**, via the **Variable Manager**. It may be necessary to toggle the deformation

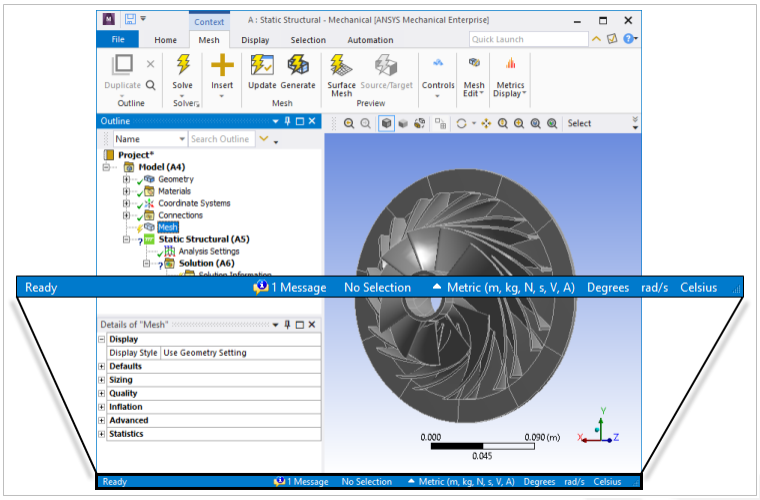
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

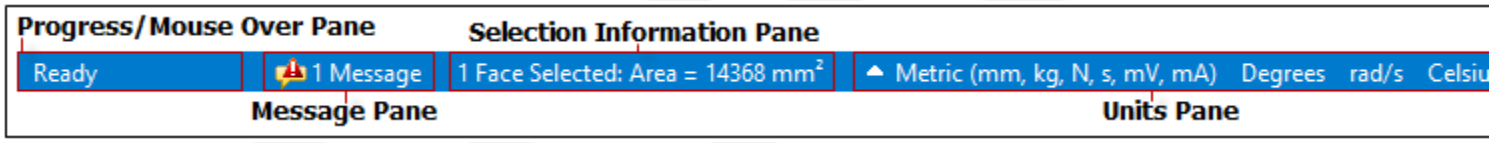
scaling (p. 63) from **True Scale** to **Undeformed** to **True Scale** again. Note that this option requires True Scaling to work properly.

Status Bar

The status bar provides information about your simulation.



Illustrated here, the status bar displays information panes. Generally, when information is available, you can single-click a pane to display an associated menu or window.



Go to a status bar component topic for more information about its purpose:

- Progress Pane (p. 135)
- Tooltip Display (p. 136)
- Application Messages (p. 137)
- Selection Information (p. 137)
- Units of Measure Pane (p. 138)
- License Read-Only Display (p. 138)
- Customize the Status Bar (p. 138)

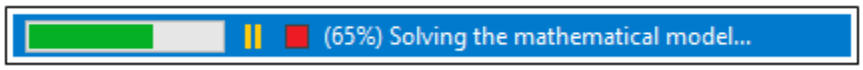
Progress Pane

This area of the status bar displays the progress of certain application processes, such as mesh generation and solution processing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

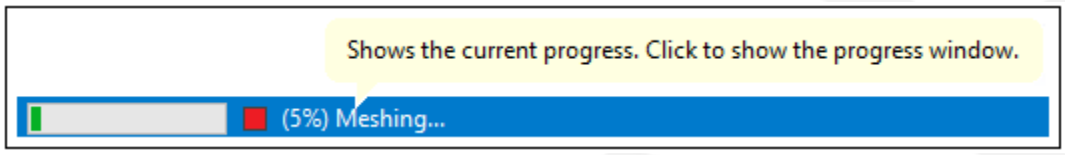
Progress Bar (**Interrupt** and **Stop** Options)

During an active process, the application displays a progress bar in the progress pane. And, depending on the process, it also provides an Interrupt (pause icon) option and/or Stop (red square icon) option. An example of the progress of a solution is illustrated blow. It includes both the Interrupt and Stop options.

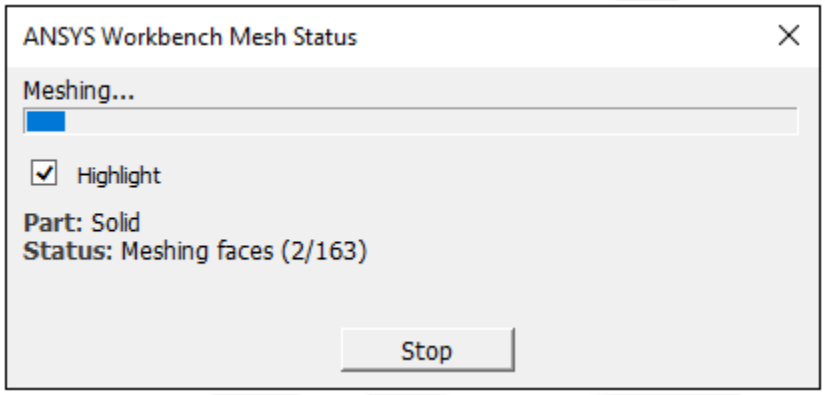


Progress Tool Tips (via Mouse Over)

During an active process, if you hover your mouse over the progress bar, a button, or the description (percentage, etc.) of the progress and process, a tool tip displays. An example of a mesh generation process is shown below.



Note that a single-click in the progress pane opens an associated progress window (shown here). This window is the legacy progress display for Mechanical.

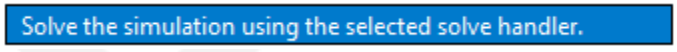


Note:

Using the **UI Options** (p. 244) preference setting **Hide Progress Window**, you can choose to always display progress windows.

Tooltip Display

When you hover your mouse over an interface option, a tooltip displays in the bottom right corner of the bar. Here is an example of the tooltip for a mouse-over of the Solve button.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

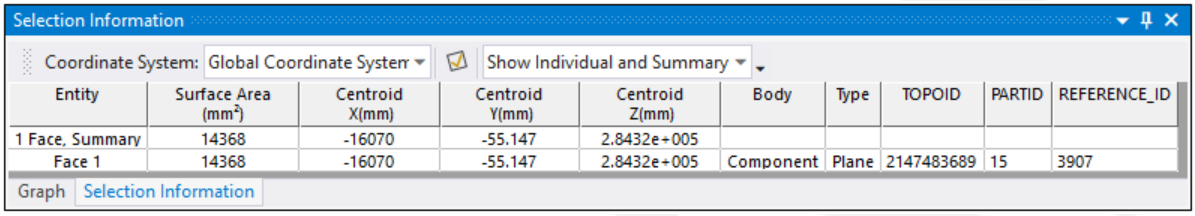
Application Messages

The status bar contains a pane that displays whether there are application generated messages (errors, warnings, and/or information). A single-click the pane to display the Message window (p. 200).

Selection Information

The **Selection Information** pane displays information about the currently selected geometric entity or entities. Such as area, length, or location. In the following example, the area of a single face is displayed.

A single-click the pane displays the Selection Information (p. 180) window.



Example Selection Information includes (but is not limited to):

- A selected node or element number if only one node/element is selected.
- The location of a node or vertex if one is selected.
- The combined volume of selected bodies, area of selected faces, and lengths of edges.
- The radius of a single circular edge or face that is selected.
- Angle between three nodes (always in degrees).
- The angle between two planar surfaces (always in degrees).
- The angle between two straight edges (always in degrees).

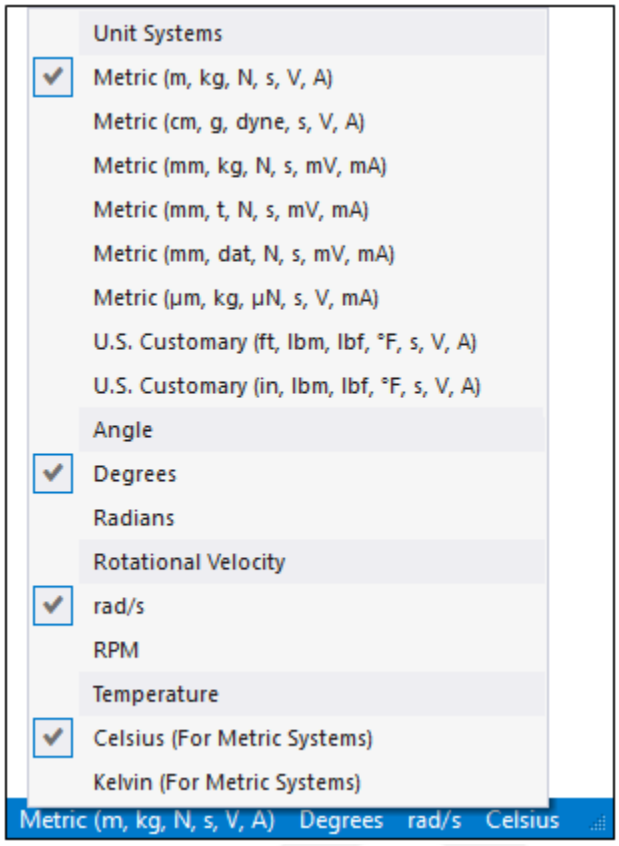
Note:

- If there is no associated information, the Selection Information pane indicates **No Selection**.
- Geometry calculations such as length and area are an approximation based on geometry information contained in either the CAD data or graphics tessellation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Units of Measure Pane

This pane displays the currently selected Unit systems. Selecting this pane displays the **Units** menu (p. 45) enabling you to change the current unit system.



License Read-Only Display

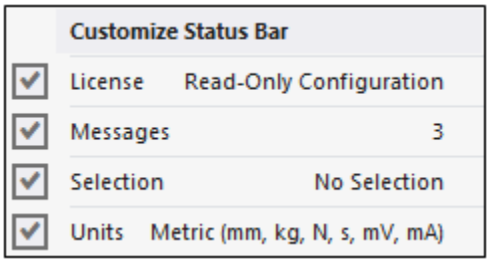
If a license is not available when you open Mechanical, the application automatically opens in **Read-Only Configuration** mode as illustrated below. Once a license becomes available and you activate it, the application automatically exits this read-only mode.

Customize the Status Bar

When you right-click the status bar, the following menu displays. Using this menu, you can select whether to hide (or show) certain status bar options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Quick Launch

The **Quick Launch** tool enables you to quickly find a desired function, feature, or interface option, and, based on the search string, automatically select, insert, or launch the pertinent interface item/feature. As you type in the Quick Launch search box, results display in the following categories:

- **Ribbon** (@rib): This category presents interface options available from all current tabs as well as the current Context tab.
- **Context Tab** (@ctx): This category presents search results for all the of the application's Context tabs (current or otherwise not displayed).
- **Pane Toolbar** (@too): This category presents search results for application features and options that are contained in all application toolbars.
- **Preferences** (@opt): This category presents selections that open the **Options** dialog (p. 217) and automatically displays the corresponding property, enabling you to modify its setting (default or current).
- **Tree** (@tre): This category presents search results for objects contained in the **Outline** pane. This option is hidden by default. Enable this option using the **Tree Items** setting of the Default Quick Launch Result Categories (p. 246) preference of the **Options** dialog.

As listed above, each category has an accompanying shortcut (@rib, etc.). You can use these entries to search within a desired category only. For example, the entry "@rib Mesh" searches for options and features of the ribbon category related to the keyword Mesh.

When you highlight a listing in the **Ribbon** category, the accompanying text string "Take me there" also displays. When selected, the "Take me there" feature tells the application to point to the option on the interface and display a pop-window that describes the option. This feature is also available for the **Pane Toolbar** and **Context Tab** categories. For the **Context Tab** category, it only displays when an action can be performed on the currently selected object. For the **Pane Toolbar** and **Tree** categories, it highlights the search item on the interface.

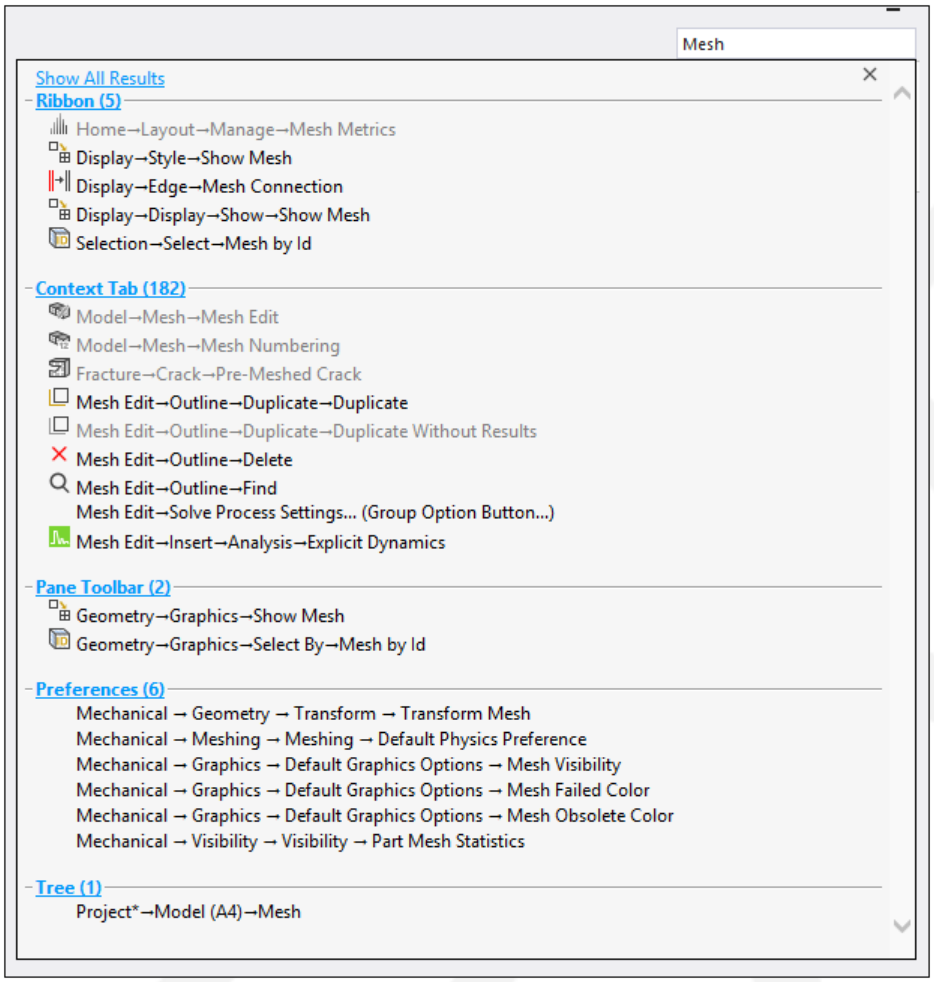
As illustrated below, some search listings display as bold and others are greyed-out. Greyed-out listing cannot be selected but suggest a potential path to your desired search item. Bold listings are selectable and cause the application to automatically take action. Example actions include the application automatically inserting or selecting an object in the **Outline** pane or highlighting a pertinent interface option. Note that bold listings may require that you have an appropriate object selected in order to successfully perform an action.

For example, if you have the Environment object selected and you search on "Pressure" and then select the Pressure listing from the Quick Launch menu, a Pressure load is automatically inserted below the

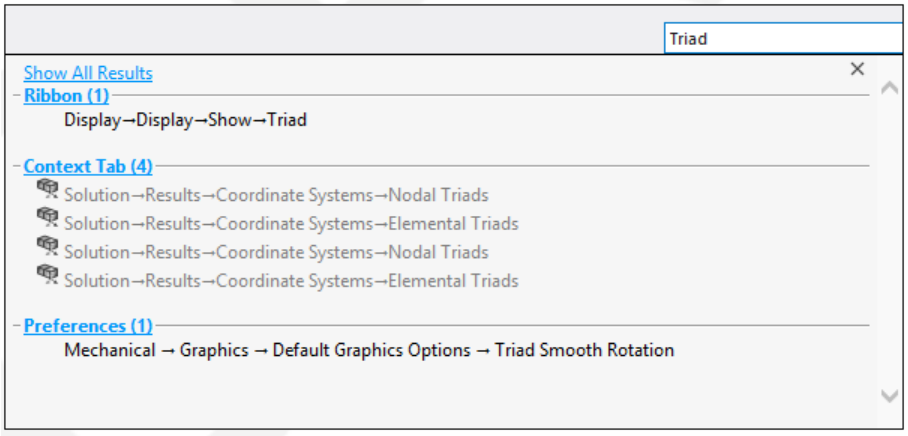
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Environment object. This is the same result as if you had performed the action via the Environment Context tab.

Bold Listings



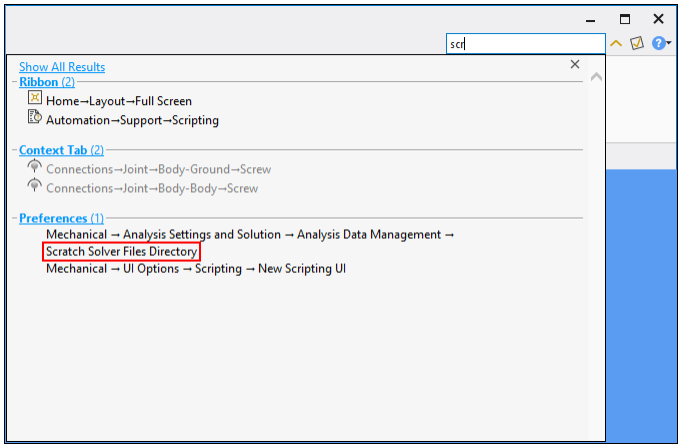
Greyed-Out Listings

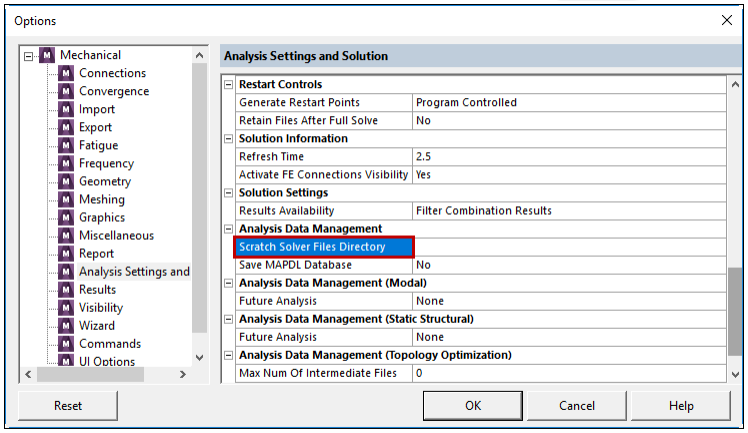


Preference Listings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

These selections open the **Options** dialog (p. 217).





Help Menu

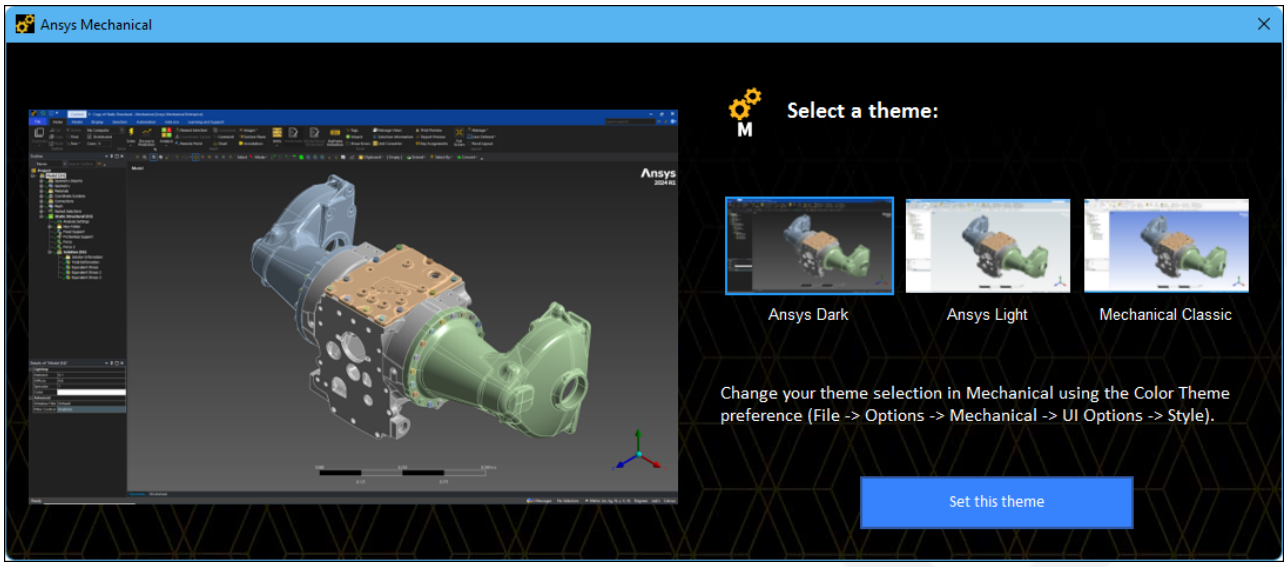
The **Help** drop-down menu provides the following options.

Function	Description
Mechanical Help	Displays the Help system in another browser window.
Mechanical Highlights - What's New? (Windows Platform Only)	This option displays an illustrated review (via the File (p. 37) tab) of the release's new features and capabilities.
Mechanical Release Notes	Open the release notes for the version of the application that you are running.
Usage Tips	This option opens a window that provides several instructional slides describing the new features and functions of the current release.
Scripting - Quick Start Guide	This option links to the introductory documentation for application scripting capabilities.
Ansys Product Improvement Program	This option launches the dialog for the Ansys Product Improvement Program and enables you to either accept or decline the invitation to participate in the program.
About Mechanical	Provides copyright and application version information.

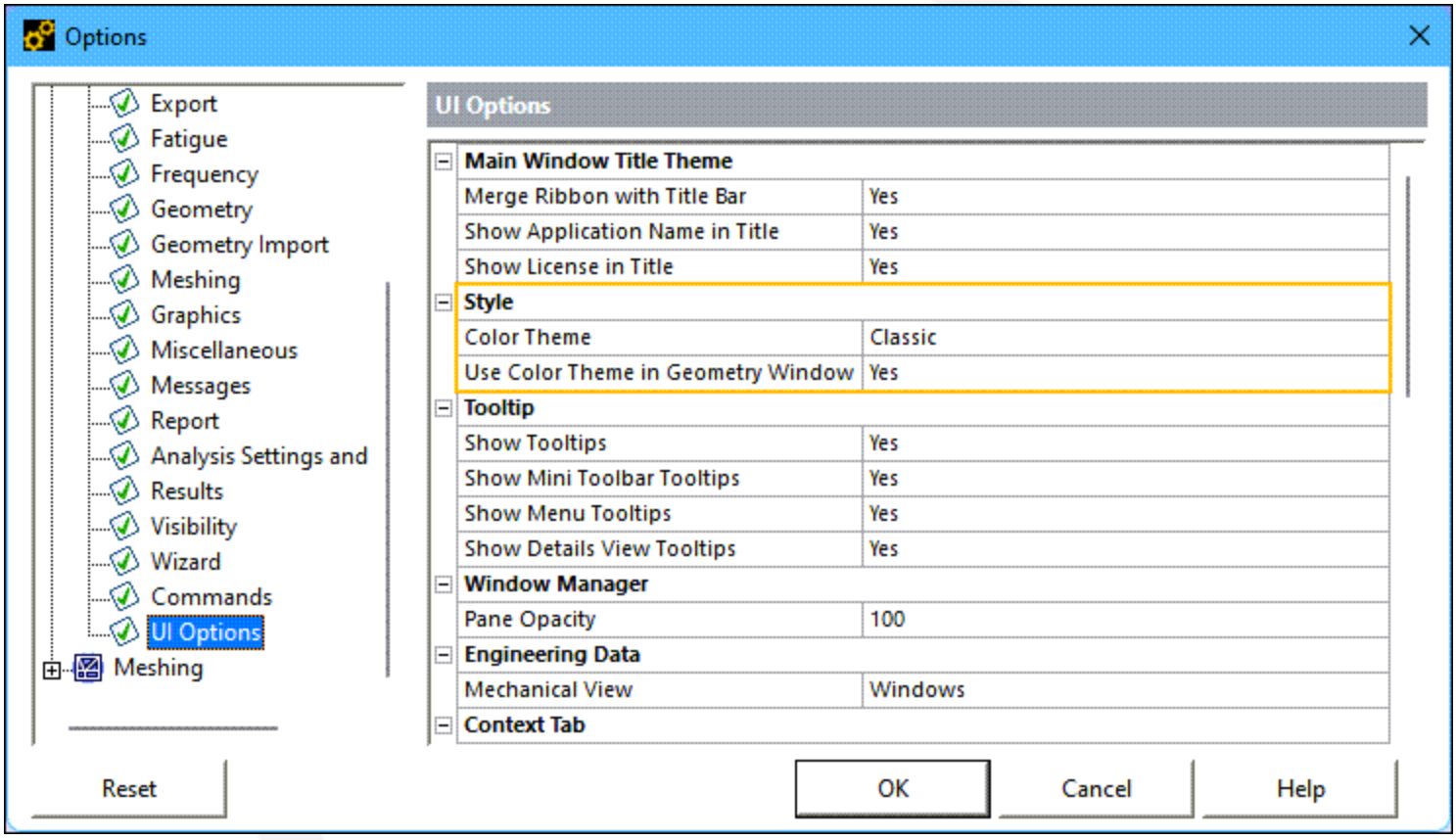
Color Theme

When you first install and launch the application, the following dialog displays so that you can select a theme. Options include **Ansys Dark**, **Ansys Light**, and **Mechanical Classic**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



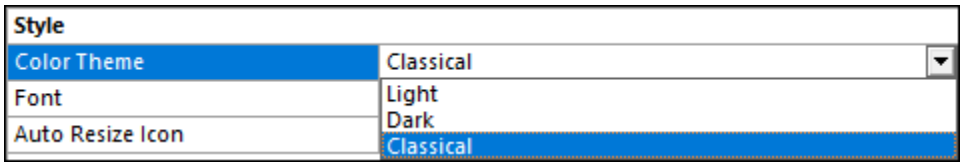
Using the Options (p. 217) dialog preferences of the Style (p. 244) category of the **UI Options** group, you can change the color theme (Dark and Light) of the interface.



Color Theme Preference

Following your initial selection, you can change the theme as desired using the **Color Theme** preference.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Use Color Theme in Geometry Window

Use this preference to select the background color of the **Geometry** window. If set to **Yes** (default), the application uses the background color of the selected **Color Theme** (light, dark, classic). Set to **No**, Mechanical uses the background colors selected in Workbench (**Tools** > **Options** > Appearance).

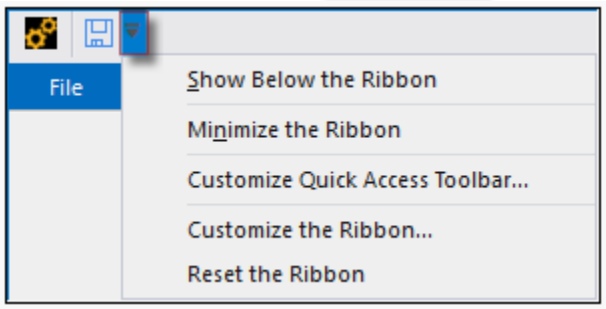
Ribbon Customization Options

The application offers features that enable you to modify the content of the tabs and the toolbar, including:

- Customizing Ribbon Content (p. 143)
- Customizing Toolbars (p. 145)

Customizing Ribbon Content

The ribbon interface includes a **Quick Access Toolbar** that you can modify. It is located in the upper felt corner of the window, right beside the Save icon (which is an option of the Quick Access Toolbar). Selecting the drop-down menu option displays the following menu.

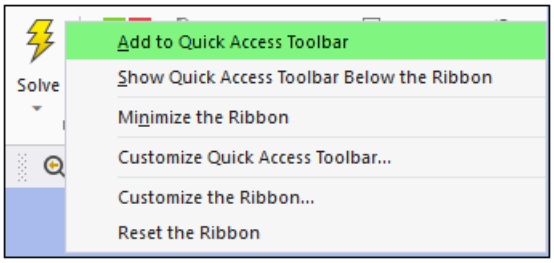


The options include:

- **Show Below the Ribbon** (or **Show Above the Ribbon**): Place the toolbar below the Tabs or return it to its default position.
- **Minimize the Ribbon**: Hide/show the ribbon.
- **Customize Quick Access Toolbar**: This option displays the **Customize** dialog. This dialog enables you to add options to the Quick Access Toolbar. See below (p. 144).
- **Customize the Ribbon**: modify the contents of the various tabs as well as create your own tabs and option groups. See below (p. 144).
- **Reset the Ribbon**: Reset the interface contents to default display settings.

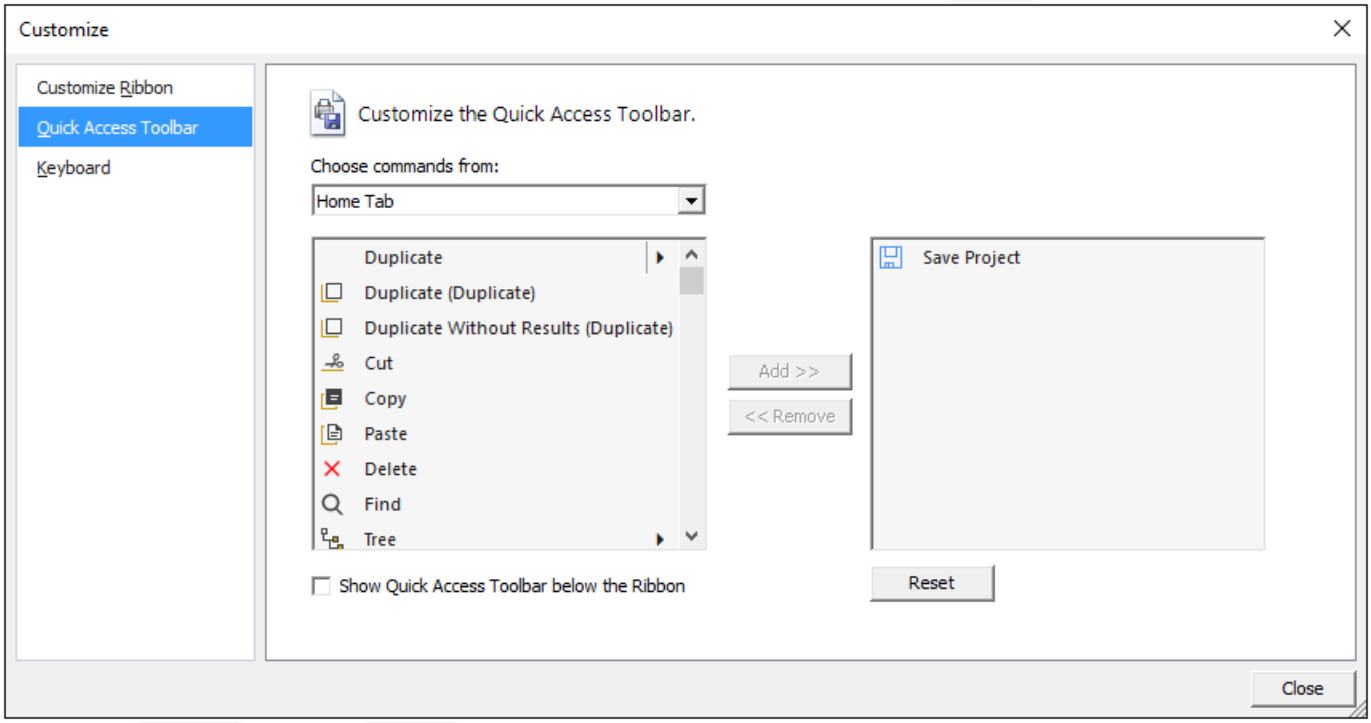
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Also note that you can highlight an interface option, right-click, and a select **Add to Quick Access Toolbar** to add the option.



Customize Quick Access Toolbar

When you select the **Customize Quick Access Toolbar** option, the following dialog appears. As you can see, the default Save option is already included. You select from the list of available commands to **Add**/**Remove** options to the Quick Access Toolbar. You can select options from all available tabs using the drop-down menu.



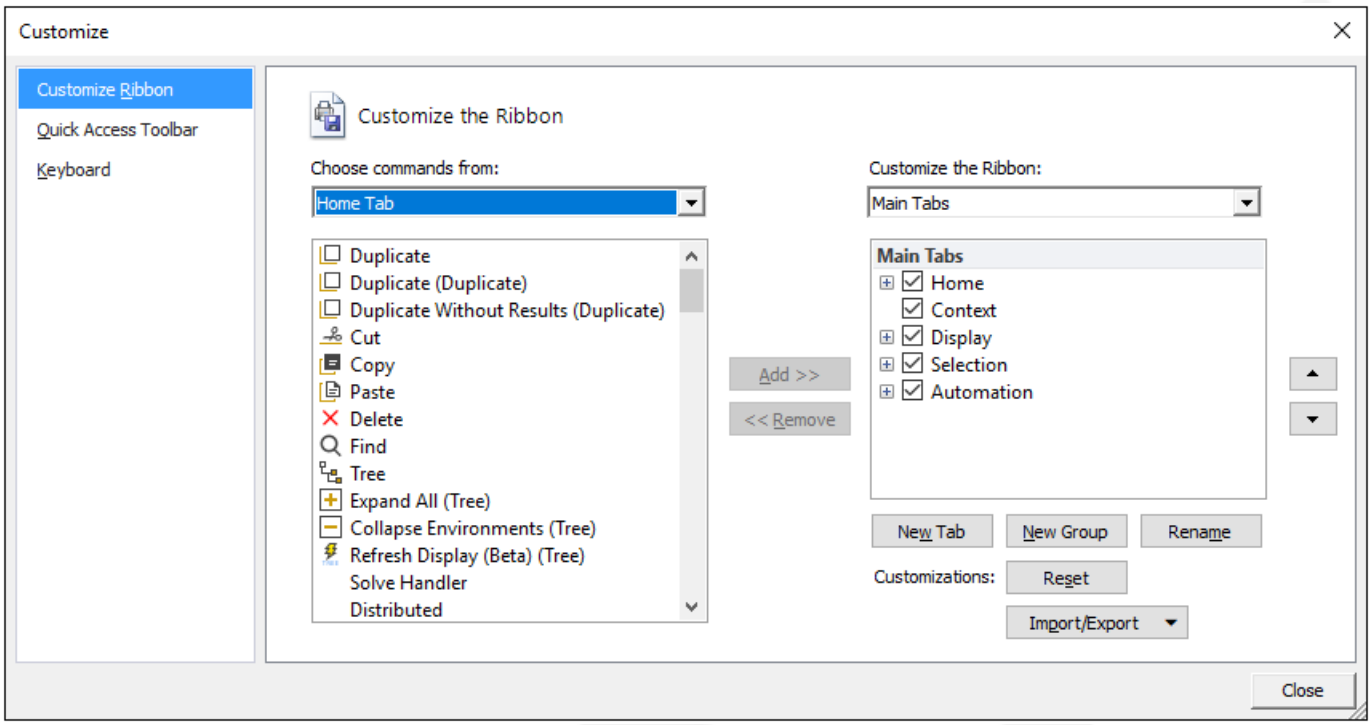
Customize the Ribbon

When you select the **Customize the Ribbon** option, the following dialog appears. A drop-down menu of tabs is available for your selection and the corresponding commands.

You can add existing options to the Groups of the Main Tabs or you can create a new customized Tab with custom Groups. Based on the Main Tab that you select in the right-side pane, if you create a **New Tab**, it is placed immediately after the currently selected option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

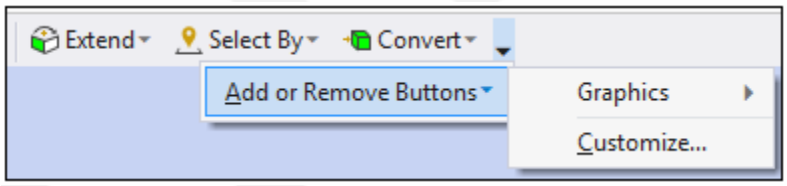


Note:

- You cannot remove system default tabs.
- If you deselected an ACT Extension in the Extension Manager or if the version of an extension is updated, any customization of the extension's Tab is automatically removed.

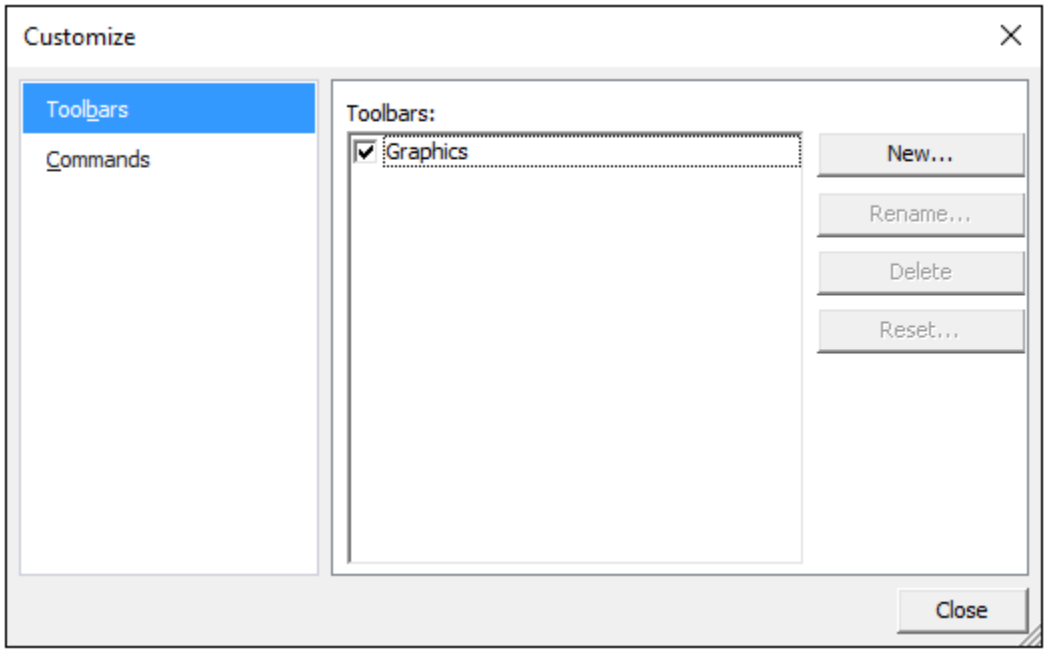
Customizing Toolbars

The various toolbars of the application, such as the Graphics Toolbar (p. 97) or the Manage Views (p. 292) window, offer an option to **Add or Remove Buttons**. You access this option using the down-arrow drop-down menu at the far end of the toolbar. An example of the Graphics Toolbar is illustrated below. When you select the **Add or Remove Buttons** option, an additional fly-out menu displays. Depending upon the toolbar or window that you have selected, the first option varies, but the common option is **Customize**.



Selecting the **Customize** option displays the following dialog.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

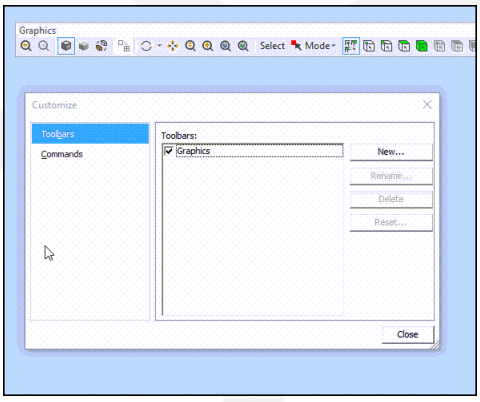


From this dialog, you create new toolbars or add options to the currently selected toolbar.

Add Options

In order to add options to the currently selected toolbar, you must first close the **Customize** dialog and detach the desired toolbar from the interface so that it is floating (as shown below). Then redisplay the **Customize** dialog, and 1) select the **Commands** option, 2) select from the available **Categories**, 3) drag-and-drop from the **Commands** pane to the toolbar.

Activating the **Show All Categories** setting of the **Options** dialog (**UI Options** (p. 244) > **Toolbar Customization**), you can display all options available in the application. This includes options from all toolbars from all panes, all commands from all Ribbon tabs, User Buttons, External ACT Extensions, etc. This gives you maximum flexibility for toolbar customization. This feature is set **No** by default. And, when turned off, any additions you make to a toolbar are cleared – the toolbar resets.

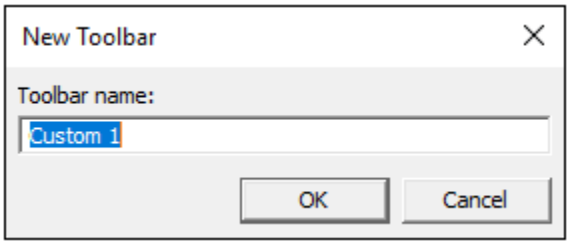


Create New Toolbar

To create a new toolbar, select the **New** button on the above dialog. An entry pane displays for you define a name for your toolbar.

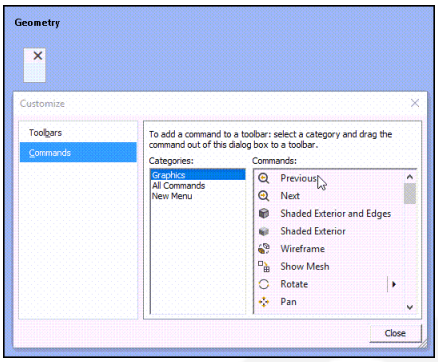
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



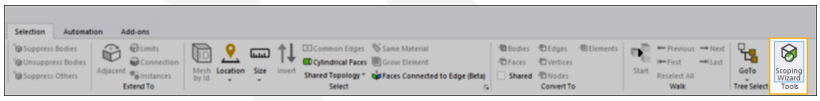
Once you define the name and click **Ok**, a blank toolbar displays (and includes down-arrow option), as illustrated bellow.

At this point you must close the **Customize** dialog and detach the new toolbar from the interface so that it is floating. Then, redisplay the **Customize** dialog, select the **Commands** option, and drag-and-drop **Commands** from the desired **Categories**. You can further build your new toolbar using the **New Menu** option that enables you to create a drop-down menu on the toolbar.



Scoping Recovery following Geometry Update

The **Scoping Wizard** option is available on the Selection (p. 88) tab of the ribbon. This feature automatically detects any geometry-based scoping that becomes lost following a geometry update. Once identified, you use the options of the feature to visualize and reestablish the original scoping. You turn the feature on (default) and off using the **Use Geometry Based Associativity** (p. 227) preference (**Options** > **Geometry**) dialog. In addition to the activation preference, there is an associated preference (**Automatic Scoping Update** (p. 227)) to have the application automatically correct geometry-based scoping that becomes lost following a geometry update.



Go to a section topic:

- Overview (p. 148)
- Feature Options (p. 149)
  - Wizard Pane Actions (p. 150)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

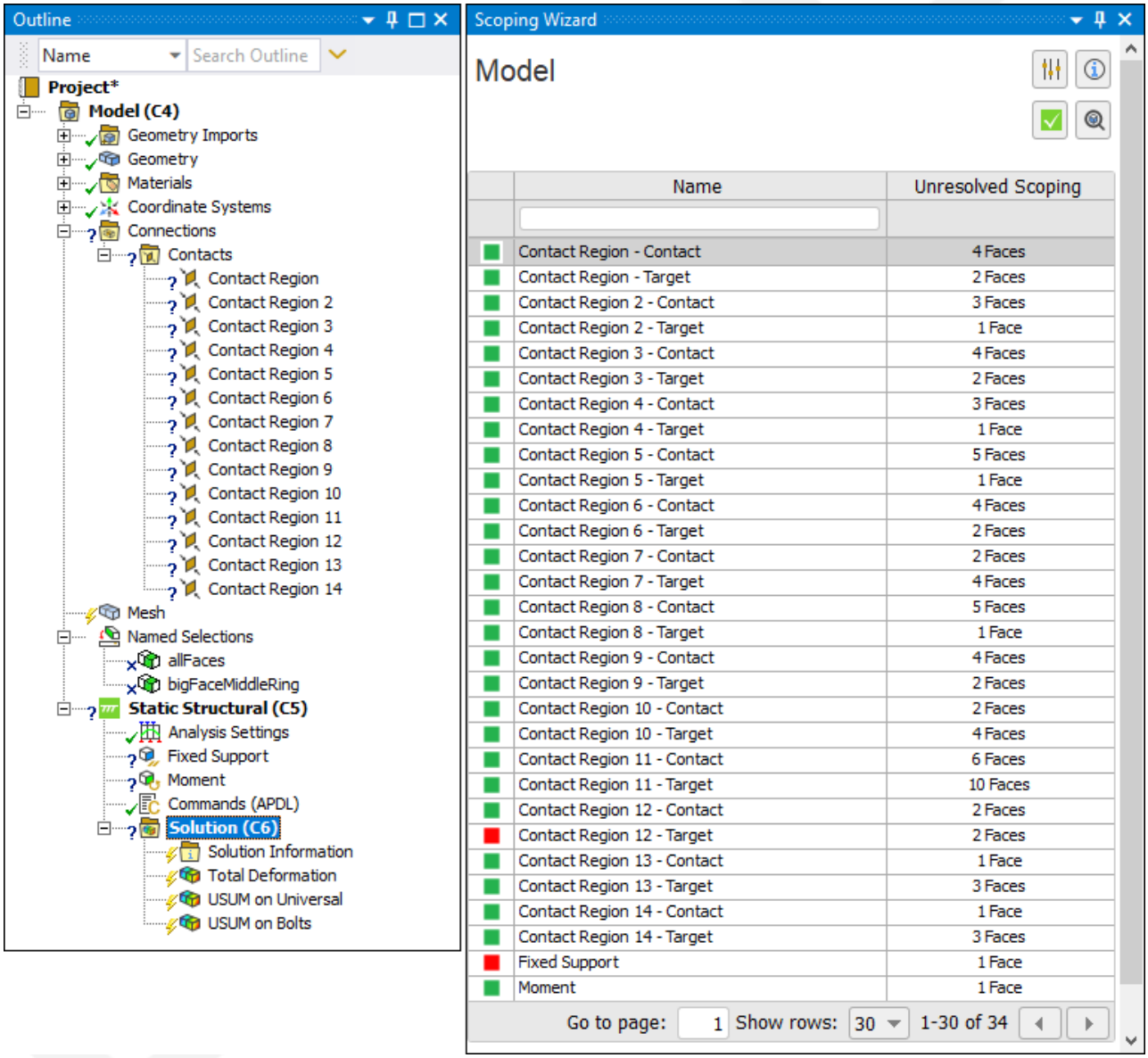
  - Context Menu Options (p. 150)
  - Color Codes and Annotations (p. 151)
- Display and Recovery Options (p. 152)

Overview

Selecting the ribbon option displays the **Scoping Wizard** worksheet pane, illustrated below, that lists all objects with unresolved scoping. Corresponding objects in the tree also display with an invalid state.

Important:

This feature does not recover mesh-based (node, element face, element) scoping.

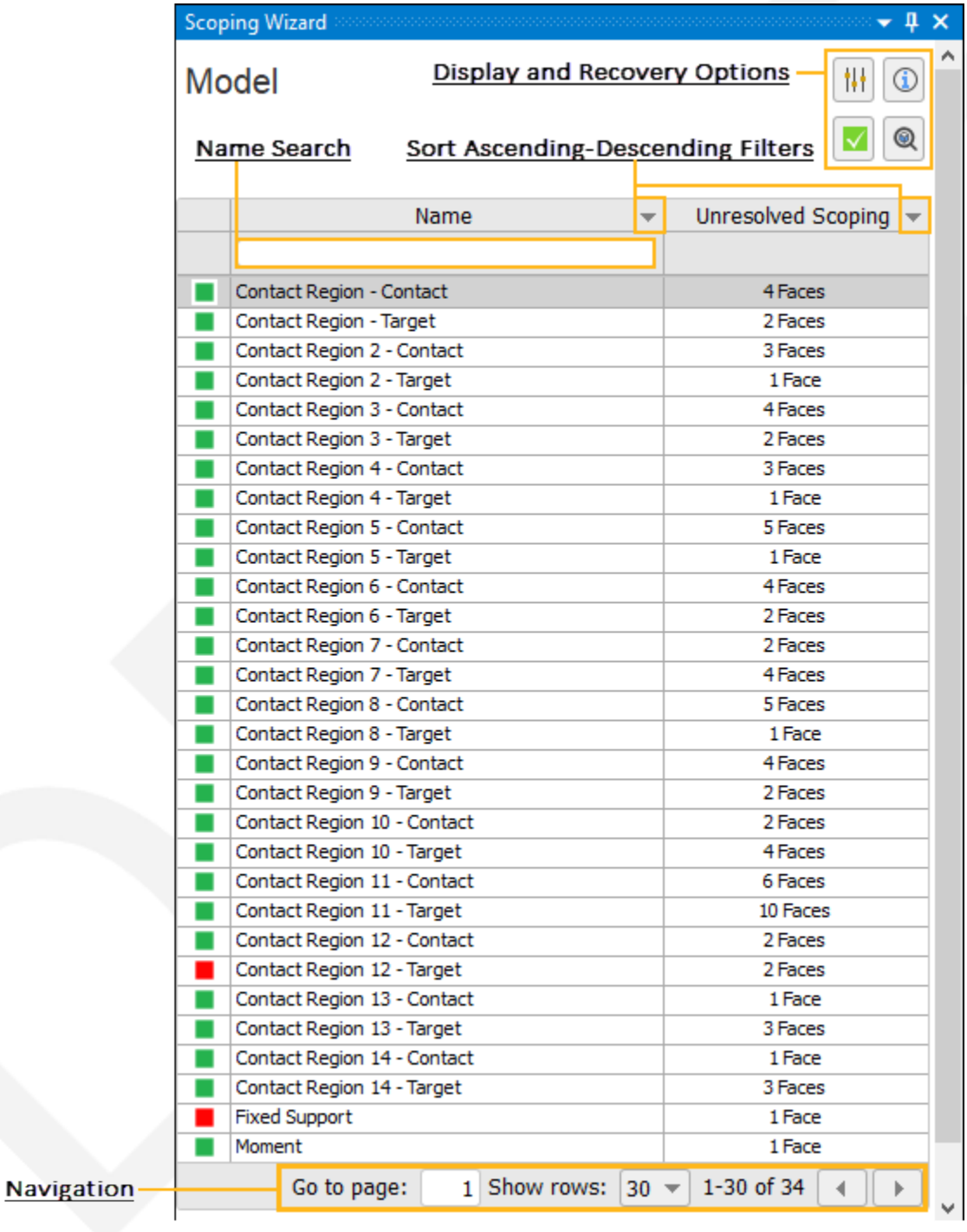


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Feature Options

The **Scoping Wizard** pane enables you to:

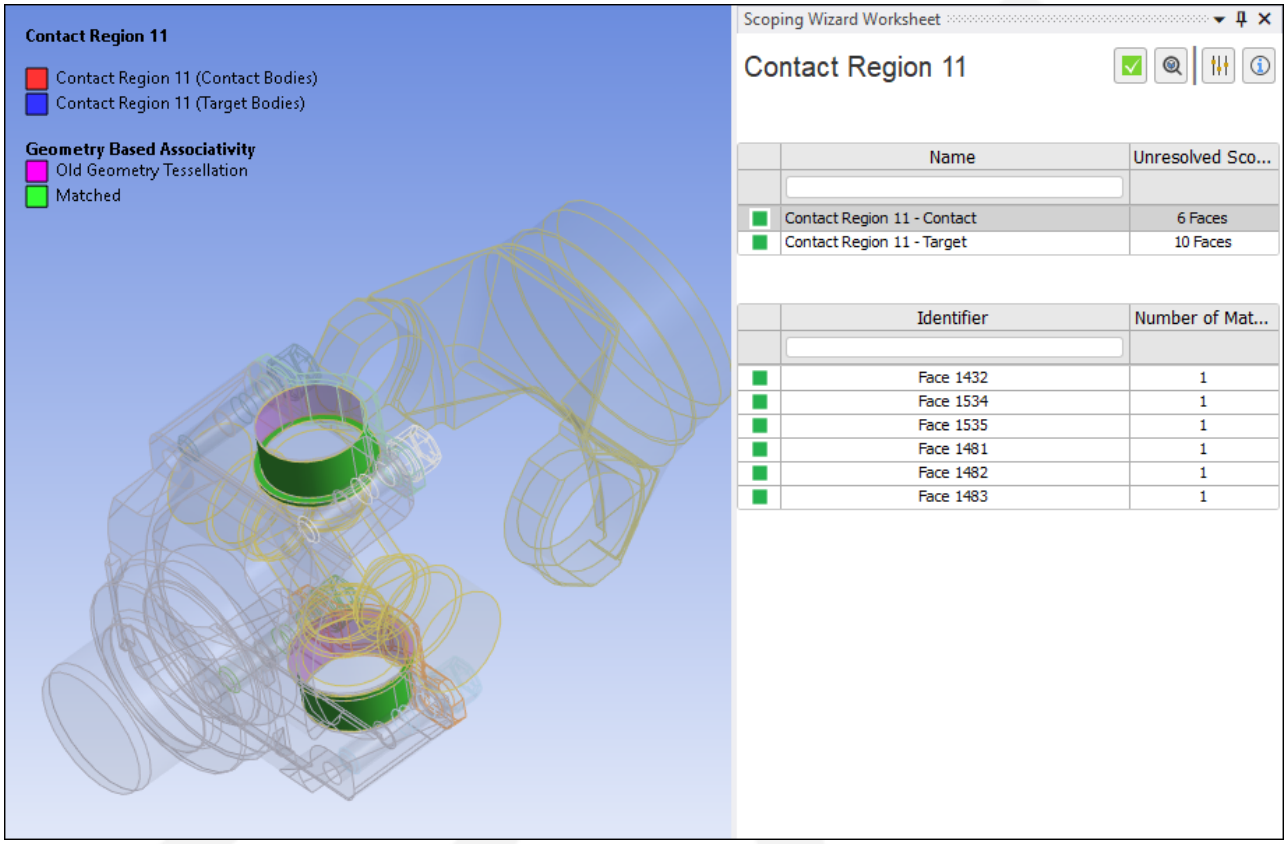
- Change display settings, fix scoping, show an information pane, and zoom on a scoping.
- View a list of unresolved objects.
- Filter the list based on object name or number of unresolved scopings (ascending/descending).
- Navigate through long lists of unresolved objects.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Wizard Pane Actions

If you double-click a table row, or select an unresolved object in the tree, the pane shows information specific to the object and its (unresolved) scoping. The following example shows a **Contact Region** that has lost scoping. The pane displays the object type/name (**Contact Region**), the number of unresolved geometric entities, whether the application identified a scoping match, and the original application-assigned ID for the entities. The first item in the worksheet is selected by default, and the parts are highlighted in the **Geometry** window.



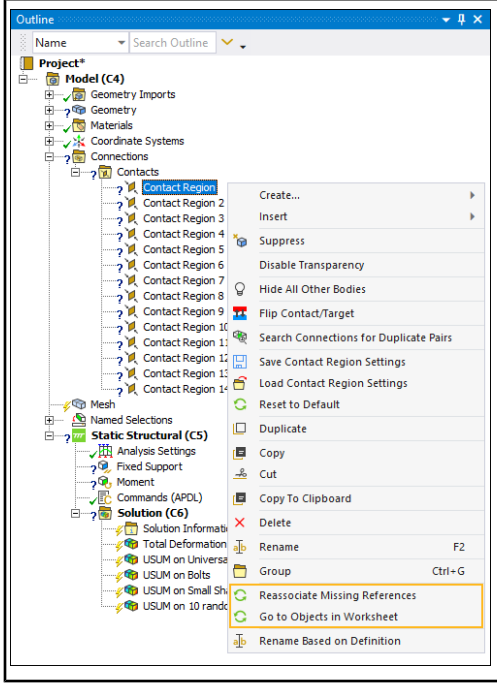
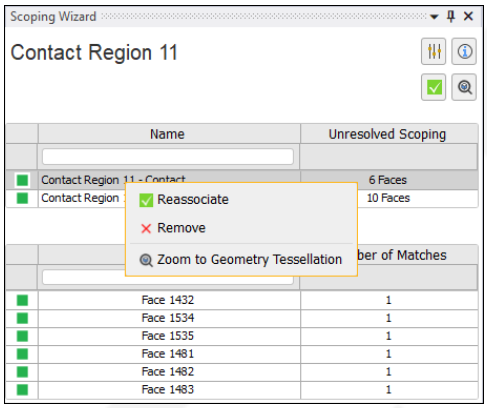
Context Menu Options

The feature also offers several context (right-click) menu options.

Outline Options	Worksheet Options
From the **Outline**, you can: • Reestablish the original scoping on individual objects using the **Reassociate Missing References** option • When the worksheet is not active/displayed, you can use the **Go to Objects in the Worksheet** option to open the worksheet and display information for the selected object.	Right-clicking on a row in the worksheet offers the following options: • **Reassociate**: Reestablish the original scoping. • **Remove**: Remove the row and its data from the worksheet. • **Zoom to Geometry Tessellation**: Zoom in on the body or part in the **Geometry** window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Outline Options	Worksheet Options
	

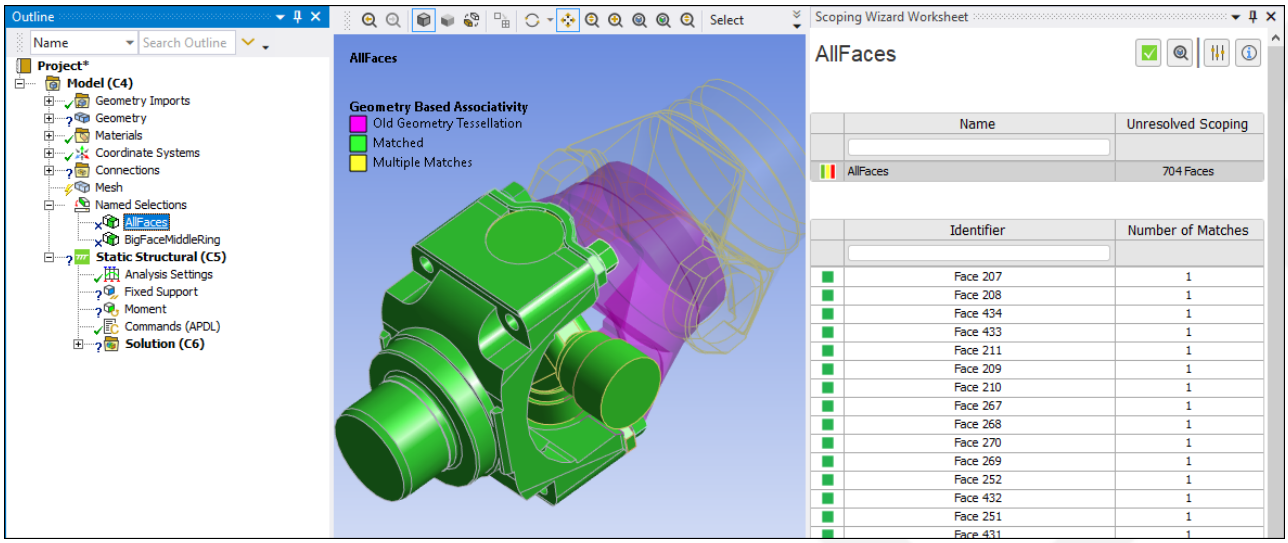
Note:

If you use the **Copy to Clipboard**/**Paste from Clipboard** options for an unresolved object, the associated data is copied and pasted between sessions and the application will automatically attempt to resolve any scoping.

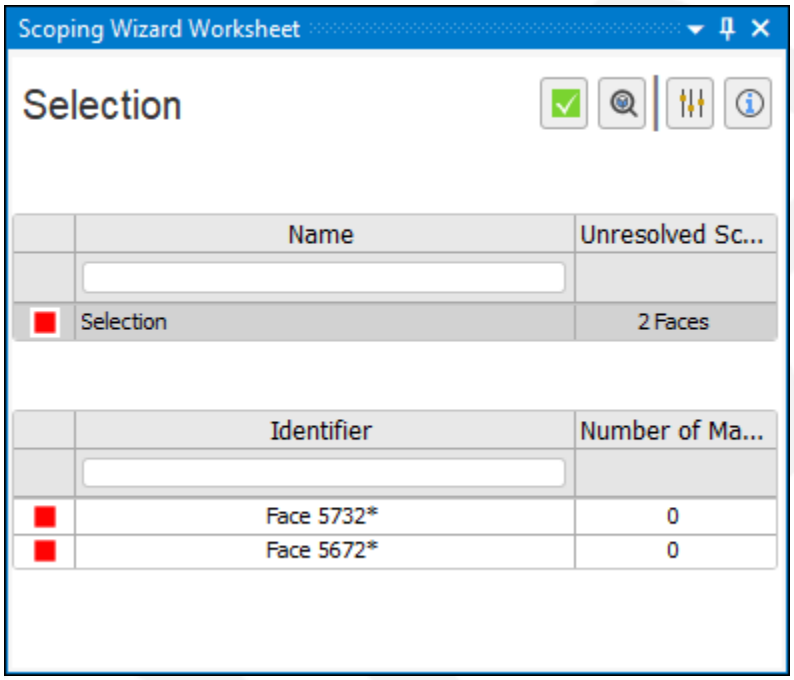
Color Codes and Annotations

The application displays reference annotations in the legend as well as highlights on the part(s) in the **Geometry** window, based on the state of the recovered reference. The application displays matched references display as green, multiple matches yellow, old geometry tessellation in purple, and if no match is available, the reference is drawn in red. Furthermore, the worksheet uses the color scheme to indicate if one or more of the conditions exists, such as the Named Selection, **AllFaces**, example shown in the table below that displays all three colors. See the Display and Recovery Options (p. 152) topic below for more information about annotations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



In addition, if a table entry appears with an asterisk, this indicates that the application could not find data associated with the reference. This can be the result of importing an older project, the use of virtual cells, or if you have turned the **Use Geometry Based Associativity** preference off.

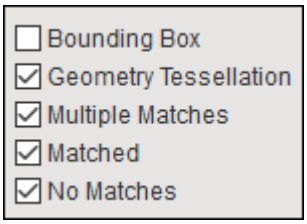


Display and Recovery Options

At the top of the pane, there are options to take corrective action, zoom in on a geometry, or modify display settings, including:

Option	Description
Settings	Illustrated below, the **Settings** drop-down menu enables you to turn annotation display options on and off. This image shows the default display settings.

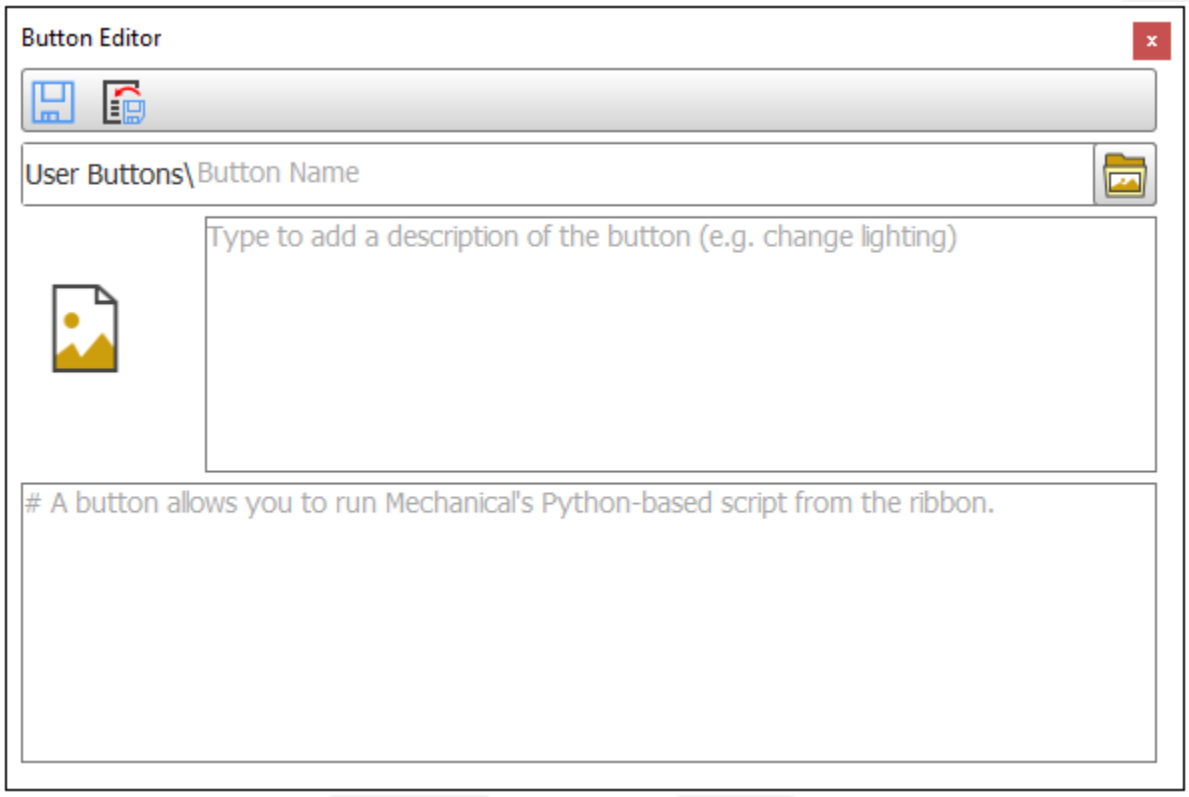
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Description
	 Options include: • **Bounding Box**: This option shows the bounding box of the old geometry tessellation for a selected item. Matched references are drawn in green, multiple matched references are drawn in yellow, and references that are not a match are drawn in red. • **Geometry Tessellation**: This option shows the full tessellation of the old geometry for a selected item. **Note:** The geometry tessellation references and the matched or multiple matched references display on top of one another and make it difficult to plainly see the old geometry tessellation. When the **Geometry** window is active, use the **[S]** key to turn off the display of matched or multiple matched references and only view the old geometry tessellation. • **Multiple Matches**: If the application determines that there are multiple potential matches, it highlights the original geometry as well as the new (multiple) geometries. For example, if two vertices in close proximity are potential matches, both will be highlighted as a Multiple Match. • **Matched**: The application determines that there is a new match for the original geometry selection. • **No Matches**: The application cannot determine a new match for the original geometry selection.
Information	This option displays a dialog that includes a description of the feature and its capabilities.
Fix All Scoping	This option automatically corrects all recovered scoping
Zoom to Geometry Tessellation	For the currently selected row in the wizard, or an object in the tree, this option zooms in on the body or part that the application has determined was associated with the original scoping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

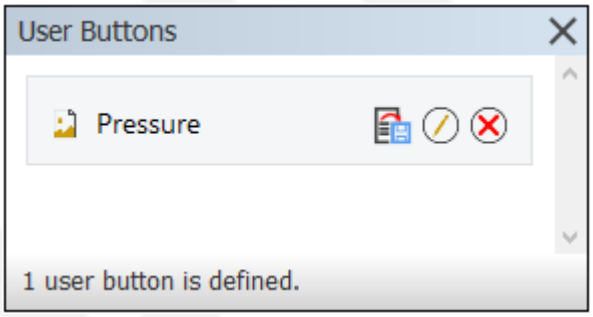
Creating User-Defined Buttons

The **User Buttons** group enables you to create, edit, manage, export and import, your own toolbar buttons. Selecting the **Manage** option opens the Button Editor.



This panel contains the following features and options:

- **Save**: This option saves button information and publishes it into the user group.
- **Import**: This option enables you to open and import an existing Python (`.py`) file that specifies a button. The content of the imported file populates the Button Editor dialog.
- **Name**: Enter a name (label) for the button.
- **Collection Editor**: The folder icon opens a panel that enables you to manage existing buttons. It contains option to edit, export, or delete the button.



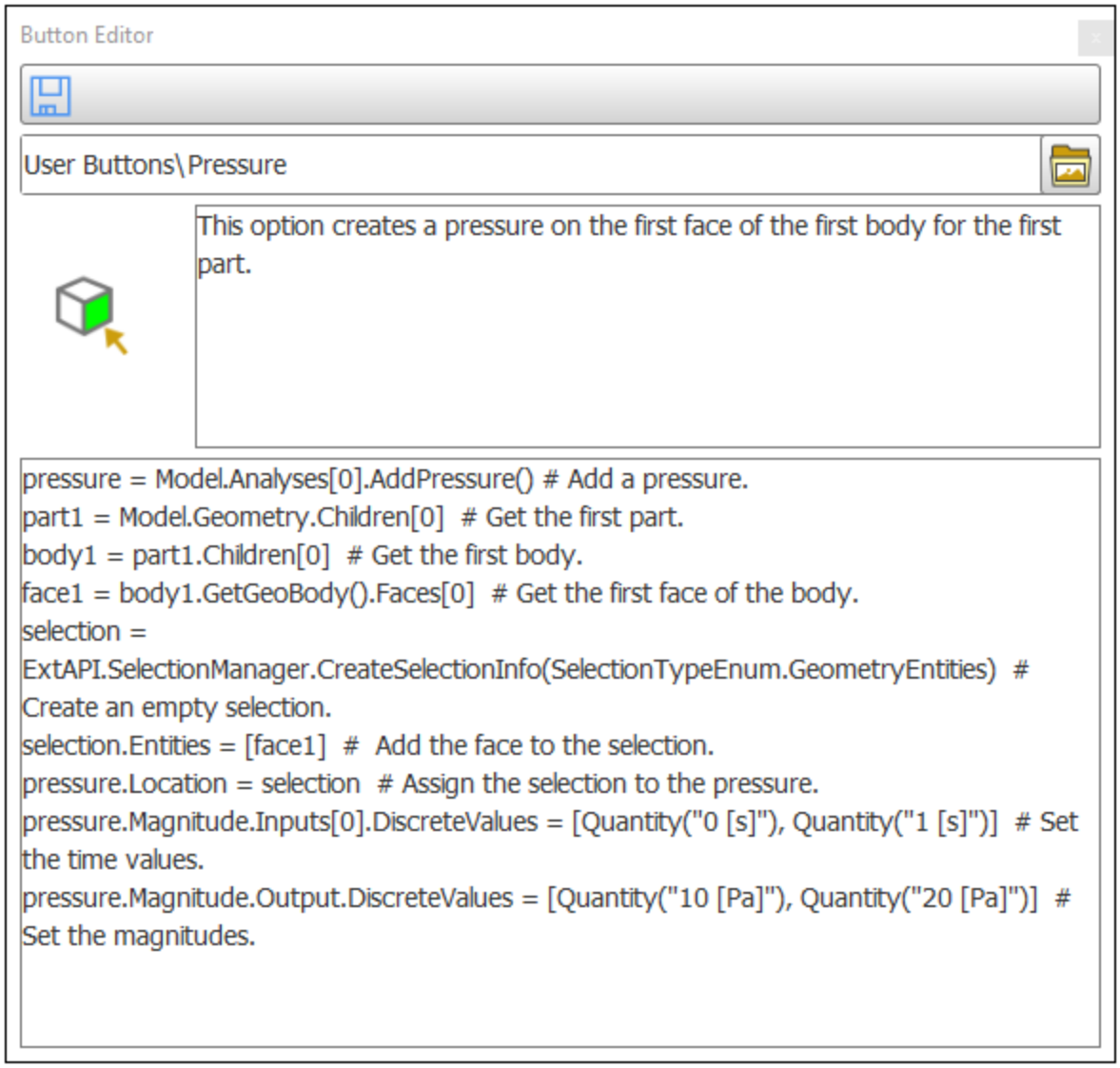
- **Image**: Select this icon to assign an image that to the button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

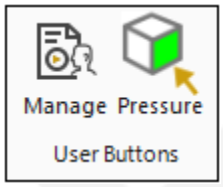
- **Description**: The description pane enables you to enter a description of what action the script performs - it is a tooltip.
- **Script**: You enter your python script into this pane.

Example

Here is an example to create of a Pressure load. You can find this example in the ACT Console.



The above example creates the following user-defined button.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

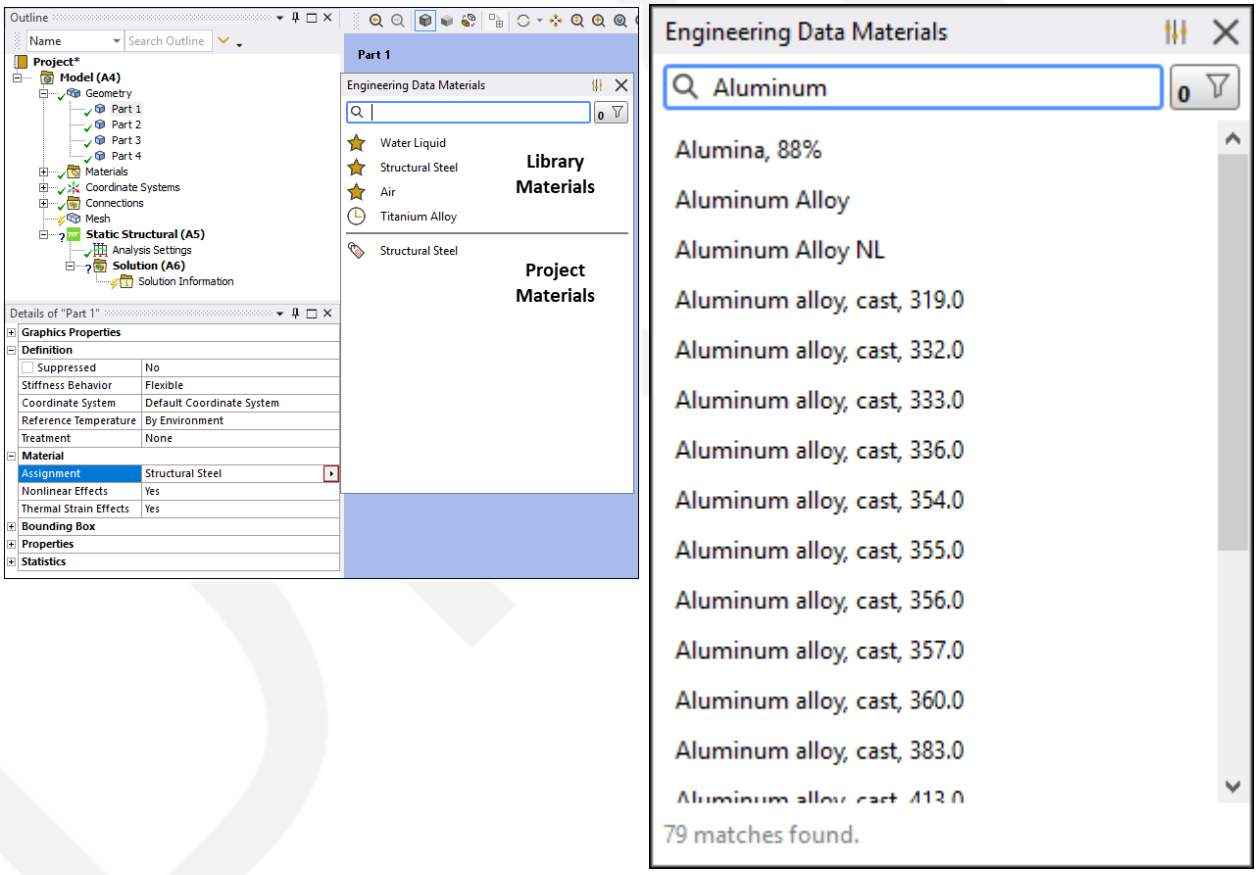
Engineering Data Material Window

The Engineering Data components of Mechanical enable Engineering Data information to be accessed and viewed within Mechanical. The available components include:

- Material Assignment (p. 156)
- Material Compare (p. 158)
- Material View (p. 165)

Material Assignment

The **Engineering Data Materials** pane is used to research and select material(s) to assign to specific application objects (p. 168) that require material assignment via the **Assignment** property. The pane can be accessed from the fly-out menu of the property. For example, as shown here, Parts (and/or Bodies) of the model require material assignment. By default, the pane lists favorites (star icon) and recent (clock icon) materials in the library portion of the pane, and project materials (material icon) from the Engineering Data workspace in the project portion of the pane. The search field (p. 159) can be used to find materials which match search criteria, in combination with filtering options (p. 159). A Material Card (p. 156) with common material properties can be viewed or used to access the complete material data to further assist in your search. When the desired material is found it can be selected for to assignment.

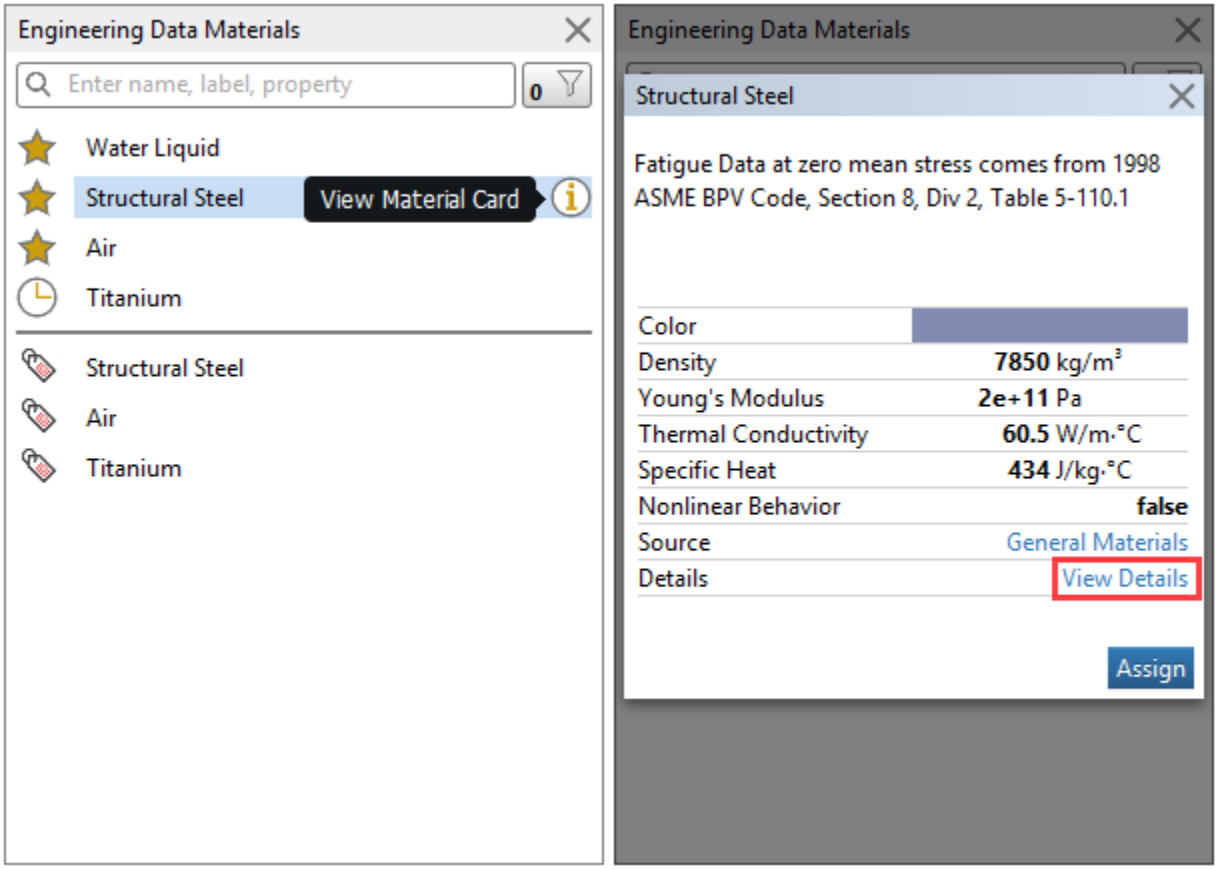


Material Card

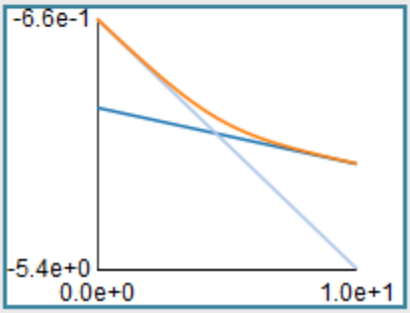
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

As illustrated below, when you highlight a material in the fly-out menu, an information icon displays. Clicking this icon will display the material card. The material card displays common properties to provide a quick overview of the material. To view the complete details of the material, click **View Details**. If you are ready to assign the material to the selected object, click **Assign**. Selecting the **View Details** option displays the Material Assignment pane combined with the **Material View**.

Material Card



Material View

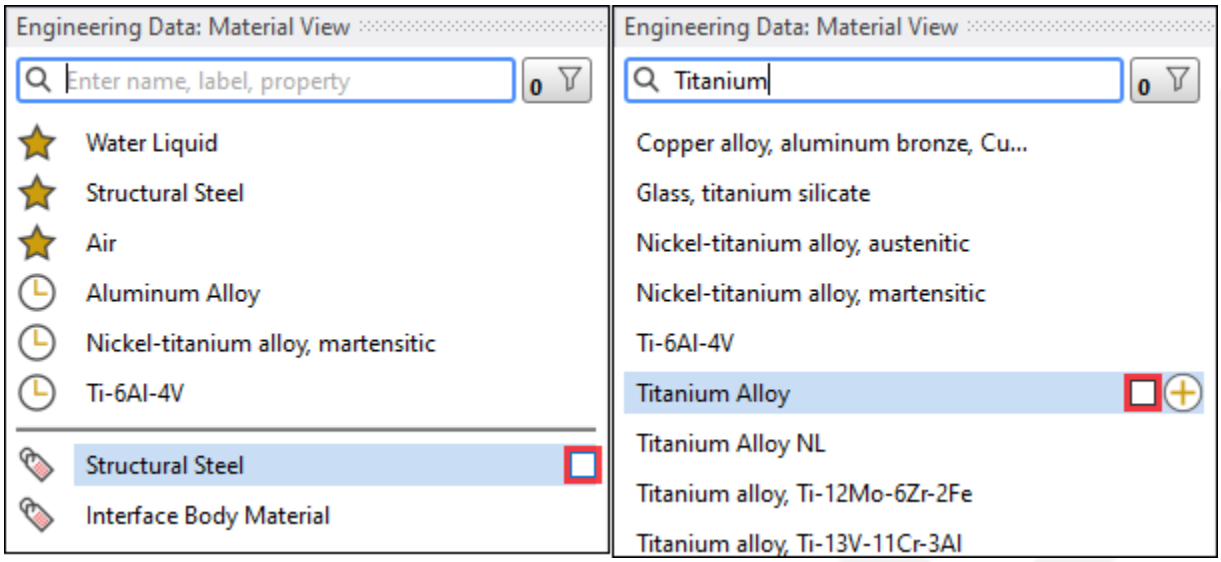
Structural Steel		
Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1		
Density	7850	kg/m³
Structural		
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	2e+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.6667e+11	Pa
Shear Modulus	7.6923e+10	Pa
Isotropic Secant Coefficient of Thermal Expansion	1.2e-05	1/°C
Compressive Ultimate Strength	0	Pa
Compressive Yield Strength	2.5e+08	Pa
Strain-Life Parameters		

Comparing Materials

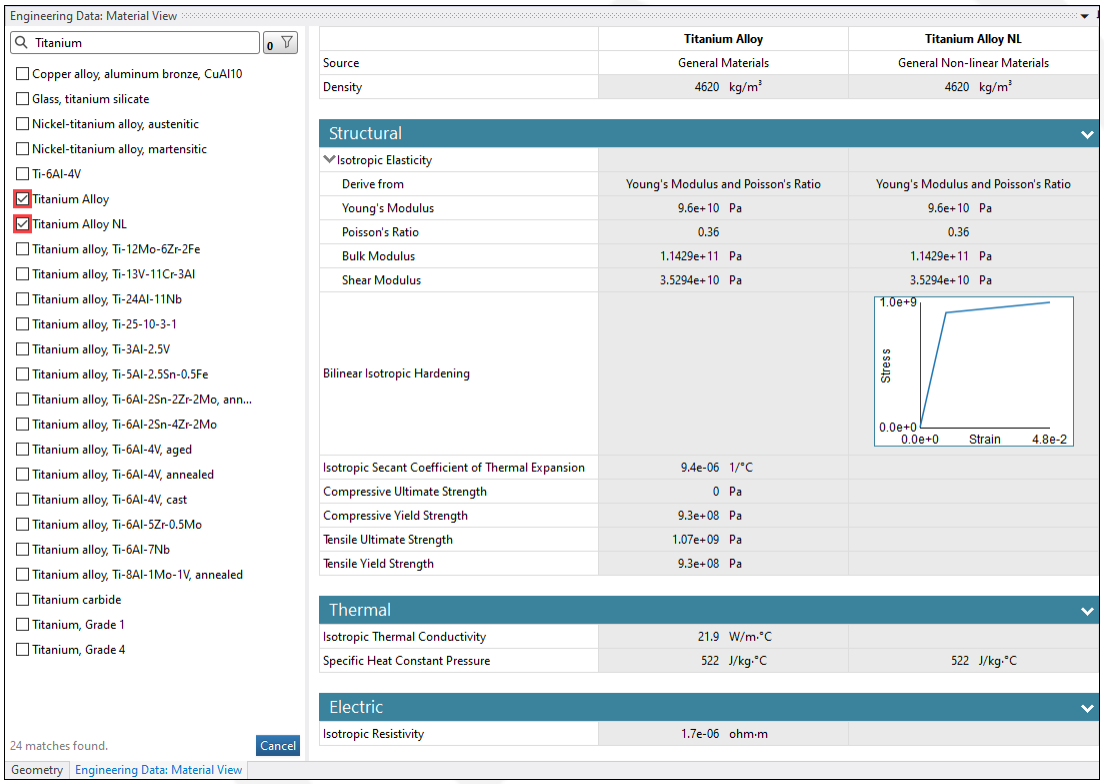
The materials added to your project, such as the default material Structural Steel, as well as the materials displayed when you perform a search, each provide a check box and a plus symbol when you hover the cursor over the material. The plus symbol (⊕) adds the material to the project.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



When you select a check box, or multiple check boxes, as show below, the application automatically displays material data to view and/or compare.



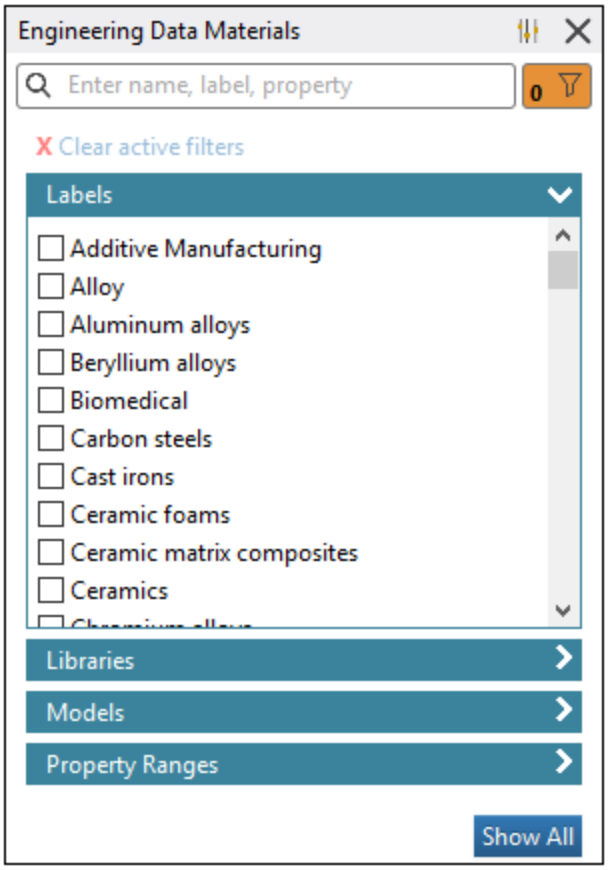
Search Field

The **Engineering Data Materials** panel provides the search field to enter filtering criteria. Each word in the field will be used to filter materials for selection (for example, criteria1 criteria2 criteria3). To return to the list of favorites, recent, and project materials; type the **[Backspace]** key in the empty field.

Filtering Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The **Engineering Data Materials** panel provides for additional filtering options by selecting the filter button (funnel icon). This displays the filtering options pane which allows selection of various filters.



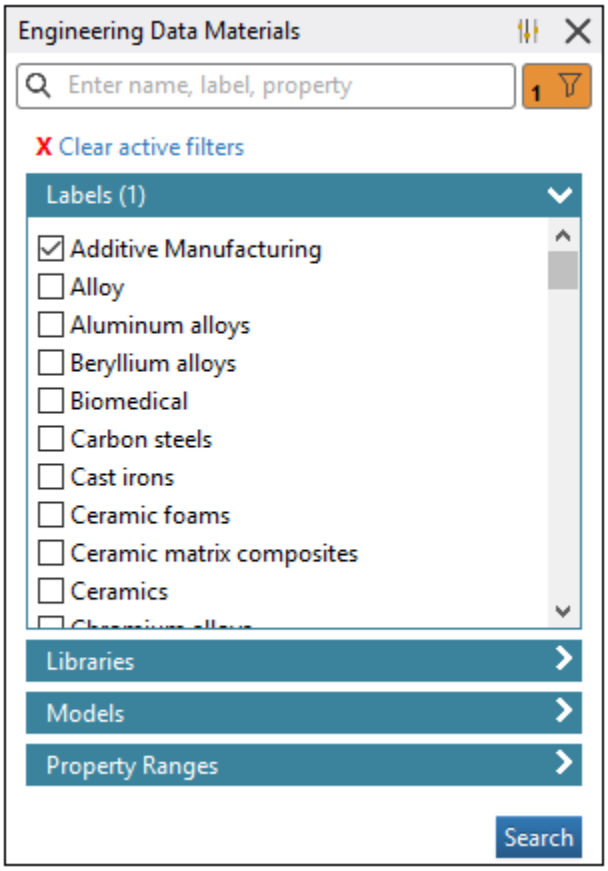
On initial entry it is possible to show all the materials available to search by choosing **Show All**. Selections can be made in each of the criteria-based drop-down groups to narrow your search and then clicking on the **Search** button at the bottom of the panel. All criteria must be matched in a material for it to be returned in the search. Note that choices in the groups will be filtered as you select to avoid choosing invalid criteria. When available, clicking on **Clear active filters** will remove all filters.

Labels

The **Labels** group displays the labels attached to materials which correspond to industry-based categories for ease of selection. These same labels can be typed into the search field. Note that not all materials will be labeled.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

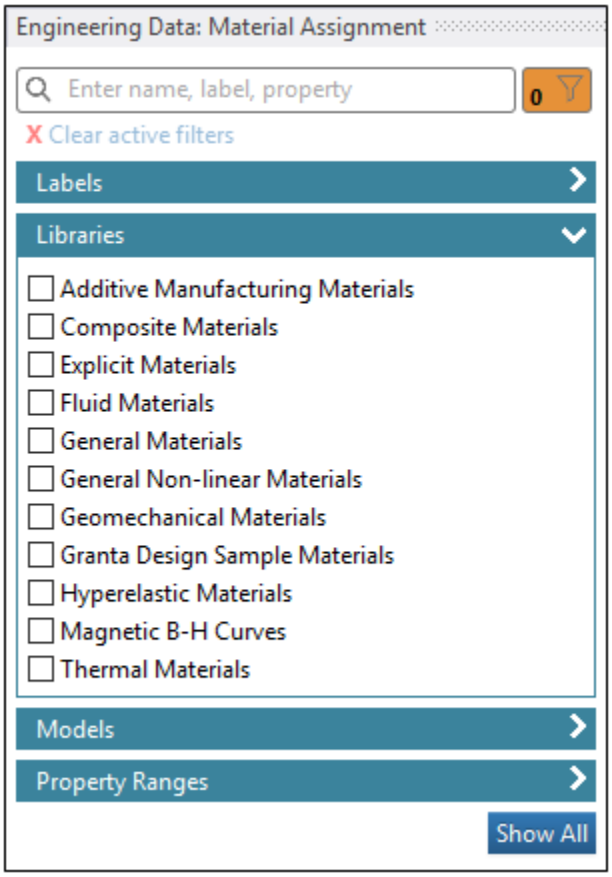


Libraries

The **Libraries** group displays those libraries available for the search. The available libraries can be modified in the Settings Panel. Note that some libraries may be not available in your product.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

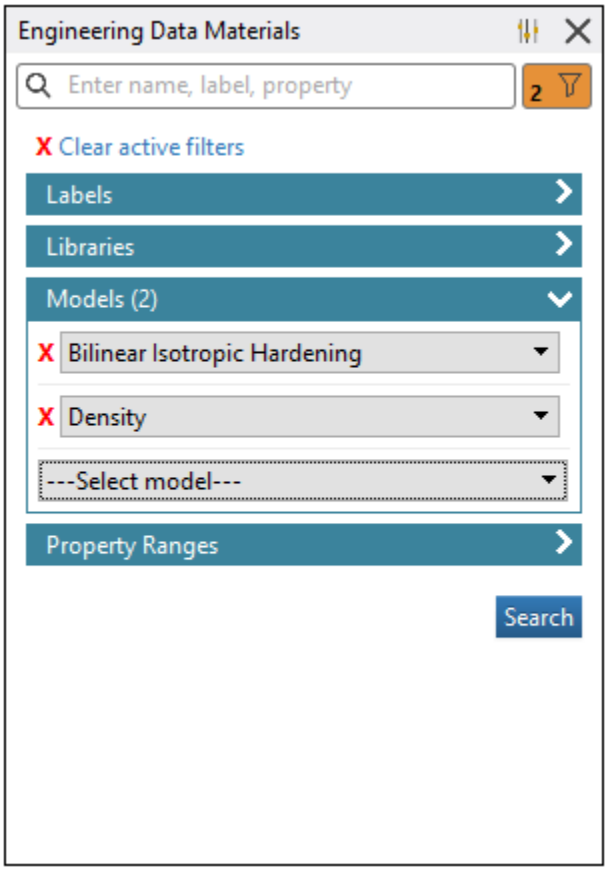


Models

The **Models** group enables you to filter to those materials which have the selected models.

DRAFT

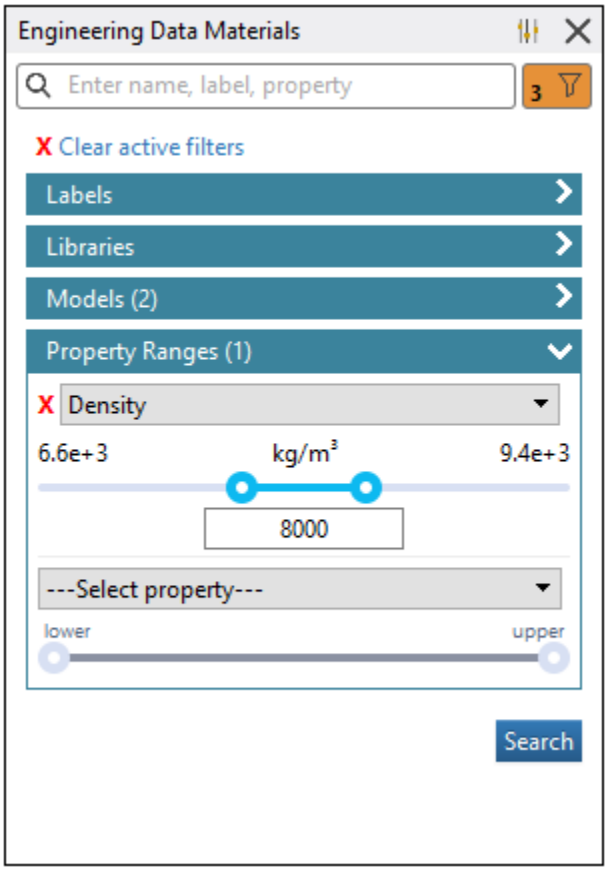
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Property Ranges

The **Property Ranges** group allows you to enter a target value for a selected property to filter to those materials having the property with a value in the range of the target. You may also drag the end points to create a custom range as well. The values on the left and right first show the lower and upper range of values for filtered materials.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

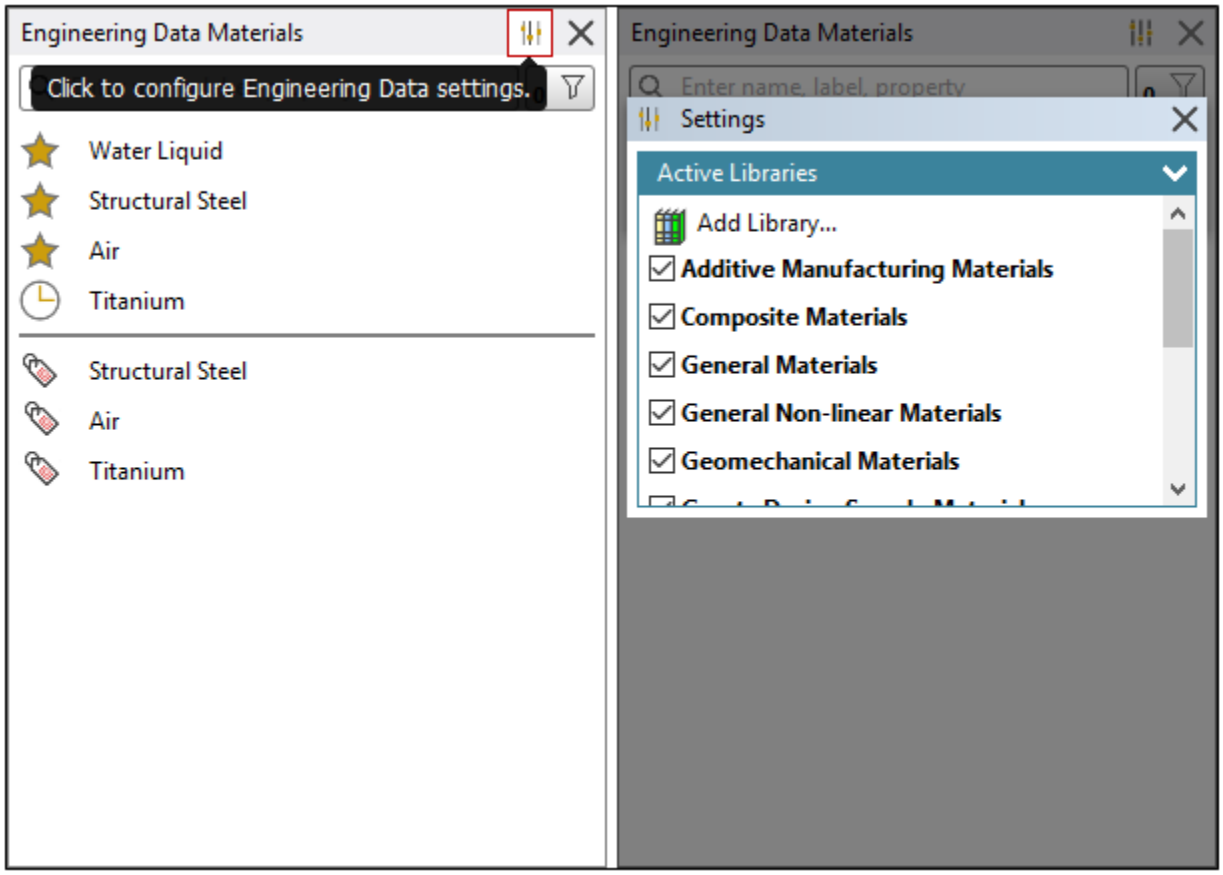


Engineering Data Settings

Selecting the Settings icon displays a **Settings** panel. The **Active Libraries** group enables you to choose which libraries are active for searches. The default installation has libraries which are not active. You can also add libraries from other locations to the active selections. The selected libraries are persisted from session to session. Note that some libraries may be not available in your product.

DRAFT

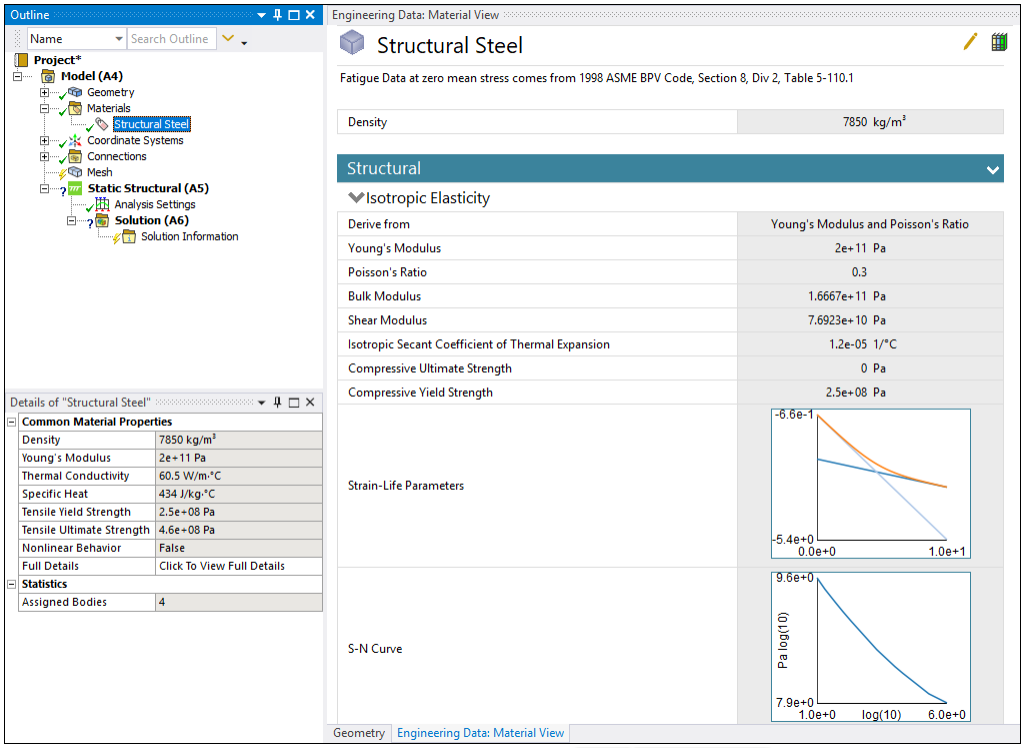
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



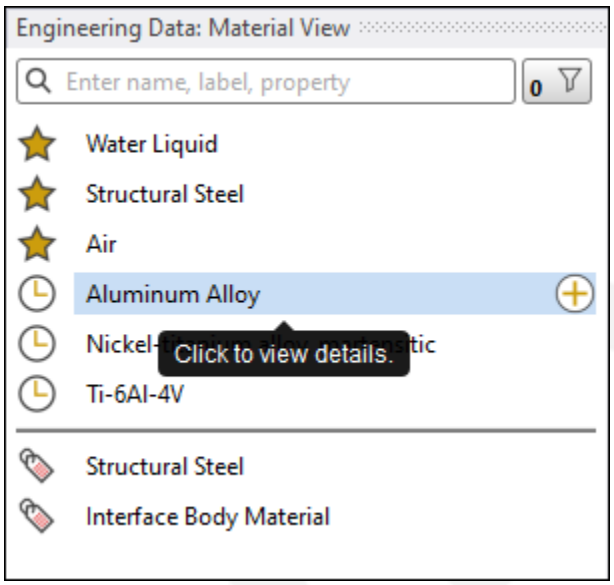
Material View

As illustrated below, when you select a Material, the **Material View** displays material data, in this example, Structural Steel. This view enables you to review the material information, for each material in Mechanical, and access edit mode.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



An option (**Click to view details.**) is also displays when you highlight a material in any of the material view panes.



Heading Components

Color Block

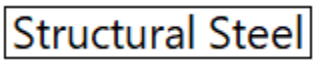
The color of the block indicates the color that will be used to display this material in a model. An imported material that doesn't have a color will be assigned a random color.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Name

The name of the material.



Edit

The edit (pencil) icon will navigate to the Engineering Data Workspace to allow data to be modified and/or parameterized. Once the edit is completed the material must be refreshed in Mechanical. The refresh can be accessed on the Materials group via right-click.

Label

The label tag will display the labels associated with a material when you hover to show the tooltip.

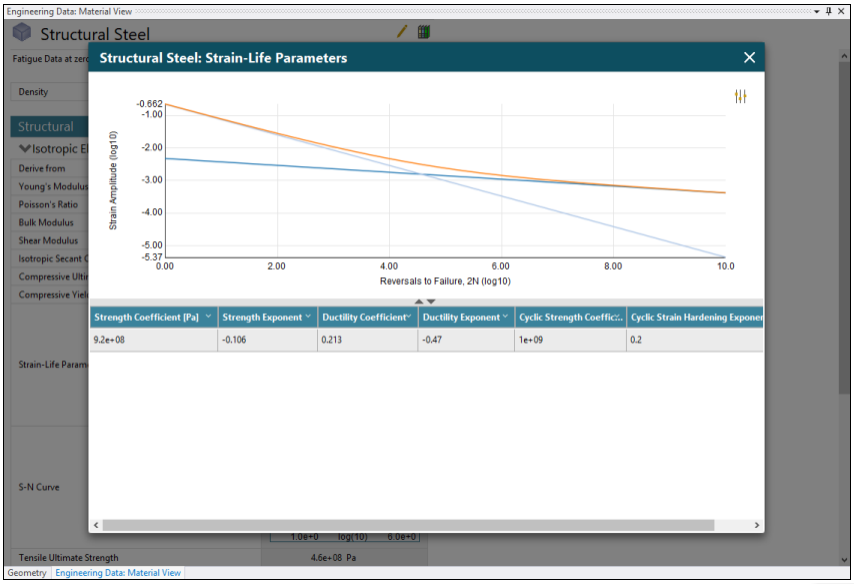
Library

The library icon will display the library that the material belongs to when you hover to show the tooltip.

Material Model Data

For each material, the window displays data for the material model. This data can be a value or charted data when there are field variables (S-N Curve). For variable data, you can select the graph to display additional data, such as the associated tabular data. The various material data is organized by physics type. Collapsible headings for Structural, Thermal, Electric, and Magnetic are common physics types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Objects Supporting Material Assignment

The following objects support material assignment:

- Analysis Ply
- Beam Connection
- Bearing
- Contact Region
- Delamination
- Joint
- Load Conditions
- Parts and Bodies
- Point Mass
- Remote Point
- Spring
- Surface Coating

Windows Management

The Mechanical interface contains window panes that house graphics, outlines, tables, object details, and other views and controls. Window management features allow you to move, resize, tab-dock, and auto-hide window panes. A window pane that is "tab-docked" is collapsed and displayed at the side of the application interface. Auto-hide indicates that a window pane (or tab-docked group of panes) automatically collapses when not in use.

Auto-Hiding

Panes are either pinned or unpinned. Toggle this state by clicking the icon in the pane title bar. A pinned pane occupies space in the window. An unpinned pane collapses to a tab on the periphery of the window when inactive.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To examine an unpinned pane, move the mouse pointer over the tab. This causes the pane to open on top of any other open window panes. Holding the mouse pointer over the tab keeps the tab visible. Clicking the tab activates the window pane (also causing it to remain visible). Pin the pane to restore it to its open state.

Moving and Docking

Drag a window's title bar to move and undock a window pane. Once you begin to drag the window, a number of dock targets (blue-filled arrows and circle) appear in the interface window. At this point you:

1. Move the mouse pointer over a target to preview the resulting location for the pane. Arrow targets indicate adjacent locations; a circular target enables tab-docking of two or more panes (to share screen space).
2. Release the button on the target to move the pane. You can abort the drag operation by pressing the **ESC** key.

Tip:

You can also double-click a window's title bar to undock the window and move it freely around the screen. Once undocked, you can resize the window by dragging its borders/corners.

Restore Original Window Layout

Select the **Reset Layout** option in the **Layout** group (p. 47) on the **Home** tab to return to the default/original pane configuration.

Preference Migration

The first time you run Ansys Workbench following the installation of a new version of the software, the application automatically migrates your user preferences to the new version. This includes preferences such as licensing settings, **Options** (p. 217) settings, solver preferences, user created buttons, Key Assignments, and Engineering Data settings.

To migrate preferences, the application creates a text file (`MigratePreferences.txt`) in the following directory:

Windows: C:\Users\John_Doe\AppData\Roaming\Ansys\<new_version>

Linux: ~/.ansys/<new_version>

If this file does not exist, the application migrates your preferences. Once the application has generated this text file, the migration no longer runs.

Turn Automatic Migration Off

You can turn migration off by:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Creating an empty file called %APPDATA%/Ansys/v<new_version>/MigratePreferences.txt before the first run.

  Or...

- Starting Workbench with (Target property) "RunWB2.exe -K Framework.MigratePreferences=off"

Remove Migrated Preference Data

You can remove the migrated user preference data by emptying the %APPDATA%/Ansys/v<new_version> directory and creating the empty file %APPDATA%/Ansys/v<new_version>/MigratePreferences.txt. Make sure that no Ansys product is running during this action.

Resource Prediction

The **Resource Prediction** option (**Home** Tab/**Solve** Group) displays a window that enables you to select an analysis and predict the computing resources needed to perform a solution for the environment. The application displays a prediction for time, memory, and disk space requirements. The prediction is displayed in green at the top of the page. When initiated, the progress of the calculation is displayed in the left corner of the Status Bar (p. 135). The progress bar includes a stop button to end the process.

Note:

- A successful prediction does not guarantee a successful solution.
- This feature presents estimates based on simulations with similar characteristics and should only be used as a reference. Your actual simulation may not solve as suggested.
- The prediction applies to the solution using distributed-memory parallel (DMP) processing.

Go to a section topic:

- Requirements and Limitations (p. 170)
- Static Structural Analysis Prediction (p. 171)
- Modal (stand-alone only) Analysis Prediction (p. 171)
- Resource Prediction under Solution Output (p. 172)

Requirements and Limitations

Review the following requirements and limitations:

- The mesh must be generated.
- Supports linear Static Structural and Modal (stand-alone) systems only.
- The resources predicted are the computational resources needed to perform one step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- This feature is not supported on the Linux platform.

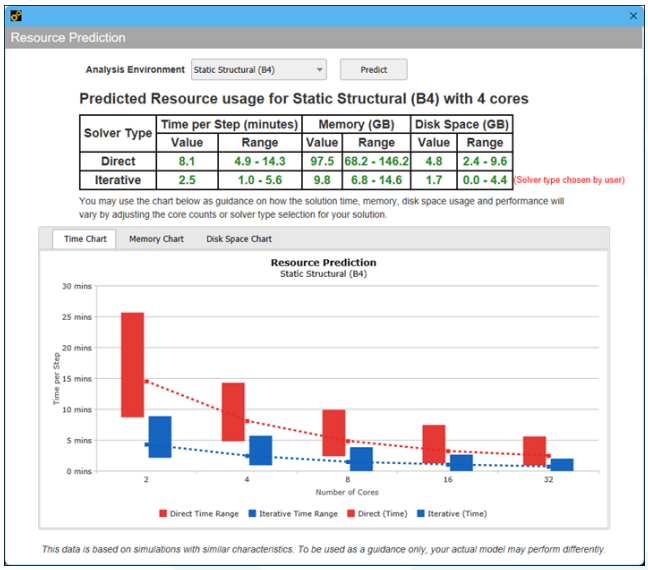
Static Structural Analysis Prediction

By default, a prediction is automatically performed for both the **Direct** and **Iterative** solver settings. The prediction display, as illustrated below, includes red text in parenthesis beside the option specified in the **Solver Type** property. If the **Solver Type** property is set to **Program Controlled**, then the red text indicates the option selected by the application.

Note:

Resource prediction using the iterative solver setting assumes convergence and it does not consider any impact if the solver setting switches from the iterative solver to the direct solver.

Time Chart Static Structural Analysis

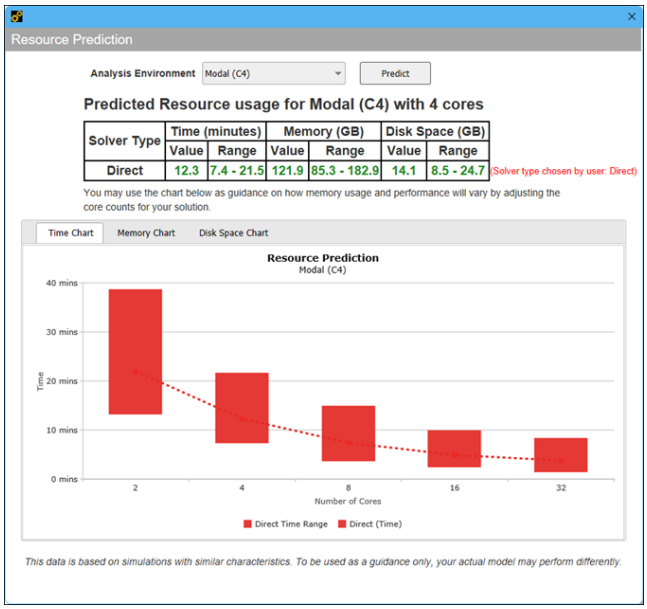


Modal (Stand-Alone Only) Analysis Prediction

A stand-alone Modal analysis supports resource prediction using the **Direct** setting for the **Solver Type** property only, either setting the option manually or through the use of the **Program Controlled** option.

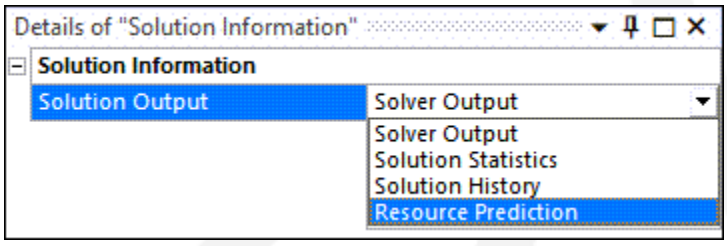
Time Chart for Modal Analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Resource Prediction under Solution Output

The **Solution Output** property of the Solution Information (p. 1833) object provides a **Resource Prediction** option. When you write an input file or preform a solution, use this option to display resource predicted values in the Worksheet.

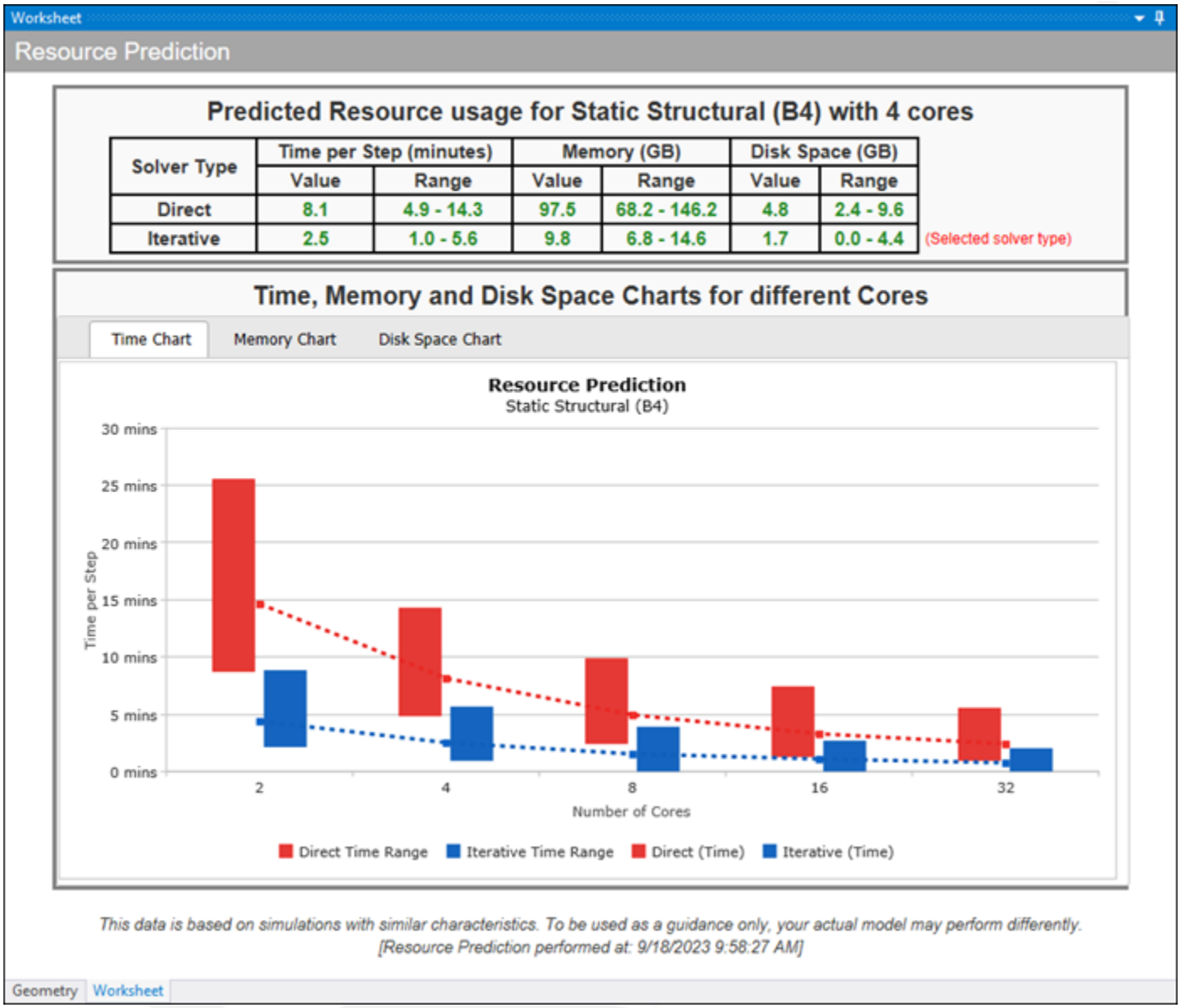


This option displays the following information:

- Predicted computational resources for the core count you specify, up to 32 cores.
- Time, Memory, and Disk Space charts that display how resources vary per core count.
- As applicable, suggestions as per predicted values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

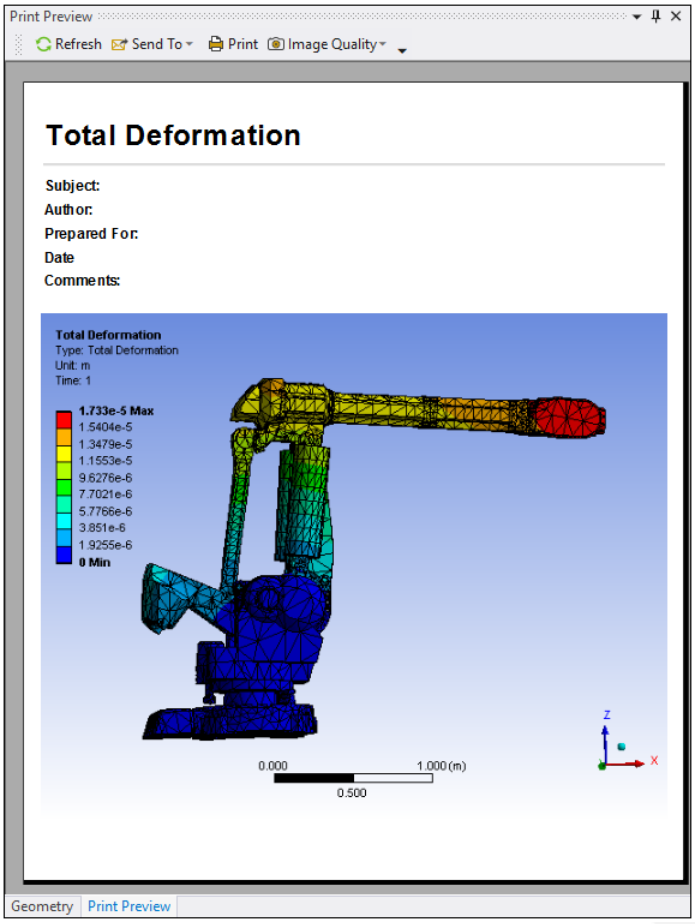
This feature only supports:

- Linear solutions for Static Structural analyses.
- Modal analyses for the **Direct Solver** setting of the **Solver Type** property (**Analysis Settings** > Solver Controls (p. 1428)), including when the **Program Controlled** setting selects **Direct Solver**.

Print Preview

The **Print Preview** option, selected from the **Tools** (p. 45) group on the **Home** tab, runs a script to generate an HTML page and an image for a selected object, such as the Deformation result shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The title block is an editable HTML table. The table initially contains the **Author**, **Subject**, **Prepared For** and **Date** information supplied from the Details of the **Project** tree node. To change or add this information, double-click inside the table. The information entered in the table does not propagate any changes back to the details view and is not saved after exiting the **Print Preview** tab.

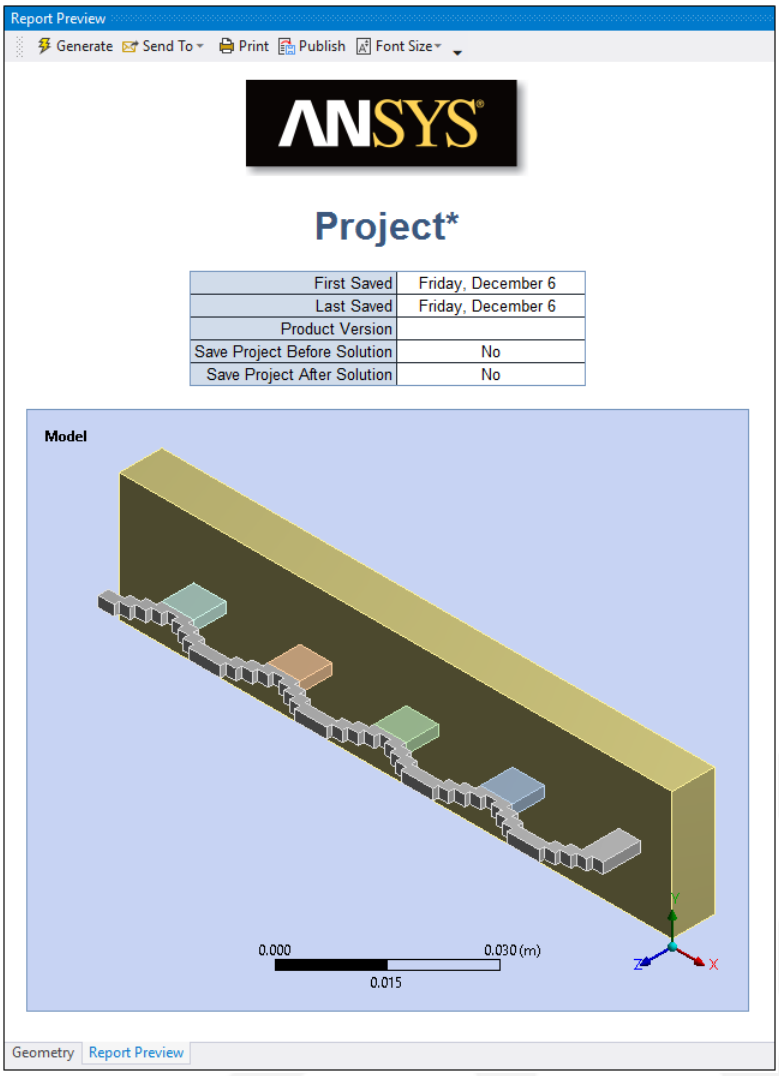
The image is generated in the same way as figures in **Report Preview**. The new **Print Preview** copies all current view settings, including those defined in the Options (p. 217), such as the **Font Magnification Factor**.

Report Preview

The **Report Preview** option, selected from the **Tools** (p. 45) group on the **Home** tab, enables you to create a report based on the analyses in the **Outline**. This report selects items in the **Outline**, examines the worksheets for it, then appends any material data used in the analysis. The report generation process starts immediately, and, once started, it must run to completion before you can begin working in the interface again.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The generation process of the feature starts as soon as you select the option. Unlike prior report generators, this system works by extracting information from the user interface. It first selects each item in the **Outline**, then examines worksheets in a second pass, and finally appends any material data used in the analysis. The material data will be expressed in the Workbench standard unit system which most closely matches Mechanical's unit system. Once started the report generation process must run to completion. Avoid clicking anywhere else in Workbench during the run because this will stop the report process and may cause an error. This approach to reporting ensures consistency, completeness, and accuracy.

Important:

When running multiple Mechanical sessions, the application automatically overwrites any existing files (MHT, HTML, Word, or PPT) if you generate report outputs without first managing them with the Publish feature or by copying files to a new location.

Note:

Not all **Report Preview** options are available on the Linux platform.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In addition to the Tables (p. 176) and Figures (p. 176) topics described below, this section examines:

Publishing the Report

Sending the Report

Comparing Databases

Customizing Report Content

Tables

Most tables in the report directly correspond to the **Details** of an object or set of related objects. **Object** names appear across the top of the tables.

By default, tables displays 12 columns. This setting increases the likelihood that tables will fit on the screen and on printed pages. In the **Report Options** (p. 237) dialog you can increase or decrease the setting for the number of columns you wish to display by default. For example, you may allow more columns if object names take up little space, if you have a high resolution screen, or print in landscape layout. The minimum is two columns, in which case no grouping of objects occurs and the Contents is equivalent to the **Outline**.

The system merges identical table cells by default. This reduces clutter and helps to reveal patterns. You can disable this feature in the **Report Options** (p. 237) dialog.

Note:

The **Report Preview** feature does not display table entries from the nonlinear joint stiffness matrix.

Figures and Images

Figures and Images appear in the report as specified in the **Outline**. The system automatically inserts charts as needed. The system creates all bitmap files in PNG format.

You may change the size of charts and figures in the **Report** (p. 237) preference in the default **Options** (p. 217) settings. For example, you may specify smaller charts due to few data points or bigger figures if you plan to print on large paper. For best print quality, increase the **Graphics Resolution** in the **Report** (p. 237) preference. In addition, you can increase legend font sizes using the **Font Magnification Factor** option under the **Graphics Options** (p. 228) preference.

Publishing the Report

Click the **Publish** option to save your report as a single HTML file that includes the picture files in a given folder, or as an HTML file with a folder containing picture files. The first option produces a single MHT file containing the HTML and pictures. MHT is the same format used by Internet Explorer when a page is saved as a "Web Archive". Only Internet Explorer 5.5 or later on Windows supports MHT. For the other two options, the HTML file is valid XHTML 1.0 Transitional.

Full support for MHT file format by any other browser cannot be guaranteed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Sending the Report

The **Send To** feature enables you to publish your report as:

- An email attachment. This option attaches a single MHT file automatically to your email application.

  Some email systems may strip or filter MHT files from incoming messages. If this occurs, email a ZIP archive of a published report or email the report from Microsoft Word.

- A Word document. This format is equivalent to opening a published HTML file in the application.

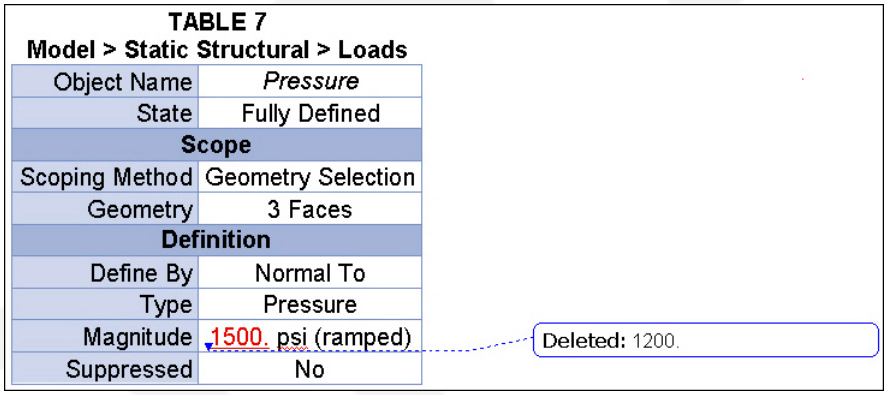
- A PowerPoint presentation (images only). A presentation is automatically created and includes the images (one per slide) of your analysis. It includes no other report information.

Note:

If you have multiple Mechanical sessions open, the application overwrites any reports that you have produced in the above formats if do not manage the reports with the **Publish** feature or by copying the files to a different location.

Comparing Databases

Because the report content directly corresponds to the user interface, it is easy to determine exactly how two databases differ. Generate a report for the first database, open it in Word, save and exit. Open the report for the second database in Word and choose **Compare Documents**. In the dialog, clear the **Find Formatting** box and select the first file. Word highlights the differences, as illustrated here:



Customizing Report Content

Report customization falls into two categories: preferences in the **Report Options** (p. 237) dialog and the ability to run a modified report generator from a local or network location. This ability to externalize the system is shared by the Mechanical Wizard (p. 321). It allows for modifications outside of the installation folder and reuse of a customized system by multiple users.

To run report externally:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Copy the following folder to a different location: `Program Files\ANSYS Inc\v241\aisol\DesignSpace\DSPages\Language\en-us\Report2006`.
2. Specify the location under **Custom Report Generator Folder** in the **Report Options** (p. 237) (for example: \\server\copied_Report2006_folder).

The easiest customization is to simply replace `Logo.png`. The system uses that image on the wait screen and on the report cover page.

The file `Template.xml` provides the report skeleton. Editing this file allows:

- Reformatting of the report by changing the CSS style rules.
- Addition of standard content at specific points inside the report body. This includes anything supported by XHTML, including images and tables.

The file `Rules.xml` contains editable configuration information:

- Standard files to include and publish with reports. The first is always the logo; other files could be listed as the images used for custom XHTML content.
- Rules for excluding or bolding objects in the **Contents**.
- Rules for applying headings when objects are encountered.
- Selective exclusion of an object's details. For example, part Color (extracted as a single number) isn't meaningful in a report.
- Exclusion of **Graph** figures for certain objects. This overrides the other four criteria used to decide if a **Graph** figure is meaningful.
- Rules against comparing certain types of objects.
- Object states that are acceptable in a "finalized" report.
- Search and replace of **Details** text. For example, the report switches "Click to Change" to "Defined". This capability allows for the use of custom terminology.
- Insertion of custom XHTML content based on object, analysis and physics types, and whether the content applies to the details table, the chart or the tabular data. For example, report includes a paragraph describing the modal analysis bar chart.

All files in the `Report2006` folder contain comments detailing customization techniques.

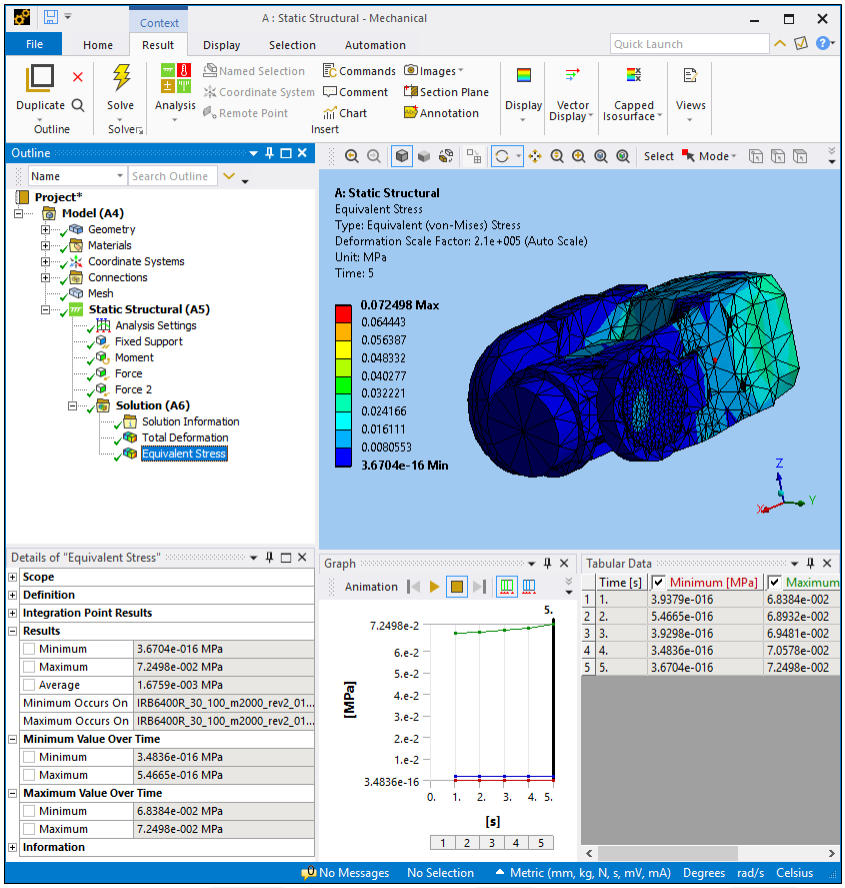
Full Screen Mode

The illustrations below depict the default layout of Mechanical as well as the full screen mode used to view/present your analysis. You activate full screen mode by selecting the **Full Screen** option from the **Layout** (p. 47) group on the **Home** tab. Or, you can use the **F11** key when the **Geometry** window, the tree **Outline**, or the **Details** view have focus. The tab option and the **F11** key toggle the display on and/or off.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

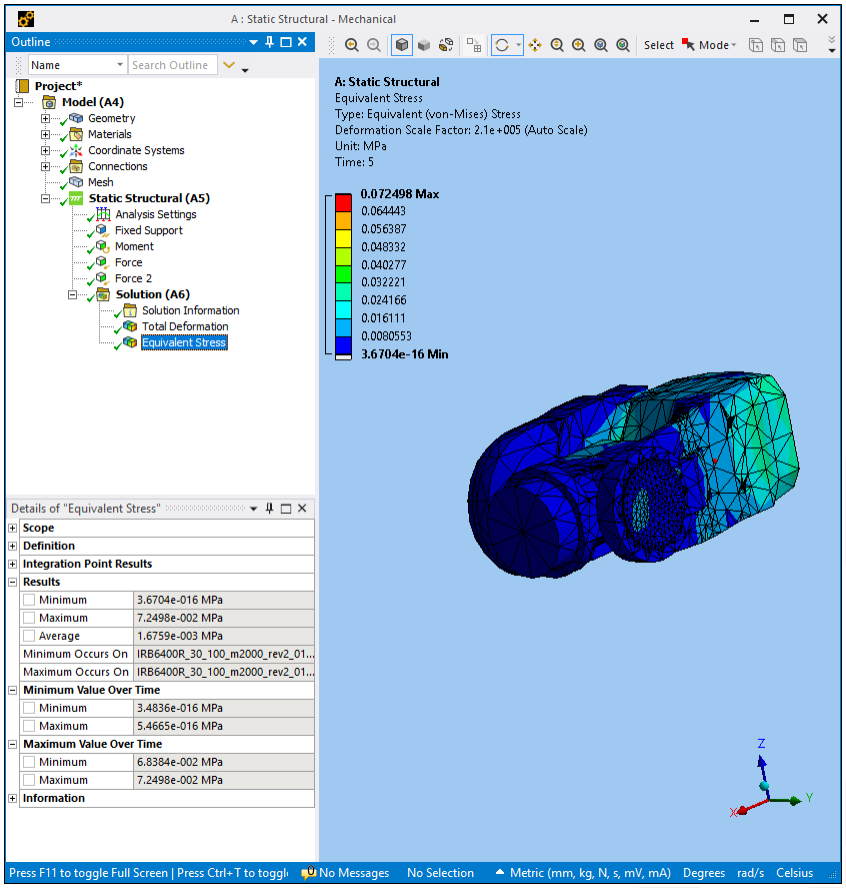
When using this mode, the presentation inherits the active selection tool. For example, if you had the **Body** selection option active, it continues to be active in full screen mode. This includes other options such as **Wireframe** and **Show Mesh**. Based on the active selection filter, you can affect the model as desired. In addition, when in the full screen mode, you may find it useful to use the available keyboard shortcuts (Key Assignments (p. 314)). These options enable you to quickly change the selection options.



By default, full screen mode maximizes the display based on the largest resolution available for your computer monitor. This is useful when you if you want to maximize the display for a model when you have a smaller screen, such as laptop.

The tree **Outline** panel displays by default while presenting your analysis. You can use the Hotkey combination **Ctrl**+**O** to toggle the **Outline** on and off. You can also use the Hotkey combination **Ctrl**+**D** to toggle the **Details** view on and off.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Contextual Windows

Mechanical provides a number of feature-based windows. This section discusses the following:

Selection Information Window

Worksheet Window

Graph and Tabular Data Windows

Messages Window

Graphics Annotations Window

Section Planes Window

Mechanical Wizard Window

Selection Information Window

The **Selection Information** window provides a quick and easy way for you to interrogate and find geometric information on items that you have selected on the model.

The following topics are covered in this section:

Activating the Selection Information Window

Understanding the Selection Modes

Using the Selection Information Window

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

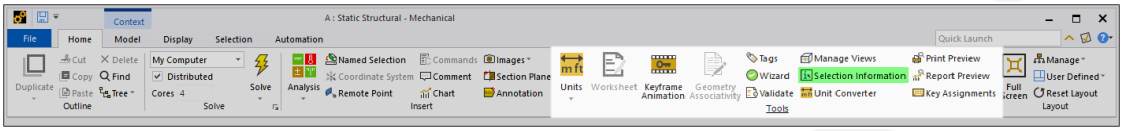
Published: 2023-10-23

Selecting, Exporting, and Sorting Data

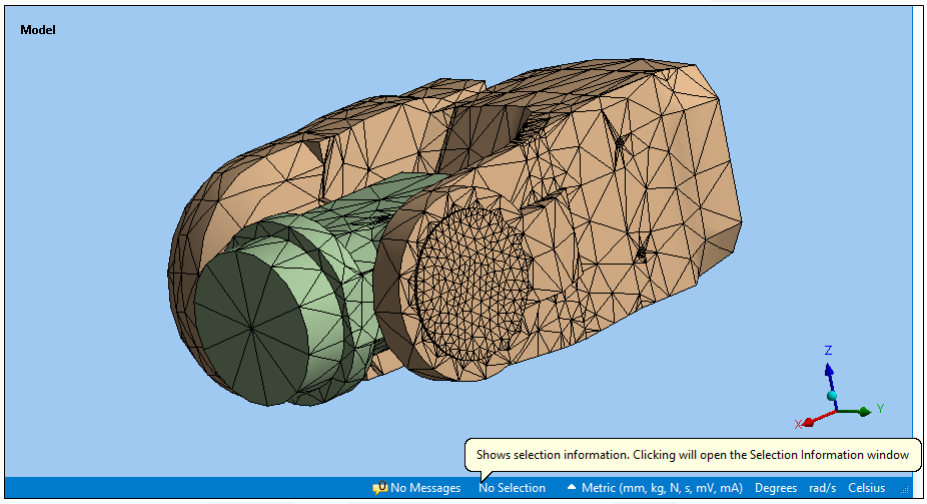
Activating the Selection Information Window

You can display the **Selection Information** window using any of the following methods:

- Select the **Selection Information** option in the **Tools** (p. 45) group on the **Home** tab (p. 40).



- Double-click the field on the status bar that displays the Selection field.



Understanding the Selection Modes

The supported selection modes are vertex, edge, face, body, node, and coordinate, as described below. Common reported information for each mode, except coordinate, includes x, y, z locations and if two selections are made, the distance between their centroids is reported.

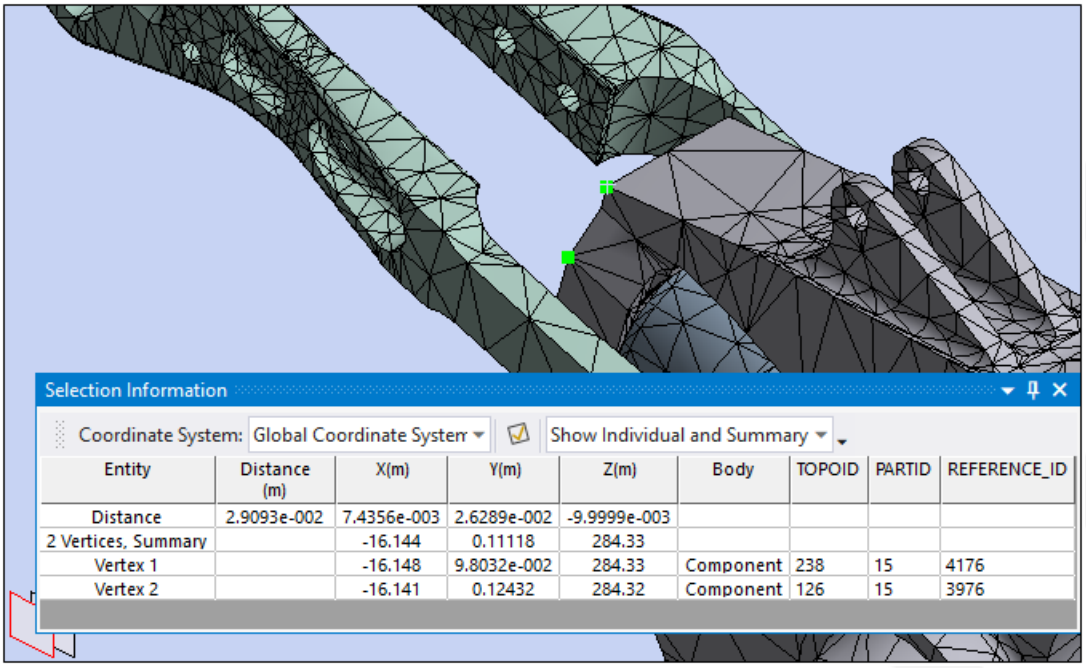
Note:

Selection Information may not be available for virtual entities.

Vertex

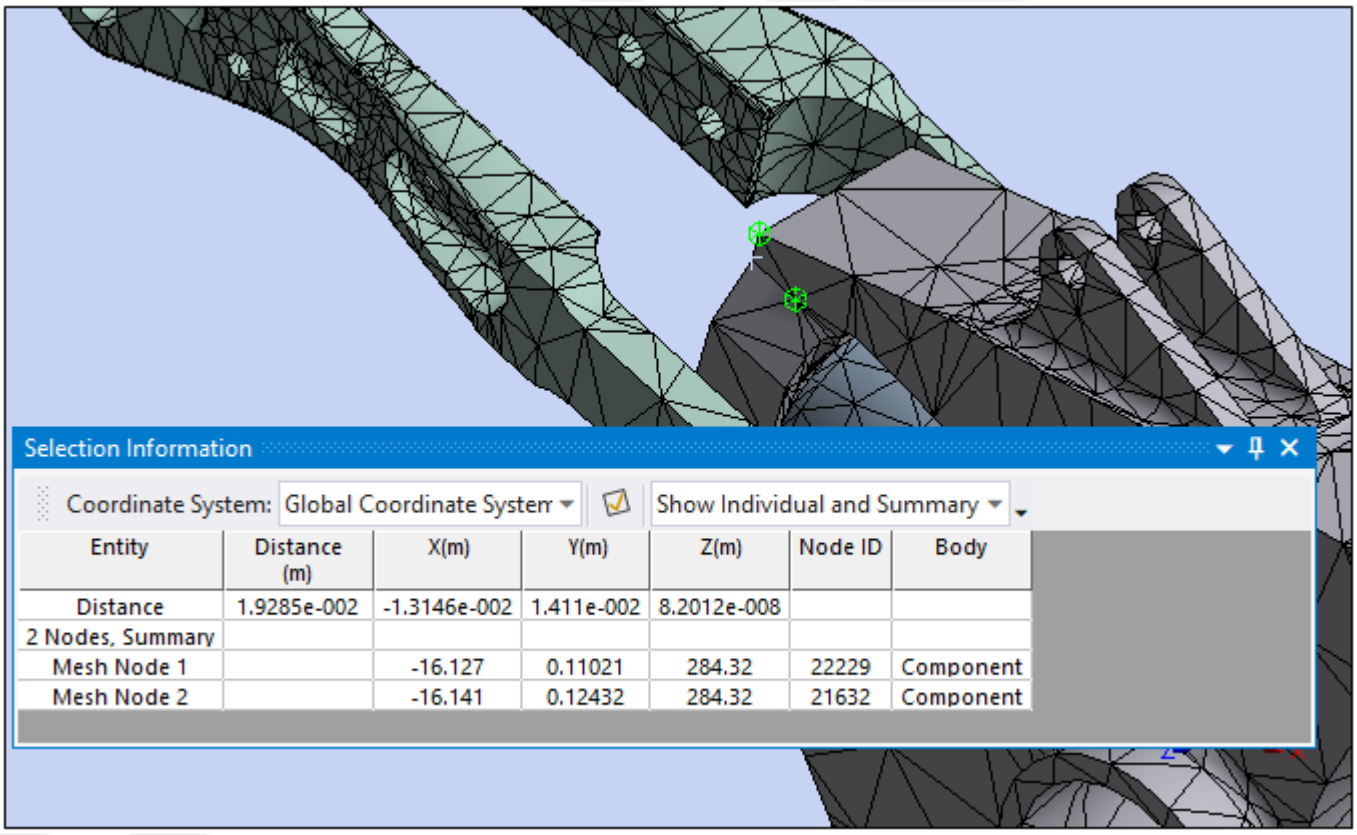
Individual vertex location and average location are reported. The bodies that the vertex attaches to are also reported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Node

The information displayed for selected nodes is similar to a vertex with the addition of the Node ID.

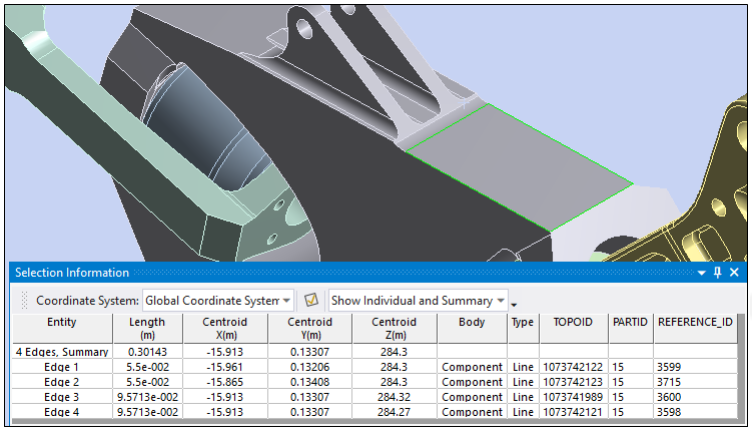


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Edge

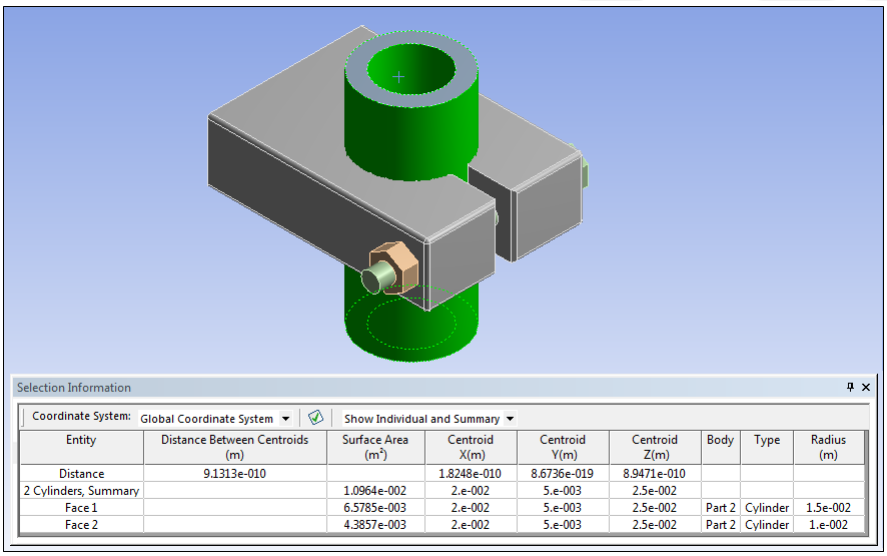
Combined and individual edge length and centroid are reported. The bodies that the edge attaches to are reported. The type of the edge is also reported. If an edge is of circle type, the radius of the edge is reported.



Entity	Length (m)	Centroid X(m)	Centroid Y(m)	Centroid Z(m)	Body	Type	TOPOID	PARTID	REFERENCE_ID
4 Edges, Summary	0.30143	-15.913	0.13307	284.3					
Edge 1	5.5e-002	-15.961	0.13206	284.3	Component	Line	1073742122	15	3599
Edge 2	5.5e-002	-15.865	0.13408	284.3	Component	Line	1073742123	15	3715
Edge 3	9.5713e-002	-15.913	0.13307	284.32	Component	Line	1073741989	15	3600
Edge 4	9.5713e-002	-15.913	0.13307	284.27	Component	Line	1073742121	15	3598

Face

Combined and individual area and centroid are reported. The bodies that the face attaches to are reported. The type of the face is reported. If a face is of cylinder type, the radius of the face is also reported.

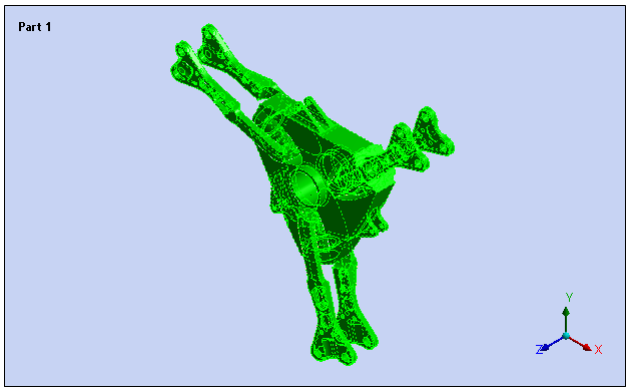


Entity	Distance Between Centroids (m)	Surface Area (m^2)	Centroid X(m)	Centroid Y(m)	Centroid Z(m)	Body	Type	Radius (m)
Distance	9.1313e-010		1.8248e-010	8.6736e-019	8.9471e-010			
2 Cylinders, Summary		1.0964e-002	2.e-002	5.e-003	2.5e-002			
Face 1		6.5785e-003	2.e-002	5.e-003	2.5e-002	Part 2	Cylinder	1.5e-002
Face 2		4.3857e-003	2.e-002	5.e-003	2.5e-002	Part 2	Cylinder	1.e-002

Body

Combined and individual volume, mass, and centroid are reported. The body name is reported. Your choice of the mass moment of inertia in the selected coordinate system or the principal is also reported. The choice is provided in the **Selection Information Column Control** (p. 187) dialog box.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Selection Information

Coordinate System: Global Coordinate System | Show Individual and Summary

Entity	Volume (m^3)	Mass (kg)	Centroid X(m)	Centroid Y(m)	Centroid Z(m)	Nodes	Elements	Body	Type	TOPOID	PARTID	REFERENCE_ID
7 Bodies, Summary	5.1249e-003	40.231	-15.987	-1.4246e-003	284.3	0	0					
Body 1	2.4173e-004	1.8976	-15.985	-0.12627	284.3	0	0	Part 12	Solid	3221225473	3	30
Body 2	2.4173e-004	1.8976	-16.096	5.9713e-002	284.3	0	0	Part 175	Solid	3221225473	5	74
Body 3	2.4173e-004	1.8976	-15.88	6.2289e-002	284.3	0	0	Part 344	Solid	3221225473	7	127
Body 4	4.3255e-004	3.3955	-15.952	-0.25875	284.3	0	0	Part 13	Solid	3221225473	9	863
Body 5	4.3255e-004	3.3955	-16.227	9.695e-002	284.3	0	0	Part 158	Solid	3221225473	11	1884
Body 6	4.3255e-004	3.3955	-15.782	0.15755	284.3	0	0	Part 326	Solid	3221225473	13	2752
Body 7	3.1021e-003	24.351	-15.987	-1.4281e-003	284.29	0	0	Component	Solid	3221225473	15	3760

Selection Information

Coordinate System: Global Coordinate System | Show Individual and Summary

Entity	Volume (m^3)	Mass (kg)	Centroid X(m)	Centroid Y(m)	Centroid Z(m)	Nodes	Elements	Body	Type	TOPOID	PARTID	REFERENCE_ID
7 Bodies, Summary	5.1249e-003	40.231	-15.987	-1.4246e-003	284.3	0	0					
Body 1	2.4173e-004	1.8976	-15.985	-0.12627	284.3	0	0	Part 12	Solid	3221225473	3	30
Body 2	2.4173e-004	1.8976	-16.096	5.9713e-002	284.3	0	0	Part 175	Solid	3221225473	5	74
Body 3	2.4173e-004	1.8976	-15.88	6.2289e-002	284.3	0	0	Part 344	Solid	3221225473	7	127
Body 4	4.3255e-004	3.3955	-15.952	-0.25875	284.3	0	0	Part 13	Solid	3221225473	9	863
Body 5	4.3255e-004	3.3955	-16.227	9.695e-002	284.3	0	0	Part 158	Solid	3221225473	11	1884
Body 6	4.3255e-004	3.3955	-15.782	0.15755	284.3	0	0	Part 326	Solid	3221225473	13	2752
Body 7	3.1021e-003	24.351	-15.987	-1.4281e-003	284.29	0	0	Component	Solid	3221225473	15	3760

Important:

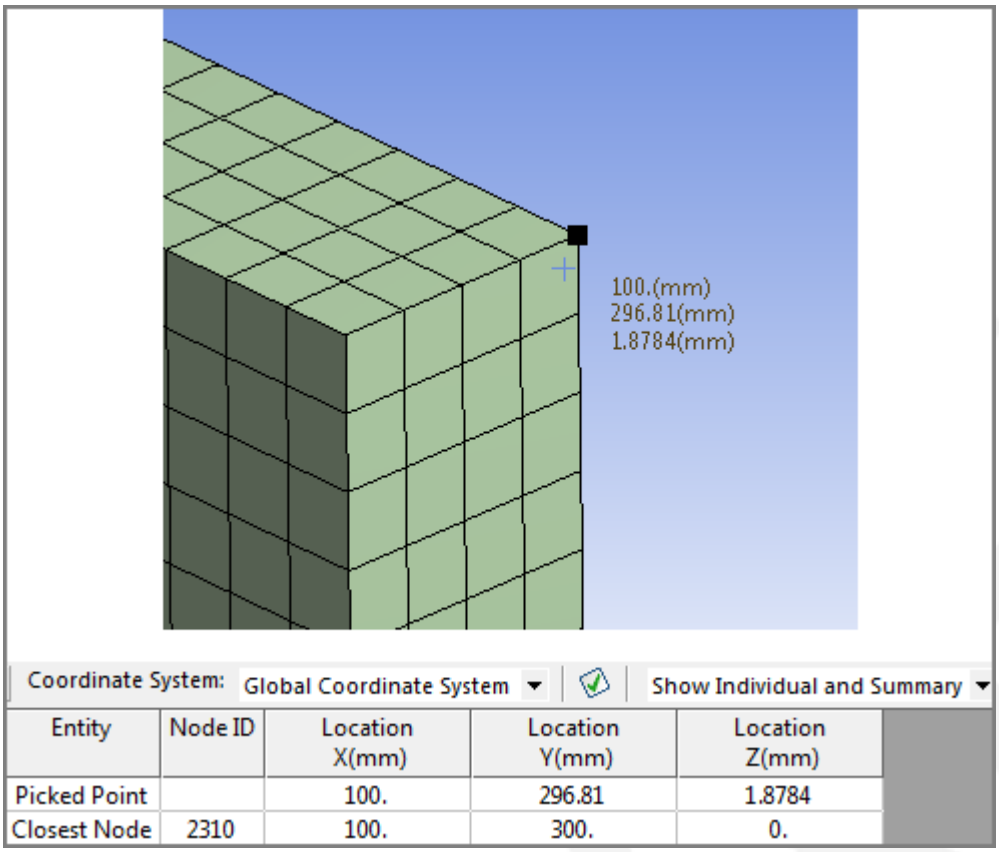
For a Surface Body, the Volume, Mass, and Moment of Inertia information for the Body selection are based on the original thickness value specified on the Surface Body object. This does not account for any Thickness object specifications. Thickness specifications overwrite the body thickness values when the application calculates thickness for any faces and/or surface bodies. Refer to the PRECISE MASS SUMMARY section from the Solution Information worksheet for solver calculated Mass values.

Coordinate

If there is a mesh present, the picked point location and the closest mesh node ID and location are reported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

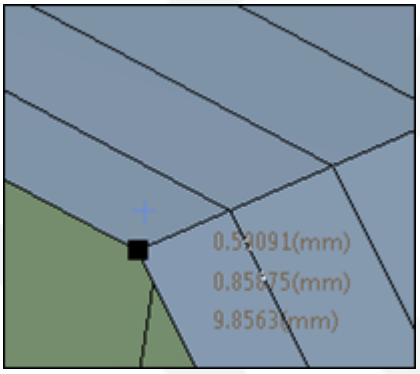
Published: 2023-10-23



Entity	Node ID	Location X(mm)	Location Y(mm)	Location Z(mm)
Picked Point		100.	296.81	1.8784
Closest Node	2310	100.	300.	0.

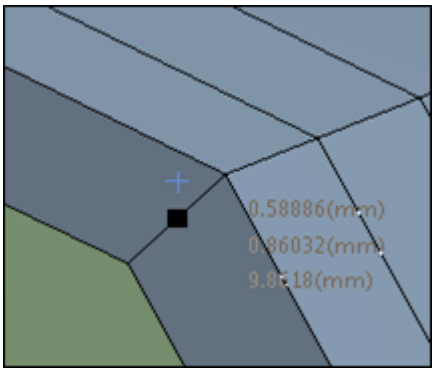
In the case of a surface body model, the closest node will be located on the non-expanded mesh (that can be seen if you select the **Thick Shells and Beams** option from the **Style** group on the **Display** tab).

Non-expanded shell view:



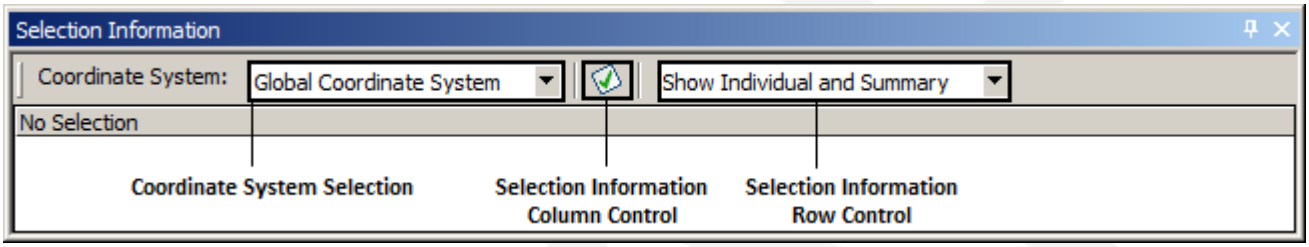
Expanded shell view:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Using the Selection Information Window

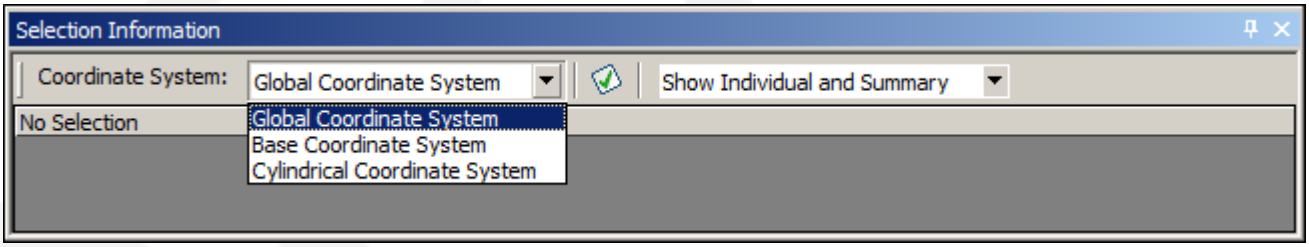
The tools located at the top of the **Selection Information** window includes the following controls:



Each of these controls is described below.

Coordinate System

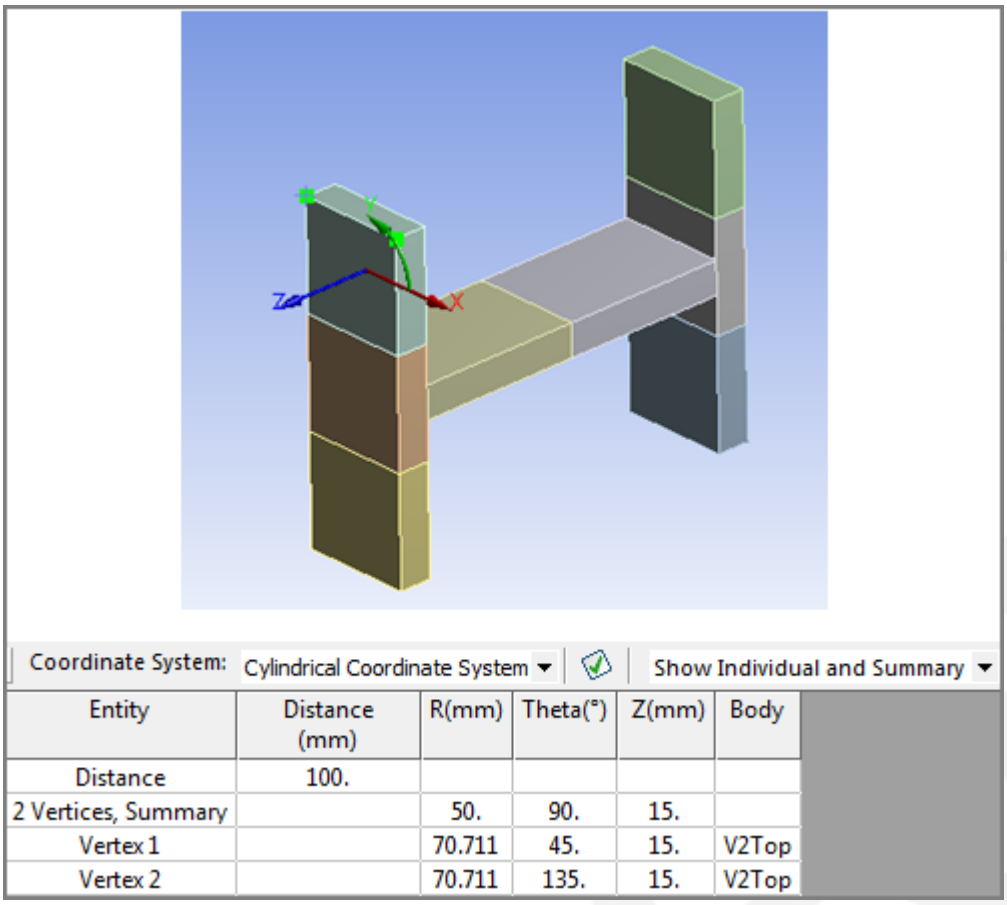
A **Coordinate System** drop-down selection box is provided. You can select the coordinate system under which the selection information is reported. The centroid, location, and moment of inertia information respect the selected coordinate system.



For example, if a cylindrical coordinate system is selected, the vertex location is reported using the cylindrical coordinates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

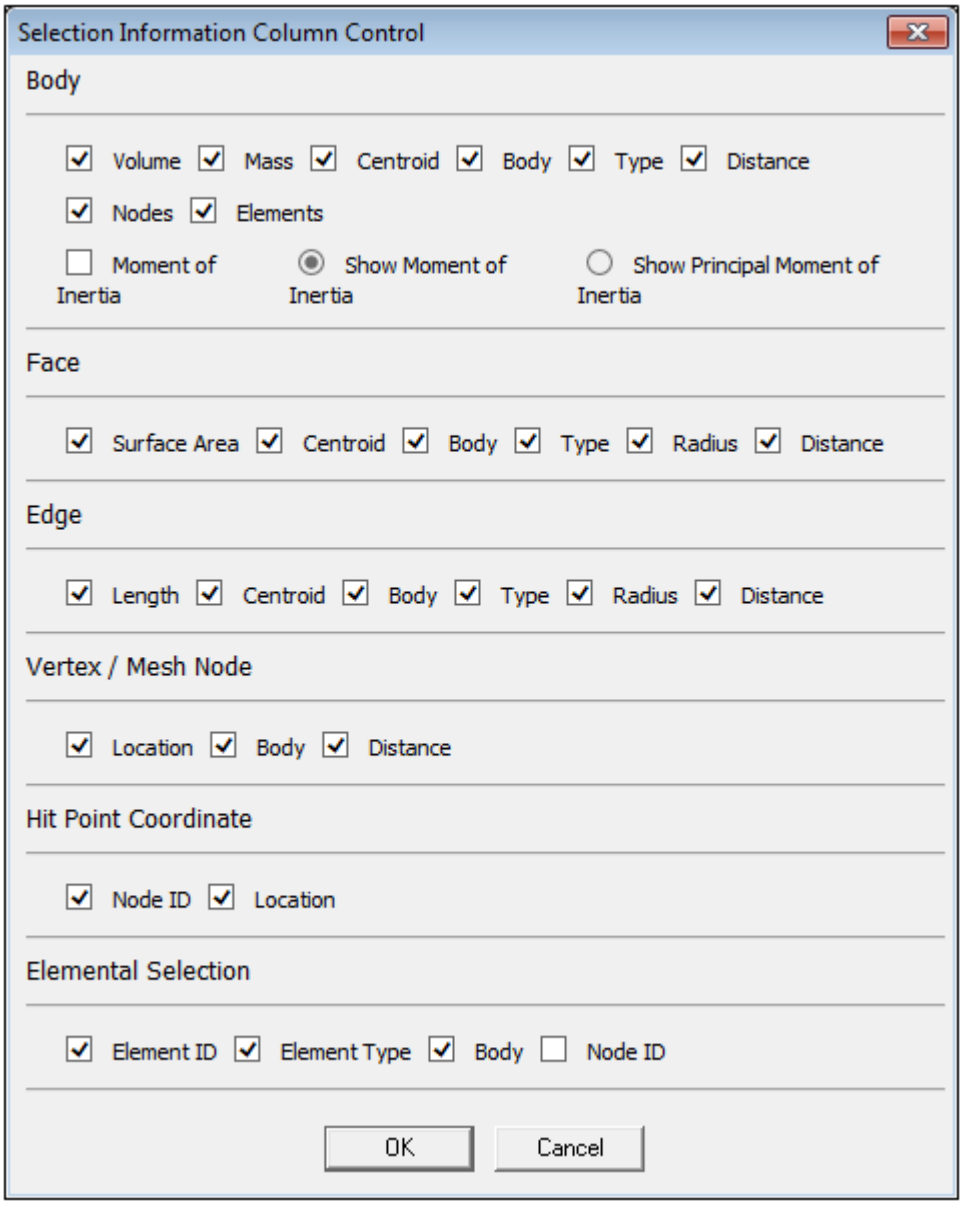


Entity	Distance (mm)	R(mm)	Theta(°)	Z(mm)	Body
Distance	100.				
2 Vertices, Summary		50.	90.	15.	
Vertex 1		70.711	45.	15.	V2Top
Vertex 2		70.711	135.	15.	V2Top

Selection Information Column Control

If you click the **Selection Information Column Control** option, a column control dialog box appears that enables you to select which columns are visible and what columns you can hide. The choices that you made with the column control are retained for the application. The default settings are illustrated below.

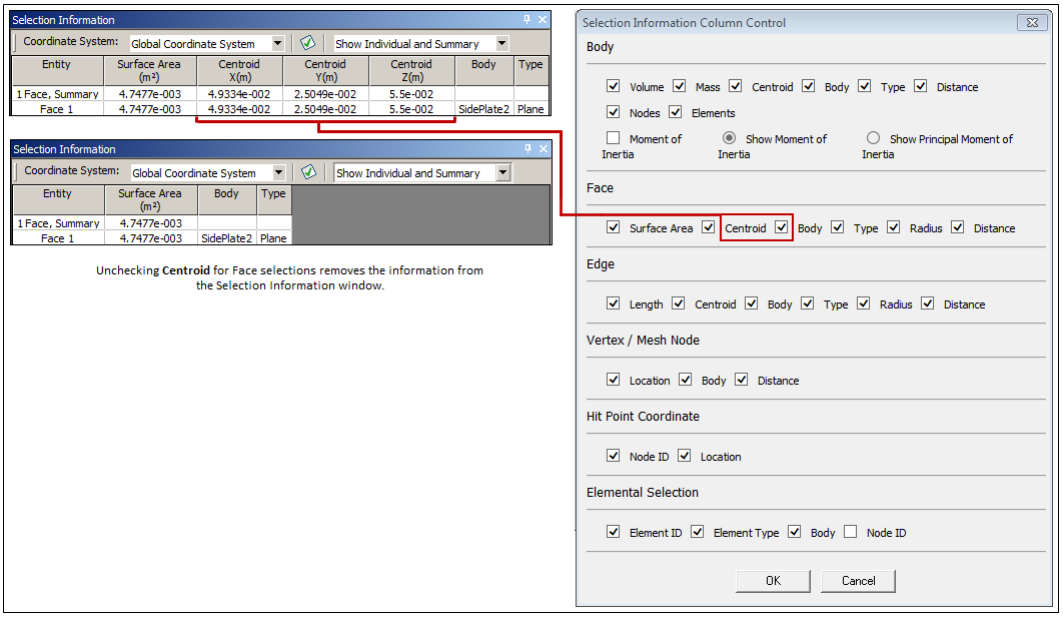
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The following example shows the effects of unchecking the centroid for face.

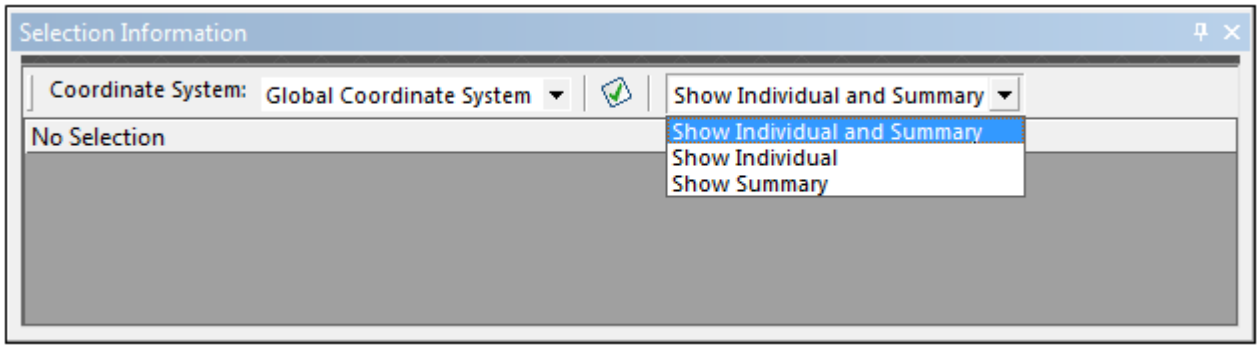
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

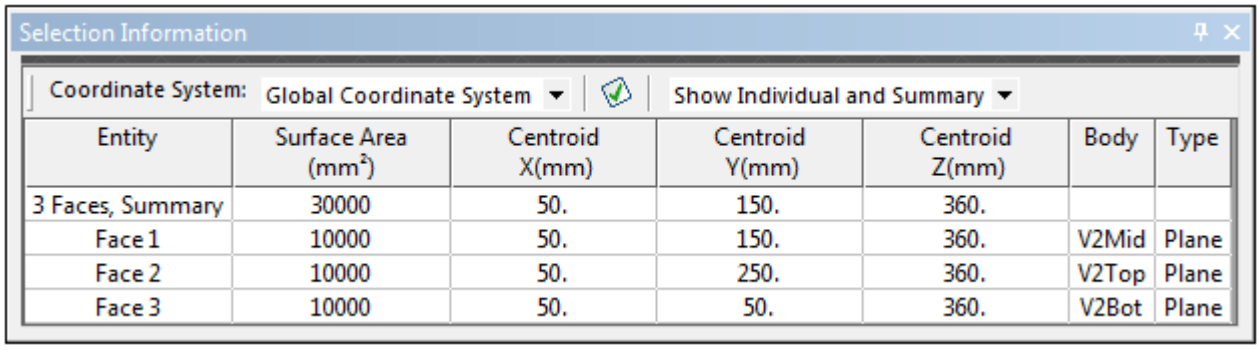
Published: 2023-10-23



Selection Information Row Control

The **Selection Information Row Control** has three options: **Show Individual and Summary**, **Show Individual**, and **Show Summary**. Depending upon your choice, the individual and/or summary information is reported.





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Selection Information

Coordinate System: Global Coordinate System | Show Individual

Entity	Surface Area (mm^2)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Type
Face 1	10000	50.	250.	360.	V2Top	Plane
Face 2	10000	50.	150.	360.	V2Mid	Plane
Face 3	10000	50.	50.	360.	V2Bot	Plane

Selection Information

Coordinate System: Global Coordinate System | Show Summary

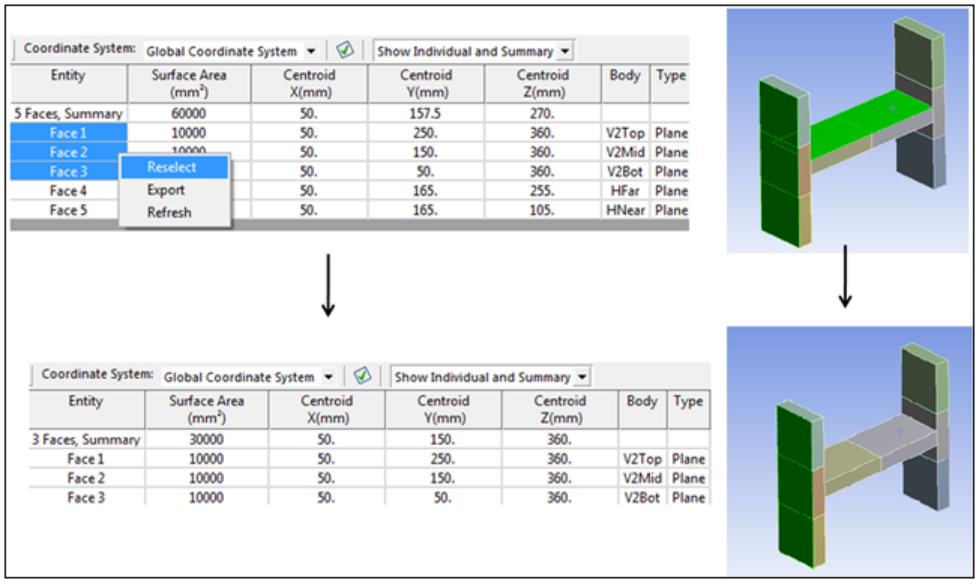
Entity	Surface Area (mm^2)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)
3 Faces, Summary	30000	50.	150.	360.

Selecting, Exporting, and Sorting Data

This section describes how you can reselect rows, export data, and sort data in the **Selection Information** window. Each function is described below.

Reselect

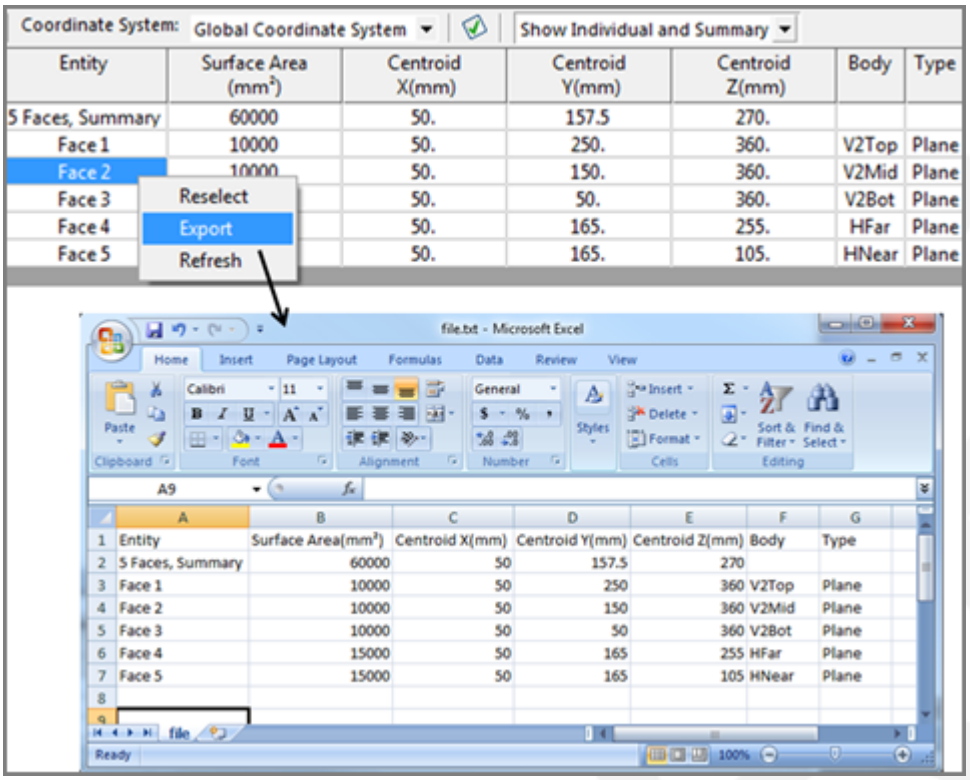
Right-click to reselect the highlighted rows.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Export

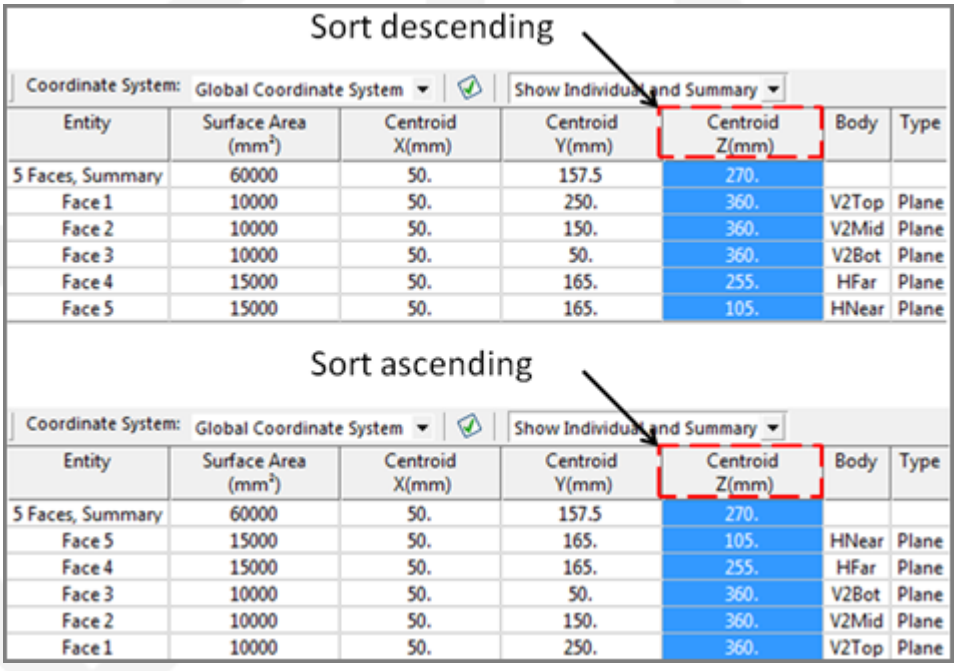
The right-click menu enables you to export the table to a text file or an Excel file.



Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Type
5 Faces, Summary	60000	50.	157.5	270.		
Face 1	10000	50.	250.	360.	V2Top	Plane
Face 2	10000	50.	150.	360.	V2Mid	Plane
Face 3	10000	50.	50.	360.	V2Bot	Plane
Face 4	15000	50.	165.	255.	HFar	Plane
Face 5	15000	50.	165.	105.	HNear	Plane

Sort

Click the column header to sort the table.



Sort descending

Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Type
5 Faces, Summary	60000	50.	157.5	270.		
Face 1	10000	50.	250.	360.	V2Top	Plane
Face 2	10000	50.	150.	360.	V2Mid	Plane
Face 3	10000	50.	50.	360.	V2Bot	Plane
Face 4	15000	50.	165.	255.	HFar	Plane
Face 5	15000	50.	165.	105.	HNear	Plane

Sort ascending

Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Type
5 Faces, Summary	60000	50.	157.5	270.		
Face 5	15000	50.	165.	105.	HNear	Plane
Face 4	15000	50.	165.	255.	HFar	Plane
Face 3	10000	50.	50.	360.	V2Bot	Plane
Face 2	10000	50.	150.	360.	V2Mid	Plane
Face 1	10000	50.	250.	360.	V2Top	Plane

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Worksheet Window

The worksheet presents you with information about objects in the tree in the form of tables, charts and text, thereby supplementing the Details view. It is typically intended to summarize data for a collection of objects (for example, the **Connections** folder worksheet reveals the inputs for all contacts, joints and others) or to receive tabular inputs (for example, to specify the coefficients and the analyses to include in **Solution Combinations**).

Behavior

- **Dockable Worksheet**

  By default, when you select an applicable object in the tree, a dockable (p. 169)**Worksheet** window displays alongside the **Geometry** window, allowing you to review both at once. You may, however, disable the display of the **Worksheet** window using the **Worksheet** option (see below). This preference is persisted in future sessions of the product. There are specific objects that ignore the preference, as outlined below.

Worksheet Function	Worksheet Behavior When Object is Selected	Example Objects
Data input and display information	Automatically appears and gains focus	**Constraint Equation**, **Solution Combination**
Display information related to object settings	Automatically appears but does not gain focus	**Analysis Settings**
Display information related to objects within a folder	Appears only if display is turned on manually using the **Worksheet** option (see below)	**Geometry** folder, **Contact** folder

- **Worksheet Option**

  For tree objects that include an associated Worksheet, the **Worksheet** option of the **Views** group on the ribbon enables you to toggle the **Worksheet** window display on or off. The option is not available (grayed out) for objects that do not include a Worksheet.

  Worksheets designed to display many data items do not automatically display the data. The data readily appears however when you click the **Worksheet** button. This feature applies to the worksheets associated with the following object folders: **Geometry**, **Coordinate System**, **Contact**, **Remote Points**, **Mesh**, and **Solution**.

Features

- **Go To Selected items**

  This useful feature enables you to find items in either the tree or **Geometry** window that match one or more rows of the worksheet. If the worksheet displays a tabular summary of a number of objects, select the rows of interest, right-click, and choose **Go To Selected Items in Tree** to instantly highlight items that match the contents of the **Name** column (leftmost column). Control is therefore transferred to the tree or **Geometry** window, as needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Viewing Selected Columns**

  When a worksheet includes a table with multiple columns, you can control which columns to display. To do so, right-click anywhere inside the table. From the context menu, check the column names of interest to activate their display. Some columns may ignore this setting and remain hidden should they be found inapplicable.

  To choose the columns that will display, right mouse click anywhere inside the worksheet table. From the context menu, click any of the column names. A check mark signifies that the column will appear. There are some columns in the worksheet that will not always be shown even if you check them. For example, if all contact regions have a **Pinball Region** set to **Program Controlled**, the **Pinball Radius** will not display regardless of the setting.

Graph and Tabular Data Windows

The **Graph** and **Tabular Data** windows enable you to review and modify the data, primarily associated with the objects and features listed below. When you select certain objects in the tree, the **Graph** window and **Tabular Data** window display beneath the **Geometry** window. Refer to the following topics for descriptions about the use of these windows as they relate to:

- Analysis Settings (p. 193)
- Loading Conditions (p. 193)
- Contour Results and Probes (p. 194)
- Display Graph Content in the Geometry Window (p. 197)
- Solution Steps and Substeps (p. 198)
- Charts (p. 199)
- Context Menu Options (p. 199)

Analysis Settings

For analyses with multiple steps, you can use these windows to select the step(s) whose analysis settings (p. 1417) you want to modify. The **Graph** window also displays all the loads used in the analysis.

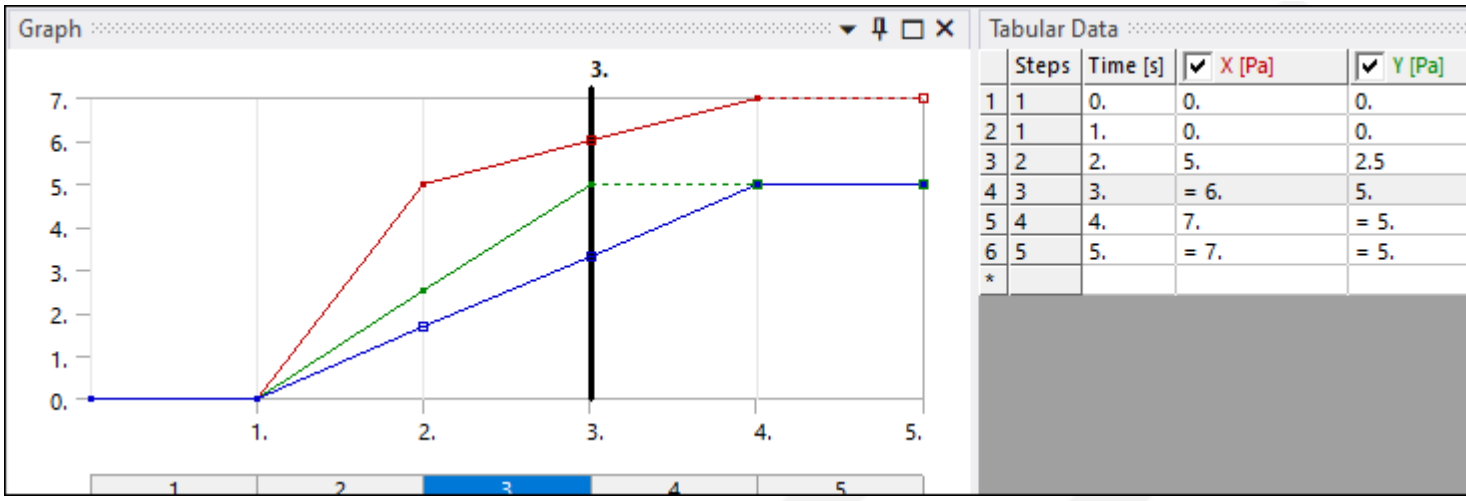
These windows are also useful when using restarts. See Using Solution Restarts (p. 1815) for more information.

Loading Conditions

Inserting a loading condition (p. 1497) updates the **Tabular Data** window with an entry table that enables you to enter data on a per-step basis. The **Graph** window updates as you make **Tabular Data** entries.

All new tabular data is entered into the row that begins with an asterisk (*) regardless of whether the time or frequency point is higher or lower than the last defined point in the table. The application automatically sorts the content of the table into ascending order. In addition, any **Tabular Data** values preceded by an equal sign (=) are not defined table values. These values are application interpolated values shown for reference.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



A check box is available in the column title for each component of a load in order to turn on or turn off the viewing of the load in the **Graph** window. Components are color-coded to match the component name in the **Tabular Data** window. Clicking on a time value in the **Tabular Data** window or selecting a row in the **Graph** window will update the display in the upper left corner of the Geometry window with the appropriate time value and load data.

As an example, if you use a Displacement (p. 1666) load in an analysis with multiple steps, you can alter both the degrees of freedom and the component values for each step by modifying the contents in the **Tabular Data** window as shown above.

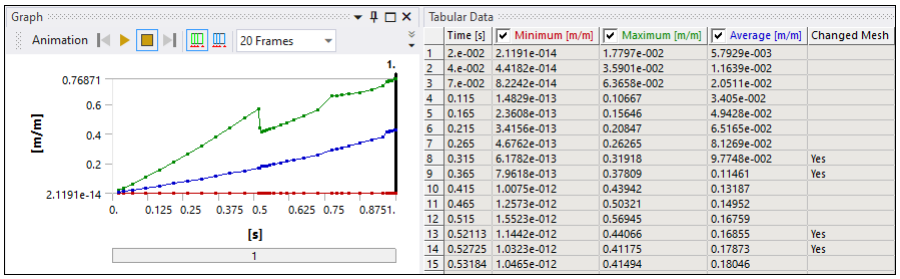
If you wish for a load to be active in some steps and removed in some other steps you can do so by following the steps outlined in Activating and Deactivating Loads (p. 1423).

Contour Results and Probes

For contour results (p. 1935) and probes (p. 1946), the **Graph** and **Tabular Data** windows display how the results vary over time. In addition, for result objects, the **Tabular Data** window usually displays the **Total**, as well as **Minimum** and **Maximum** values calculated on the specified geometry. The color of the column headers for these values corresponds to the colors displayed in the **Graph** window, red and green as illustrated below. You can animate your results in the **Graph** window for the specified result set domain. And, you can further specify a specific range to animate by dragging your mouse across graph content.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

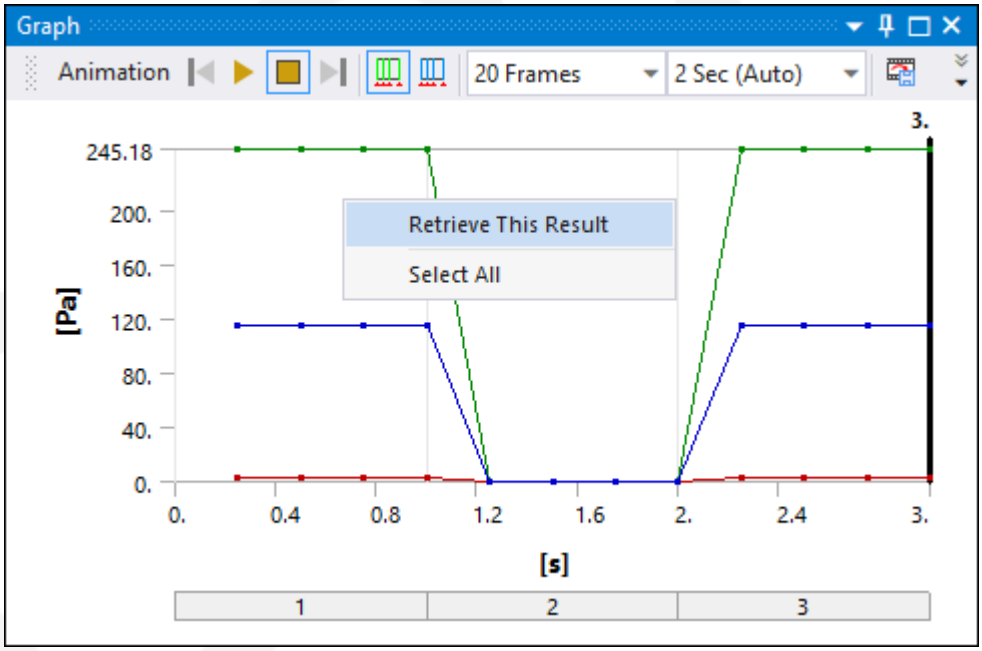
If you refine the mesh using the **Nonlinear Adaptive Region** (p. 1707) condition, the **Changed Mesh** column displays and indicates when mesh regeneration took place.

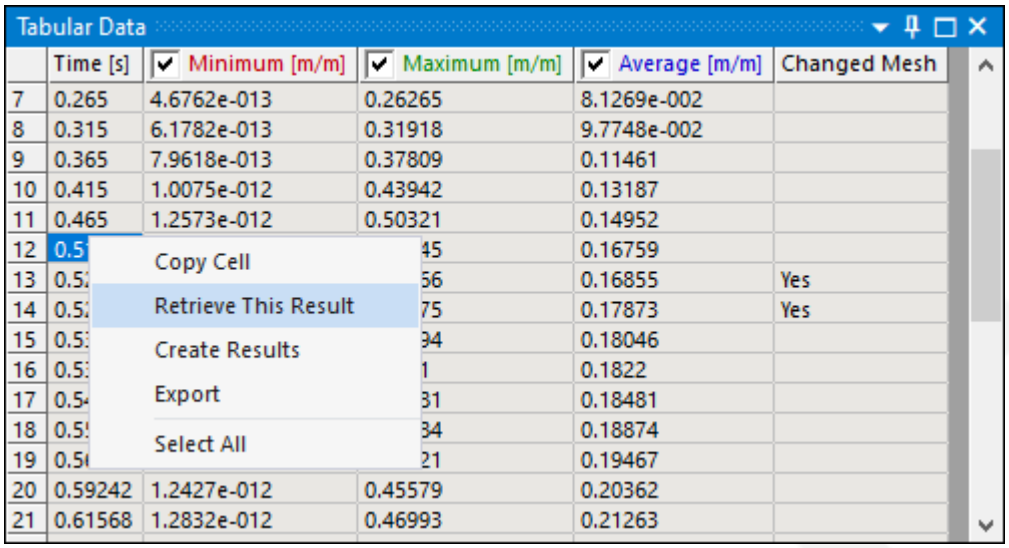
Important:

For results displayed in the **Tabular Data** window, if **0** (zero) displays for both the **Minimum** and **Maximum** values of a row, the result set may not contain result data. You can use the **Retrieve** option, discussed below, to view a result set in order to determine if any data exists for the set. If no data is available, the result contours in the **Geometry** window display as fully transparent.

Retrieving Results

To view the results in the **Geometry** window for a desired time point, select the time point in the **Graph** window or **Tabular Data** window, then right-click the mouse and select **Retrieve This Result**. The **Details** for the chosen result object also updates to the selected step.



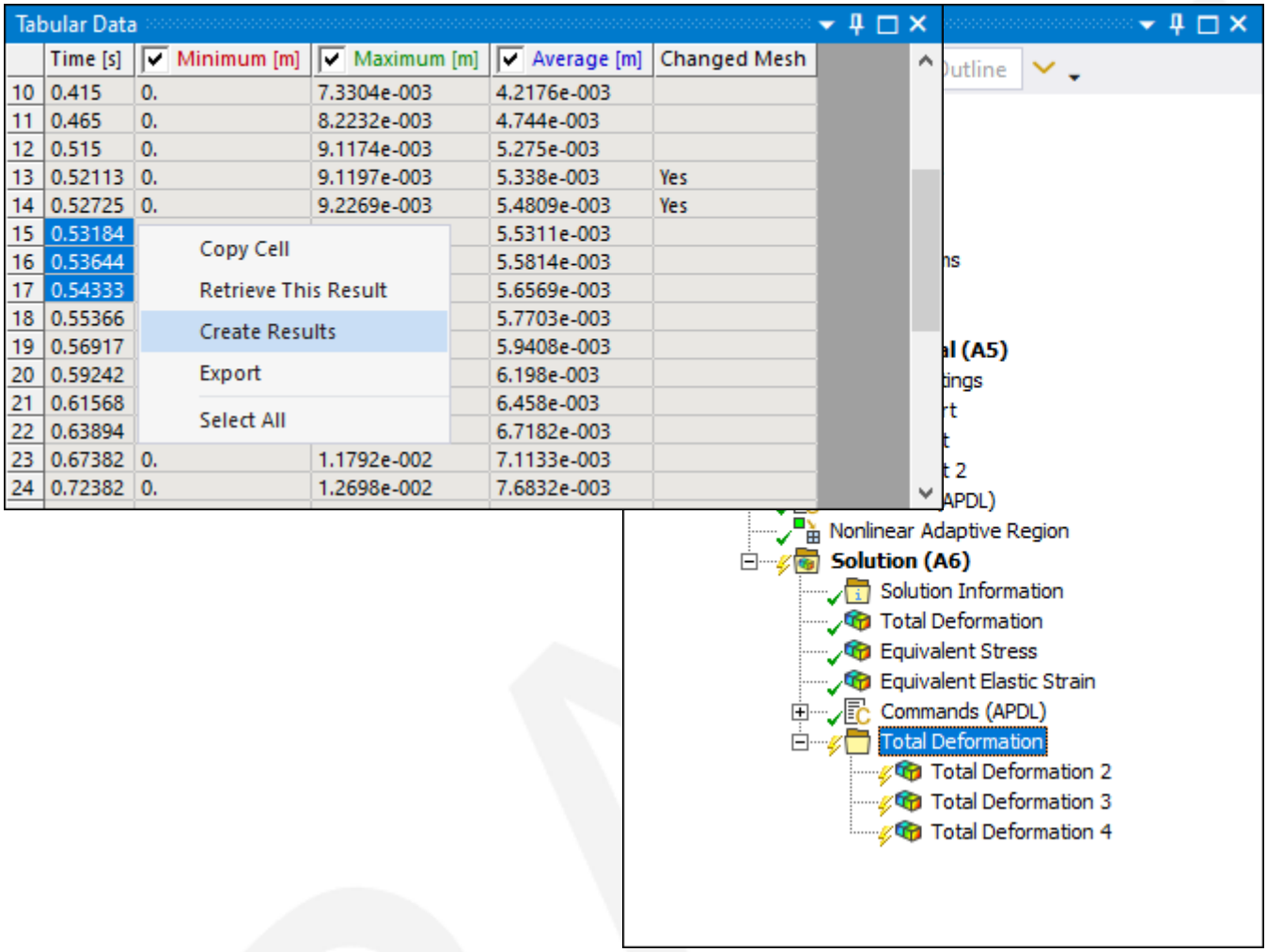


Creating Results

The contextual (right-click) menu of the **Tabular Data** window also includes an option to **Create Results**. This feature enables you to select multiple rows in the table and create individual results for each selection. These new results are placed in a **Group** folder (p. 208) in the tree. The **Group** folder has the same name as the original result. Or, in the event the originally result was already grouped, the new results are added to this existing group.

Create Results Selection **Generated Results**

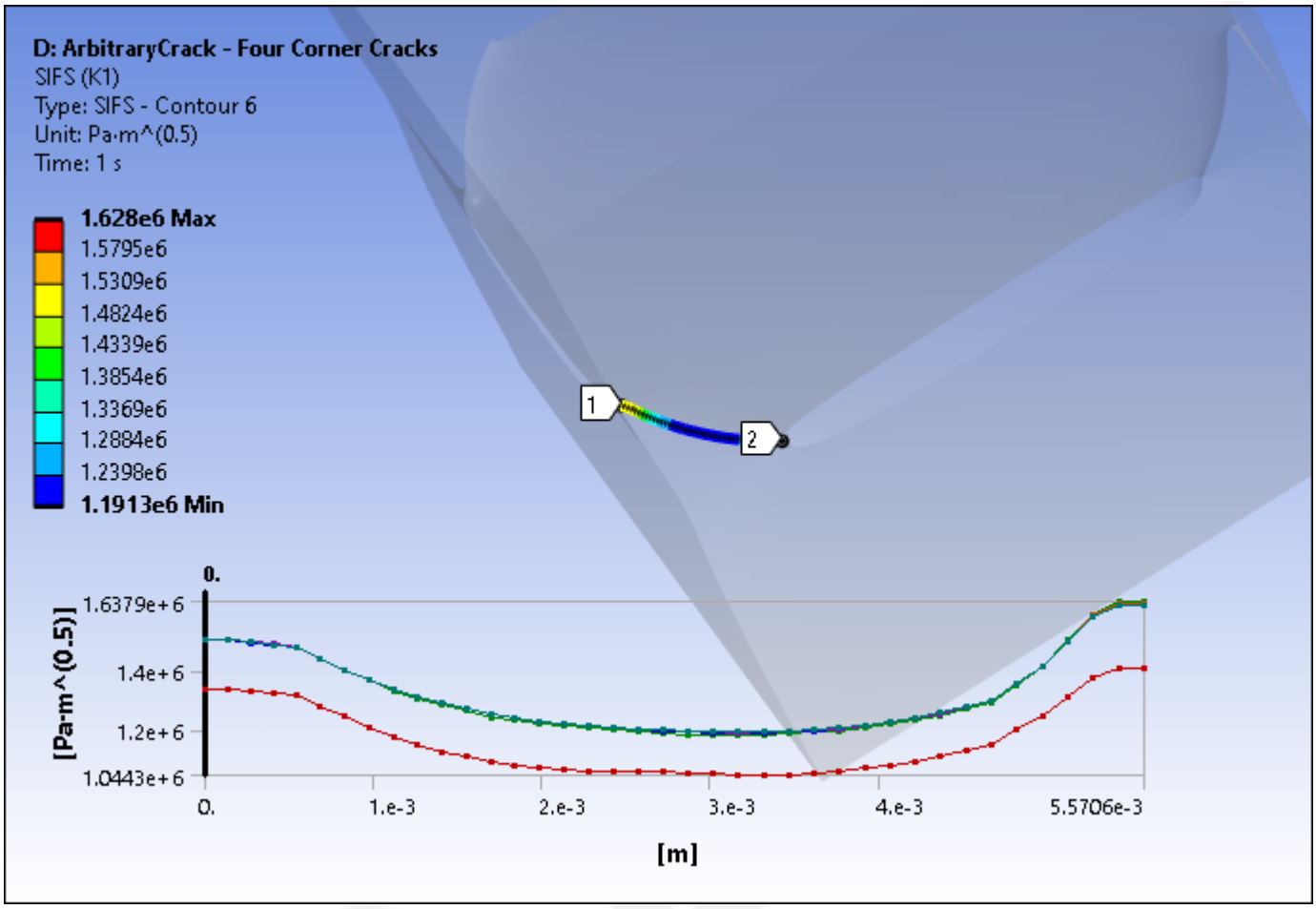
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Display Graph Content in the Geometry Window

When the **Geometry** window has focus, you can select the **Display Graph** option on the Graphics Toolbar (p. 97), or use the keyboard key **G** or the context (right-click) menu option, to display the content (load and result data) of the **Graph** window in the **Geometry** window. This enables you to include the graph in image captures of a viewport. Once displayed, you can drag and drop the graph within the window. You can activate this feature in multiple viewports and it remains active until you select one of the **Display Graph** options to remove the graph. In addition, the animation and animation export options support this feature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



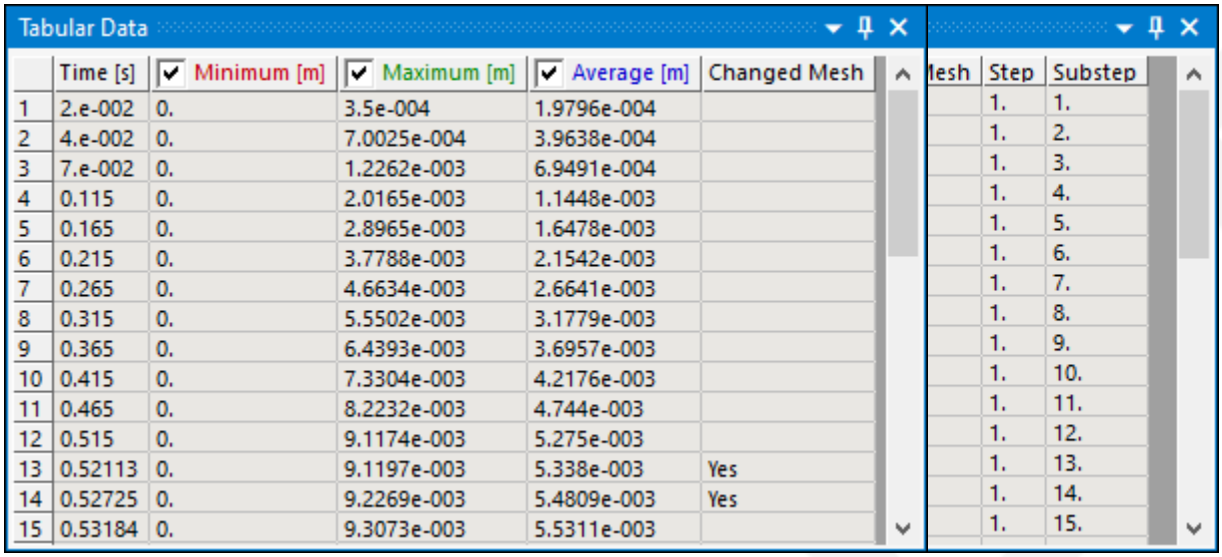
Solution Steps and Substeps

During analyses that are using the Mechanical APDL solver, selecting the **Solution** object following the solution process for an analysis that includes multiple steps, the **Tabular Data** window displays the **Time** associated with each **Step** of the analysis as well as each **Substep** as applicable. The following examples of the **Tabular Data** window show these options for a deformation result.

Total Deformation Result Tabular Data **Solution Object Tabular Data**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23


Tabular Data

	Time [s]	Minimum [m]	Maximum [m]	Average [m]	Changed Mesh	lesh	Step	Substep
1	2.e-002	0.	3.5e-004	1.9796e-004			1.	1.
2	4.e-002	0.	7.0025e-004	3.9638e-004			1.	2.
3	7.e-002	0.	1.2262e-003	6.9491e-004			1.	3.
4	0.115	0.	2.0165e-003	1.1448e-003			1.	4.
5	0.165	0.	2.8965e-003	1.6478e-003			1.	5.
6	0.215	0.	3.7788e-003	2.1542e-003			1.	6.
7	0.265	0.	4.6634e-003	2.6641e-003			1.	7.
8	0.315	0.	5.5502e-003	3.1779e-003			1.	8.
9	0.365	0.	6.4393e-003	3.6957e-003			1.	9.
10	0.415	0.	7.3304e-003	4.2176e-003			1.	10.
11	0.465	0.	8.2232e-003	4.744e-003			1.	11.
12	0.515	0.	9.1174e-003	5.275e-003			1.	12.
13	0.52113	0.	9.1197e-003	5.338e-003	Yes		1.	13.
14	0.52725	0.	9.2269e-003	5.4809e-003	Yes		1.	14.
15	0.53184	0.	9.3073e-003	5.5311e-003			1.	15.

Charts

With charts (p. 1929), the **Graph** and **Tabular Data** windows can be used to display loads and results against time or against another load or results item.

Context Menu Options

Presented below are some of the commonly used options available in a context menu that displays when you click the right mouse button within the **Graph** window and/or the **Tabular Data** window. The options vary depending on how you are using these windows (for example, loads vs. results).

- **Retrieve This Result**: As discussed above, for a selected object, this option retrieves and presents the result data at the selected time point you have selected in the **Graph** window or **Tabular Data** window.
- **Create Results**: As discussed above, this option create result objects for the rows that you select in the **Tabular Data** window and places the new results in a group folder.
- **Create Total Deformation Results**/**Create Equivalent Strain Results**/**Create Equivalent Stress Results**/**Create Temperature Results**: These options are available for **Tabular Data** content when the **Solution** object is selected for a solved analysis. These options enable you to create results for a selected row or multiple rows of data.
- **Insert Step**: Inserts a new step at the currently selected time in the **Graph** window or **Tabular Data** window. The newly created step will have default analysis settings. All load objects in the analysis will be updated to include the new step.
- **Delete Step**: Deletes a step.
- **Copy Cell**: Copies the cell data into the clipboard for a selected cell or group of cells. The data may then be pasted into another cell or group of cells. The contents of the clipboard may also be copied into Microsoft Excel. Cell operations are only valid on load data and not data in the **Steps** column.
- **Paste Cell**: Pastes the contents of the clipboard into the selected cell, or group of cells. Paste operations are compatible with Microsoft Excel.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Delete Rows**: Removes the selected rows. In the **Analysis Settings** object this will remove corresponding steps. In case of loads this modifies the load vs time data.
- **Select All Steps**: Selects all the steps. This is useful when you want to set identical analysis settings for all the steps.
- **Select All Highlighted Steps**: Selects a subset of all the steps. This is useful when you want to set identical analysis settings for a subset of steps.
- **Activate/Deactivate at this step!**: This enables a load to become inactive (deleted) in one or more steps. By default any defined load is active in all steps.
- **Zoom to Range**: Zooms in on a subset of the data in the **Graph** window. Click and hold the left mouse at a step location and drag to another step location. The dragged region will highlight in blue. Next, select **Zoom to Range**. The chart will update with the selected step data filling the entire axis range. This also controls the time range over which animation takes place.
- **Zoom to Fit**: If you have chosen **Zoom to Range** and are working in a zoomed region, choosing **Zoom to Fit** will return the axis to full range covering all steps.

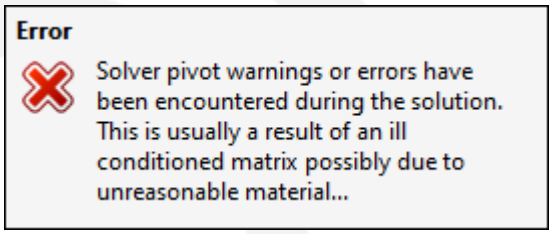
Messages Window

The **Messages Window** is a Mechanical application feature that prompts you with feedback concerning the outcome of the actions you have taken in the application. For example, Messages display when you resume a database, Mesh (p. 348) a model, or when you initiate a Solve (p. 355).

Messages come in three forms:

- **Error**
- **Warning**
- **Information**

As illustrated below, when the application issues a message, a window first displays the message for five seconds and then the pop-up is automatically hidden. You may change the default setting for the these messages in the Messages (p. 236) category of the Options preferences (p. 217). Change the **Pop-up Messages** setting to **No**. The default setting is **Yes**.

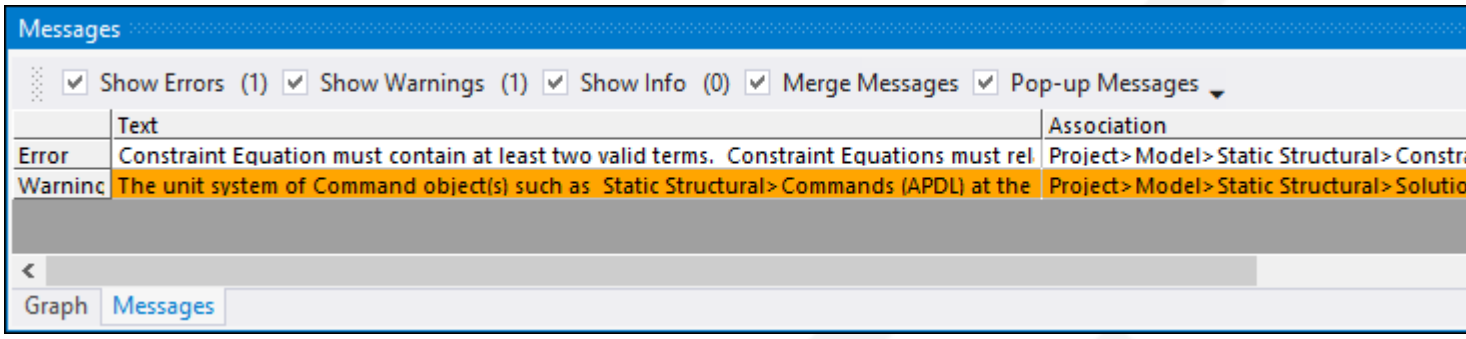


Note:

The Linux platform does not support the message pop-up feature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

By default the **Messages** window is hidden. To display the window, open the **Manage** drop-down menu from the **Layout** (p. 47) group of the **Home** tab and select **Messages**. An example of the **Messages** window is shown below.



As shown here, the Status Bar provides a dedicated area to alert you should one or more messages become available to view. You can double-click this field to display messages.

Highlighted Messages

As illustrated above, messages may display with an orange highlight to indicate that there is a potential problem related to an object. The object corresponding to this message is also highlighted in the tree (p. 110). You can turn this feature off using the Options (p. 217) dialog box (see the Messages (p. 236) category).

Window Options

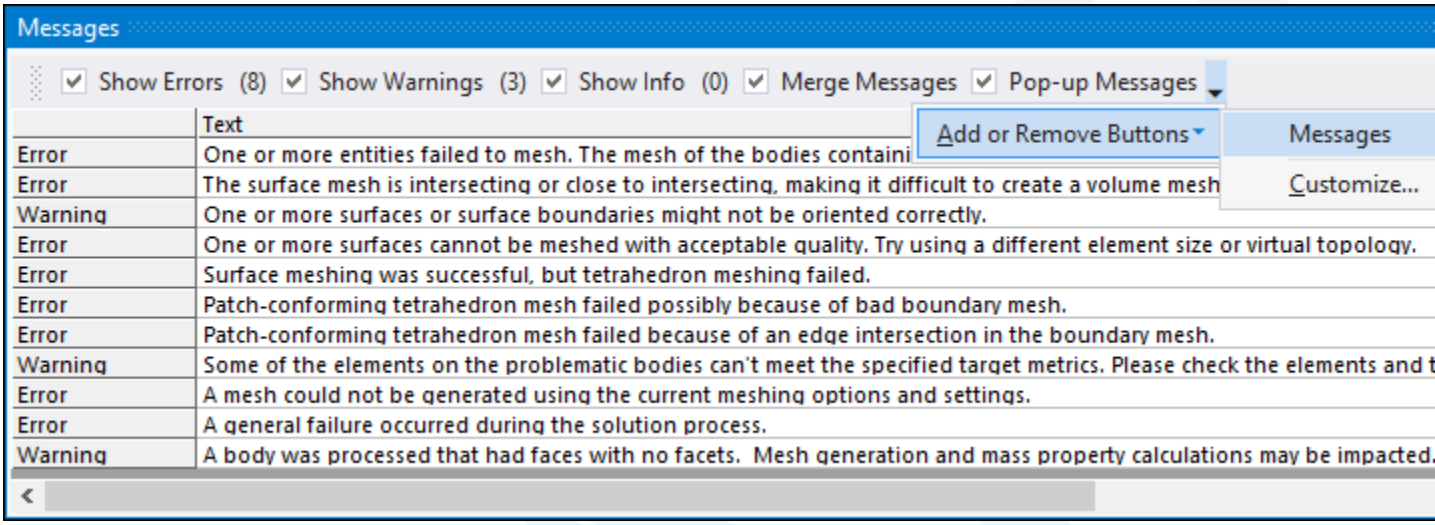
Once messages are displayed in the **Messages** window, you can:

- Double-click a message to display its contents in a pop-up window.
- You can filter messages using the display options listed below. Checking and clearing the options may change the order in which the window displays the messages. These options have associated preferences settings in the Messages (p. 236) category of the Options (p. 217) dialog that enables you to define the default setting.
  - **Show Errors**: Display or hide error messages. This field also lists the number of different error messages displayed in the Messages window. When you merge/unmerge messages, this value changes accordingly.
  - **Show Warnings**: Display or hide warning messages. This field also lists the number of different warning messages displayed in the Messages window. When you merge/unmerge messages, this value changes accordingly.
  - **Show Info**: Display or hide information-based messages. This field also lists the number of different information messages displayed in the Messages window. When you merge/unmerge messages, this value changes accordingly.
  - **Merge Messages**: Merge similar messages into one row. Unchecking this option will unmerge messages - all original messages are then displayed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– **Pop-up Messages**: When selected, pop-ups display as expected. When you unselect this option, system generated messages do not display as a pop-up, but are included in the list of messages in the **Messages** window. You can double-click messages to display them as a pop-up dialog.

The filter toolbar also contains a drop-down menu option that enables you to hide/display any or all of the above filter options as well as reset the toolbar.



- Right-click one or more messages to display the following context menu options:

  – **Go To Object**: Selects the object in the tree which is responsible for the message.

  – **Show Problematic Geometry**: Highlights the geometries in the Geometry window that are causing the error or warning.

  – **Show Intersecting Surface Mesh**: This option creates a Named Selection of the problematic bodies and elements for your review.

  – **Show Message**: Displays the selected message in a popup window.

  – **Copy**: Copies the selected messages to the clipboard. You can also highlight a message and then press the key combination **Ctrl** + **C** to copy its contents to the clipboard.

  – **Delete**: Removes the selected messages. You can also use the **Delete** key to remove a selected message from the window.

  – **Refresh**: Refreshes the contents of the **Messages Window** as you edit objects in the Mechanical application tree.

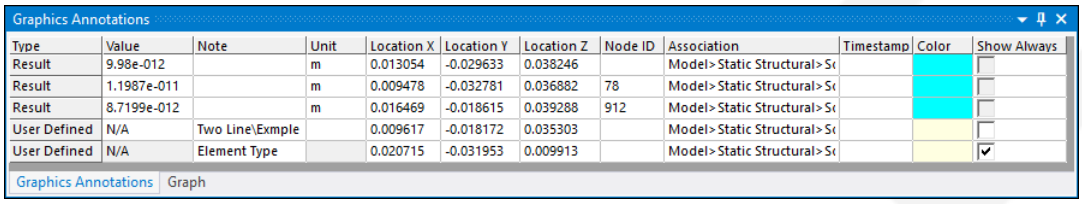
Graphics Annotations Window

The **Graphics Annotations** window, illustrated below, displays a table of user-defined annotations. The annotations are either a note that you place on your model using the **Annotation** (p. 42) option or the display of a specific result value at its coordinate location on your model, using the Probe (p. 71) option.

These annotations are labels that you place on your model that include a/an:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Value** (**Probe** only)
- **Note**
- Coordinates of the label (**Location X/Y/Z** columns)
- **Node ID** as applicable. **Node ID** values are included in the table for probe selections when the Snap (p. 71) option is selected (active by default).
- Object associated with the label (**Association** column).
- **Timestamp**
- Label color (**Color** column)
- Display option for user-defined annotations (not probes) that you can select to have that label remain displayed (**Show Always** column).



Go to a section topic:

- Application (p. 203)
- Annotation Types (p. 204)
- Graphics Annotations Window Features (p. 204)
  - Write Notes (p. 205)
  - Highlight Values (p. 205)
  - Display Node IDs (p. 206)
  - Move Labels (p. 206)
  - Change Probe Line Coloring (p. 207)
  - Change Label Color (p. 207)
  - Display Obscured Labels (p. 207)
  - Display Annotation Labels Continuously (p. 207)
  - Select Context Menu Options (p. 207)
  - Manually Delete Annotations (p. 208)

Application

The **Graphics Annotations** window displays automatically when you:

1. Place an annotation on your model. Select the **Annotation** option from the **Insert** group (p. 42) on the Home (p. 40) tab and then click a location on your model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Or...

Note:

The **Insert** group displays on numerous Context tabs.

2. Place a probe on your model. Select a result object, select the **Probe** option on the Result Context Tab (p. 62), and then click a location on your model.

Note:

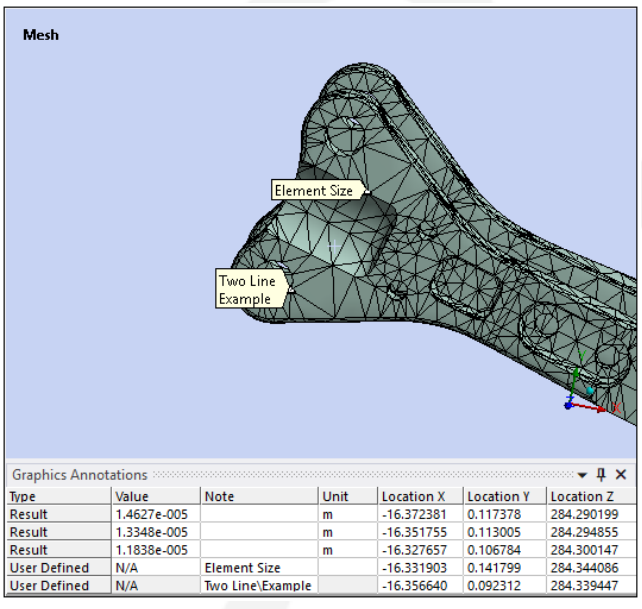
For the **Probes** feature:

- Annotations are not supported for results scoped to edges and vertices.
- Values are cleared if you re-solve your simulation.
- Values for positions inside of an element may differ from hand-calculated values, especially for certain user defined result expressions. See the limitations topic in the Interpolation of Result Values (p. 1937) section.

Annotation Types

Examples of the annotation types as displayed in the **Geometry** window are shown below.

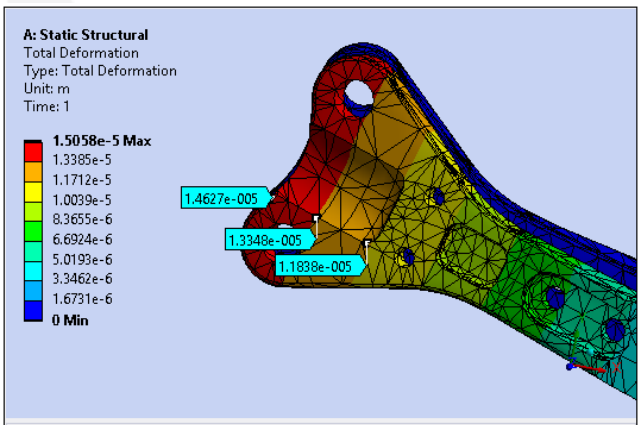
User Defined Graphics Annotation



Graphics Annotations

Type	Value	Note	Unit	Location X	Location Y	Location Z
Result	1.4627e-005		m	-16.372381	0.117378	284.290199
Result	1.3348e-005		m	-16.351755	0.113005	284.294855
Result	1.1838e-005		m	-16.327657	0.106784	284.300147
User Defined	N/A	Element Size		-16.331903	0.141799	284.344086
User Defined	N/A	Two Line\Example		-16.356640	0.092312	284.339447

Probe Annotations for Result



Graphics Annotations

Type	Value	Note	Unit	Location X	Location Y	Location Z
Result	1.4627e-005		m	-16.372381	0.117378	284.290199
Result	1.3348e-005		m	-16.351755	0.113005	284.294855
Result	1.1838e-005		m	-16.327657	0.106784	284.300147
User Defined	N/A	Element Size		-16.331903	0.141799	284.344086
User Defined	N/A	Two Line\Example		-16.356640	0.092312	284.339447

Graphics Annotations Window Features

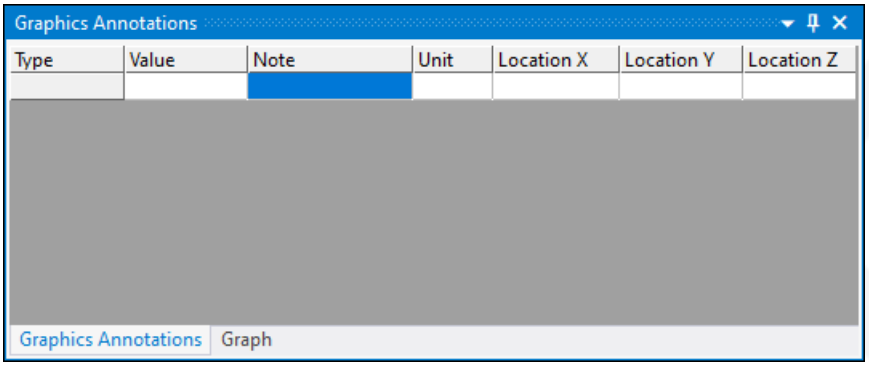
The **Graphics Annotations** window provides the following options once you have created annotations:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

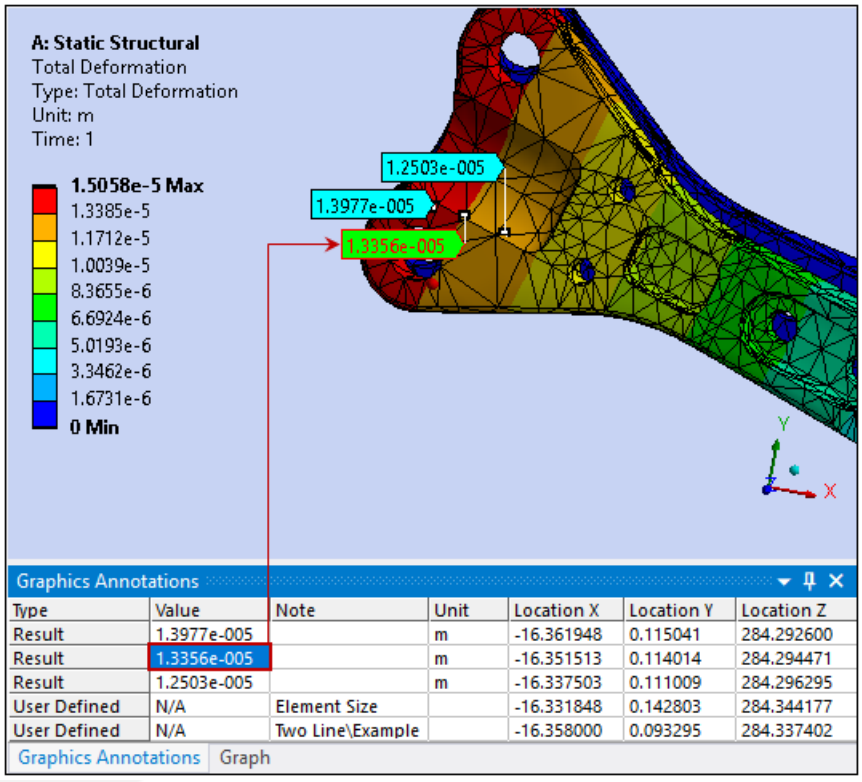
Write Notes

When you select the **Annotation** option in the **Insert** group (p. 42) and place the annotation on a point of your model, the **Graphics Annotations** window displays as shown below. The **Note** cell of the table is active and you can being to type your note. You can edit the text entry for a user-defined annotation by double-clicking the annotation's **Note** cell. Your text entries can span more than one line using the backslash (\) keyboard character. Note the "Two Line" note example above.



Highlight Values

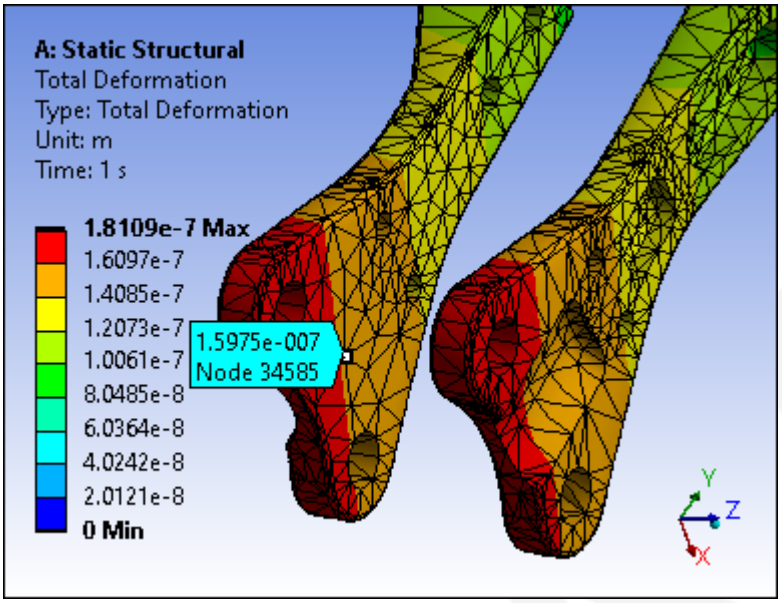
Selecting a table cell in the **Value** column (or a cell of the **Association** column) or selecting the annotation label in the **Geometry** window, highlights the annotation in the table as well as in the **Geometry** window. Note that for a user-defined annotations, the **Geometry** display switches to the corresponding object of the tree that includes the annotation, such as the Element Size example shown above.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Display Node IDs

Using the **Show Node ID in Snap-Probe Labels** preference of the **Graphics** (p. 228) category in the **Options** dialog, you can include the Node ID in the probe label, as illustrated below.



Move Labels

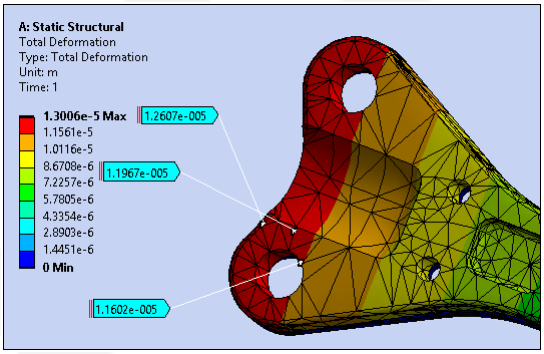
Selecting the **Label** option on the Graphics Toolbar (p. 97) enables you select and then drag-and-drop an annotation.

Moving a Text Label

You can only reposition a text label on the model. Once repositioned, the label will rotate and move in space with the model.

Moving a Probe Label

Probe labels can be moved anywhere within the **Geometry** window. As illustrated below, a white line directs you to the probe's location on the model. Also, two vertical red lines appear beside a label when it is moved. You can then freely drag-and-drop the annotation to a different location. Once moved, the label is stationary. If you rotate or move the model, the label remains in the same position in the window. As needed, you can simply return the annotation to the original position or drag-and-drop it to an new position in order to reposition it along with the model.



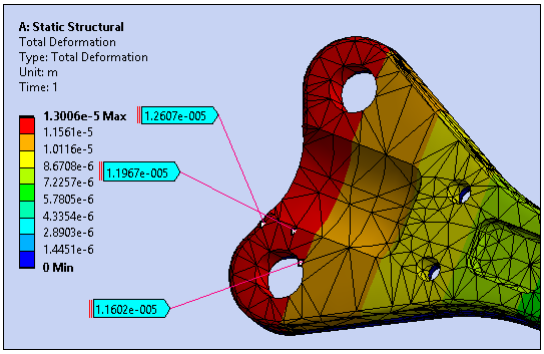
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To return probe labels to their original position (the anchor of the probe always remains in the original position), select the annotation in the **Geometry** window or in the window and press the **Esc** key. You can select multiple probes using the **Ctrl** key.

Change Probe Line Coloring

You can change the color of the line connecting the probe label to its location on the model using the **Probe Line Color** property in the Graphics (p. 228) category of the Options (p. 217) dialog.



Change Label Color

When you double-click a cell in the **Color** column, a color selection pane displays that enables you to change the color of the label.

Display Annotation Labels Continuously

For user-defined annotations (not probes), you can select the check box under the **Show Always** column to have that label remain displayed regardless of the object or interface selection you make.

Display Obscured Labels

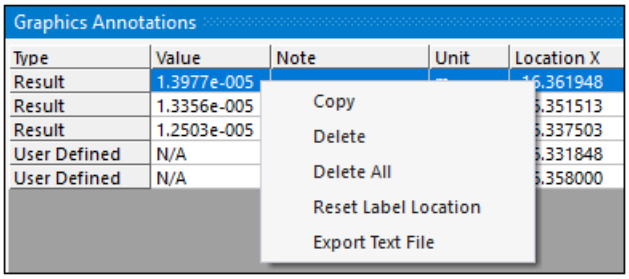
The Options (p. 217) dialog contains a Graphics (p. 228) category preference, **Obscured Label Display Style**, that enables you to change the display of an annotation label and/or probe that is not visible because of the orientation of your model. The options include **Solid** (default), **Translucent**, and **Hidden**. When a label or probe is on the opposite side of the model's current orientation, you can choose whether to have it show "through" the model using the Solid and Translucent options or not at all using the **Hidden** option.

Also note that the **Graphics** category includes an associated preference: **Unobscured Label Tolerance Factor**. For the **Translucent** or **Hidden** options, this tolerance enables you make small adjustments to make sure that a label behaves as expected in the event a label displays as obscured when it should not. For example, when a label's position is located slightly within a curved surface rather than directly on the surface. This is most likely a result of how the application calculates the location of a label. The tolerance range is between 0 and 1. Small incremental changes, such as 0.0001 can be appropriate to properly display obscured labels.

Select Context Menu Options

As illustrated below, the window provides a context menu when you make a table selection and then right-click the mouse.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Context Menu Options	Option Descriptions
	• **Copy**: copy the content contained in the row into the clipboard. • **Delete**: delete table selections. • **Delete All**: delete all table content. • **Reset Label Location**: return all selected probes to their original position. • **Export Text File**: export table content to a text file.

Manually Delete Annotations

To delete an annotation, select a row in the window or select the annotation label in the **Geometry** window, and then press the **Delete** key. You can select multiple probe labels or table cells using the **Ctrl** key.

Section Planes Window

The **Section Plane** window gives you access to the functionality for creating a cut or slice on your model so that you can view internal geometry, or mesh and results displays. For more information on this feature, see Creating Section Planes (p. 294).

Mechanical Wizard Window

The **Mechanical Wizard** window appears in the right side panel whenever you click the Mechanical Wizard button on the Home Tab (p. 40). For details, see Mechanical Wizard (p. 321).

Group Tree Objects

Mechanical enables you to organize and group certain objects in the **Outline** pane. Using context menu (right-click) options, the application provides a number of different options that you use to group objects.

Go to a section topic:

- Group Multiple Objects (p. 209)
- Group, Hide, and Suppress Geometry Objects (p. 211)
- Group All (p. 212)
- Group Loads by Load Vector (**Substructure Generation** Analysis Only) (p. 214)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

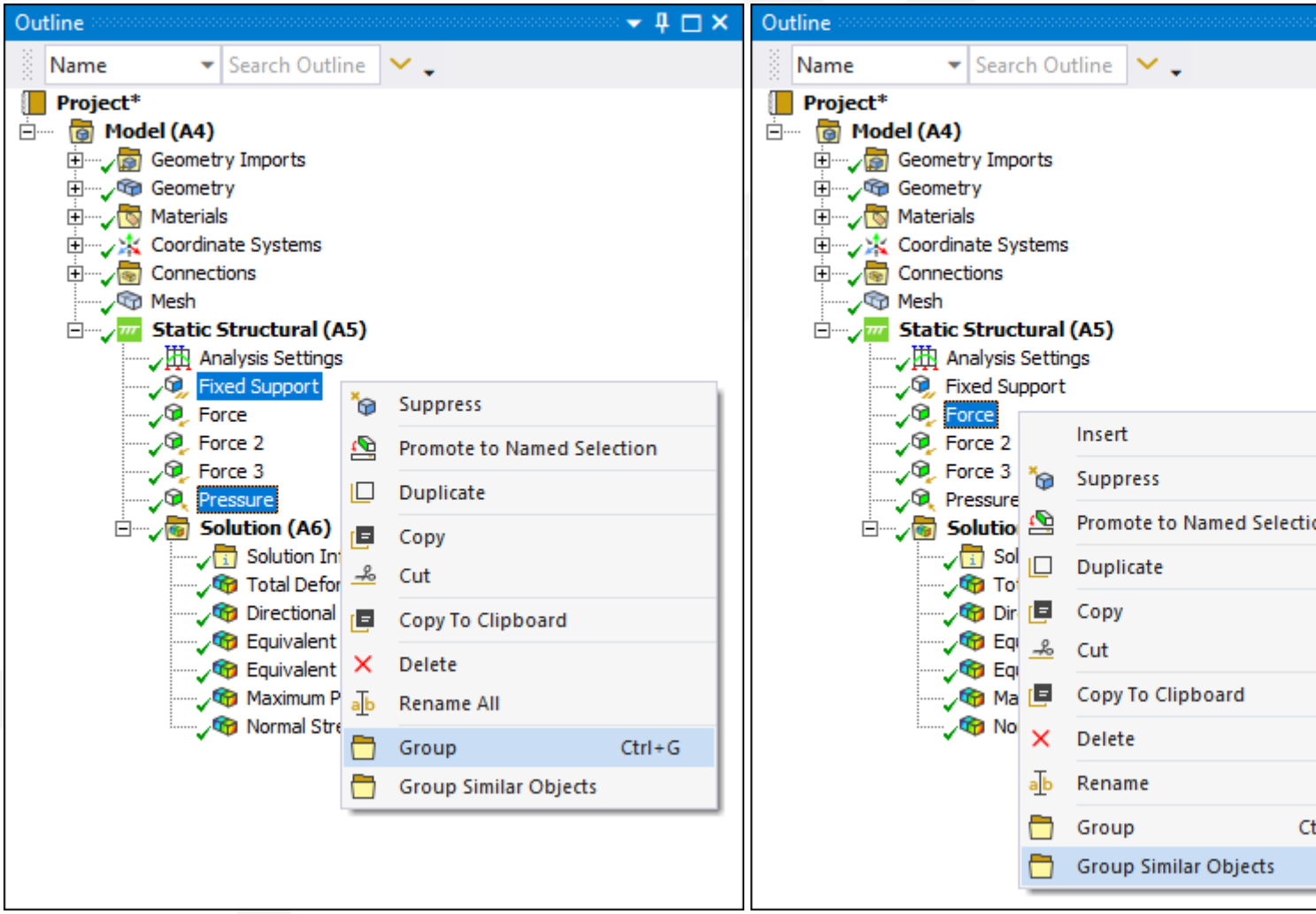
- Objects with Limited Grouping (p. 215)

Note:

For CAD files that include a hierarchy structure, the **Options** dialog preference setting, **Assembly Hierarchy**, enables you to automatically group parts and bodies under the **Geometry** object upon import. If you use this preference, you cannot change the setting once you have imported your geometry. This is due to a geometry update limitation.

Group Multiple Objects

Use the **Group** option when you individually select multiple objects to be grouped. The **Group Similar Objects** groups together objects of the same type (for example, Pressure, Displacement, etc.) and renames the group folder according to that type.

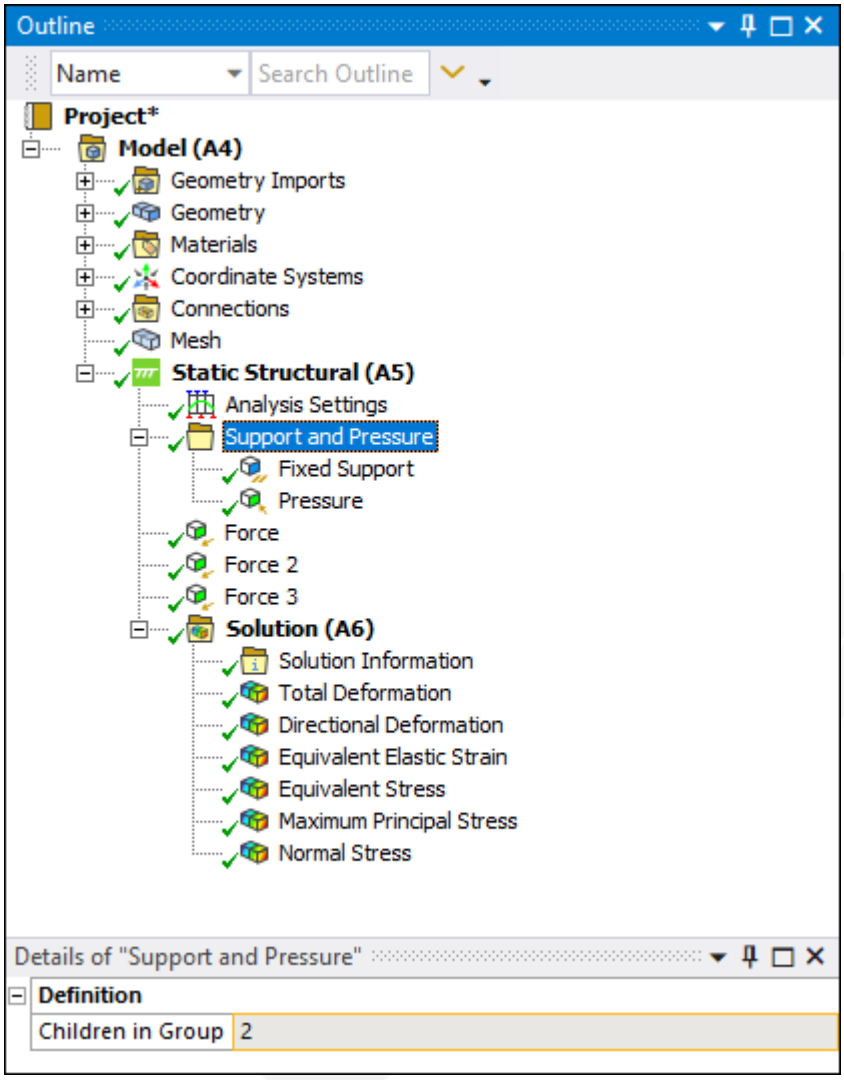


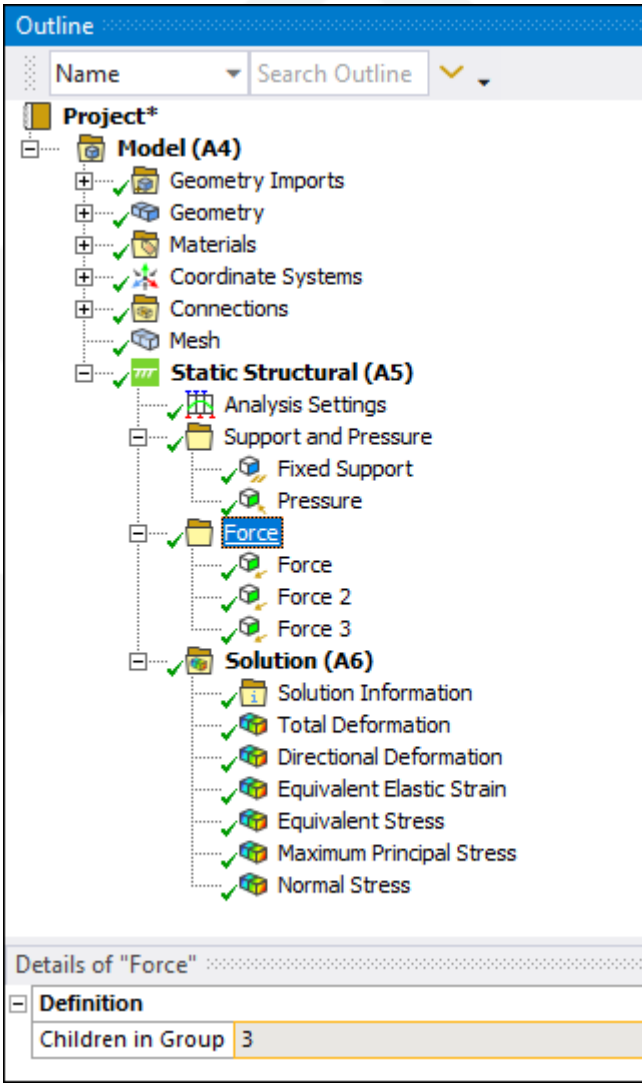
Note:

If you change and update your geometry, always verify your groupings. For example, using the Explode Part feature in DesignModeler alters Part IDs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Once you create groups, you are prompted to rename the folder or you may accept the default name (**New Folder**). In the following example, the folder was named "Support and Pressure." The similar objects folder name is automatically created based on the object type, in this case, **Force**. Also note that this new object provides the Details view property **Children in Group** that displays the total number of objects contained in the new group.

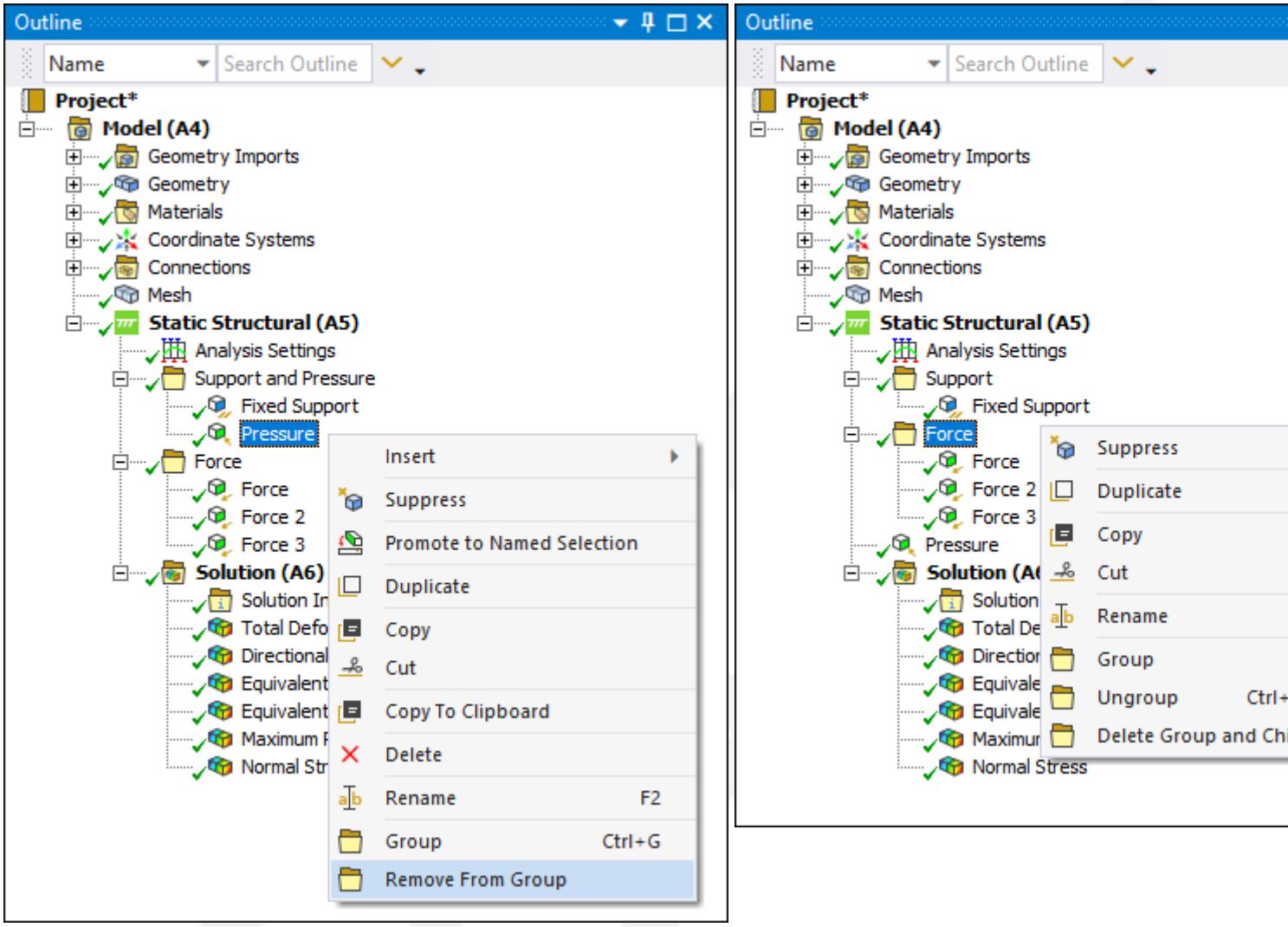




Once defined, options become available that enable you to **Suppress** (and **Unsuppress**) a group as well as remove objects from a group (**Remove From Group**), further group objects into sub-folders/groups (**Group**), **Ungroup** a particular folder, as well as delete a folder and its sub-folders (**Delete Group and Children** option or **[Delete]** key). You can also **Cut**, **Copy**, and **Duplicate** the content of a group folder. And in addition to the context menu options, you can drag-and-drop objects between folders.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

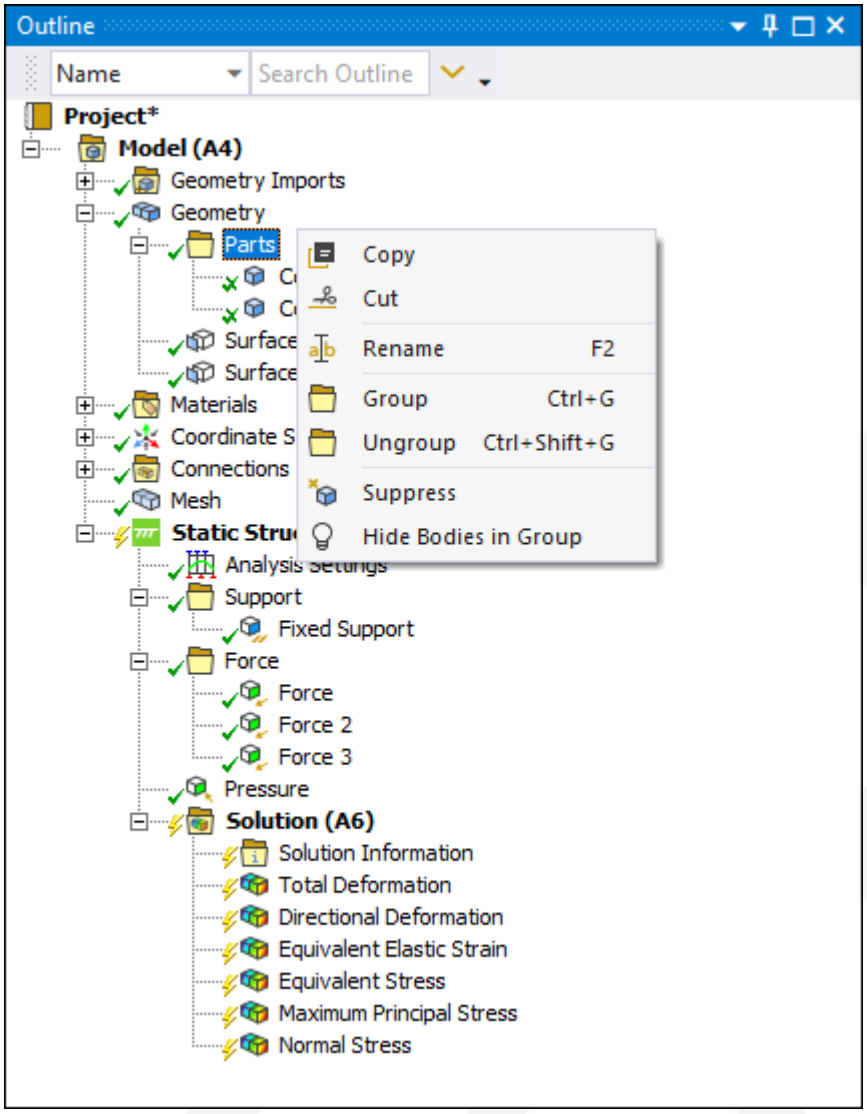
Published: 2023-10-23

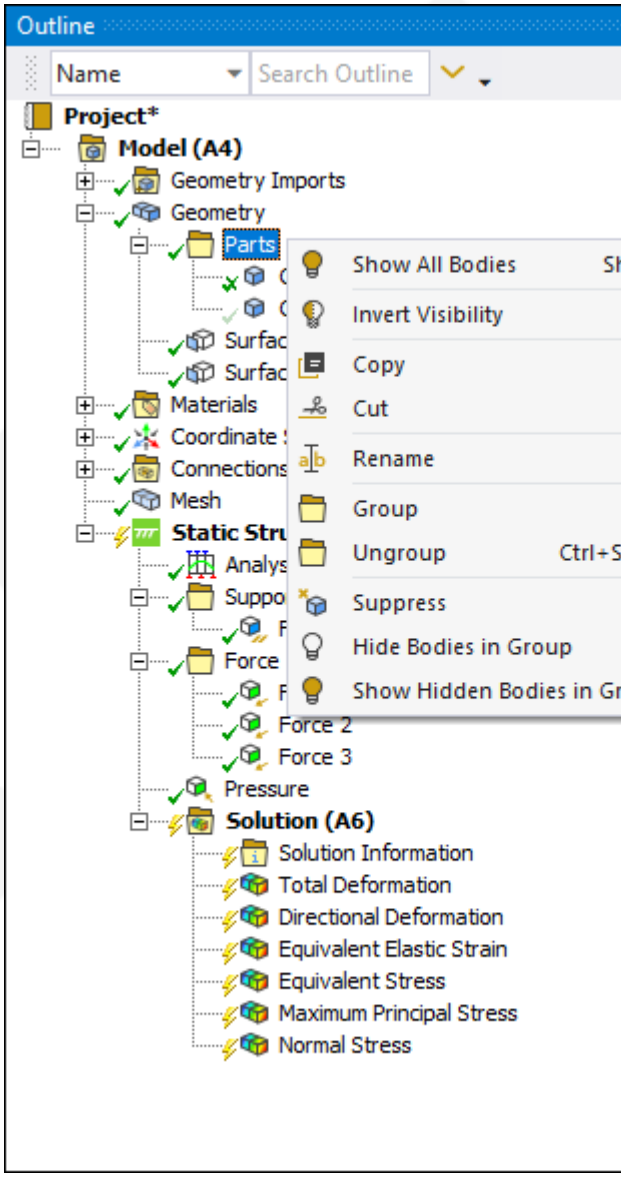


Group, Hide, and Suppress Geometry Objects

As illustrated below, the **Geometry** object offers additional grouping options that enable you to Hide and Show bodies inside and outside of a group as well as Suppress/Unsuppress (p. 113) bodies. The **F9** hotkey (p. 314) also enables you to hide selected bodies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



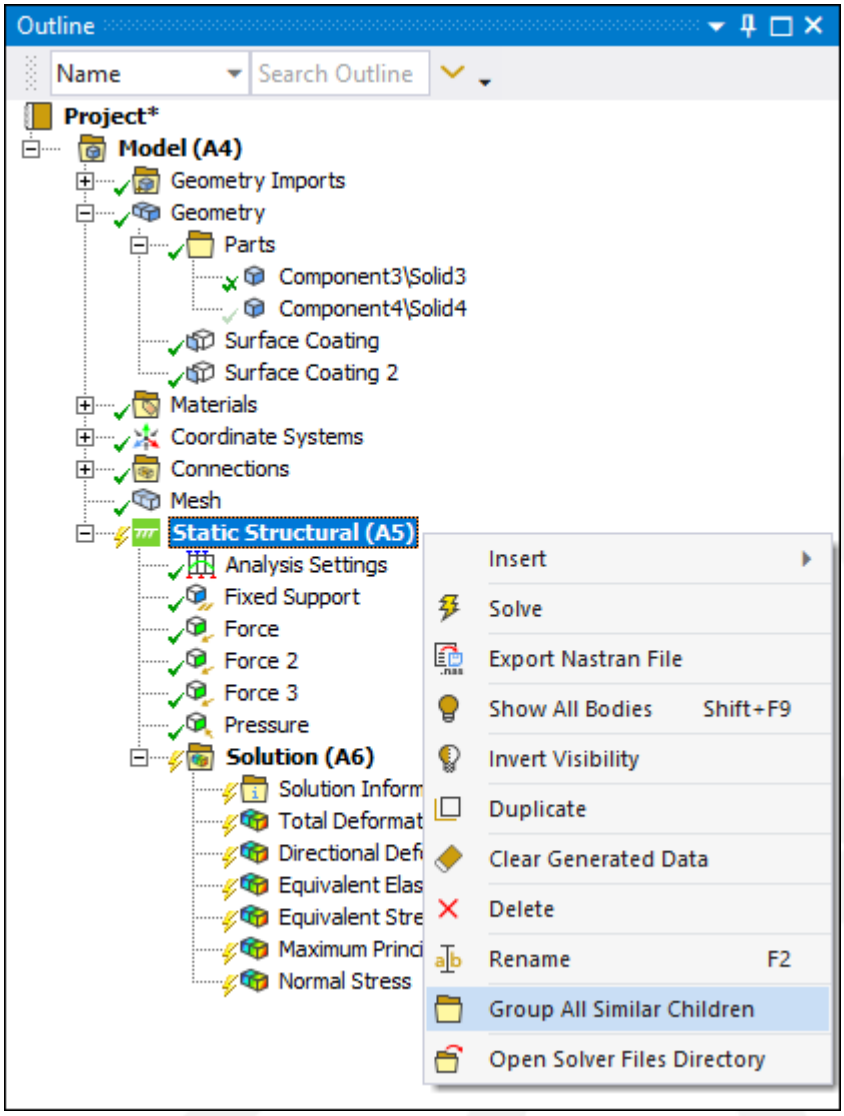


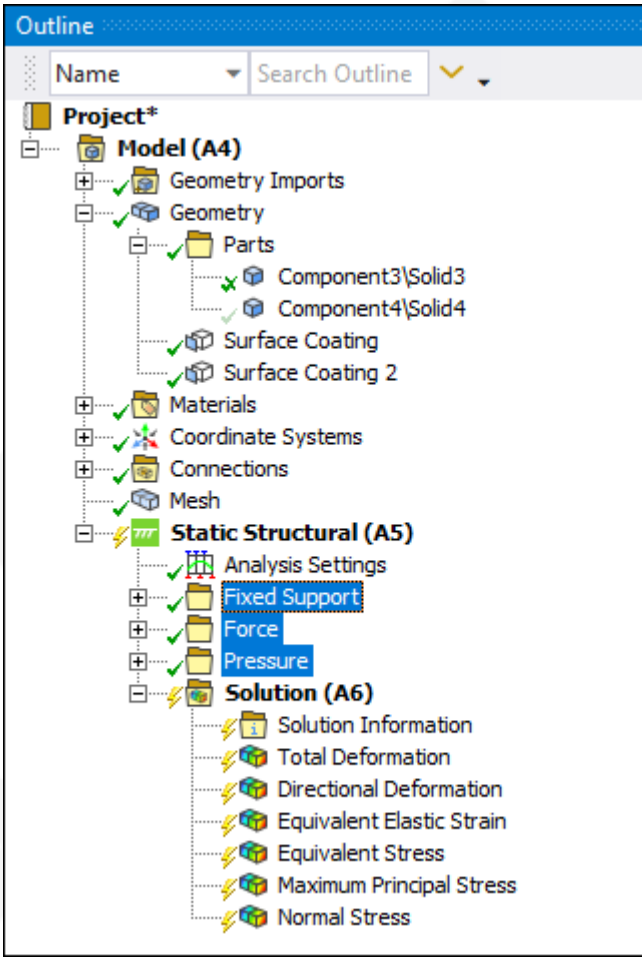
Group All

The **Mesh**, **Named Selection**, **Environment**, and **Solution** objects provide an option to **Group All Similar Children**. This option groups together the same type (for example, Mesh Method, Pressure, Stress result, etc.) of objects (that are not already included in a grouping) and automatically names the folder based on that type.

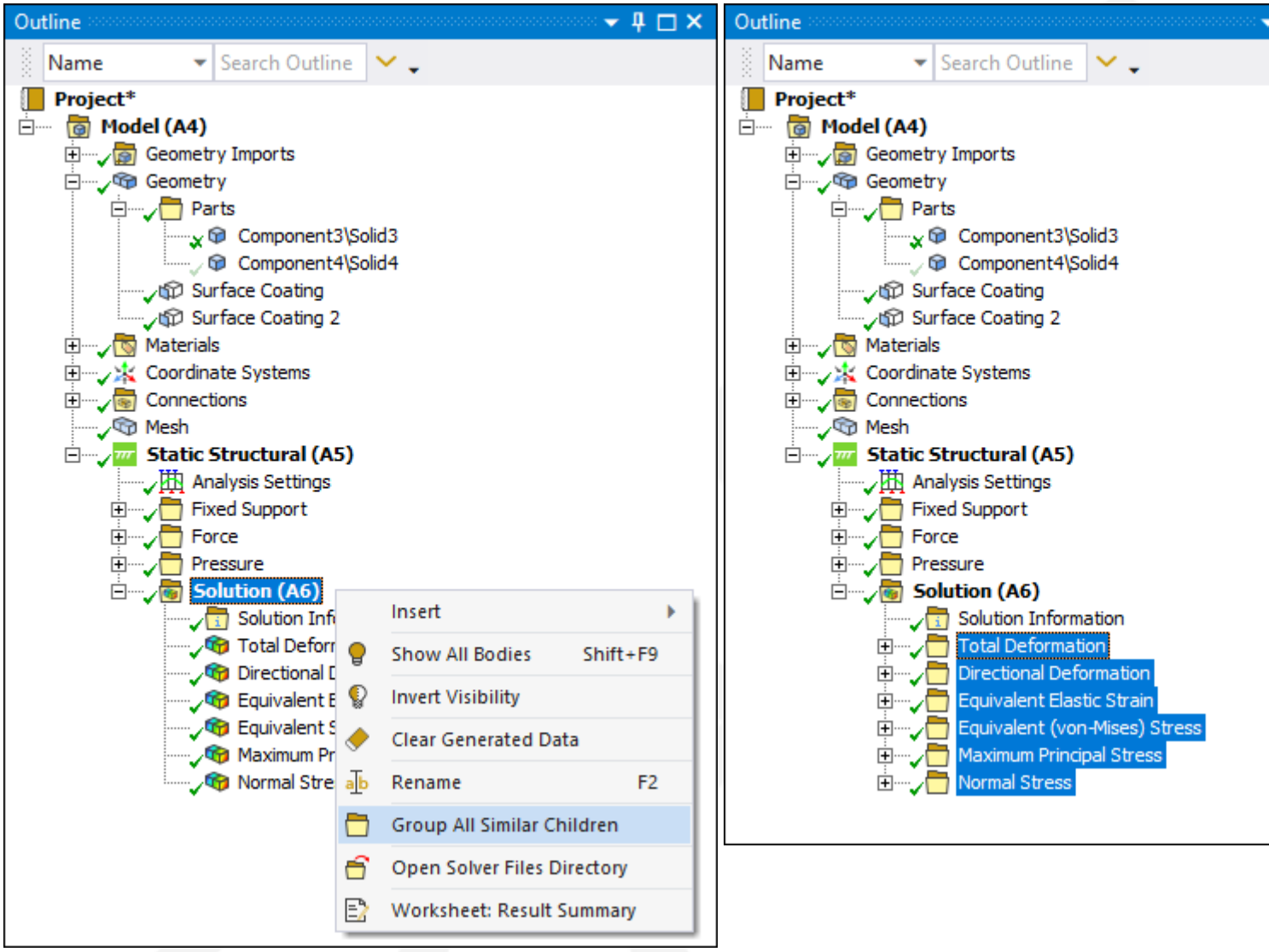
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

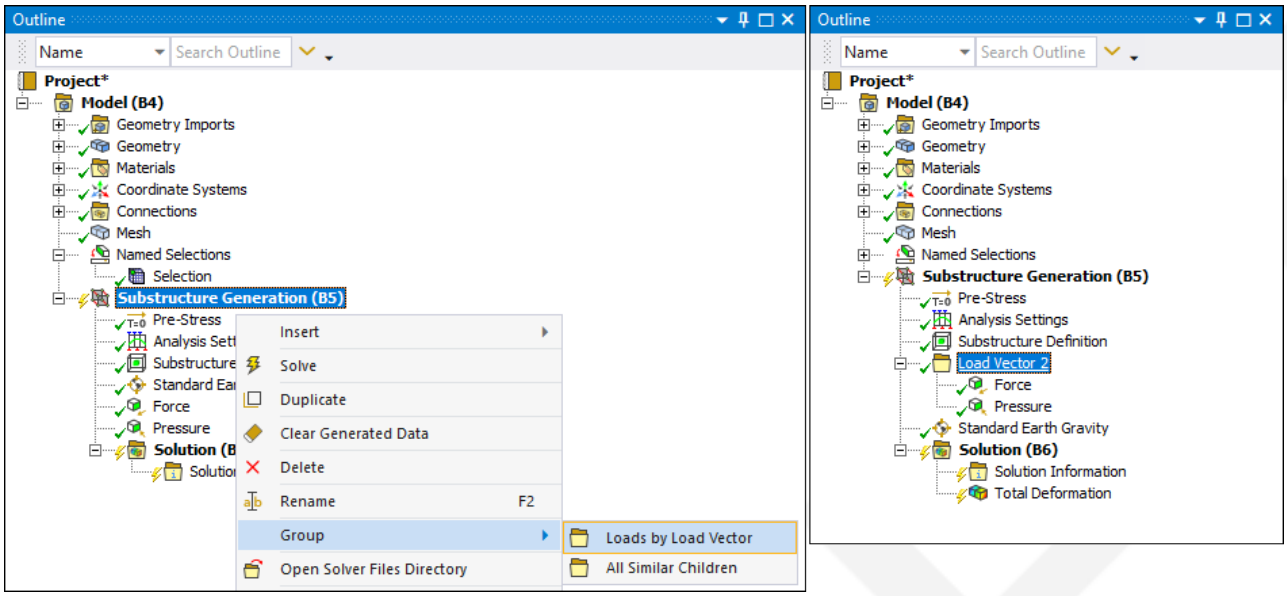


Group Loads by Load Vector (Substructure Generation Analysis Only)

For the boundary conditions supported by the Substructure Generation (p. 540) analysis, the Environment object provides the grouping option: **Loads by Load Vector**. This option automatically groups environment load objects that have the same **Load Vector Number**, as specified by the **Load Vector Controls** properties available from the boundary condition object. The application generates a folder based on the corresponding **Load Vector Number**, as illustrated below for load vector number **2**. Note that loads with the same load vector number that are already included in a group, are not added to the new group.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Objects with Limited Grouping

The following objects have limitations regarding grouping. This includes several objects that cannot be grouped.

- **Model** object children (except **Chart**) cannot be grouped. However, the child objects of these model-level children may be grouped.
- System generated Named Selections under the **Fracture** object cannot be grouped.
- **Gasket Mesh Control** objects under a **Part** object cannot be grouped.
- **Global Coordinate System**: this is the application defined Coordinate System and does not support grouping.
- **Reference Coordinate Systems**: These coordinate systems are *not* defined by the **Coordinate Systems** object and do not support grouping.
- **Analysis Settings**: grouping not supported.
- **Initial Conditions** (including child objects): grouping not supported.
- **Solution Information**: grouping not supported.

Interface Behavior Based on License Levels

The licensing level that you choose automatically enables you to exercise specific features and blocks other features that are not allowed. Presented below are descriptions of how the interface behaves when you attempt to use features not allowed by a license level.

- If the licensing level does not allow an object to be inserted, it will not show in the **Insert** menus.
- If you open a database with an object that does not fit the current license level, the database changes to "read-only" mode.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If a Details view option is not allowed for the current license level, it is not shown.
- If a Details view option is not allowed for the current license level, and was preselected (either through reopening of a database or a previous combination of settings) the Details view item will become invalid and shaded yellow.

Note:

When you attempt to add objects that are not compatible with your current license level, the database enters a read-only mode and you cannot save data. However, provided you are using *any* license, you can delete the incompatible objects, which removes the read-only mode and enables you to save data and edit the database.

Environment Filtering

The Mechanical interface includes a filtering feature that only displays model-level items applicable to the particular analysis type environments in which you are working. This provides a simpler and more focused interface.

The environment filter has the following characteristics:

- Model-level objects in the tree that are not applicable to the environments under a particular model are hidden.
- The user interface inhibits the insertion of model-level objects that are not applicable to the environments of the model.
- Model-level object properties (in the Details view of objects) that are not applicable to the environments under the model are hidden.

The filter is enabled by default when you enter the Mechanical application. You can disable the filter by highlighting the **Model** object, clicking the right mouse button, and choosing **Disable Filter** from the context menu. To enable the filter, repeat this procedure but choose **Auto Filter** from the context menu. You can also check the status of the filter by highlighting the **Model** object and in the Details view, noting whether **Control** under **Filter Options** is set to **Enabled** or **Disabled**.

The filter control setting (enabled or disabled) is saved when the model is saved and returns to the same state when the database is resumed.

Application Preferences and Default Behaviors

You can specify certain application default settings and behaviors as well as create unique options. Select from and review the following for more information:

- Specifying Application Defaults and Preferences (p. 217)
- Creating User-Defined Buttons (p. 154)
- Setting Variables (p. 247)
- Using Macros (p. 248)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

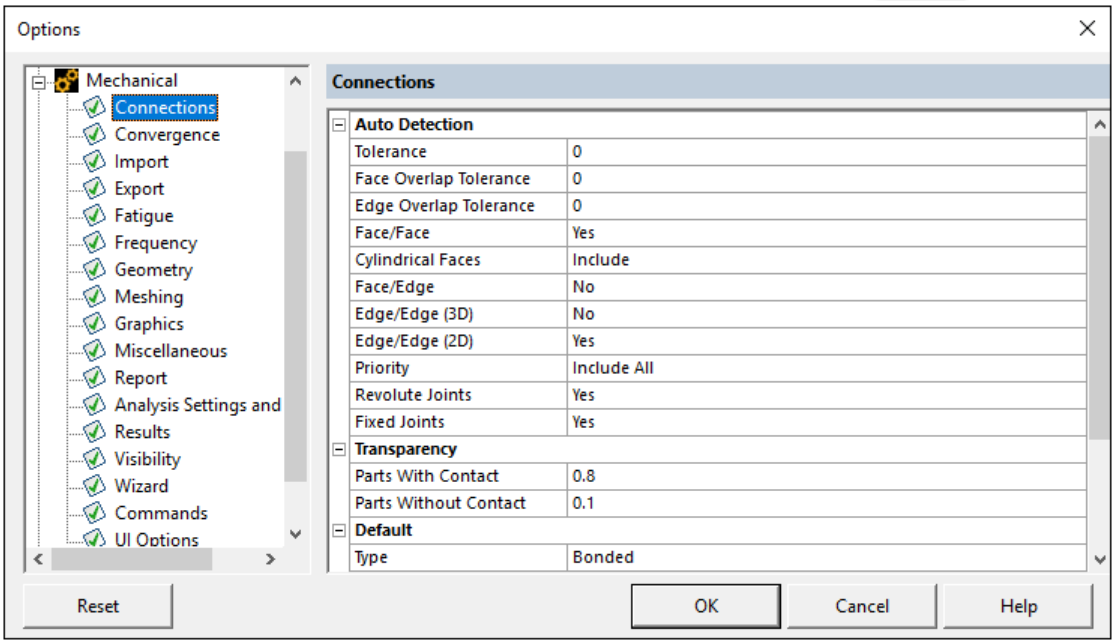
Specifying Application Defaults and Preferences

Using the **Options** dialog, you can control various behaviors and default functions of the application to better suit your uses. This feature essentially enables you to establish preferences for application behaviors and property settings.

Application

To open and make changes to **Options** settings:

1. Select the **File** tab and then **Options** (an option is also available beside the **Quick Launch** field on the title bar). A dialog box titled **Options** displays. Groupings associated with default behaviors for the application display under the Mechanical heading. These groups are referred to as categories. Within each category are various properties that you can change the settings for. For example, and as illustrated below, the **Connections** category is highlighted by default. Here you can see that you can specify a value for the **Face Overlap Tolerance** from the default of zero (0). These types of customizations can be very beneficial.



2. Select a desired category. Each category has associated properties.

3. Change any of the property settings by clicking directly in the field. You will first see a visual indication for the kind of interaction required in the field (examples are drop-down menus, secondary dialog boxes, direct text entries).

4. Click **OK**.

Important:

Option settings within a particular language are independent of option settings in another language. If you change any options from their default settings, then start a new Workbench

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

session in a different language, the changes you made in the original language session are not reflected in the new session. You are advised to make the same option changes in the new language session.

Note:

User Preferences File

The Mechanical application stores the configuration information from the **Options** dialog box in a file called a User Preference File on a per user basis. This file is created the first time you start the Mechanical application. Its default location is:

```
%APPDATA%\Ansys\v241\%AWP_LOCALE241%\dsPreferences.xml
```

Mechanical Options

Select a link below to go to the topic concerned with the desired application preference:

Connections (p. 219)
Convergence (p. 221)
Import (p. 222)
Export (p. 222)
Fatigue (p. 223)
Frequency (p. 224)
Geometry (p. 225)
Geometry Import (p. 227)Meshing (p. 227)
Graphics (p. 228)
Miscellaneous (p. 235)
Messages (p. 236)
Report (p. 237)
Analysis Settings and Solution (p. 238)
Results (p. 242)
Visibility (p. 243)
Wizard (p. 243)
Commands (p. 244)
UI Options (p. 244)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Connections

The **Auto Detection** category enables you to change the default values for the following:

Note:

The auto contact detection on geometry attach can be turned on/off from the Workbench Options dialog box for the Mechanical application. See the Mechanical part of the Setting Ansys Workbench Options section of the Help.

- **Tolerance**: Sets the default for the contact detection slider; that is, the relative distance to search for contact between parts. The *higher* the number, the *tighter* the tolerance. In general, creating contacts at a tolerance of 100 finds fewer contact surfaces than at 0. The default is **0**. The range is from **-100** to **+100**.
- **Face Overlap Tolerance**: Sets the default tolerance for overlap of faces in contact; that is, the minimum percentage of overlap at which a contact pair is created for two overlapping faces. For example, if **Face Overlap Tolerance** is set to 25, a contact pair is created for each pair of faces for which at least 25% of one face overlaps the other. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a tolerance that is appropriate for your simulation type. The default is **0**, which means overlap checks are turned off. The range is from **0** to **100**.

  Note:

  The **Face Overlap Tolerance** value and the **Tolerance** value are evaluated together to determine which faces are considered to be in contact.

- **Face/Face**: Sets the default preference[1] (p. 220) for automatic contact detection between faces of different parts. The choices are **Yes** or **No**. The default is **Yes**.
- **Cylindrical Faces**: Set the default for separating flat surfaces from cylindrical faces for face/face contact. Options include **Include** (default), **Exclude**, and **Only**.
- **Face/Edge**: Sets the default preference[1] (p. 220) for automatic contact detection between faces and edges of different parts. The choices are:
  - **Yes**
  - **No** (default)
  - **Only Solid Edges**
  - **Only Surface Edges**
  - **Only Beam Edges**
- **Edge Overlap Tolerance**: Sets the default tolerance for overlap of an edge and a face in contact; that is, the minimum percentage of overlap at which a contact pair is created for an edge and a face that overlap. For example, if **Edge Overlap Tolerance** is set to 25, a contact pair is created

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

for an edge and a face when at least 25% of the edge overlaps the face. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a tolerance that is appropriate for your simulation type. The default is **0**, which means overlap checks are turned off. The range is from **0** to **100**.

- **Edge/Edge (3D)**: Sets the default preference[1] (p. 220) for automatic contact detection between edges of different parts in a three dimensional model. The choices are **Yes** or **No**. The default is **No**.

- **Edge/Edge (2D)**: Sets the default preference[1] (p. 220) for automatic contact detection between edges of different parts in a two dimensional model. The choices are **Yes** or **No**. The default is **Yes**.

- **Priority** (p. 1217): Sets the default preference[1] (p. 220) for the types of contact interaction priority between a given set of parts. The choices are:

  – **Include All** (default)

  – **Face Overrides**

  – **Edge Overrides**

- **Revolute Joints**: Sets the default preference for automatic joint creation (p. 1359) of revolute joints (p. 1299). The choices are **Yes** and **No**. The default is **Yes**.

- **Fixed Joints**: Sets the default preference for automatic joint creation (p. 1359) of fixed joints (p. 1298). The choices are **Yes** and **No**. The default is **Yes**.

[1] Unless changed here in the **Options** dialog box, the preference remains persistent when starting any Workbench project.

The **Transparency** category includes the following exclusive controls for this category. There are no counterpart settings in the Details view.

- **Parts With Contact**: Sets transparency (p. 1269) of parts in selected contact region so the parts are highlighted. The default is **0.8**. The range is from **0** to **1**.

- **Parts Without Contact**: Sets transparency of parts in non-selected contact regions so the parts are not highlighted. The default is **0.1**. The range is from **0** to **1**.

The **Default** category enables you to change the default values for the following:

- **Type**: Sets the definition type of contact (p. 1234). The choices are:

  – **Bonded** (default)

  – **No Separation**

  – **Frictionless**

  – **Rough**

  – **Frictional**

- **Behavior** (p. 1236): Sets the contact pair. The choices are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Program Controlled** (default)
  - **Asymmetric**
  - **Symmetric**
  - **Auto Asymmetric**
- **Formulation**: Sets the type of contact formulation method (p. 1241). The choices are:
  - **Program Controlled** (default)
  - **Augmented Lagrange**
  - **Pure Penalty**
  - **MPC**
  - **Normal Lagrange**
- **Small Sliding**: This setting sets the default behavior for the **Small Sliding** (p. 1244) property. The options include **Program Controlled** (default), **Yes**, and **No**.
- **Update Stiffness**: Enables an automatic contact stiffness update (p. 1249) by the program. The choices are:
  - **Program Controlled** (default)
  - **Never**
  - **Each Iteration**
  - **Each Iteration, Aggressive**
- **Shell Thickness Effect** (p. 1232): This setting enables you to automatically include the thickness of surface bodies during contact calculations. The default setting is **No**.
- **Auto Rename Connections**: Automatically renames joint, spring, contact region, and joint condition objects when **Type** or **Scoping** are changed. The choices are **Yes** and **No**. The default is **Yes**.
- **Bushing Joint Worksheet View**: Enables you to set the default display (on or off) of the Worksheet for a Bushing Joint (p. 1303). Options include **Yes** (default) and **No**.

Convergence

The **Convergence** category enables you to change the default values for the following:

- **Target Change**: Change of result from one adapted solution to the next. The default is **20**. The range is from **0** to **100**.
- **Allowable Change**: This should be set if the criteria is the max or min of the result. The default is **Max**.

The **Solution** category enables you to change the default values in the Details view for the **Max Refinement Loops** property. This property enables you to change the number of refinement loops the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

application performs. The default is **1**. The range is from **1** to **10**. When performing an out of process (p. 1803) solution asynchronously, wherein the solve may finalize during another Workbench session, the application performs only one maximum refinement loop. As necessary, you must manually perform additional loops. To solve with a single user action, solve synchronously.

Import

The **Import** category enables you to specify preferences for when you import data into Mechanical. Currently, these preferences are for importing delamination interfaces from the Ansys Composite PrepPost (ACP) application.

- **Create Delamination Objects**: This option controls the automatic creation of Interface Delamination objects in Mechanical when importing layered section data from ACP. When Interface layers are specified in ACP, Interface Delamination objects corresponding to Interface Layers are automatically inserted into the Mechanical Tree Outline under the Fracture object. The default setting is **Yes**.
- **Delete Invalid Objects**: This option controls the deletion of Invalid Interface Delamination objects scoped to Interface Layers from ACP. When an Interface Layer specified in ACP is deleted, the corresponding Interface Delamination object is deleted in Mechanical when the project is refreshed. The default setting is **No**. This default setting suppresses invalid objects instead of automatically deleting them.

Export

The **Text File Export** category provides the following exclusive settings. There are no counterpart settings in the Details view.

- **File Encoding**: select either **ASCII** (default) or **UNICODE (Windows only)** as the encoding to use for exporting data.
- **Automatically Open Excel**: Excel will automatically open with exported data. The default is setting **Yes**.
- **Remove Duplicate Nodes**: When set to **Yes**, the application automatically removes any duplicate nodes when you export data to a text file (p. 249). The default setting is **No**.
- **Include Node Numbers**: Node numbers will be included in exported file. The default is setting **Yes**.
- **Include Node Location**: Node location can be included in exported file. The default is setting **No**.
- **Show Tensor Components**: Options include **Yes** and **No** (default). For the default setting **No**, the export data contains the principal stresses and strains (1, 2, and 3) as well as the three Euler angles. The export data for the **Yes** setting contains raw components of stress and strain (X, Y, Z, XY, YZ, XZ).
- **Show Additional Header**: Options include **Yes** and **No** (default). Prepend the exported file with the following header information:
  - Result File Name
  - Unit
  - Coordinate System

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Environment Name
  - Time
  - Result Name
  - Result Type

The **STL Export** category provides the setting **Export Format**. This property sets the default for how STL files are exported, using either **Binary** (default) or **ASCII** format.

The **AVZ Viewer Option** category provides the property **Open AVZ Viewer**. When this property is set to **Yes**, the application automatically opens the Ansys Viewer (after you have saved the file) when you are exporting a result object using the contextual menu option **Ansys Viewer File (AVZ)**. When set to **No**, the application simply prompts you to save the AVZ file. This feature applies to result objects only.

The **Views** category provides the property **File Directory**. This property enables you to specify a default location to where you will export and/or import the graphical views (p. 292) that you have created, exported, or imported. Using the property's field, you enter a folder location, such as **`C:\Mechanical\Manage_Views`**. This location becomes the default folder location. By default, the application uses the automatically generated **`user_files`** folder.

Fatigue

The **General** category enables you to change the default values for the following:

- **Design Life**: Number of cycles that indicate the design life for use in fatigue calculations. The default is **1e9**.
- **Analysis Type**: The default fatigue method for handling mean stress effects. The choices are:
  - **SN - None** (default)
  - **SN - Goodman**
  - **SN - Soderberg**
  - **SN - Gerber**
  - **SN - Mean Stress Curves**

  The Goodman, Soderberg, and Gerber options use static material properties along with S-N data to account for any mean stress while Mean-Stress Curves use experimental fatigue data to account for mean stress.

The **Cycle Counting** category enables you to change the default values for the following:

- **Bin Size**: The bin size used for rainflow cycle counting. A value of 32 means to use a rainflow matrix of size 32 X 32. The default is **32**. The range is from **10** to **200**.

The **Sensitivity** category enables you to change the default values for the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Lower Variation**: The default value for the percentage of the lower bound that the base loading will be varied for the sensitivity analysis. The default is **50**.
- **Upper Variation**: The default value for the percentage of the upper bound that the base loading will be varied for the sensitivity analysis. The default is **150**.
- **Number of Fill Points**: The default number of points plotted on the sensitivity curve. The default is **25**. The range is from **10** to **100**.
- **Sensitivity For**: The default fatigue result type for which sensitivity is found. The choices are:
  - **Life** (default)
  - **Damage**
  - ***Factor of Safety***

Frequency

The **Modal** category enables you to change the Modal Analysis default values for the following:

- **Max Number of Modes**: The number of modes that a newly created frequency branch will contain. The default is **6**. The range is from **1** to **200**.
- **Limit Search to Range**: You can specify if a frequency search range should be considered in computing frequencies. The default is **No**.
- **Min Range (Hz)**: Lower limit of the search range. The default is value is **0.01** for Modal Acoustic analyses and **0.0** all other analysis types.
- **Max Range (Hz)**: Upper limit of the search range. The default is value is **100000000**.
- **Cyclic Phase Number of Steps**: The number of intervals to divide the cyclic phase range (0 - 360 degrees) for frequency couplet results in cyclic modal analyses.

The **Eigenvalue Buckling** category enables you to change the Eigenvalue Buckling Analysis default values in the Details view for the **Max Modes to Find** property. This property defines the number of buckling load factors and corresponding buckling mode shapes. The default value is **2**.

The **Harmonic** category enables you to change the default setting for the **Frequency Spacing** property. The options include:

- **Linear** (default)
- **Logarithmic**
- **Octave Band**
- **1/2 Octave Band**
- **1/3 Octave Band**
- **1/6 Octave Band**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **1/12 Octave Band**
- **1/24 Octave Band**

Geometry

The **Import** category provides the following properties:

- **Assembly Hierarchy**: Options include **Yes** and **No** (default). If your CAD file includes a hierarchy structure, the **Yes** setting enables Mechanical to automatically group the parts and bodies under the **Geometry** object.

  Note:

  If you group parts and bodies in SpaceClaim, you need to account for shared topology and the fact that shared topology creates multi-body parts. In this instance, the application groups multi-body parts together regardless of the assembly structure. You can use the **View Assembly Structure** tool to see how the SpaceClaim assembly structure is affected by Shared Topology in Mechanical. See the *SpaceClaim Documentation* (Workbench > Shared Topology > Viewing Tools > View Assembly Structure) for more information.

  Caution:

  You cannot change the **Assembly Hierarchy** setting once you have imported your geometry. This is due to a geometry update limitation.

- **Volume Calculation**. Options include **Analytical** (default) and **Faceted**. Using the **Faceted** option, the volume is calculated using the faceted (graphical) representation of the volume. This improves computation times. Note, however, that this setting could be less accurate (~0.1% depending on rendering quality or facet quality value) than the default setting.
- **Delete Missing Named Selections from CAD**: Options include **Yes** (default) and **No**. This setting deals with the interaction between the Named Selections you create in your CAD application and import into Mechanical. For either setting, you need to refresh your geometry from the source or refresh your project for Mechanical to be aware of any upstream changes.

  For the default **Yes** setting, if you delete a Named Selection in CAD and refresh your project, Mechanical automatically deletes the Named Selection from the Outline. In addition, any object scoped to the Named Selection becomes invalid.

  For the **No** setting, for any upstream deletion, instead of deleting the object from the Outline, it suppresses. Objects scoped to the Named Selection become invalid, however: the scoped object maintains the reference to the deleted Named Selection. That is, Mechanical keeps the name of the Named Selection in the appropriate property of the scoped object. This capability enables you to recreate the Named Selection in CAD and the application automatically reproduces downstream scoping once you refresh from the source application.

The **Geometry** category enables you to change the default values for the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Beam Cross Section (For Solver)**: Define the default setting to send user-defined cross-sections, to the Mechanical APDL solver, as either a **Pre-Integrated** (default) cross-section or as a **Mesh** section.
- **Nonlinear Material Effects**: Indicates if nonlinear material effects should be included (**Yes**), or ignored (**No**). The default is **Yes**.
- **Thermal Strain Calculation**: Indicates if thermal strain calculations should be included (**Yes**), or ignored (**No**). The default is **Yes**.
- **2D Tolerance**: For surface bodies, you use this property to set the tolerance used to validate that the imported geometry is two-dimensional (2D) by checking the value of the **Length Z** property, using CAD units, in the **Bounding Box** category. The default value for this property is 0.00001.
- **2D Axisymmetric Check**: When you import a 2D geometry (p. 878) and set its **Behavior** property to **Axisymmetric**, Mechanical automatically performs a check to make sure that the geometry lies only on the positive X axis. Certain CAD applications can automatically increase bounding box values and cause a geometry to appear in the negative X plane. This generates a system error that prohibits you from executing a solution. This property enables you to change a system generated error to a warning. **Options** include **Error** (default) and **Warning**. A **Warning** setting allows the application to attempt a solution.
- **Clear Mesh on Dimension Change**: When you change the setting (**3D** or **2D**) of the **Dimension** property (Body object) or if you change the setting of the **Analysis Type** property of the **Geometry** cell on the Workbench project page, also **3D** or **2D**, this preference instructs the application whether to automatically clear and regenerate the mesh. Options include **Yes** (default) and **No**.

Certain CAD applications automatically increase the bounding box size beyond the exact limits of the geometry and can cause the geometry to appear in the negative X plane. This causes Mechanical to generate an error and prohibit a solution. In this scenario, you can change the error setting to a warning in order to perform a solution. You use the **Geometry** (p. 225) preference **2D Axisymmetric Check** in the **Options** dialog to change this setting.

Note:

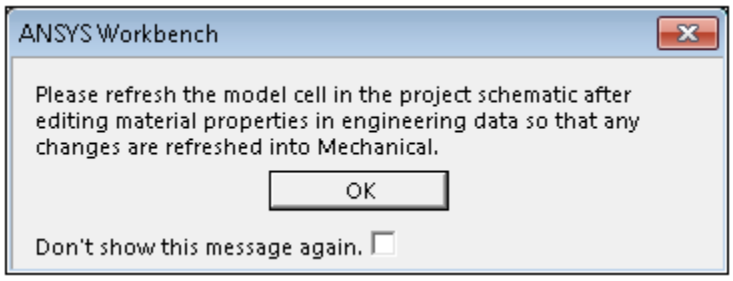
This setting applies only to newly attached models, not to existing models.

The **Material** category enables you to change the default values for the following:

- **Prompt for Model Refresh on Material Edit**: This setting relates to the material **Assignment** (p. 345) property. If you choose to edit a material or create/import a new material via this property, the application displays a message (illustrated below) reminding you to refresh the Model cell in the Workbench Project Schematic. The default setting is **Yes**. The message in Mechanical provides you with the option to not show the message again. This option is in addition to this method of changing this setting to **No**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- **Assign Default Material to New Bodies Based on Update**: This setting relates to the default setting of the **Assign Default Material** property. The **Assign Default Material** property controls default material assignment when geometry is updated in Mechanical.

The **Geometry Based Associativity** category enables you to change the default values for the following:

- **Use Geometry Based Associativity**: Options include **Yes** (default) and **No**. When set to **Yes**, this preference instructs the application to retain available geometry data prior to a geometry update.
- **Automatic Scoping Update**: Options include **Yes** and **No** (default). When set to **Yes**, this preference enables the application to automatically correct geometry-based scoping that becomes lost following a geometry update.

Geometry Import

The **Geometry Import** category enables you to change the default settings of the properties associated with importing geometry into Mechanical. These are the same preferences that are available for the **Geometry** cell of an analysis system on the Workbench Project Schematic before you have specified a model to import. You should review the Geometry Preferences section of the *CAD Integration* documentation as well as the Geometry Import section of the *Workbench User's Guide* for additional information.

Meshing

The **Meshing** category enables you to change the value of **Default Physics Preference**. The physics preference that you choose here will be the default for all Mechanical systems, regardless of whether they are analysis systems or component systems. The choices are:

- **Mechanical** (default)
- **Nonlinear Mechanical**

Note:

The default physics preference that you can set in the Meshing application's **Options** dialog box has no effect on the default that is set for Mechanical systems.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Graphics

The **Default Graphics Options** category enables you to change the default values for the following:

- **Reset Views on Geometry Refresh**: Select whether geometry refreshes will reset the graphical view in Mechanical. The default setting is **No**.
- **Max Number of Annotations to Show**: A slider that specifies the number of annotations that are shown in the legend of the **Geometry** window. The range is adjustable from 0 to 50. The default is **10**.
- **Show Min Annotation**: Indicates if Min annotation will be displayed by default (for new databases). The default setting is **No**.
- **Show Max Annotation**: Indicates if Max annotation will be displayed by default (for new databases). The default setting is **No**.
- **Obscured Label Display Style**: Options include **Solid** (default), **Translucent**, and **Hidden**. When you have an annotation label applied on either the **Mesh** object or on a result object, or you have a probe label applied to a result, this preference lets you choose whether to have the label/probe show "through" the model when a label or probe is not visible because of the model's current orientation.
- **Unobscured Label Tolerance Factor**: There may be times when a label displays as obscured when it should not. This is most often a result of how the application calculates the location of a label. There are instances when a label's position will be located beneath the surface of a body. For the Translucent or Hidden options, this tolerance enables you make small adjustments to the tolerance factor to make sure that the label behaves as expected. The tolerance range is between 0 and 1. Small incremental changes, such as 0.0001, can be appropriate to properly display obscured labels.
- **Number of Local Min/Max Probes**: Specify the number of Min/Max probe labels (p. 2278) you wish to display for your result data. The default setting is **6**. The supported range is 1-20.
- **Contour Option**: Selects default contour option. The options include:
  - **Smooth Contour**
  - **Contour Bands** (default)
  - **Isolines**
  - **Solid Fill**
- **Flat Contour Tolerance**: Flat contours (no variation in color) display if the minimum and maximum results values are equal. The comparison of the minimum and maximum values is made using scientific notation with the number of significant digits to the right of the decimal point as specified with the flat contour tolerance setting (**3** to **9**). Increasing this tolerance enables you to display contours for an otherwise too narrow range of values. Decreasing this tolerance prevents insignificant range variations from being contoured. This setting has a default value of **3**.
- **Edge Option**: Selects default edge option for result display. The choices are:
  - **No Wireframe**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Show Undeformed Wireframe**
  - **Show Undeformed Model**
  - **Show Elements** (default)
- **Highlight Selection**: Indicates default face selection. The choices are:
  - **Single Side** (default)
  - **Both Sides**
- **Number of Circular Cross Section Divisions**: Indicates the number of divisions to be used for viewing line body cross sections (p. 877) for circular and circular tube cross sections. The range is adjustable from 6 to 360. The default is **16**.
- **Mesh Visibility**: Indicates if mesh is automatically displayed when the **Mesh** object is selected in the Tree Outline, or if it's only displayed when you select the **Show Mesh** button. The default is **Automatic**.
- **FE Annotation Color**: This option enables you to change the default coloring for FE related annotations (FE-based Named Selections and/or Objects scoped to Nodes (p. 269) or Elements (p. 276)). It also changes the color of the elements displayed for an Analysis Ply object.
- **Mesh Failed Color**: Set the color of the of the **Mesh Failed** annotation.
- **Mesh Obsolete Color**: Set the color of the of the **Mesh Obsolete** annotation.
- **Geometry Highlight Color**: Specify the default color used when a part or body is selected from the **Geometry** folder.
- **Level of Detail (Beta)**: This selection defines two separate behaviors: 1) the level of complexity for the graphical display of the model in the **Geometry** window and 2) the speed it to takes select objects in the **Outline**. See the beta documentation for this option.
- **Model Rotation Center**: This option enables you to change how the rotation feature behaves. Selections include:
  - **Click to Set** (default): Select a location on the model to be the center of rotation.
  - **On Mouse Down**: Select a location on the model to be the center of rotation. Rotation is available immediately - no additional mouse selections ("clicks") are required.
- **Shell Expansion Edge Angle**: This option enables you to change the setting for the angle used to determine whether adjacent element normals are averaged. This is applicable when shell thickness is being applied to the mesh to represent the actual thickness. The range is adjustable from 0 to 180. The default is **180**.
- **Line Body Thickness**: This option enables you to change how line bodies are displayed in the **Geometry** window. Selections include **Thin** (default) and **Thick**.
- **Mouse Rotation Mode**: This option enables you to change cursor rotation behaviors (p. 280) in the **Geometry** window when you are using the **Rotate** option on the Graphics toolbar (p. 97). Selections include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - **Free Rotate Only** (default): using this setting, the cursor provides free 360° model rotation.
  - **Axis Rotation Available**: specifying this setting activates the roll, yaw, and pitch cursor options.

- **Triad Smooth Rotation**: Active by default, this option instructs the application to compute the shortest path between model positions when using the **Triad** feature in order to facilitate smoother model rotations. Options include **Yes** (default) and **No**.
- **Show Coupled Physics Analysis**: For analyses that support the use of the Physics Region object, this preference enables you to display the bodies and/or parts associated with each properly defined **Physics Region** as a different color when the Environment object is selected. Options include **Yes** and **No** (default).
- **Consider Section Planes When Mesh Picking**: Options include **Yes** and **No** (default). Activating this preference enables you to select newly displayed finite element entities (nodes, elements, and element faces) from geometries within the model that were hidden before adding the specified Section Plane (p. 294). See the Selecting Mesh Entities Displayed by the Section Plane (p. 301) topic for additional information.
- **Animation Draw Option**: Options include **Yes** (default) and **No**. In older releases, the application first processed result animations and then displayed them instead of displaying the animation as it is being processed. As needed, you can revert to the previous display method by setting the this property to **No**.
- **Use Deformed Edge for Slice ISO Option**: Options include **Yes** (default) and **No**. This property applies to the **IsoSurfaces**, **Capped IsoSurfaces**, and **Section Planes** options of the **Geometry** menu (p. 67) and the **Show Undeformed Wireframe** and **Show Undeformed Model** options of the **Edges** menu, both of the Result (p. 62) Context tab, and how they work together. When you set this property to **No**, the **IsoSurfaces**, **Capped IsoSurfaces**, and **Section Planes** options display the selected result in a deformed state but a wireframe overlay (**Show Undeformed Wireframe**) of the model or a translucent overlay (**Show Undeformed Model**) of the model in an undeformed state.
- **Disable 2D Overlays** (Linux Platform Only): Options include **Yes** and **No** (default). When active, this option stabilizes the graphical display by preventing your model from disappearing during mouse movements. This option also turns off a variety of display features, such as the ability to highlight geometry selections (single, box select, lasso, etc.) prior to selecting a geometric entity, as well as graphics labels (such as interactive probe labels). These display and selection features operate properly, but do not provide pre-selection highlights and labels.
- **Mesh Translucency**: This preference defines the translucency of the displayed mesh. The slider option has a range of **0** (no translucency) to **1** (totally transparent). The default setting is **0**.
- **Maximum Face Angle to Preserve Edges**: When you have the **Edges** (p. 68) result display set to **Show Undeformed Wireframe**, you could experience display issues for edges when the tolerance angle between two faces is insufficient. This preference enables you to increase this tolerance. The default setting is **`120°`**. Supported entry range is **`0`** to **`180°`**.
- **Anti-Aliasing (MSAA)** (Windows Only): Options include **Off**, **Low (2x)**, **Medium (4x)**, and the default **High (8x)**. You use this preference to specify the desired sampling level for the display of the model in the **Geometry** window. The sampling process smooths the display of jagged edges. You can cycle through these settings using the key combinations **Shift**+**[1]** through **Shift**+**[4]**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Contour Edge Line Weight**: For results scoped to edges, this preference enables you to specify the thickness of edge result contours. It provides an entry field combined with a slider option. The setting range is **0** to **8**. The default setting is **5**.
- **Mesh Edge Coloring**: This preference enables you to change the color of mesh lines/edges from the default black to the mesh body's color in order to improve edge visibility for dense meshes. Options include **Yes** and **No** (default).

The **Interactive and Min/Max Probes** category enables you to change how Probe annotations (p. 71) behave and display. Options include:

- **Move Probe Labels with Deformation Scale Factor**: Options include **Yes** (default) and **No**. Enables probe labels to move with a deformed mesh when you make a change to the Deformation Scale Factor (p. 63). If set to **No**, the application deletes probe labels whenever you change to the scale factor.
- **Show Mesh Entity ID in Snap-Probe Labels**: Options include **Yes** (default) and **No**. Enables you to automatically display Node or Element IDs on probe annotations (p. 71) when the **Snap** option is selected.
- **Probe Line Color**: Set the color of the line that connects a probe label to its location on the model.
- **Show Detailed Information in Min/Max Probe Labels**: Options include **Yes** and **No** (default). When active, the application displays the min/max values as well as Node or Element ID on the probe annotation label.

The **Probe Tool and Hit Point Coordinate (Windows Only)** category enables you to change the default values for the attributes (font, size, style, color) of the font displayed for the **Probe** (p. 71) annotations. Options include:

- **Font**: Arial is the default font. Ten font types are available for selection.
- **Size**: The font size is in pixels. 18 is the is the default size. The size range is 10 to 30.
- **Style**: Styles include Regular (default), Bold, Italic, and Bold Italic.
- **Color**: White is the default color. A color palette displays when you select this option.

The **Lighting** category enables you to change the default values for the following properties of the Model object:

- **Ambient**: Default value is 0.1.
- **Diffuse**: Default value is 0.6.
- **Specular**: Default value is 1.
- **Color**: The default is white.
- **Force White Lighting for Results**. When you change the **Color** property of the Model object from white, this property enables you to make sure that load variation contours and result

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

contours will correspond to the color contours of the legend. Options include **Yes** and **No** (default).

Important:

Lighting preferences are project-based, therefore, when you change one of these default settings, you must close Mechanical and Workbench and then begin a new project. Once established, all future saved projects will include the preferences. Any previously saved projects will have the lighting defined when the project was saved.

The **Image Export** category defines the resolution, image content, background characteristics, and font size contained on the image when you save it as a file using the **Image to File** option in the **Image** drop-down menu on the **Insert** (p. 42) group Home (p. 40) tab, or when you create a **Figure** (p. 313), or when you prepare a **Print Preview** (p. 173) of an object or a **Report Preview** (p. 174). It includes the following properties:

- **Current Graphics Display**: Specify that the option is turned on or off: **Yes** (default) or **No**.
- **Graphics Resolution**: Defines default resolution setting. Options include:
  - **Optimal Onscreen Display (1:1)** (default)
  - **Enhanced Print Quality (2:1)**
  - **High Resolution Print Quality (4:1)**

Note:

The Ansys logo does not scale at higher resolution settings.

- **Image to Clipboard Uses Graphics Resolution**: Instructs the application to use the setting of the **Graphics Resolution** preference (described above) when you use the **Image to Clipboard** option of the **Images** drop-down menu (**Home** tab/Insert (p. 42) group). Settings for this preference include **Yes** and **No** (default).
- **Capture**: Defines whether the legend is included in the image. Options include:
  - **Image and Legend** (default)
  - **Image Only**
- **Background**: Defines the background coloring. Options include:
  - **Graphics Appearance Setting** (default): Mechanical setting or user-defined background color.
  - **White**
- **Show Preferences Dialog**: **Yes** (default) or **No**. This option determines whether a dialog box automatically displays when you select the **Image to File** option. The dialog box contains all of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the above options. If disabled using this option, the application saves the most recent settings that you have used.

- **Font Magnification Factor**: This setting changes the font size of the textual content contained in the legend. The magnification range for the font size is **0.5** to **1.5**. If you enter a value less than or greater than this range, the application will default to the corresponding minimum (**0.5**) or maximum (**1.5**) value. The default value is **1**. This setting also scales the contour color band.

- **Append Graph**: Options include **Yes** and **No** (default). When set to **Yes**, the application adds the content of the **Graph** window in to the captured image of the viewport. If there is no corresponding data in the **Graph**, the captured image is appended with a graph that includes no data. When set to **Yes**, the corresponding setting on the Image to File Preferences (p. 42) dialog is activated.

- **Composite Image Borders**: Options include **None** (default), **Interior**, **Exterior**, and **Interior and Exterior**. The application automatically adds the specified borders around/in-between the captured viewports or includes no borders. When you select one of the border options, the corresponding setting(s) on the Image to File Preferences (p. 42) dialog is/are automatically activated.

- **Probe Label Offset**: The options for this setting relate to how the application captures and saves images of results displayed in the **Geometry** window as well as how the application presents images on the **Print Preview** (p. 173) tab, when you have inserted Probe labels and then moved those probe labels (p. 207) on the screen. Options include:

  – **Respect User-Defined Offset** (default): This options ensures that probe labels maintain their location in the **Geometry** window when you are using the **Image to File** option contained in the **Image** drop-down menu on the **Insert** (p. 42) group Home (p. 40) tab or if you are using the **Print Preview** feature.

  – **Reset to Probe Anchor**: This options ensures that probe labels maintain their location in the **Geometry** window when you are using the **Image to File** option but only when the **Graphics Resolution** preference is a 1:1 ratio. Otherwise, the application places the probe labels in their original position. This option also places the probe labels in their original position if you employ the **Print Preview** feature.

Animation Export: This category contains the property **Legacy Animation Export**. Setting this property to **Yes** enables you to export AVI animation files using Microsoft Windows API on the Windows platform. The default setting is **No**.

The **Legend** category provides the control of the following:

- **Orientation**: Select a desire display orientation for the legend. Options include **Vertical** (default) and **Horizontal**.

- **Show Min/Max on Color Bar**: You use this setting to either display or hide (default) the legend's context menu (right-click) option **Show Min/Max on Color Bar**. Options include **Yes** and **No** (default).

- **Show Deformation Scale Factor**: You use this setting to either display (default) or hide the legend's context menu (right-click) option **Show Deformation Scale Factor**. Options include **Yes** (default) and **No**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Show Date and Time**: Options include **Yes** (default) and **No**. This control enables you turn off the display of the date and time in the **Geometry** window. The context (right-click) menu option **Date and Time** also changes this default setting.
- **Max Number of Labels to Shown in Legend**: A slider that specifies the number of annotations that are shown in the legend of the **Graph** window when you are using Charts (p. 1929). The range is adjustable from 0 to 50. The default is 10.

The **Performance** category provides the control of the following:

- **Accelerated Graphics for Mesh/Result Display**: See the Accelerated Graphics (p. 289) section for more information. Property options include:
  - **Program Controlled** (default): The application selects the graphical mode based on application settings and support status, such as analysis settings, the specified result, view options, etc. This is the recommended setting.
  - **Yes, if possible**: This setting forces the use of **Accelerated Graphics** wherever possible, even if only partially supported.
  - **No**: This setting disables the feature.
- **Animation Mode** (p. 2268). Options include:
  - **Program Controlled**: The application selects the animation mode based on the given analysis to balance performance and display features.
  - **Standard** (default): This setting uses the traditional animation process. It supports all display options without processing capabilities.
  - **Accelerated (Beta)**: This setting offers improved processing capabilities.
- **Use Faceted Lighting for Accelerated Graphics/Animation (Beta)**: This beta property enables you to modify the lighting. Options include **No** (default) and **Yes**. The **No** option specifies the standard lighting mode. This setting can cause issues with large deformation but maintains smooth lighting and is suitable for most models. The **Yes** setting uses an alternate faceted lighting mode. Lighting is done on a per-triangle basis resulting in a more jagged rendering of some models, but the lighting should be correct for any deformation.
- **Show Why Accelerated Graphics is Not Used**: Display an Information message for why Accelerated Graphics (or Accelerated Animation) is not used for the mesh and results when it is not possible. Options include **No** (default) and **Yes**.

Note:

Informational messages may display even when the Accelerated Graphics feature is disabled because this preference also supports Accelerated Animation. Accelerated Animation is controlled by the **Animation Mode** preference described above.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Edge Connectivity Display Mode**: This preference controls the internal algorithm used to compute and highlight the edges of your model when the By Connection (p. 84) option is selected. Options include **Accelerated** (default) and **Legacy**.

  Important:

  When set to **Accelerated**, hidden faces are not taken into consideration when computing the number of shared edges.

- **Show Accelerated Display Message in Graphics Window**: Turn the "Accelerated Display" label on and off. Options include **No** (default) and **Yes**.
- **Varying Loads (Optimization Options)**: Specify how varying loads display in the **Geometry** window. Options include **Accuracy** (default) and **Performance**. The **Accuracy** setting displays variable load contours normally. The **Performance** option displays colored discrete points on the model, based on legend colors, of the load variation. This option provides significantly faster redrawing times. The computational improvement may be desirable for models with a large number of parts/bodies.

Miscellaneous

The **Miscellaneous** property enables you to change the default values for the following:

- **Load Orientation Type**: Specifies the orientation input method for certain loads. This input appears in the **Define By** option in the Details view of the load, under **Definition**.
  - **Vector** (default)
  - **Component**
- **Reverse Displacement for Inverse Steps**: Options include **No** (default) and **Yes**.

Cross Sections

Unique Names:

The **Named Selections** category includes the preference **Unique Names (MAPDL Solver)**. Options include **Warning** (default) and **Error**. When the Mechanical APDL solver encounters a duplicate named selection it issues a Warning. This preference enables you to change the issuance to an **Error**. If set to **Error**, the application will stop the solution process and allow you to address the naming issue.

The **Image** category includes the **Image Transfer Type** control. There are no counterpart settings in the Details view. Using this control, you define the type of image file created when you send an image to Microsoft Word or PowerPoint,or when you select Print Preview. Options include:

- **PNG** (default)
- **JPEG**
- **BMP**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The **Post Processing (MAPDL Only)** category includes the control **Result File Caching**. This control is for results files written by the Mechanical APDL solver only. By holding substantial portions of a file in memory, caching reduces the amount of I/O associated with result file reading. The cache can, however, reduce memory that would otherwise be used for other solutions. Control options include:

- **System Controlled** (default): The operating system determines whether or not the result file is cached for reading.
- **Off**: There is no caching during the reading of the result file.
- **Program Controlled**: The Mechanical application determines whether or not the result file is cached for reading.

Note:

You need to close and then reopen Mechanical in order for changes to this preference to take effect.

The **Save Options** category includes the following controls for this category.

- **Save Project Before Solution**: Sets the **Yes** / **No** default for the **Save Project Before Solution** setting located in the Project Details panel. Although you can set the default here, the solver respects the latest **Save Project Before Solution** setting in the Details panel. The default for this option is **No**. Selecting **Yes** saves the entire project immediately before solving (after any required meshing). If the project had never been previously saved, you can now select a location to save a new file.
- **Save Project After Solution**: Sets the **Yes** / **No** default for the **Save Project After Solution** setting in the Project Details panel. The default for this option is **No** Selecting **Yes** Saves the project immediately after solving but before postprocessing. If the project had never been previously saved, nothing will be saved.

Note:

The save options you specify on the Project Details panel override the options specified in the Options dialog box and will be used for the current project.

Clipboard

Find Objects by Name on Paste:

The **Legend** category of the Miscellaneous option provides the control **Show Date and Time**. Options include **Yes** (default) and **No**. This control enables you turn off the display of the date and time in the **Geometry** window. The context (right-click) menu option **Date and Time** also changes this default setting.

Messages

The **Messages** category enables you to change the default values for the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Report Performance Diagnostics in Message**: Turn on messaging that reports the time it takes for certain processes to execute, such as the time it takes for contact detection, mesh generation, writing the input file, solution, etc. Options include **Yes** and **No** (default).
- **Pop-up Messages**: Turn pop-up messages in the Message Window (p. 200) on (**Yes**, default) or off (**No**).
- **Message Coloring**: Set whether to have the application highlight objects in the tree that are experiencing an issue. The objects as well as the corresponding message in the Messages window can be highlighted or you can select to highlight only Message window content. The available options include **On** (default), **Off**, and **Messages Window Only**.

  Note:

  You need to close and then reopen Mechanical in order for changes to this preference to take effect.

- **Show Info Messages**: Turn **Information** messages on or off. Options include **Yes** (default) and **No**.
- **Show Warning Messages**: Turn the automatic display of **Warning** messages on or off. Options include **Yes** (default) and **No**.
- **Merge Similar Messages**: Merge similar messages of the Messages (p. 200) window into one row. Options include **Yes** (default) and **No**. Changing this preference changes the setting of the **Merge Message** option of the **Messages** window if you close the application.

Report

The **Figure Dimensions (in Pixels)** category includes the following controls that allow you to make changes to the resolution of the report for printing purposes.

- **Chart Width** - Default value equals **600** pixels.
- **Chart Height** - Default value equals **400** pixels.
- **Graphics Width** - Default value equals **600** pixels.
- **Graphics Height** - Default value equals **500** pixels.
- **Graphics Resolution** - Resolution values include:
  - **Optimal Onscreen Display (1:1)**
  - **Enhanced Print Quality (2:1)**
  - **High-Resolution Print Quality (4:1)**

The **Customization** category includes the following controls:

- **Maximum Number of Table Columns**: (default = **12** columns) Changes the number of columns used when a table is created.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Merge Identical Table Cells**: merges cells that contain identical values. The default value is **Yes**.
- **Omit Part and Joint Coordinate System Tables**: chooses whether to include or exclude Coordinate System data within the report. This data can sometimes be cumbersome. The default value is **Yes**.
- **Include Figures**: specifies whether to include **Figure** objects as pictures in the report. You may not want to include figures in the report when large solved models or models with a mesh that includes many nodes and elements are involved. In these cases, figure generation can be slow, which could significantly slow down report generation. The default value is **Yes**.

  Note:

  This option applies only to **Figure** objects as pictures. **Graph** pictures, Engineering Data graphs, and result graphs (such as phase response in a harmonic analysis) are not affected and will appear regardless of this option setting.

- **Custom Report Generator Folder**: reports can be run outside of the Workbench installation directory by copying the Workbench **Report2006** folder to a new location. Specify the new folder location in this field. See the Customize Report Content (p. 177) section for more information.

Analysis Settings and Solution

The **Solver Controls** category enables you to change the default values for the following:

- **Solver Type**: Specifies which Mechanical APDL solver will be used. The choices are:
  - **Program Controlled** (default)
  - **Direct**
  - **Iterative**
- **Use Weak Springs**: specifies whether weak springs are added to the model. The **Program Controlled** setting automatically enables weak springs to be added if an unconstrained model is detected, if unstable contact exists, or if compression only supports are active. The choices include:
  - **Program Controlled**
  - **On**
  - **Off** (default)
- **Solver Pivot Checking (p. 1434)** : Sets the default for all new analyses created. Options include:
  - **Program Controlled** (default): enables the solver to determine the response.
  - **Warning**: Instructs the solver to continue upon detection of the condition and attempt the solution.
  - **Error**: Instructs the solver to stop upon detection of the condition and issue an error message.
  - **Off**: Pivot checking is not performed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The **Solver Control (Eigenvalue Buckling)** category provides the **Include Negative Load Multiplier** property. Options include **Program Controlled** (default), **Yes**, and **No**. This option enables you to evaluate either negative and positive load multipliers or only positive load multipliers. The **No** setting evaluates positive load multipliers given the load directions. The **Yes** setting evaluates positive load multipliers given the load directions as well as the negative load multipliers by flipping the load directions.

The **Output Controls (Static and Transient)** category enables you to change the default values for the following:

- **Stress** (Default setting = **Yes**)
- **Strain** (Default setting = **Yes**)
- **Nodal Forces** (Default setting = **No**)
- **Contact Miscellaneous** (Default setting = **No**)
- **General Miscellaneous** (Default setting = **No**)
- **Calculate Reactions** (Default setting = **Yes**)
- **Calculate Thermal Flux** (Default setting = **Yes**)

The **Output Controls (Modal)** category enables you to change the default value for the following:

- **Stress**: Writes stress results to the file, `file.mode`. Options include **Yes** (default) and **No**.
- **Strain**: Writes strain results to the file, `file.mode`. Options include **Yes** (default) and **No**.
- **Store Modal Results**: Options include **Program Controlled** (default), **No**, or **For Future Analysis**.

The **Options (Modal, Harmonic and Transient Mode Superposition)** category enables you to change the default settings for the following:

- **On Demand Expansion Option** property: This property is contained in the Options (p. 1447) category of the **Analysis Settings** object. Options include **Program Controlled** (default), **Yes**, and **No**. You use this property to improve solution performance and reduce result file sizes.
- **On Demand Mode Shapes** preference: Options include **Yes** (default) and **No**. When you set this preference and the **On Demand Expansion** property to **Yes**, the application automatically removes mode shape data from result files.

The **Options (Random Vibration)** category enables you to change the default value for the following:

- **Exclude Insignificant Modes**: When set to **Yes**, this property enables you to exclude modes for the mode combination based on the entry of the **Mode Significance Level** property. The default setting is **No**.
- **Mode Significance Level**: This property defines the threshold for the numbers of modes for mode combination. The default setting is **0** (all modes selected). Supported entries are between **0.0** and **1**. Displayed only when **Exclude Insignificant Modes** is set to **Yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The **Output Controls (Random Vibration)** category enables you to change the default value for the following:

- **Keep Modal Results**: include or remove modal results from the result file of Random Vibration analysis. The default setting is **No**.
- **Calculate Velocity**: Write Velocity results to the results file. The default setting is **No**.
- **Calculate Acceleration**: Write Acceleration results to the results file. The default setting is **No**.

The **Restart Controls** category enables you to change the default value for the following:

- **Generate Restart Points**: **Program Controlled** (default setting) automatically generates restart points. Additional options include **Manual**, that provides user-defined settings, and **Off**, which restricts the creation of new restart points.
- **Retain Files After Full Solve**: when restart points are requested, the necessary restart files are always retained for an incomplete solve due to a convergence failure or user request. However, when the solve completes successfully, you have the option to request to either keep the restart points by setting this field to **Yes**, or to delete them by setting this field to **No**.

Note:

You can control these settings in the Details view of the **Analysis Settings** object under Restart Controls (p. 1438), or in the **Options** dialog under the **Analysis Settings and Solution** preferences list. The setting in the Details view overrides the preference setting.

The **RSM** category includes the preference, **RSM Polling Delay (sec)**. This preference controls how often the processing from a remote solution is polled or queried for result data. The default setting is 5 seconds. The supported range is 1 to 6000 seconds. This feature does not prolong solution processing. Solution completion ends the polling process.

The **Solution Information** category enables you to change the default value of the following Details pane properties:

- **Refresh Time**: specifies how often any of the result tracking items under a Solution Information (p. 1833) object get updated while a solution is in progress. The default is **2.5 s**.
- **Activate FE Connection Visibility**: specifies the value of the **Activate Visibility** property. The default setting is **Yes**.

The **Analysis Data Management** category enables you to specify default settings for the following:

- **Scratch Solver Files Directory**: Use this option to specify a unique disk drive that the application will use to process the solution. Using this entry field, you must specify an existing disk location. If the entry is invalid, the application uses the default disk.
- **Save MAPDL Database**: Use this option to set the default value for the **Save MAPDL Database** control. Selections include **No** (default) or **Yes**. The setting of the **Future Analysis** control (see **Analysis Data Management** (p. 1484) Help section) can sometimes require the database file to be written. In this case, the **Save MAPDL Database** control is automatically set to **Yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Solver File Verbosity**: For loading conditions imported from External Model, this preference enables you to control how much information is written to the input file (`ds.dat`) and output file (`solve.out`) during the solution process. Options include:

  – **Minimum** (default): The application does not write comments to the input and solver file. This option provides faster processing time from the Mechanical APDL solver.

  – **Medium**: The application writes comments to the input file, but no comments are written (echoed) to the solver file.

  – **Verbose**: The application writes comments to the input and solver file.

The **Analysis Data Management (Modal)** category enables you to set the default value for the **Future Analysis** property. The options include **None** (default), **MSUP Analyses**, and **Topology Optimization**. If this property is set to **MSUP Analyses** or **Topology Optimization**, the application creates the files needed for future MSUP analyses or Topology Optimization. If this property is set to **None**, the files are not created in order to improve solution time and reduce file size.

The **Analysis Data Management (Static Structural)** category enables you to set the default value for the **Future Analysis** property. The options include **None** (default) or **Topology Optimization**. If this property is set to **Topology Optimization**, the application creates the files needed for Topology Optimization. If this property is set to **None**, the files are not created in order to improve solution time and reduce file size.

The **Analysis Data Management (Topology Optimization)** category enables you to set the default value for the **Max Num of Intermediate Files** property. It specifies the number of intermediate topology files you wish to retain for all iterations solved. A value of 1 indicates that the generated file is overwritten each iteration. The default value for the property is set to the text string "All Iterations" that equals a setting of zero (0).

The **Post Processing** category contains the **Distributed Post Processing** property. Options for the property include **Program Controlled** (default), **Yes**, and **No**.

The **Advanced (Static Structure)** category contains the **Inverse Solving** property. Options for the property include **No** (default) and **Yes**. This property species the default setting for the **Inverse Option** property of the **Advanced** (p. 1461) category in the **Analysis Settings** object.

The **Advanced** category contains the following properties:

- **Contact Split (DMP)**: Options for the property include **Program Controlled** (default), **On**, and **Off**. This property species the default setting for the **Contact Split (DMP)** property of the **Advanced** (p. 1461) category of the **Analysis Settings** object.

- **Contact Diagnostics for Modal and Harmonic**: Options for the property include **No** (default) and **Yes**. When you set this property to **Yes** and solve a stand-alone Modal or Harmonic Response analysis, Mechanical creates the CND file, `file.cnd`, that includes all the contact diagnostics information.

The **Solution History** category pertains to the content presented in the Worksheet when the Solution Output (p. 1833) property is set to **Solution History**. This category contains the following properties:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Maximum Solutions to Store**. This property specifies the default setting for the number of the solutions to be tracked. The default setting for this property is **10**. The minimum value is **1** and the maximum value is **50**.
- **Track Results**. This property specifies whether or not to collect and present result data. Options include **Yes** (default) and **No**.

Results

The **Default** category of the Results option provides the following controls:

- **Calculate Time History**: Sets the default value for calculation of time history. The default is **Yes**.
- **Auto Rename Results**: Automatically renames a result when the result **Type** is changed. The choices are **Yes** and **No**. The default is **Yes**.
- **Average Across Bodies**: Change the default setting of the property **Average Across Bodies**. The options are **Yes** and **No**. The default setting is **No**.
- **Prompt Before Deleting Results on Solve**: This control enables you to activate a confirmation prompt for the **Solve** option on the Solution folder's right-click context menu as well as the child object of the Solution folder (for example, result objects). The prompt only displays for analyses with existing solution data. The available options include:
  - **Failed and Restart Solution** (default): You are prompted when you attempt to re-solve a failed solution or for a solution with restart points.
  - **Never**: the confirmation prompt feature is turned off.
  - **Always**: you are always prompted when you select the right-click **Solve** option.

  Note:

  The **F5** hotkey does not support this function.

- **Update Dependent Expressions on Identifier Change**: Options include **Yes** and **No** (default). When you set this control to **Yes**, the application will attempt to automatically update downstream **Expression** property entries that include an Identifier (p. 1978) that was modified in an upstream system.

The **Cyclic Result Option** category of the Results option provides the **Allow Phase Sweep** control. Options include **No** (default) and **Yes**.

The **RSM File Manager** category of the Results option provides the **RSM Output Files Download** control. Options include **Show** and **Hide** (default).

The **Cyclic Solution Display** category of the Results option provides the following controls.

- **Number of Sectors**: For an analysis that includes a **Cyclic Region** object, this controls specifies the default setting of the **Number of Sectors** property of the **Solution** object. The default setting is 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Starting at Section**: For an analysis that includes a **Cyclic Region** object, this controls specifies the default setting of the **Starting at Section** property of the **Solution** object. The default setting is 1.

Visibility

The **Visibility** selection and category provides the **Part Mesh Statistics** setting. This setting enables you to display or hide the **Statistics** category in the Details view for Body and Part objects.

The **Tree Filtering** category provides the following controls:

- **Graphics**: **Never** (default) or **On Hide/Show Bodies**. Setting this option to **On Hide/Show Bodies** when the **Graphics** filter is active and set to **Visible Bodies**, causes the tree to automatically filter using that option whenever a body is hidden or shown so that only visible bodies and objects associated with any visible body display.
- **Expand**: **Yes** or **No** (default). Change the default setting of the **Expand on Refresh** button on the Filtering (p. 114) feature.

Wizard

The **Wizard Options** category includes the following exclusive controls for this category. There are no counterpart settings in the **Details** view.

- **Default Wizard**: This is the URL to the XML wizard definition to use by default when a specific wizard isn't manually chosen or automatically specified by a simulation template. The default is `StressWizard.xml`.
- **Flash Callouts**: Specifies if callouts will flash when they appear during wizard operation. The default is `Yes`.

The **Skin** category includes the following exclusive controls for this category. There are no counterpart settings in the **Details** view.

- **Cascading Style Sheet**: This is the URL to the skin (CSS file) used to control the appearance of the Mechanical Wizard. The default is `Skins/System.css`.

The **Customization Options** category includes the following exclusive controls for this category. There are no counterpart settings in the **Details** view.

- **Mechanical Wizard URL**: For advanced customization. See *Appendix: Workbench Mechanical Wizard Advanced Programming Topics* for details.
- **Enable WDK Tools**: *Advanced*. Enables the Wizard Development Kit. The WDK adds several groups of tools to the Mechanical Wizard. The WDK is intended only for persons interested in creating or modifying wizard definitions. The default is **No**. See the *Appendix: Workbench Mechanical Wizard Advanced Programming Topics* for details.

Note:

- URLs in the Mechanical Wizard follow the same rules as URLs in web pages.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- *Relative URLs* are relative to the location of the Mechanical Wizard URL.
- *Absolute URLs* may access a local file, a UNC path, or use HTTP or FTP.

Commands

The **Command Editor Options** category includes the following controls that enable you to change the presentation and operation of the **Commands (APDL) Object Worksheet**:

- **Font Size**: Specify the desired font size of the text in the window. The default setting is **11**.
- **Show Invisibles**: Show or hide formatting marks (spaces, paragraph symbol, etc.). The default setting is **No**.
- **Show Line Numbers**: Show or hide line numbers. The default setting is **Yes**.
- **Syntax Highlighting** (Mechanical APDL or Rigid Dynamics solvers only): Turn syntax highlighting on/off. The default setting is **On**.
- **Interactive Tooltips** (Mechanical APDL solver only): Turn the tooltip feature on/off. The default setting is **On**.

UI Options

The **UI Options** group includes the following categories that enable you to change certain interface display default settings.

Main Window Title Theme

For the Windows platform only, this category includes the following preferences:

- **Merge Ribbon with Title Bar**. Options include **Yes** (default) and **No**.

  This setting merges the title bar of the Mechanical interface with the ribbon in order to save space. When active, there is a potential display issue. If you are working in full screen mode (only) and you are running a lengthy process, it is possible that the application becomes unresponsive ("not responding"). In this situation, the Mechanical interface enlarges and covers your entire screen – including the Task Bar. This is undesirable. To eliminate this display limitation, given the required situation, disable the preference.

- **Shown Application Name in Title**: Options include **Yes** (default) and **No**. Turn the display of the application name shown in the title bar on and off.
- **Show License in Title**: Options include **Yes** (default) and **No**. Turn the display of the license currently in use in the title bar on and off.

Style

For the Windows platform only, this category includes the following preferences:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Color Theme**: When you first install and launch the application, you are prompted to select a desired theme. Options include **Light**, **Dark**, and **Classic**. At any time after this initial selection, you can change the theme using this preference.
- **Use Color Theme in Geometry Window**: Options include **Yes** (default) and **No**. Use this preference to select the background color of the **Geometry** window. If set to **Yes**, the application uses the background color of the selected **Color Theme**. Set to **No**, Mechanical uses the background colors selected in Workbench (**Tools** > **Options** > Appearance).
- **Font Size**: You use this preference to change the size of the font used in the Outline pane, Details pane, and the Ribbon. Options include **System Default**, **Normal**, **Large**, and **Extra Large**. The default setting is **Normal**.
- **Auto Resize Icons**: You use this preference to automatically change the size of ribbon icons when you modify the above **Font Size** preference. The default setting for this preference is **Yes**.

Tooltip

This category includes default settings for the following:

- **Show Tooltips**: Show/hide all tooltips for all available options. Options include **Yes** (default) and **No**. Setting this option to **No** hides all tooltips and the options below become ineffective.
- **Show Mini Toolbar Tooltips**: Show/hide tooltips for the Mini Selection Toolbar (p. 264). Options include **Yes** (default) and **No**.
- **Show Menu Tooltips**: Show/hide tooltips for options on right-click context menu. Options include **Yes** (default) and **No**.
- **Show Details View Tooltips**: Show/hide tooltips for the properties contained in the **Details** pane. Options include **Yes** (default) and **No**.

Window Manager

This category contains the **Pane Opacity** setting. This setting enables you to modify the transparency of the interface panes. The default setting is 100.

Engineering Data

This category contains the **Mechanical View** setting. Options include **None** and **Windows** (default). Selecting **None** returns the material assignment display and menu options to the previous layout and behavior.

Context Tab

This category includes the following settings:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Common Groups Visibility**. This setting enables you to hide the tab groups **Outline**, **Solve**, and **Insert** from Context tabs. By default, these groups are displayed on the various Context tabs. The options for this setting are **Show** (default) and **Hide**.

  Note:

  – This setting does not apply to the **Project**, **Named Selection**, **Remote Point**, and **Convergence** Context tabs as they only display these three groups.

  – If you select multiple objects in the **Outline**, these groups automatically display regardless of this setting. When the **Hide** setting is specified, the groups again become hidden once a single object is selected.

- **Edit and Combo Box Visibility**. Options include **Show** (default) and **Hide**. If you select the **Hide** option, the application hides all the editable and combo boxes available in context tabs and places them in a newly displayed ribbon tab titled "**Miscellaneous**". For this change to take place, you need to restart the Mechanical application.

- **Refresh Wait for Controls Deletion** (Windows Only). When you switch between objects in the **Outline** that have an associated context tab (Model, Mesh, Solution, etc.), the application uses a specific sequence for displaying the options and controls on the ribbon for each associated context tab. This preference optimizes this presentation sequence. Options include **Yes** (default) and **No**.

Mini Selection Toolbar

This category includes the **Mini Selection Toolbar Visibility** setting. This setting enables you to hide or display (default) the Mini Selection Toolbar (p. 264).

Progress

This category includes the setting: **Hide Progress Window**. This setting enables you to hide (default) or display the progress windows of certain application processes, such as attaching geometry, mesh generation, and the solution process. The progress for these processes is also displayed in the status bar (p. 135).

Scripting

This category includes the **New Scripting UI** setting. Based on a scripting interface update, the **Mechanical Scripting** interface is now the default. If desired, you can revert to the **ACT Console** interface using this setting. Mechanical must be restarted to implement the change.

Default Quick Launch Result Categories

The settings of this category, listed below, enable you to display or hide the Quick Launch pane groupings. These settings affect only the default search results. Results using the shortcut options "(@" symbol) are not affected. Setting options include **Show** and **Hide**. All settings are set to **Show** by default, except for the **Tree Items** setting that is set to **Hide**.

- **Ribbon Items**
- **Context Items**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Preference Items**
- **Pane Toolbar Items**
- **Tree Items**

Delay Loading

This category includes the **Tree** setting. Options include **Enable** and **Disable** (default). This preference tells the application to import, but not display, all upstream data associated with the geometry, contact conditions, and Named Selections. You simply expand (plus symbol) the corresponding object to display the geometry, etc. This can be a useful feature for models that include many parts, contact conditions, or Named Selections.

Pause View Update

This category includes the **Details** setting. This setting enables you to halt **Details** pane updates. The options of this preference are **Yes** and **No** (default).

Toolbar Customization

This category includes the **Show All Categories** setting. This setting enables you to display all of the options available in the application. All toolbars from all panes, all commands from all Ribbon tabs, User Buttons, External ACT Extensions, etc. This gives you maximum flexibility for toolbar customization. The options of this preference are **Yes** and **No** (default).

UI Controls

This category includes the following preferences:

- **Ribbon/Toolbar Refresh**. Options include **Automatic** (default) and **Manual**. Using the **Automatic** setting, the application refreshes the ribbon and toolbar controls every 100ms. If set to **Manual**, the application does not refresh automatically but rather refreshes only as a result of certain actions taken in the application. Refreshing the options of the application helps to ensure their proper display and operation.
- **Details View Combo Boxes**. This property enables you to change the default setting for the ability to search drop-down lists (p. 123) in the Details view. Options include **Searchable** (default) and **Non-Searchable**.

Setting Variables

Variables enable you to override default settings.

To set a variable:

1. Choose **Variable Manager** from the **Tools** menu.
2. Right-click in the row to add a new variable.
3. Enter a variable name and type in a value.
4. Click **OK**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Variable name	Supported Values	Description
`DSMESH OUTPUT`	filename	Writes mesher messages to a file during solve (default = no file written). If the value is a filename, the file is written to the temporary working folder (usually `c:\temp`). To write the file to a specific location, specify the full path.
`DSMESH DEFEATUREPERCENT`	a number between 1e-6 and 1e-3	Tolerance used in simplifying geometry (default = .0005).
`TreatModalAsComplex`	1	Mode shapes and contour colors are in sync for animated Modal results.
`contactAllowEmpty`	1	Allows the solution to proceed even if no contact elements are generated for a given Contact Region.
`UsePseudoShellDisp`	1	Display expanded shell thickness based on solver updated nodal locations (for shells with large deformations (p. 134)).
`edge contact type`	CONTA175 or CONTA177	Forcibly specifies the element type of edge contact (both for real contact and Remote Boundary Conditions (p. 1735)).

Status

The status box indicates if a particular variable is active or not. Checked indicates that the variable is active. Unchecked indicates that the variable is available but not active. This saves you from typing in the variable and removing it.

Using Macros

The Mechanical application enables you to execute custom functionality that is not included in a standard Mechanical application menu entry via its **Scripting** feature. The functionality is defined in a *macro* - a script that accesses the Mechanical application programming interface (API).

Macros can be written in the Python (.py) programming language. For additional information, refer to the Script Examples section of the Scripting in Mechanical Quick Start Guide. Macros cannot currently be recorded from the Mechanical application.

To access a macro from the Mechanical application:

1. Choose **Scripting** option (p. 94) from the **Support** group on the **Automation** tab.
2. Navigate to the directory containing the macro.
3. Select the macro.

Note:

Note: For the current release. Mechanical also supports macros written in the legacy languages Microsoft's JScript and VBScript. Several macro files are provided with the Ansys Workbench installation under **`\ANSYS Inc\v190\aisol\DesignSpace\DSPages\macros`**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Data Export

Mechanical enables you to export specifically supported analysis data to one or more of the following file types. Review the following topics based upon the desired export file format.

- General Export Procedure (p. 249)
- Exporting to Text and Delimited Files (p. 249)
- Stereolithography (STL) (p. 250)
- Ansys Viewer File (AVZ) (p. 251)
- Geometry (Part Manager Database) (p. 251)
- NASTRAN Bulk Data (p. 252)
- Ansys Sound (p. 253)
- Waveform Audio File (WAV) (p. 253)

See the Options Settings (p. 255) topic at the end of the section for some general export settings that are available using the **Options** dialog.

General Export Procedure

1. Select an object in the tree.
2. Click the **Worksheet** to give it focus (if applicable).
3. Right-click the selected object in the tree or on a selection in the **Tabular Data** window, select **Export**, and then select a file type as required.
4. Specify a file name and save the file. Based on the object type, the application may automatically open Excel, assuming you have the application.

Note:

You must right-mouse click the selected *object in the tree* to use this **Export** feature. On Windows platforms, if you have the Microsoft Office 2002 (or later) installed, you may see an **Export to Excel** option if you right-mouse click in the **Worksheet**. This is not the Mechanical application **Export** feature but rather an option generated by Microsoft Internet Explorer.

Exporting to Text and Delimited Files

You can export a variety of analysis data to a tab-delimited text file. This file format enables you to view the data in a text editor as well as Microsoft Excel. Mechanical supports exporting data from the following object types (without access to worksheet data):

- Contour Results (p. 2276)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Node-Based Named Selections (p. 1085)
- Element-Based Named Selections (p. 1085)
- Imported Loads (p. 1736)

Additionally, data from the following additional objects can be exported but requires worksheet data to be active:

Connections	Convergence	Geometry
Contact Group	Coordinate Systems	Mesh
Contact Initial Information	Fatigue Sensitivities	Solution
Contact Tool	Frequency Response	Thermal Condition

Stereolithography (STL)

The following objects enable you to export object information in **STL** file format, either as Binary (default) or ASCII. File size is the primary difference between the file formats. The binary format generates smaller files, however; it does not include information for the bodies of your model. The ASCII format preserves all body information during export. Using the Options (p. 217) dialog box, under the Export (p. 222) category, you can change the default setting for exporting in STL format.

- Geometry (ASCII format only (p. 223))
- Mesh
- Results and Result Tools (Group)

Important:

Mechanical obtains unit data from imported CAD models and displays the unit in the **Length Unit** property of the Geometry object. This is the unit system used by the STL export feature. When opening your exported STL file in a CAD application, make sure that the application is also using this unit system. For example, in SpaceClaim, set your unit system by selecting **File** > **SpaceClaimOptions** > **File Options** > **STL**, and then specify the appropriate unit system from the **Units** drop-down menu.

Note:

- When a model contains multiple bodies, Mechanical uses a nonstandard file format for the ASCII representation. In this case, the application separates the bodies.
- Files saved in the STL format can be viewed in appropriate STL supported applications, such as SpaceClaim. Currently for the **Mesh** object and for results-based objects (not including contour data), files exported in the ASCII format enable you to render individual parts of your model in SpaceClaim. Files in the binary format do not support this display capability.
- The display of an exported STL file, regardless of viewer type, is based on the scale you specify in the **Result** (p. 63) drop-down menu, on the **Solution** Context tab. That

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

is, if the scale is set to show deformations, the model is exported in the deformed shape.

- When you select **Top/Bottom** as the **Shell** setting in the Details view for a surface body and export the result contours (such as stresses and strains), the export file contains two results for every node on a shell element (p. 1993). The first result is for the bottom face and the second result is for the top face.

Ansys Viewer File (AVZ)

Mechanical enables you to export your mesh and/or a result object as an AVZ (.avz) file. The export operation creates a 3D model representation that you can display in the Ansys Viewer (installation required). The Ansys Viewer is a WebGL based 3D image viewer that you can use to visualize the exported mesh and results of your analysis without opening the Mechanical application. For results, you can inspect result values at specific locations by hovering the mouse over a point on your model. For result objects and as desired, you can automatically launch the Ansys Viewer by changing the default setting under the **Export** preference (p. 222) of the **Options** dialog.

Note:

Exporting results that include a customized legend may present legend/contour display inconsistencies in the viewer.

Geometry (Part Manager Database)

You can export the geometry (entirely or as parts) to a binary Part Manager Database (.pmdb) file by:

- Right-clicking on the **Geometry** object and then selecting **Export**>**Geometry**. The application writes the entire geometry to the `.pmdb` file. This option also writes any Named Selections created in Mechanical into the `.pmdb` file.

  Or...

- Right-clicking one or more bodies/parts, and then select **Export**>**Geometry**. The application writes the selected parts to the `.pmdb` file. If a selected body is part of a multi-body part, then the entire part is written to the file.

  Or...

- Right-clicking one or more bodies in the Geometry window and then selecting **Export**>**Geometry**. The application writes the selected parts to the `.pmdb` file.

Part Manager Database Requirements and Limitations

- Exporting the Geometry as a `.pmdb` file facilitates future geometry import into SpaceClaim, DesignModeler, as well as re-importing the file back into Mechanical.
- When exporting a geometry to a `.pmdb` file, the application exports all bodies, including suppressed bodies, to the file without maintaining their suppression status. Therefore, when you re-import the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

geometry, all of the bodies are unsuppressed. However, the application **does** export whether or not a you have hidden bodies. This means that suppressed bodies, which are typically hidden, appear hidden when you re-import the geometry. As a result, your geometry may have bodies that are hidden, because they were suppressed on export, but that are no longer suppressed.

- The application does not export supplemental model data created after the geometry was imported. This includes coordinate systems, work points, spot welds, or materials that you manually added during your Mechanical session.
- When defined, `.pmdb` files include geometry cross sections for line bodies in the exported file. Subsequent SpaceClaim and Mechanical sessions import the line body cross section data accordingly. However, DesignModeler does not support importing line body cross section data and as needed, requires you to redefine the cross sections if imported into DesignModeler.
- For any of the following that are exported as a Part Manager Database (.pmdb) file and then imported into a new Mechanical session, the application does not currently support re-meshing using normal mesh methods:
  - Geometries created from External Model
  - Deformation results
  - Parts created using the Mesh **Pull** feature

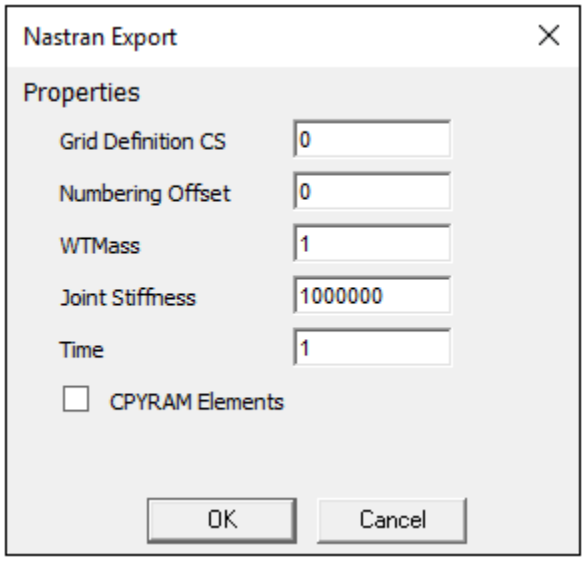
NASTRAN Bulk Data

For Static Structural and Modal analyses, you can export your simulation as a NASTRAN Bulk Data (.bdf, .dat, .nas) file.

When you select the Environment object, the option, **Export NASTRAN File** is available in the **Tools** group of the Environment Context Tab (p. 61). Based on your analysis type, one of the following dialogs displays. You use these property options to further define how you wish to export your simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

Also see the Writing NASTRAN Files (p. 1867) section for the steps to export your analysis as a NASTRAN (.nas) file.

Ansys Sound

Mechanical enables you to export a result to Ansys Sound software for sound synthesis, sound design, or psychoacoustics analysis. When exporting the result, the data is stored in an XML file, then Ansys Sound SAS is opened (if installed on the machine) and automatically opens this file. The option is supported for the following acoustics results:

- Equivalent Radiated Power Level
- ERP Level Waterfall Diagram
- Frequency Response Sound Power Level
- Far-field SPL Mic (when the **Definition Method** property is set to **Coordinates**)
- Far-field Sound Power Level
- Far-field Mic Waterfall Diagram
- Far-field Sound Power Level Waterfall Diagram

For more information, see the Ansys Sound documentation.

Waveform Audio File (WAV)

You can export the following acoustic-based result types to a Waveform Audio File (.wav) file:

- Equivalent Radiated Power Level

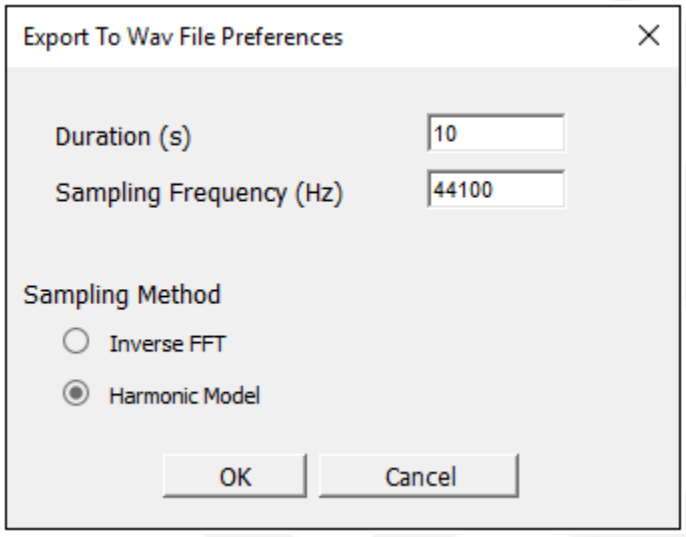
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- ERP Level Waterfall Diagram
- Frequency Response Sound Power Level
- Far-field SPL Mic (when the **Definition Method** property is set to **Coordinates**)
- Far-field Sound Power Level
- Far-field Mic Waterfall Diagram
- Far-field Sound Power Level Waterfall Diagram

Note:

Waterfall result types are only supported by the **Harmonic Model** sampling method described below.

For the above result types, the context (right-click) menu option **Export** > **Export to WAV File** opens the preference window shown below. The default settings are displayed.



The **Duration** and **Sampling Frequency** options are basic settings that enable you to specify the duration and sampling frequency of the audio file.

The **Sampling Method** options include:

- **Inverse FFT** (Fast Fourier Transform): This method generates a sound from a given spectrum using a Fourier transform. This method creates a sound sample based on the given input spectrum (level vs. frequency). This resulting sound sample has the same spectrum as the input. This method is recommended for broadband noise spectrum.
- **Harmonic Model**: This method generates a sound from a spectrum using sound synthesis from sinusoidal patterns. This method creates a sound sample based on the given input spectrum (level vs. frequency) that contains harmonic components at the same exact frequencies specified in the input spectrum. Each frequency has the same level as the specified input. This method is recommended for pure tones sounds.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Options Dialog Settings

The **Export** (p. 222) category of the **Options** dialog includes the following preferences that apply to exporting data.

Automatically Open Excel (**Yes** by default)

Remove Duplicate Nodes (**No** by default)

Include Node Numbers (**Yes** by default)

Include Node Location (**No** by default)

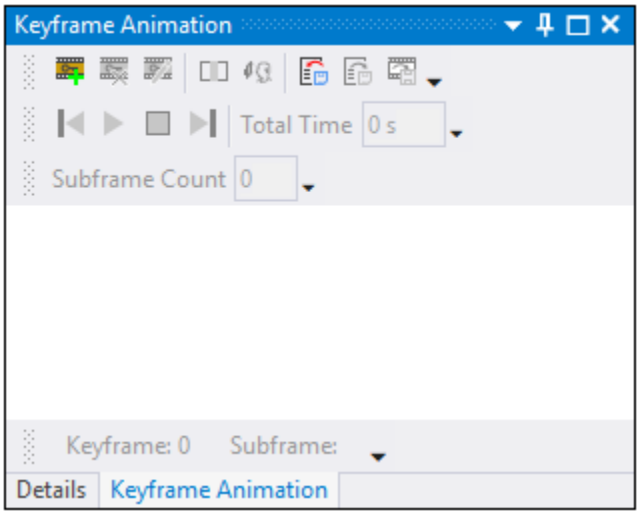
Keyframe Animation

Overview

The Keyframe animation feature enables you to string together snapshots of your model in the **Geometry** window to create an animation. Each Keyframe is a Start and End point that the application then links together by drawing Subframes (by default 30 Subframes) to create the animation. The application interpolates the transition from frame to frame to create a smooth animation. For example, you can create an animation of your model rotating.

Application

You create Keyframes using the **Keyframe Animation Views** window. To display (or close) the window, select the **Keyframe Animation** option from the **Tools** group (p. 45) on the **Home** tab. The window, as shown here, provides the interface for using the feature.



Create a Keyframe

To create Keyframes, you need to first position your model as desired in the **Geometry** window. Then, select the **Create a Keyframe** button in the **Keyframe Animation** window. A new entry displays in the window. The application assigns a numerical value to each Keyframe (**Keyframe 0**, **Keyframe1**, etc.). Each Keyframe is a snapshot of the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Once you define a Keyframe animation in the window, you can double-click the Keyframe to view its position.

Modify a Keyframe

To change an existing Keyframe, select the Keyframe in the window, position your model as desired, and select the **Modify a Keyframe** button.

Delete a Keyframe

Select the Keyframe in the window and click the **Delete** button.

Save/Load/Export Animations

The window provides options to Save your defined Keyframes as an XML file, to load a saved XML files of Keyframes, and an export option that enables you to save your Keyframe animation as a video file (AVI, MP4, WMV, or GIF). Any **Subframe Count** and **Total Time** specifications apply to exported files types except the GIF format.

Insert Keyframe

Insert a new keyframe before the currently selected keyframe.

Apply Keyframe

Set the graphics window's camera settings to the currently selected keyframe's camera settings.

Note:

- When exporting your animation as a video file, make sure to keep the Mechanical application window is in front of any other desktop windows until the export process is complete. Opening other windows in front of the application window before the export process is complete could include those windows in the video capture.
- The GIF file format does not support the Workbench Gradient settings (the default setting). For this file format, the application automatically changes the Workbench appearance setting to Uniform. As a result, exported GIF files have a plain background compared to exported videos.

Frame Animation Control Options and Displays

The window provides the following options/displays:

- **Play**: Start or Resume the animation.
- **Pause**: Pause current animation
- **Stop**: Stop the animation
- **Previous**/**Next Frame**: These options move the animation backwards or forwards one frame at a time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Subframe Count**: Specify the desired number of subframes (0 to 200) between each Keyframe. Subframes define the number of interpolations performed between each frame. This affects the smoothness or lack thereof of the animation. This setting applies to exported animations.
- **Total Time**: Specify a desired amount of time for your animation. This property defines presentation speed. This setting applies to exported animations.
- **Keyframe**: Read-only field that displays the Keyframe being displayed.
- **Subframe**: Read-only field that displays the subframes (per Subframe Count property) as the animation progresses. You can use the Previous/Next Frame options to view specific frames/subframes. Otherwise these fields automatically cycle through the animation.

Graphical Selection and Display

Here are some tips for working with graphics:

- You can use the ruler, shown at the bottom of the **Geometry** (p. 129) window, to obtain a good estimate of the scale of the displayed geometry or results (similar to using a scale on a geographic map). The ruler is useful when setting mesh sizes.
- You can activate the **Consider Section Planes When Mesh Picking** Graphics (p. 228) preference in order to select newly displayed finite element entities (nodes, elements, and element faces) from geometries within the model that were hidden before adding the specified Section Plane (p. 294). See the Selecting Mesh Entities Displayed by the Section Plane (p. 301) topic for additional information.
- Hold the control key to add or remove items from a selection. You can *paint select* faces on a model by dragging the left mouse button.
- Use the stack of rectangles in the lower left corner of the **Geometry** (p. 129) window to select faces hidden by your current selection.
- To multi-select one or more faces, hold the **Ctrl** key and click the faces you wish to select, or use **Box Select** to select all faces within a box. The **Ctrl** key can be used in combination with **Box Select** to select faces within multiple boxes.
- Use the options of the Selection Tab (p. 88) to make or manipulate geometry selections.
- Click the Using Viewports (p. 284) icon to view up to four images in the **Geometry** (p. 129) window.
- Controls are different for Graphs & Charts (p. 291).
- Mechanical supports 3Dconnexion devices. See the Platform Support section of Ansys.com for a complete list of 3Dconnexion products certified with the current release of Ansys applications.

More information is available in the following topics:

Selecting Geometry

Selecting Nodes

Selecting Elements and Element Faces

Selecting Nodes and Elements by ID

Manipulating the Model in the Geometry Window

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Defining Direction
Using Viewports
Accelerated Graphics
Controlling Graphs and Charts
Managing Graphical View Settings
Creating Section Planes
Viewing Annotations
Controlling Lighting
Inserting Comments, Images, and Figures

Selecting Geometry

This section discusses cursor modes and how to select and pick geometry in the **Geometry** window. It includes information on the following:

Pointer Modes (p. 258)
Highlighting (p. 259)
Picking (p. 259)
Blips (p. 259)
Painting (p. 260)
Depth Picking (p. 261)
Selection Filters (p. 261)
Selection Modes (p. 262)
Mini Selection Toolbar (p. 264)
Extend To (p. 264)

For Help on how to select mesh nodes and elements, see the Selecting Nodes (p. 269) and Selecting Elements (p. 276) sections. Many of the same selection and picking tools are employed for mesh selections.

Pointer Modes

The pointer in the Geometry window is always either in a picking filter mode or a view control mode. When in a view control mode the selection set is locked. To resume the selection, repress a picking filter button.

The Graphics Toolbar (p. 97) offers several geometry filters and view controls as the default state, for example, face, edge, rotate, and zoom.

If a **Geometry** field in the Details Pane (p. 119) has focus, inappropriate picking filters are automatically disabled. For example, a pressure load can only be scoped to faces.

If the **Direction** field in the Details Pane (p. 119) has focus, the only enabled picking filter is **Select Direction**. **Select Direction** mode is enabled for use when the **Direction** field has focus; you never choose **Select Direction** manually. You may manipulate the view while selecting a direction. In this case the **Select Direction** button enables you to resume your selection.

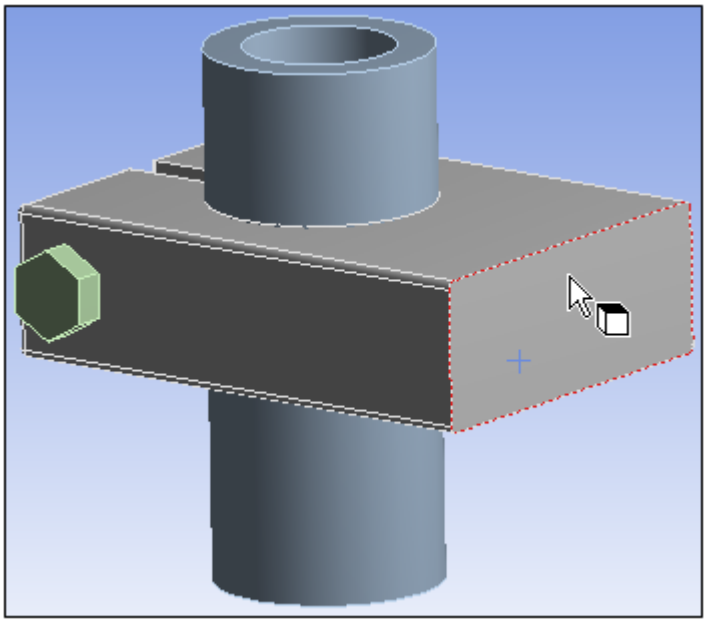
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Highlighting

Hovering your cursor over a geometry entity *highlights* the selection and provides visual feedback about the current pointer behavior, for example select faces, and location of the pointer, for example, over a particular face.

As illustrated here, the face edges are highlighted in colored dots.



Picking

A *pick* means you select or "click" on a visible geometric entity. A pick becomes the current selection, replacing previous selections. A pick in empty space clears the current selection.

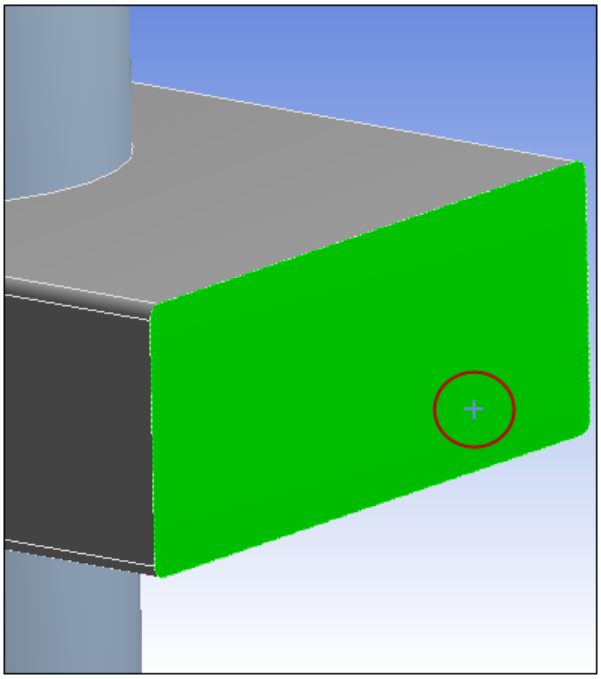
By holding the **Ctrl** key down, you can add additional selections or remove existing selections. Clicking in empty space with **Ctrl** depressed does not clear current selections.

For information on picking nodes, see Selecting Nodes (p. 269).

Blips

As illustrated below, when you make a selection on a model, a cross-hair “blip” appears.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The blip serves to:

- Mark a picked point on visible geometry.
- Represent a ray normal to the screen passing through all hidden geometry.

When you make multiple selections using the **Ctrl** key, the blip is placed at the last selection entity. Clicking in empty space clears your current selection, but the blip remains in its last location. Once you have cleared a selection, hold the **Ctrl** key down and click in clear space again to remove the blip.

Note:

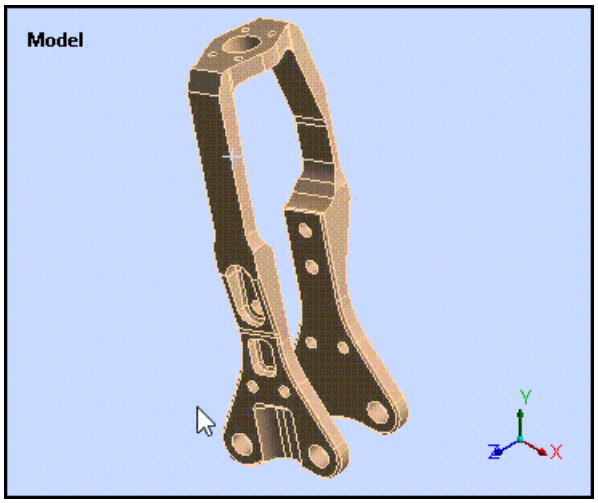
This is important for depth picking, a feature discussed below.

Painting

Painting means dragging the mouse on visible geometry to select more than one entity. A pick is a trivial case of painting. Without holding the **Ctrl** key down, painting picks all appropriate geometry touched by the pointer.

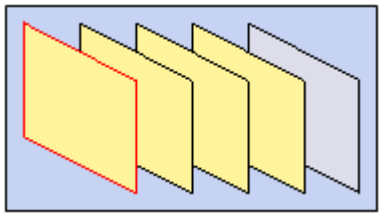
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Depth Picking

Depth Picking enables you to pick geometry through the Z-order behind the blip.



Whenever a blip appears above a selection, the Geometry window displays a stack of rectangles in the lower left corner. The rectangles are stacked in appearance, with the topmost rectangle representing the visible (selected) geometry and subsequent rectangles representing geometry hit by a ray normal to the screen passing through the blip, front to back. The stack of rectangles is an alternative graphical display for the selectable geometry. Each rectangle is drawn using the same edge and face colors as its associated geometry.

Highlighting and picking behaviors are identical and synchronized for geometry and its associated rectangle. Moving the pointer over a rectangle highlights both the rectangle its geometry, and vice versa. **Ctrl** key and painting behaviors are also identical for the stack. Holding the **Ctrl** key while clicking rectangles picks or unpicks associated geometry. Dragging the mouse (Painting (p. 260)) along the rectangles picks geometry front-to-back or back-to-front.

Selection Filters

When you are using your mouse pointer in the **Geometry** window, you are often selecting or viewing geometry entities or mesh selections. The Graphics Toolbar (p. 97) provides the geometry and mesh selection filters listed below. When you activate a filter, the specific entities (vertex, edge, face, body, node, or element) that you can select highlight as you pass your cursor over the entity. This helps you to make desired selections. You can use the filters with the options of the **Select Mode** drop-down list (that is, Single Select, Box Select, Box Volume Select, etc.).

Depressing the **Ctrl** key enables you to make multiple selections for a specific entity type. Furthermore, you can switch between modes (single, box, lasso, etc. as supported) and continue to add to your selection using the **Ctrl** key. You can release the **Ctrl** key while you change selection modes.

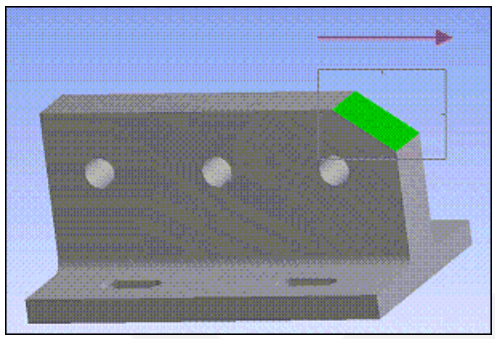
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Smart Select**
- **Vertex**
- **Edge**
- **Face**
- **Body**
- **Node**
- **Element Face**
- **Element**

Selection Modes

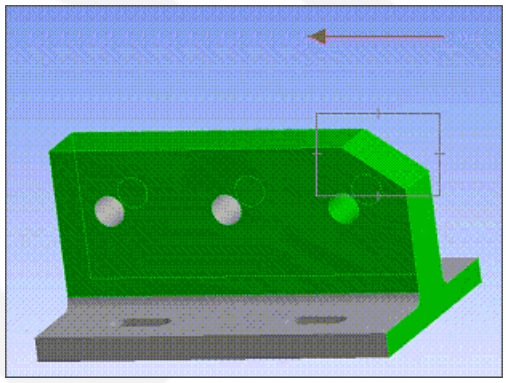
The **Select Mode** option enables you to select items designated by the Selection Filters (p. 261) through the **Single Select** or **Box Select** drop-down menu options.

- **Single Select** (default): Click an item to select it.
- **Box Select**: Define a box that selects filtered items. When defining the box, the *direction* that you drag the mouse from the starting point determines what items are selected, as shown in the following figures:



  – Dragging to the *right* to form the box selects entities that are *completely* enclosed by the box.

  – **Visual cue:** 4 tick marks completely inside the box.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Dragging to the *left* to form the box selects all entities that touch the box.
  - **Visual cue:** 4 tick marks that cross the sides of the box.

- **Box Volume Select**: Available for node-based Named Selections (p. 275) only. Selects all the surface and internal node within the box boundary across the cross-section. The line of selection is normal to the screen.
- **Lasso Select**: Available for node-based Named Selections (p. 275) only. Selects surface nodes that occur within the shape you define.
- **Lasso Volume Select**: Available for node-based Named Selections (p. 275) only. Selects nodes that occur within the shape you define.

Note:

Selection shortcuts:

- You can use the **Ctrl** key for multiple selections in both modes.
- You can change your selection mode from **Single Select** to **Box Select** by holding the right mouse button and then clicking the left mouse button.
- Given a generated mesh and that the **Mesh Select** option is active, holding the right mouse button and then clicking the left mouse button scrolls through the available selection options (single section, box selection, box volume, lasso, lasso volume).

Extend To (p. 264)

Selection Modes (p. 262)

For Help on how to select mesh nodes and elements, see the Selecting Nodes (p. 269) and Selecting Elements (p. 276) sections. Many of the same selection and picking tools are employed for mesh selections.

Pointer Modes

The pointer in the **Geometry** window is always either in a picking filter mode or a view control mode. When in a view control mode the selection set is locked. To resume the selection, repress a picking filter button. The Graphics Toolbar (p. 97) offers several geometry filters and view controls as the default state, for example, face, edge, rotate, and zoom.

If a **Geometry** field in the Details Pane (p. 119) has focus, inappropriate picking filters are automatically disabled. For example, a pressure load can only be scoped to faces.

If the **Direction** field in the Details Pane (p. 119) has focus, the only enabled picking filter is **Select Direction**. **Select Direction** mode is enabled for use when the **Direction** field has focus; you never choose **Select Direction** manually. You may manipulate the view while selecting a direction. In this case the **Select Direction** button enables you to resume your selection.

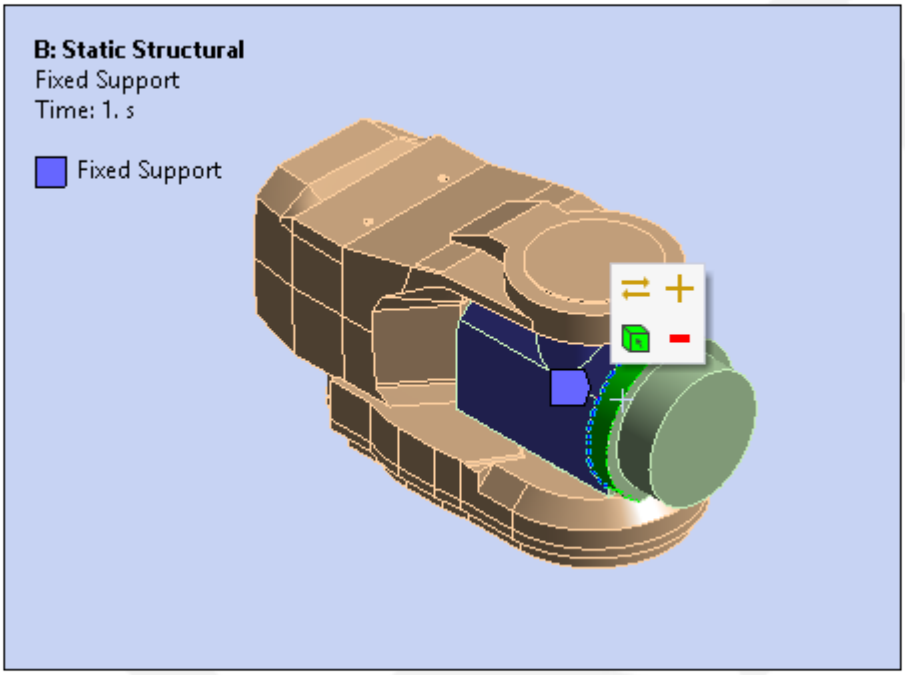
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mini Selection Toolbar

When you are making geometric selections on your model, such as scoping contact conditions, boundary conditions, and/or results, a mini toolbar automatically displays in the **Geometry** window. This toolbar enables you to make selection changes "on the fly." Toolbar options include:

- **Apply Selection**: Replace scoping with the current geometry selection.
- **Add to**: Add the current geometry selection to the existing scoping.
- **Remove from**: Remove the current geometry selection from the existing scoping.

In addition, when you are using the Smart Select option (p. 97) option, an option to select the parent body of your current selection is also available on the toolbar.



Extend To

The **Extend To** group (p. 89), on **Selection** tab as well as the **Extend** drop-down menu on the Graphics Toolbar (p. 97), is enabled only for edge or face selection modes and only with a selection of one or more edges or faces. The following options are available in the drop-down menu:

Note:

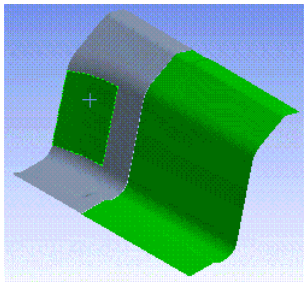
For all options, you can modify the angle used to calculate the selection extensions in the **WorkbenchOptions** dialog setting **Extend Selection Angle Limit** under Graphics Interaction.

Adjacent

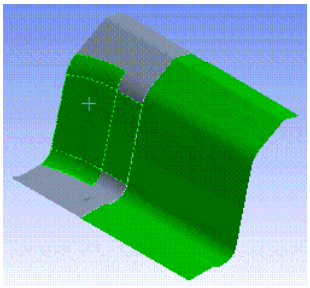
- For *faces*, the **Adjacent** option searches for faces adjacent to faces in the current selection that meet an angular tolerance along their shared edge.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

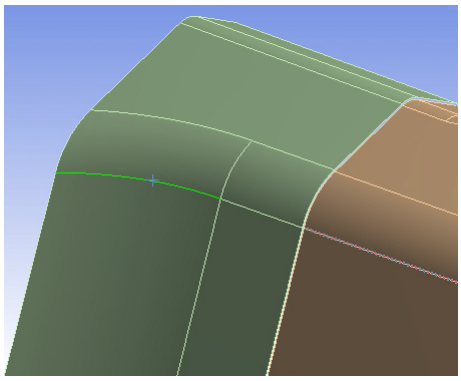


Single face selected in part on the left.

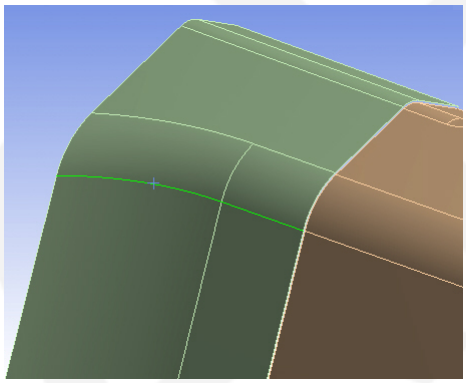


Additional adjacent faces selected after **Extend to Adjacent** option is chosen.

- For *edges*, the **Adjacent** option searches for edges adjacent to edges in the current selection that meet an angular tolerance at their shared vertex.



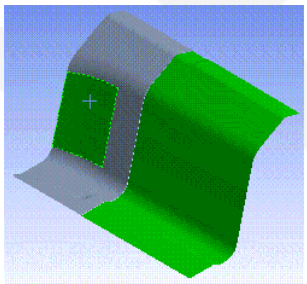
Single edge selected in part on the left.



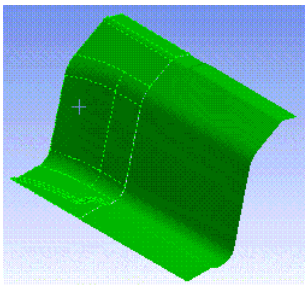
Additional adjacent edges selected after **Extend to Adjacent** option is chosen.

Limits

- For *faces*, the **Limits** option searches for faces that are tangent to the current selection as well as all faces that are tangent to each of the additional selections within the part. The selections must meet an angular tolerance along their shared edges.



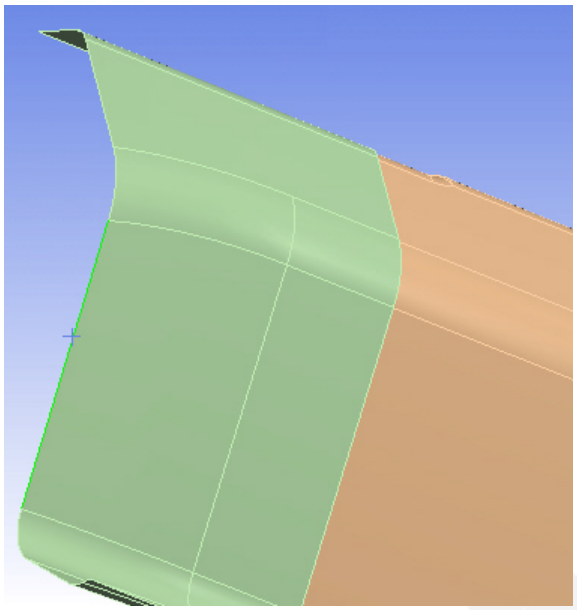
Single face selected in part on the left.



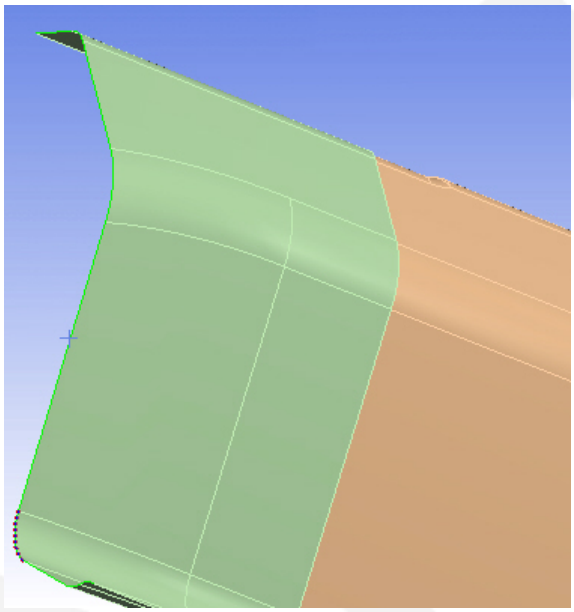
Additional tangent faces selected after **Extend to Limits** option is chosen.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For *edges*, the **Limits** option searches for edges that are tangent to the current selection as well as all edges that are tangent to each of the additional selections within the part. The selections must meet an angular tolerance along their shared vertices.



Single edge selected in part on the left.



Additional tangent edges selected after **Extend to Limits** option is chosen.

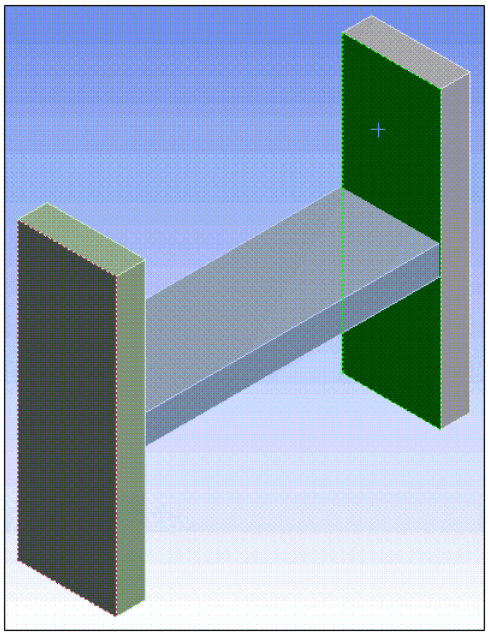
Instances

(Available only if CAD pattern instances are defined in the model): When a CAD feature is repeated in a pattern, it produces a family of related topologies (for example, vertices, edges, faces, bodies) each of which is named an "instance". Using **Instances**, you can use one of the instances to select all others in the model.

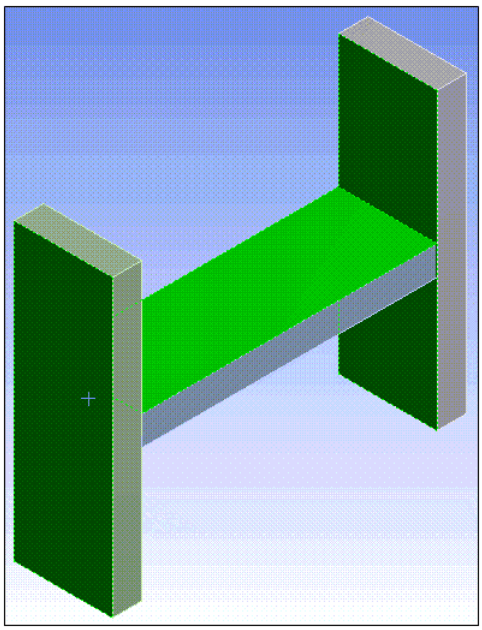
As an example, consider three parts that are instances of the same feature in the CAD system. First select one of the parts.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Then, choose **Instances**. The remaining two part instances are selected.



See CAD Instance Meshing for further information.

Connection

As described in Define Connections (p. 348), connections can be contact regions, joints, and so on.

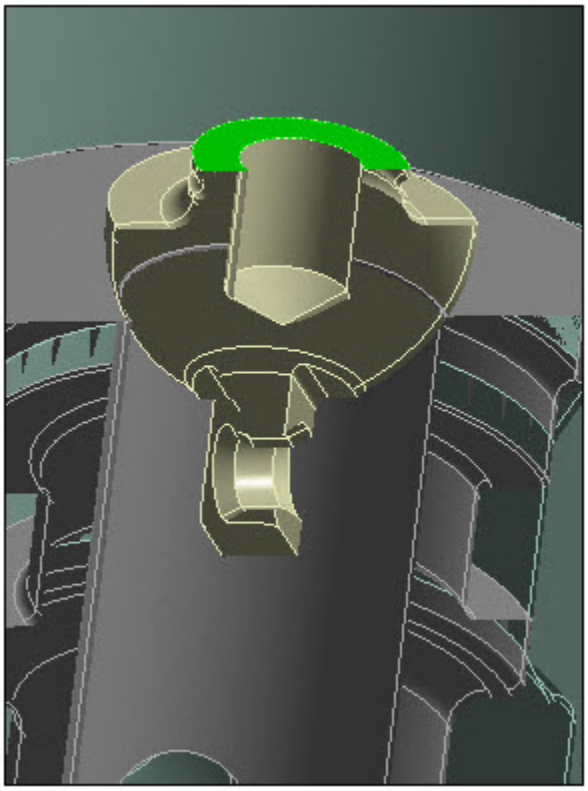
Connection searches for faces that are adjacent to the current selection as well as all faces that are adjacent to each of the additional selections within the part, up to and including all connections on the selected part. This does not include *all* faces that are part of a connection—it includes only those faces that are part of a connection and are also on the selected part.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

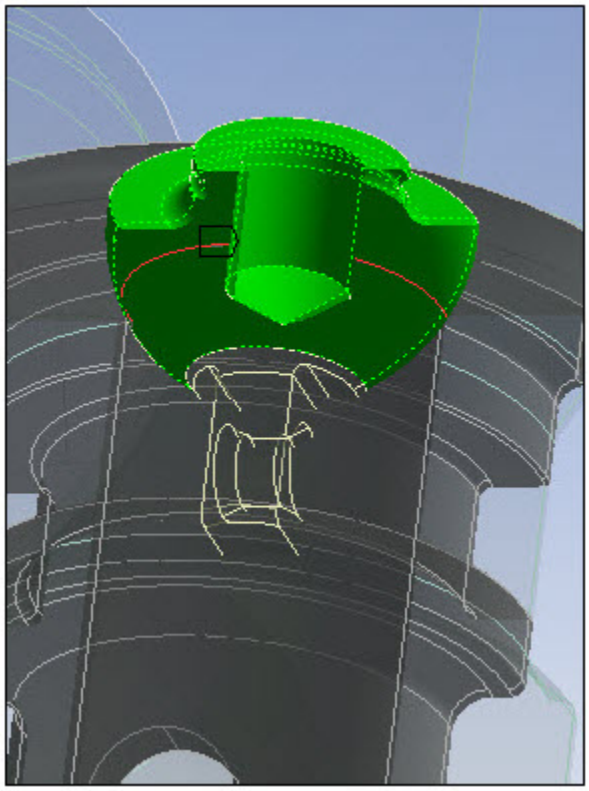
If an edge used by a connection is encountered, the search stops at the edge; a face across the edge is not selected. If there are no connections, all adjacent faces are selected. If the current selection itself is part of a connection, it remains selected but the search stops.

Note:

- The extent of the faces that will be included depends greatly on the current set of connections, as defined by the specified connections criteria (for example, **Connection Type**, **Tolerance Value**, and so on). By modifying the criteria and regenerating the connections, a different set of faces may be included. Refer to Connection Features and Operations (p. 1218) for more information.
- The figures below illustrate simple usage of the **Connection** option.



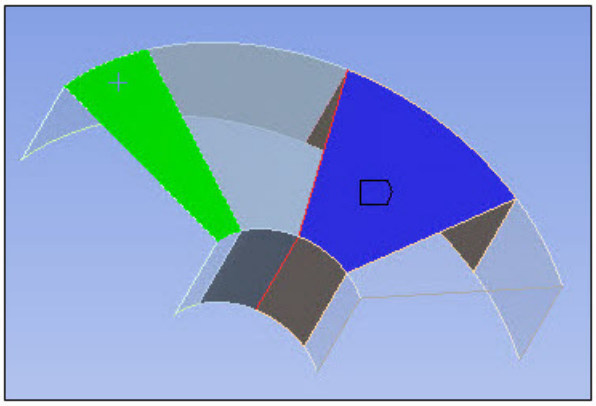
Single face selected in part.



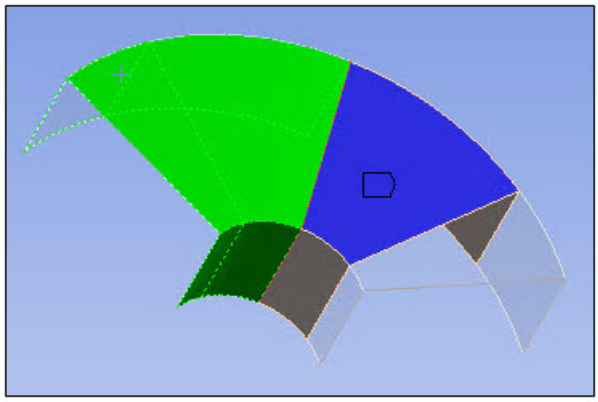
Additional connected faces selected after **Connection** option is chosen.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Single face selected in part. In this example, a multiple edge to single face connection is defined.



Additional connected faces selected after **Connection** option is chosen. When the connection is encountered, search stops at edge.

Selecting Nodes

As with geometry selection, you use many of the same selection tools for mesh nodes. Once you have generated the mesh on your model, you use picking tools to select individual or multiple nodes on the mesh. You use node selections to define objects such as a node-based coordinate system or node-based Named Selections (p. 1057) as well as examining solution information about your node selections. This section describes the steps to create node-based objects in Mechanical.

The following additional topics included in this section describe the uses for the node selection feature:

- Node Selection (p. 269)
- Selection Modes for Node Selection (p. 270)
- View Node Information (p. 272)
- Select Mesh Nodes on a Result Contour (p. 273)

Also see these sections for the steps to create node-based coordinate systems and Named Selections.

- Creating a Coordinate System by Direct Node Selection (p. 274)
- Specifying Named Selections by Direct Node Selection (p. 275)

Node Selection

To select individual nodes:

1. Generate a mesh by highlighting the **Mesh** object and clicking the **Generate Mesh** button.
2. From the Graphics Toolbar (p. 97), select the **Node** filter option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. As needed, choose the appropriate selection tool from the **Select Mode** drop-down list. For more information on the node-based selection modes, see Selection Modes for Node Selection (p. 270).

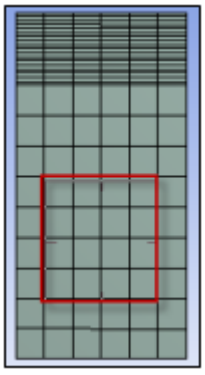
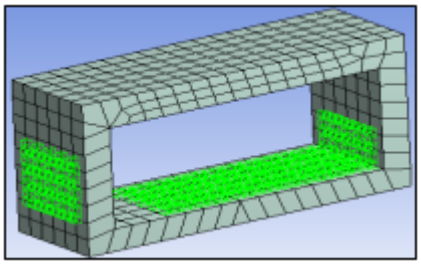
Note:

- When working with **Line Bodies**: Nodes can be selected using volume selection modes only (**Box Volume Select** or **Lasso Volume Select**).
- When working with **Line Bodies** and **Surface Bodies**: Ansys recommends that you turn *off* the **Thick Shells and Beams** option (**Display** tab (p. 78)). This option changes the graphical display of the model's thickness and as a result can affect how your node selections are displayed.

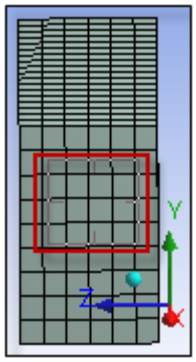
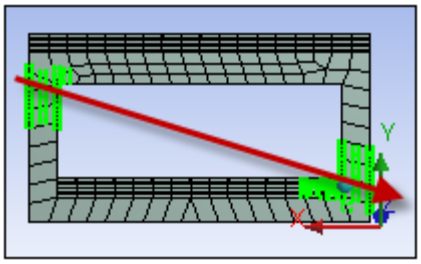
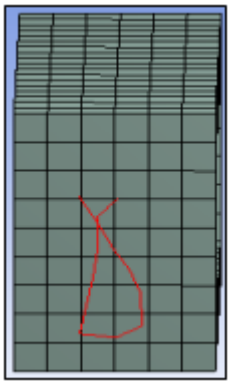
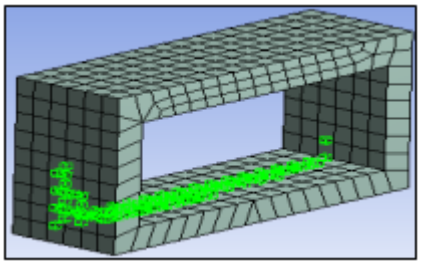
---

4. Select individual nodes or define the shape to select nodes. With your selections active, you can now define a coordinate system (p. 274) or named selection (p. 275) from selected nodes.

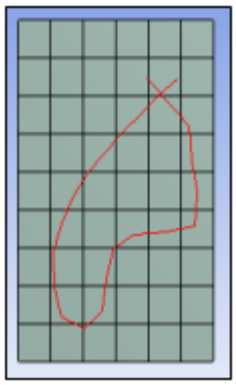
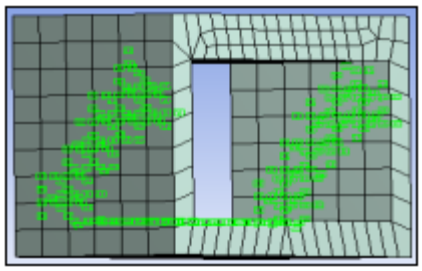
Selection Modes for Node Selection

Single Select	Selects individual nodes or a group of nodes on the surface.
Box Select	Selects all the surface nodes within the box boundary for all the surfaces oriented toward the screen.  

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

 Box Volume Select	Selects all the surface and internal nodes within the box boundary across the cross-section. The line of selection is normal to the screen.  
 Lasso Select	Is similar to the **Box Select** mode. Selects surface nodes that occur within the shape you define for surfaces oriented toward the screen.  

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Lasso Volume Select	Similar to **Box Volume Select** mode. Selects the nodes that occur within the shape you define.  

Tip:

- To select multiple nodes, press the **Ctrl** key or press the left mouse and then drag over the surface. You can also create multiple node groups at different locations using the **Ctrl** key.
- To select all internal and surface nodes, use the **Box Volume Select** or **Lasso Select** tool and cover the entire geometry within the selection tool boundary.
- The **Select All** (**Ctrl**+**A**) option is not available when selecting nodes.

View Node Information

You can view information such as the location of each selected node and a summary of the group of nodes in the **Selection Information** window. A brief description of the selected nodes is also available on the status bar of the application window. To view node id and location information:

1. Select the nodes you wish to examine.
2. Select the **Selection Information** option from the **Tools** (p. 45) group on the **Home** tab.

The following options are available as drop-down menu items in the **Selection Information** window.

Selection Information	Description
Coordinate System	Updates the **X**, **Y**, and **Z** information based on the selected coordinate system.
Show Individual and Summary	Shows both the node Summary and information on each node.
Show Individual	Shows information related to each node.
Show Summary	Shows only a summary of selected nodes.

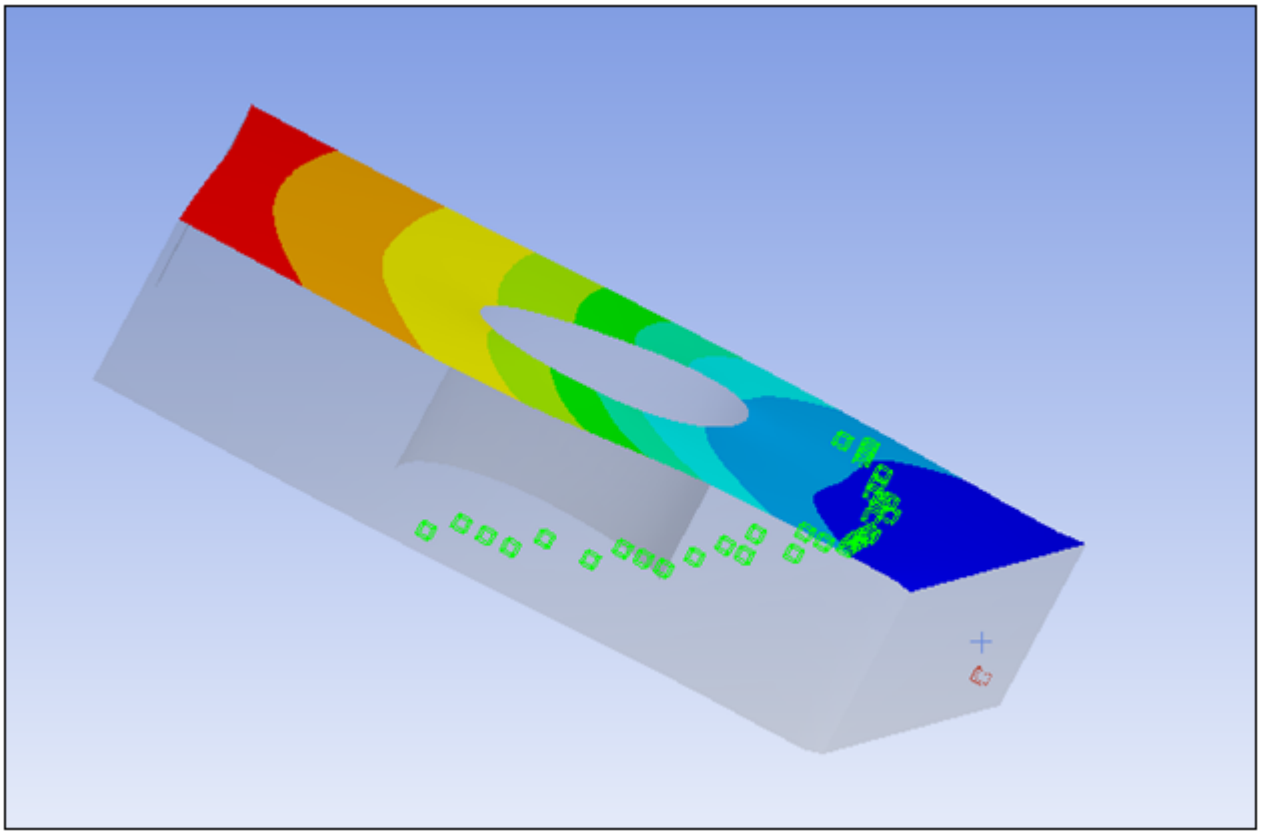
For more information see the **Using the Selection Information Window** (p. 186) section.

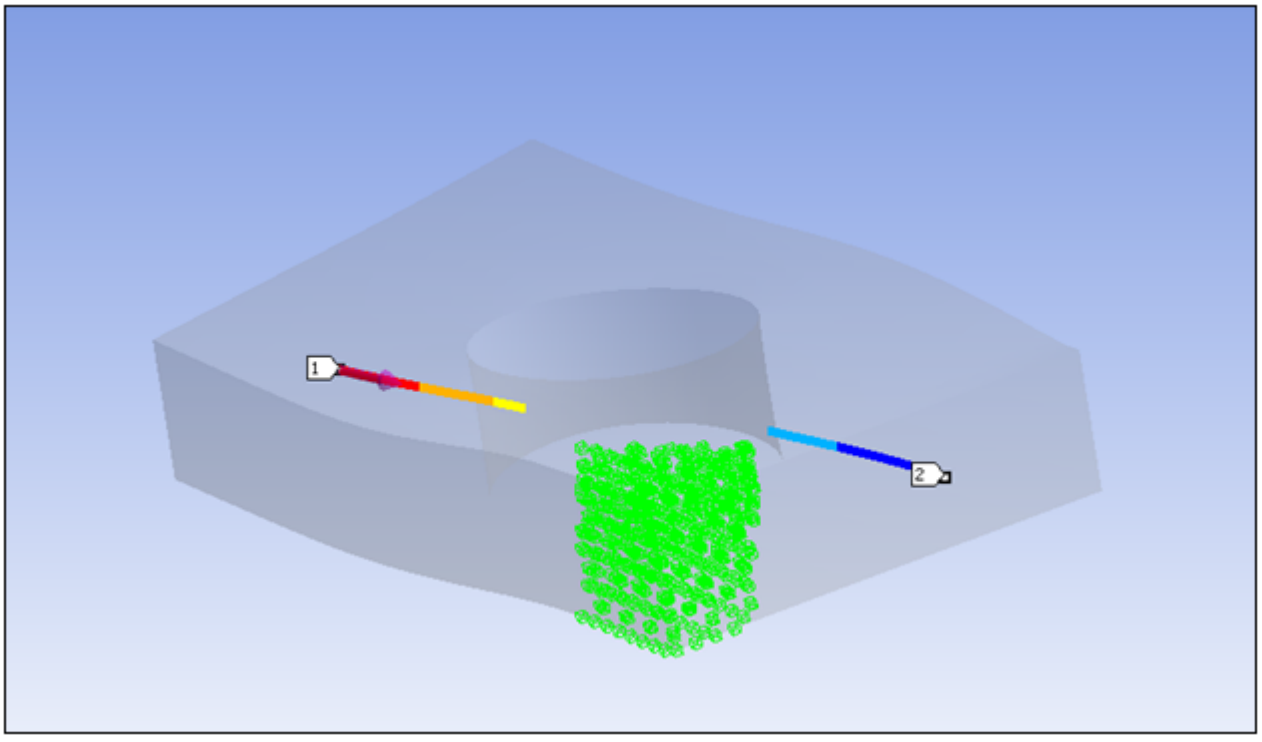
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Select Mesh Nodes on a Result Contour

Nodes (from the original mesh) can be selected even if they don't have values for the selected result, as in a path or surface scoped result.





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The positions of selected nodes reported in the **Selection Information** window are those from non-deformed mesh.

Note:

If the graphics expansion is used (for shells and cyclic expansion, for example), the selection will work on the expanded graphics, while the reported node ID and position will be those in the non-expanded mesh. To eliminate confusion, switch the expansion off.

Creating a Coordinate System by Direct Node Selection

You can select one or more nodes and then create a coordinate system directly in the Geometry window. The new coordinate system is created at the location of the selected node or the centroid of multiple nodes using the (X, Y, Z) locations, rather than the nodes themselves, to ensure that the location does not change upon re-meshing.

To create a coordinate system from nodes in the Geometry window:

1. Using the **Node** selection filter on the Graphics Toolbar (p. 97), select a node.
2. Right-click the selected nodes and select **Create Coordinate System**.

   A new coordinate system is created at the location of the selected node or the centroid of multiple nodes.

If you re-mesh the body at this point, you will see that the coordinate system remains in the same location, as it is based on node location rather than node number.

Creating an Aligned Coordinate System

You can also select an individual node and create an aligned coordinate system on a solved vector principal stress or strain result.

Note:

While you cannot create an aligned coordinate system based on multiple nodes, you can create a local coordinate system at the centroid with an axis oriented in the direction of the global coordinate system.

To create an aligned coordinate system:

1. From the **Tree Outline**, select a **Vector Principal Stress** or **Vector Principal Strain** result.
2. Using the **Node** selection filter on the Graphics Toolbar (p. 97), select one or more nodes.
3. Right-click in the Geometry window and select **Create Aligned Coordinate System**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

A coordinate system is created. The Y-axis of the local coordinate system is oriented in the direction of *S1* (*direction of max. principal stress*).

Note:

Vector Principal Stress and **Vector Principal Strain** results cannot be applied to line bodies or a node located on a line body. As a result, any automatically generated (aligned) coordinate system would be incorrect.

Specifying Named Selections by Direct Node Selection

You create node-based Named Selections in the graphical viewer by scoping selections to single nodes, a group of surface nodes, or a group of nodes across a geometry cross-section.

Note:

You can make direct node selections when working with beams (line bodies) using the Worksheet (p. 1061). Direct graphical selection is also available using the **Node** selection filter on the Graphics Toolbar (p. 97).

To define node-based Named Selections:

1. Using the **Node** selection filter on the Graphics Toolbar (p. 97), select one or more nodes.

   Note:

   For accuracy, ensure that the selected node lies within the scoped area of the result

2. In the Geometry window, right-click the selected node or nodes and select **Create Named Selection**.
3. Enter a name for the Named Selection and click **OK**.

Note:

- If you select a large number of nodes (order of magnitude: 10,000), you are prompted with a warning message regarding selection information time requirements.
- Following a remesh or renumber, all nodes are removed from named selections. If named selections were defined with **Scoping Method** set to **Worksheet** and if the **Generate on Remesh** field was set to **Yes** in the **Details** view of the **Named Selection** folder, then the nodes are updated. Otherwise, node scoping does not occur and the named selection will be empty.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Selecting Elements and Element Faces

Once you have generated the mesh on your model, you can select one or more elements or element faces on your mesh using the **Element** or **Element Face** selection filter on the Graphics Toolbar (p. 97) as well as the options of the **Select Mode** drop-down menu. The following topics describe element-based selection methods and features:

- Selecting Elements or Element Faces (p. 276)
- Specifying Element and Element Face-Based Named Selections (p. 276)
- Viewing Element Information (p. 278)

Selecting Elements or Element Faces

To select elements/element faces:

1. Generate the mesh by highlighting the **Mesh** object and clicking the **Generate Mesh** button.
2. From the Graphics Toolbar (p. 97), select the **Element** or **Element Face** filter option.
3. As needed, choose the desired selection tool from the **Select Mode** drop-down menu on the Graphics Toolbar (p. 97).
4. Select an individual element or multiple elements. To select multiple elements:
   - Hold the **Ctrl** key and click the desired elements/element faces individually. You can also deselect elements/element faces by holding down the **Ctrl** key clicking an already selected element/element face.
   - Hold the left mouse button and drag the cursor across multiple elements/element faces.
   - Use the **Box Select** tool to select all elements/element faces within a box. The **Ctrl** key can also be used in combination with **Box Select** to select multiple boxes of elements/element faces.

Specifying Element and Element Face-Based Named Selections

To create an element or element face-based Named Selection (p. 1057):

1. Select individual or multiple elements/element faces as described above.
2. With your desired element/element face selections highlighted, right-click the mouse and select **Create Named Selection** from the context menu.
3. Enter a name for the Named Selection and click **OK**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

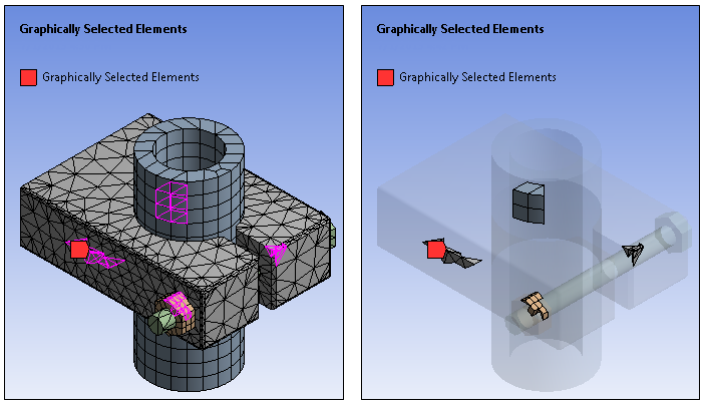
Published: 2023-10-23

Element-based Named Selections are written into the Mechanical APDL input file and this data can be used by the Command (p. 2303) object for further processing.

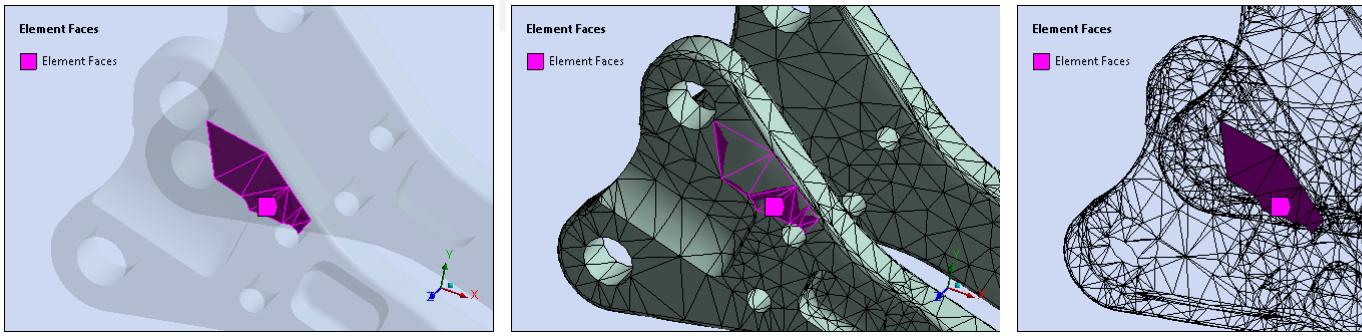
Note:

- An example element-based Named Selection is illustrated below. The example is named **Graphically Selected Elements**. When the **Show Mesh** feature is active, as illustrated in the first image, the elements of a named selection (or multiple named selections) are highlighted. Otherwise, the elements are drawn and the remained of the model is transparent, as illustrated in the second image.

Show Mesh On **Show Mesh Off**



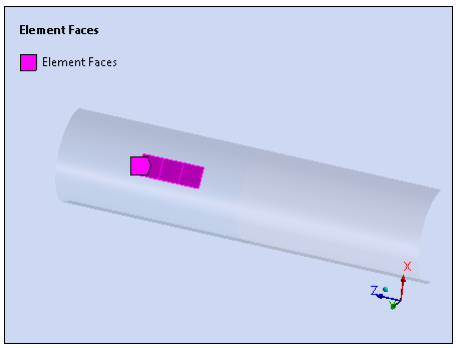
- For Element Face-based Named Selections, each selected face is displayed and each face is "filled" as illustrated in the first image shown below. This is different from Element-based selections that only highlight/display edges. When the **Show Mesh** feature is active, the element faces of a Named Selections may present "bleeding" on the annotation as illustrated in the second image shown below. You can turn on Wireframe mode to accurately display annotations when Show Mesh is selected, as illustrated in the third image.

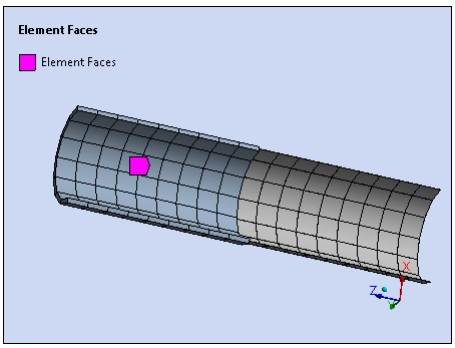


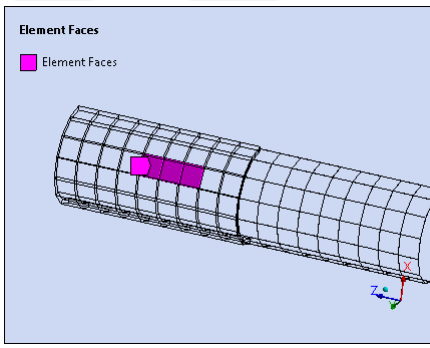
- When working with **Line Bodies** and **Surface Bodies**: Ansys recommends that you turn off the **Thick Shells and Beams** option (**Style** group of the **Display** tab). This option changes the graphical display of the model's thickness and as a result can affect how your element selections are displayed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

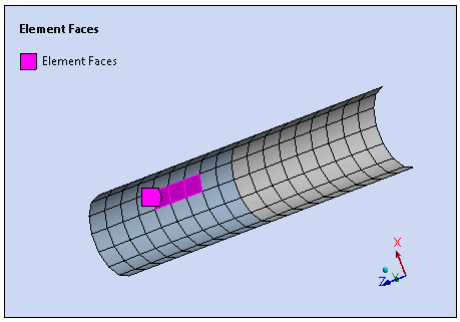
- For surface body Element Face-based Named Selections, the selected element faces can become invisible as a result of being hidden behind an expanded mesh as illustrated below in the first two images below. This issue can again be remedied using Wireframe mode to accurately display annotations when **Show Mesh** is selected, as illustrated in the third image.

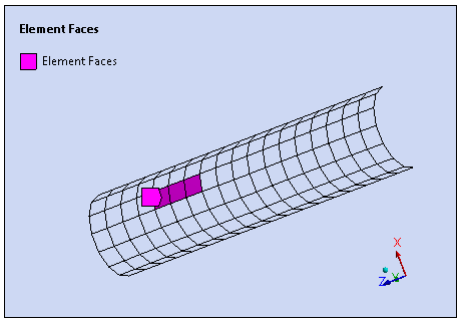






In addition, and as illustrated below in the first image below, not expanding the mesh (turn **Thick Shells and Beams** option off) displays the annotations properly. You can also use Wireframe mode, as illustrated in the second image below.

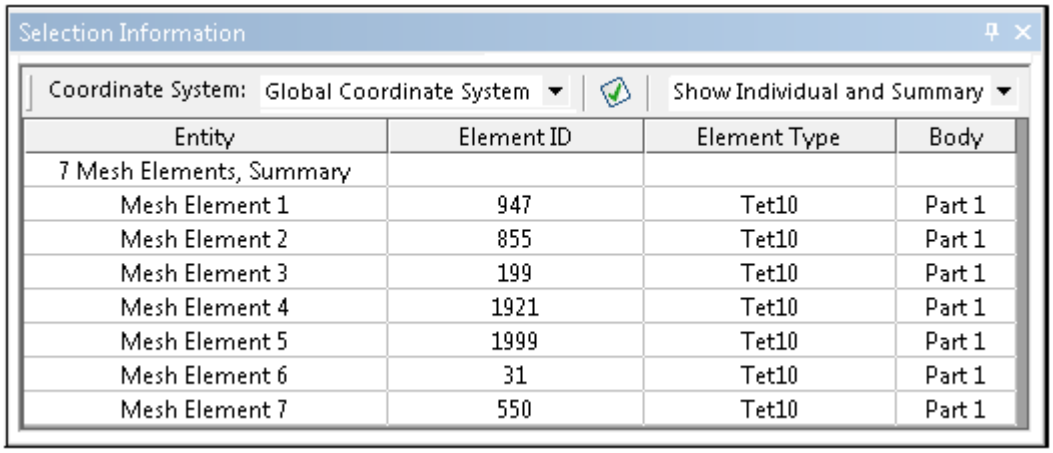




Viewing Element Information

As illustrated below, you can view information about your element/element face selections, such as Element Type, Element ID, as well as the body that the element is associated with using the **Selection Information** window. Once you have selected your desired element or elements, display the **Selection Information** window by selecting the **Selection Information** option from the **Tools** group (p. 45) on the **Home** tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Entity	Element ID	Element Type	Body
7 Mesh Elements, Summary			
Mesh Element 1	947	Tet10	Part 1
Mesh Element 2	855	Tet10	Part 1
Mesh Element 3	199	Tet10	Part 1
Mesh Element 4	1921	Tet10	Part 1
Mesh Element 5	1999	Tet10	Part 1
Mesh Element 6	31	Tet10	Part 1
Mesh Element 7	550	Tet10	Part 1

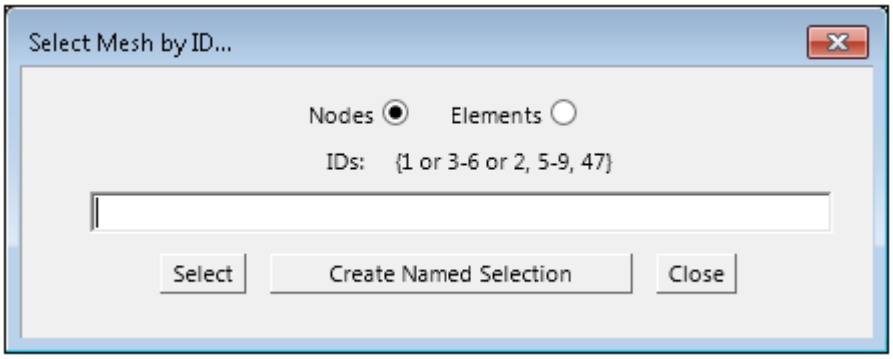
Note:

The status bar at the bottom of the application window also displays the number of elements/element faces you currently have selected.

For additional information, see the **Using the Selection Information Window** (p. 186) section.

Selecting Nodes and Elements by ID

Once you have generated the mesh for your model, the contextual menu (right-click) option, **Select Mesh by ID (M)**, is available from the **Geometry** window. You can also activate the feature using the **M** (p. 314) key, when the **Geometry** window has focus. As illustrated below, this dialog enables you to select mesh nodes and mesh elements using their IDs. This feature is modeless and therefore enables you to work with the user interface while the dialog box is displayed.



From this entry window, you can make comma separated entries of individual nodes (or elements), range entries by using a dash, and/or a combination of the two and then click the **Select** button. Range entries must increase in the appropriate order (for example, 1-10, not 10-1). The dialog displays messages regarding incorrect criteria as needed.

This feature works in tandem with the **Selection Information** window (p. 180), enabling you to view and verify your entries. Open the window to display information about your selections as you make them. In addition, the status bar displays your selections. You can double-click the status bar pane to activate the **Selection Information** window (p. 181). You can refer to the **View Node/Element In-**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

formation topics in the Selecting Nodes (p. 269) and Selecting Elements (p. 276) sections of the Help for additional information.

Important:

- The application does not verify your ID entries. The application ignores any specified ID that does not exist.
- Exponential (for example, 1e6) notation is not supported.

Once you make your selections, you can create an associated Named Selection (p. 1057) by selecting the **Create Named Selection** button. For the newly created Named Selection, there will be a Worksheet (p. 1074) entry for each delimiter-separated set of nodes or elements.

Manipulating the Model in the Geometry Window

This section describes the tools to manipulate (rotate, pan, and zoom) your model in the Geometry window.

Panning the Model

Selecting the **Pan** option on the Graphics toolbar (p. 97) enables you to vertically and horizontally reposition your model in the **Geometry** window. Simply select the **Pan** button on the toolbar and click your mouse within the **Geometry** window to give it focus. Then, hold the left mouse button and reposition your model. In addition, once you have given the **Geometry** window focus, you can use the arrow key to reposition your model.

Note:

At any time while the **Pan** option is active, you can use the middle mouse button (or **[Ctrl]**+**[Arrows Keys]**) to rotate your model.

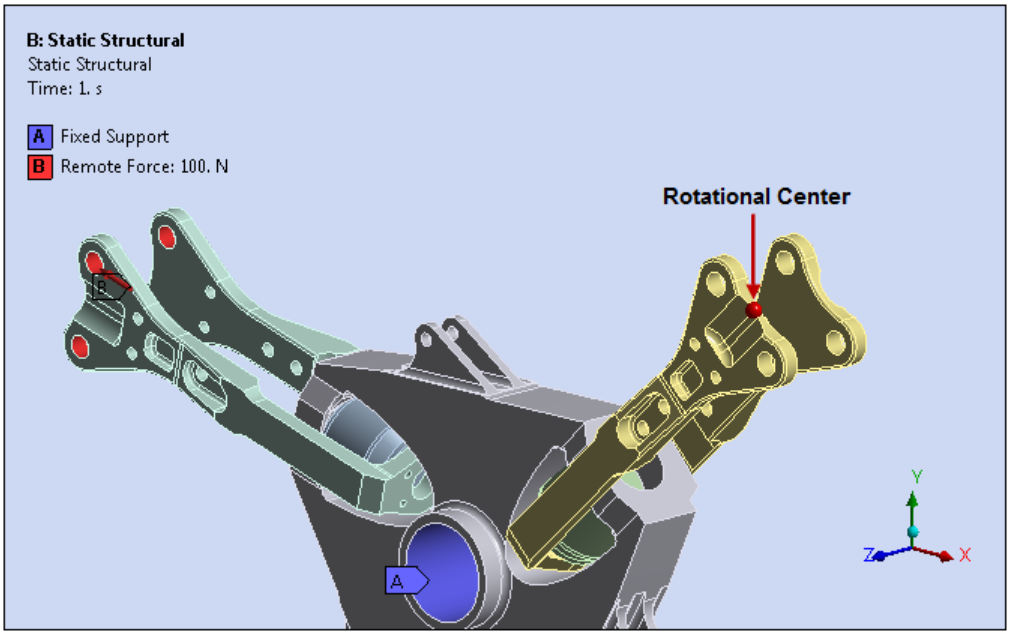
Rotating the Model

Selecting the **Rotate** option on the Graphics toolbar (p. 97) enables you to turn your model about a default or user-selected center using the left mouse button. This is a common application feature. By default, the rotational center is the center of your model.

To rotate about a specific point on the model, select a new point of rotation on your model with the left mouse button. This action recenters your model in the **Geometry** window and displays a red sphere that indicates the newly selected center of rotation. From this position, you can rotate your model freely about the new rotation point. To restore the default rotation point, simply click off of the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Rotation Using the Middle Mouse Button

You can also, at any time, rotate your model in the **Geometry** window using the middle mouse button. With the middle mouse button, you can select a location on you model, hold the middle mouse button, and rotate your model about that point. Clicking off of the model returns the rotational center to the application default. Using this method, the application does not recenter the model in the window based on the newly selected center of rotation.

Note:

These middle mouse button options are always available when the Geometry window has focus.

On Mouse Down Set

For the left mouse button, you may change the default setting for the **Model Rotation Center** option in the Graphics (p. 228) category of the Options preference (p. 217) to **On Mouse Down Set**. Using this setting, with the **Rotate** feature active, the application does not recenter your model and you can immediately rotate it around the new point. When in this mode, your new selection becomes the default.

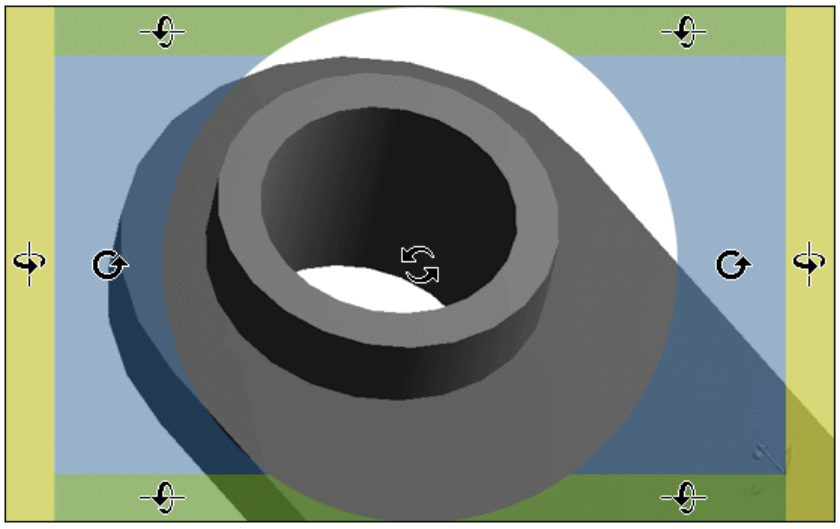
Rotation Behavior Based on Cursor Location

You may change the default setting for the **Mouse Rotation Mode** option in the Graphics (p. 228) category of the Options preference (p. 217) to **Axis Rotation Available**. Using this setting, with the **Rotate** feature active, the application activates the roll, yaw, and pitch cursor options in the **Geometry** window, as described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Geometry Window Cursor Icon	Description
	Free rotation.
	Rotation around an axis that points out of the screen (roll).
	Rotation around a vertical axis relative to the screen (yaw axis).
	Rotation around a horizontal axis relative to the screen (pitch axis).

As illustrated below, the application displays these icons based upon where you position the cursor in the **Geometry** window.



Model Orientation using the Triad

The Triad, located in the lower right corner of the Geometry window, enables you to reorient the position of your model based on a desired axis as well as reset the isometric view (light blue ball). If you move your cursor around the triad, you will see an arrow appear that shows the direction that corresponds to the position of your cursor (+x, -x, +y, -y, +z, -z). If you click the arrow, it changes your view so that the axis indicated by the arrow is facing outward.

Zooming In and Out on Your Model

There are a number of Zoom options available in the Graphics toolbar:

- Selecting the **Zoom** button enables you to drag your left mouse button up and down in the **Geometry** window to zoom in and out on your model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Selecting the **Box Zoom** button displays a box selection area when you to drag your left mouse button. The application zooms in on this area in the **Geometry** window. Note that the smaller area that you select with this tool, the smaller area that is zoomed in upon.

Note:

- You can hold the **Shift** key and use the up and down arrow buttons respectively, to zoom in and out on your model.
- You can zoom in or out by rolling the mouse wheel.

Defining Direction

Orientation may be defined by any of the following geometric selections:

- A planar face (normal to).
- A straight edge.
- Cylindrical or revolved face (axis of).
- Two vertices.

This section discusses the following topics:

Direction Defaults (p. 283)
Highlighting Geometry in Select Direction Mode (p. 283)
Selecting Direction by Face (p. 284)

Direction Defaults

If you insert a load on selected geometry that includes both a magnitude and a direction, the **Direction** field in the **Details view** states a particular default direction. For example, a force applied to a planar face by default acts normal to the face. One of the two directions is chosen automatically. The load annotation displays the default direction.

Highlighting Geometry in Select Direction Mode

Unlike other picking filters (where one specific type of geometry highlights during selection) the **Select Direction** filter highlights any of the following during selection:

- Planar faces
- Straight edges
- Cylindrical or revolved faces
- Vertices

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

If one vertex is selected, you must hold down the **Ctrl** key to select the other. When you press the **Ctrl** key, only vertices highlight.

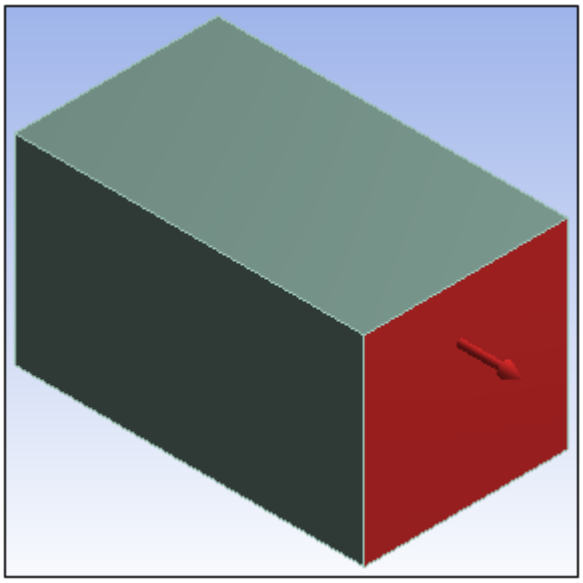
Selecting Direction by Face

The following figure shows the graphic display after choosing a face to define a direction. The same display appears if you edit the **Direction** field later.

- The selection blip indicates the hit point on the face.
- Two arrows show the possible orientations. They appear in the lower left corner of the Geometry Window (p. 129).

  If either arrow is clicked, the direction flips.

When you finish editing the direction, the hit point (initially marked by the selection blip) becomes the default location for the annotation. If the object has a location as well as a direction (e.g. Remote Force), the location of the annotation will be the one that you specify, not the hit point.



Note:

The scope is indicated by painting the geometry.

Using Viewports

The **Viewports** feature enables you to split the **Geometry** window into multiple windows, up to four, and perform independent actions in each window.

Got to a section topic:

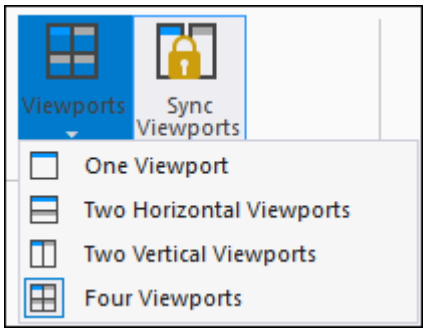
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Display Options (p. 285)
- Synchronize Viewports (p. 286)
- Capture Viewport Images (p. 286)
- Append Graph Window Data (p. 288)

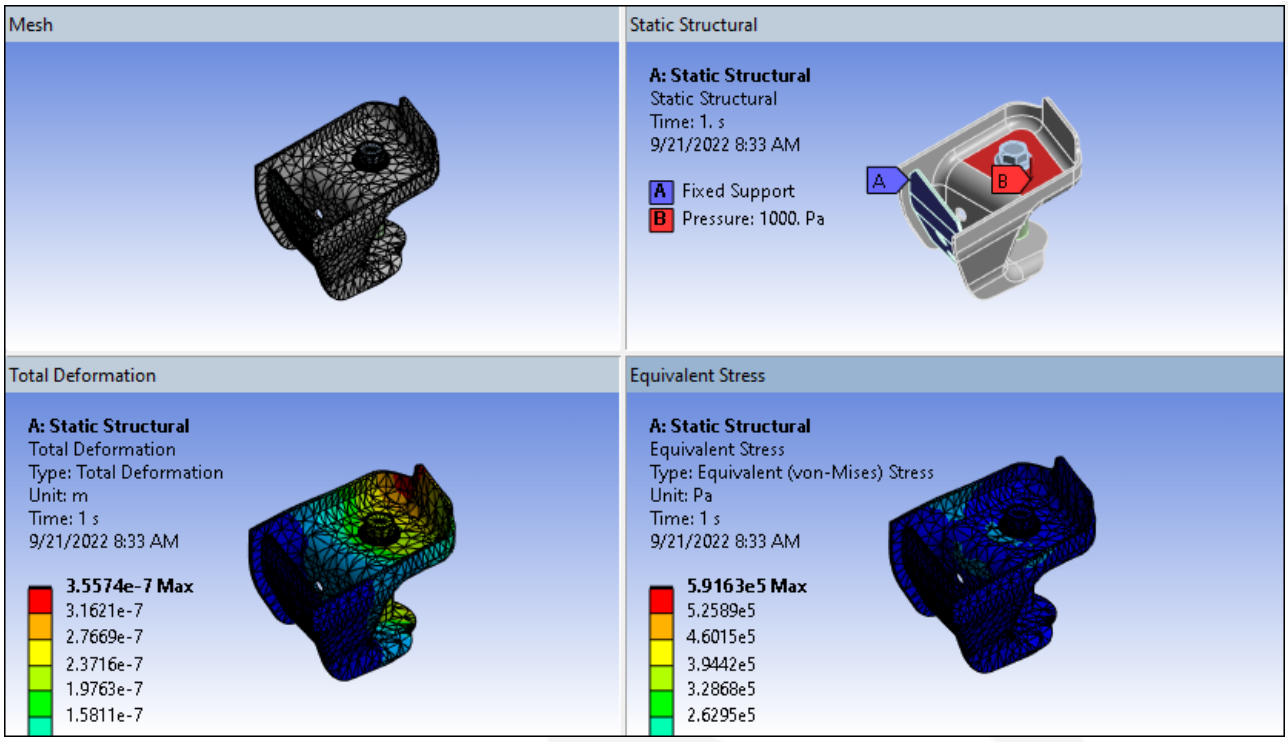
Display Options

As illustrated, the **Viewports** drop-down menu enables you to display one, two (horizontally or vertically), or four viewports in the **Geometry** window.



By default, the **Geometry** window displays one viewport/window. Once you select to display two or more viewports, the currently selected object displays in all viewport windows. You activate a viewport window by selecting it (clicking in the window or on the title bar). Once active, you can change the selected object, move your model, change property settings or scoping for the selected object, etc., *independent* of the other viewport windows displayed. For example, you can view loading conditions, results, Named Selections, contact conditions, and make changes to each separately.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

A figure can be viewed in a single viewport only. If multiple viewports are created with the figure in focus, all other viewports display the parent of the figure.

Synchronize Viewports

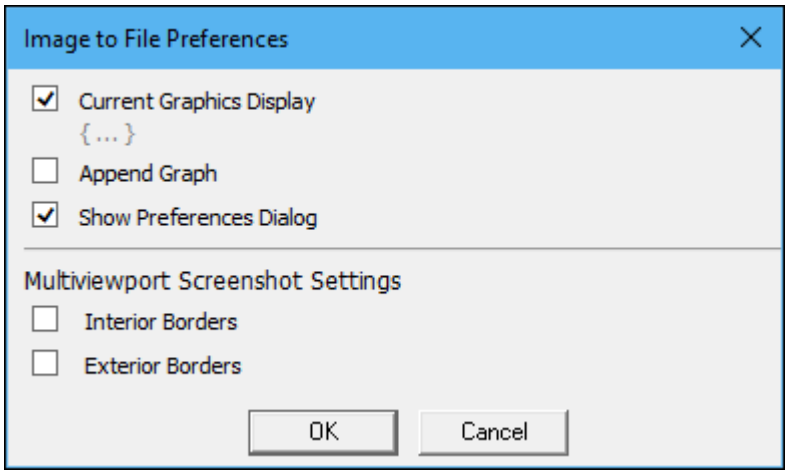
The ribbon group also includes the **Sync Viewports** option. The default setting is active. It enables you to synchronize the display in each window to reorient/move (pan, zoom, rotate) your model in each window simultaneously. When you have multiple orientations configured with the **Sync Viewports** option *inactive* and then you activate it, the viewport window that you select and manipulate first, becomes the window that all the other windows synchronize to.

Capture Viewport Images

The **Images** drop-down menu of the Insert (p. 42) group on the **Home** tab includes the option **Composite image to File**. This option acts the same as the Image to File (p. 42) option. It enables you to save the contents (viewports) of the **Geometry** window as an image file. When you select the option, a dialog displays that enables you to change image details such as resolution, appearance, and magnification.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

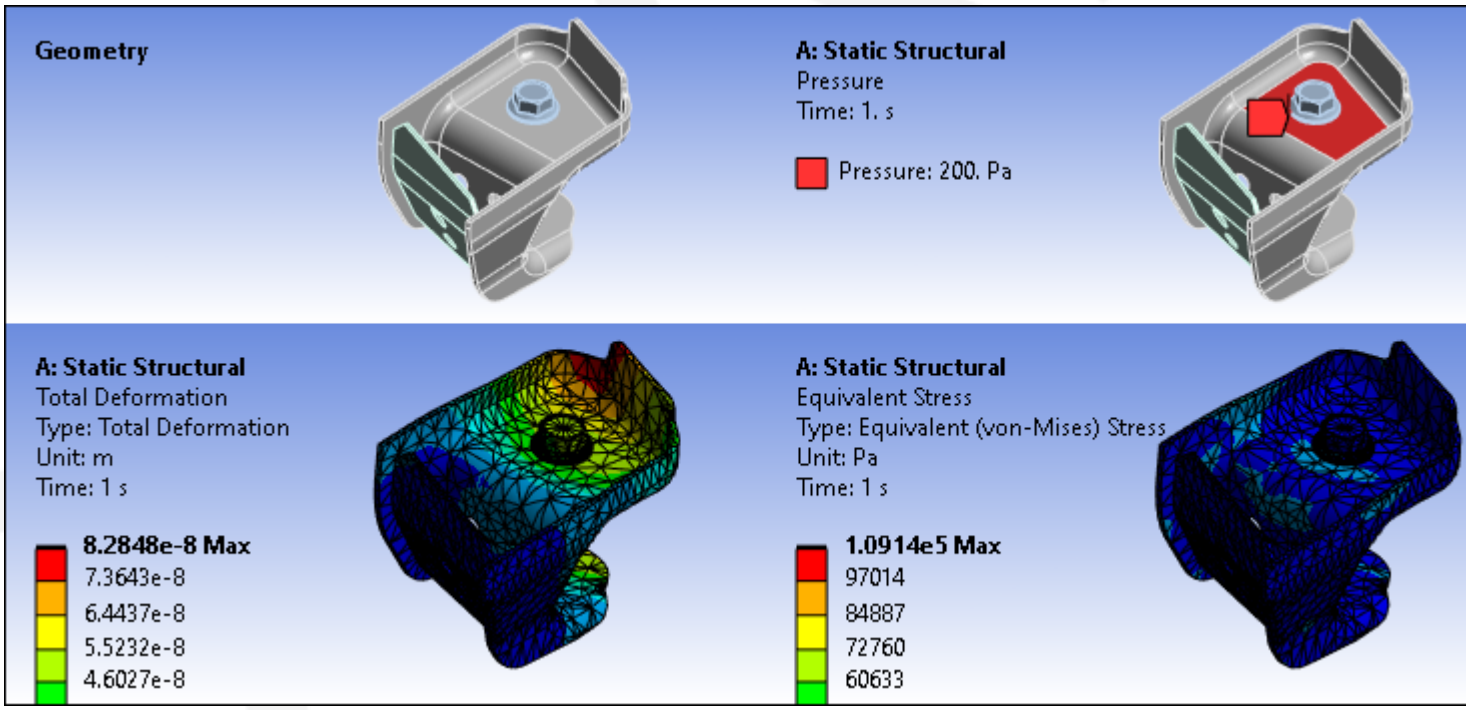
Published: 2023-10-23



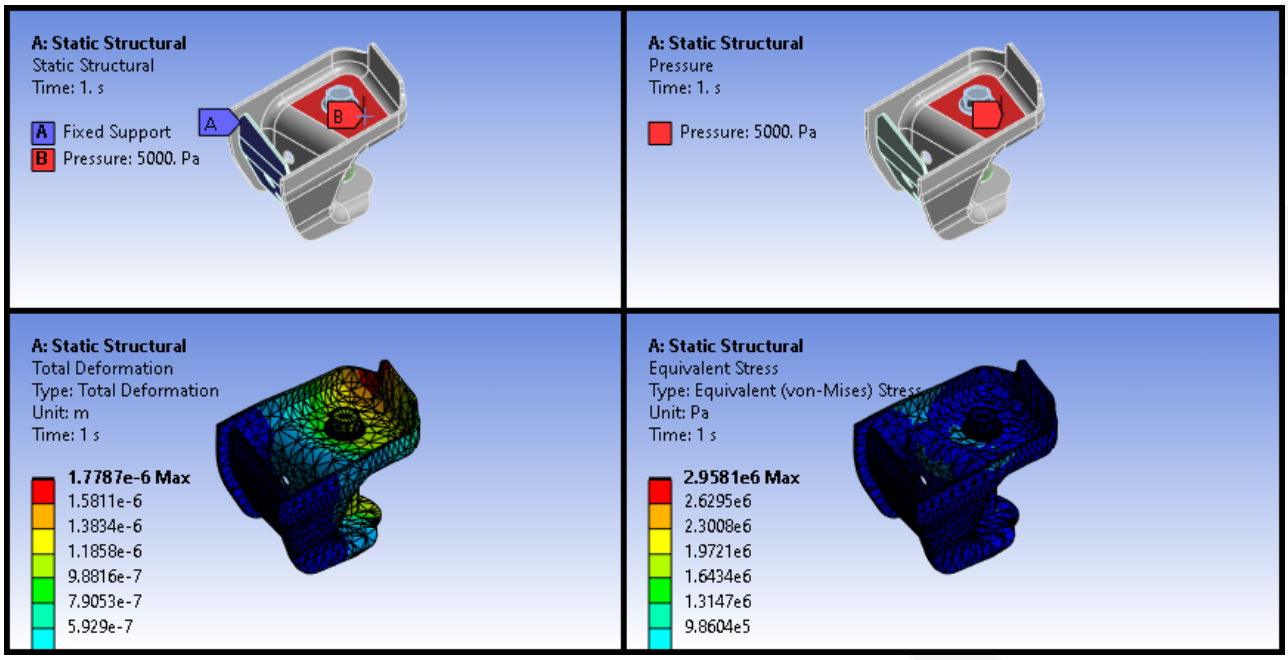
Note:

You can change the default settings for the dialog's options using the properties of the Image Export (p. 232) group of the **Graphics** category of the Options (p. 217) dialog.

The dialog also includes the options **Interior Borders** and **Exterior Borders** under the **Multiviewport Screenshot Settings** title that enables you to add borders around and in-between the viewports you capture. The following images show a comparison of these options when turned on and off.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

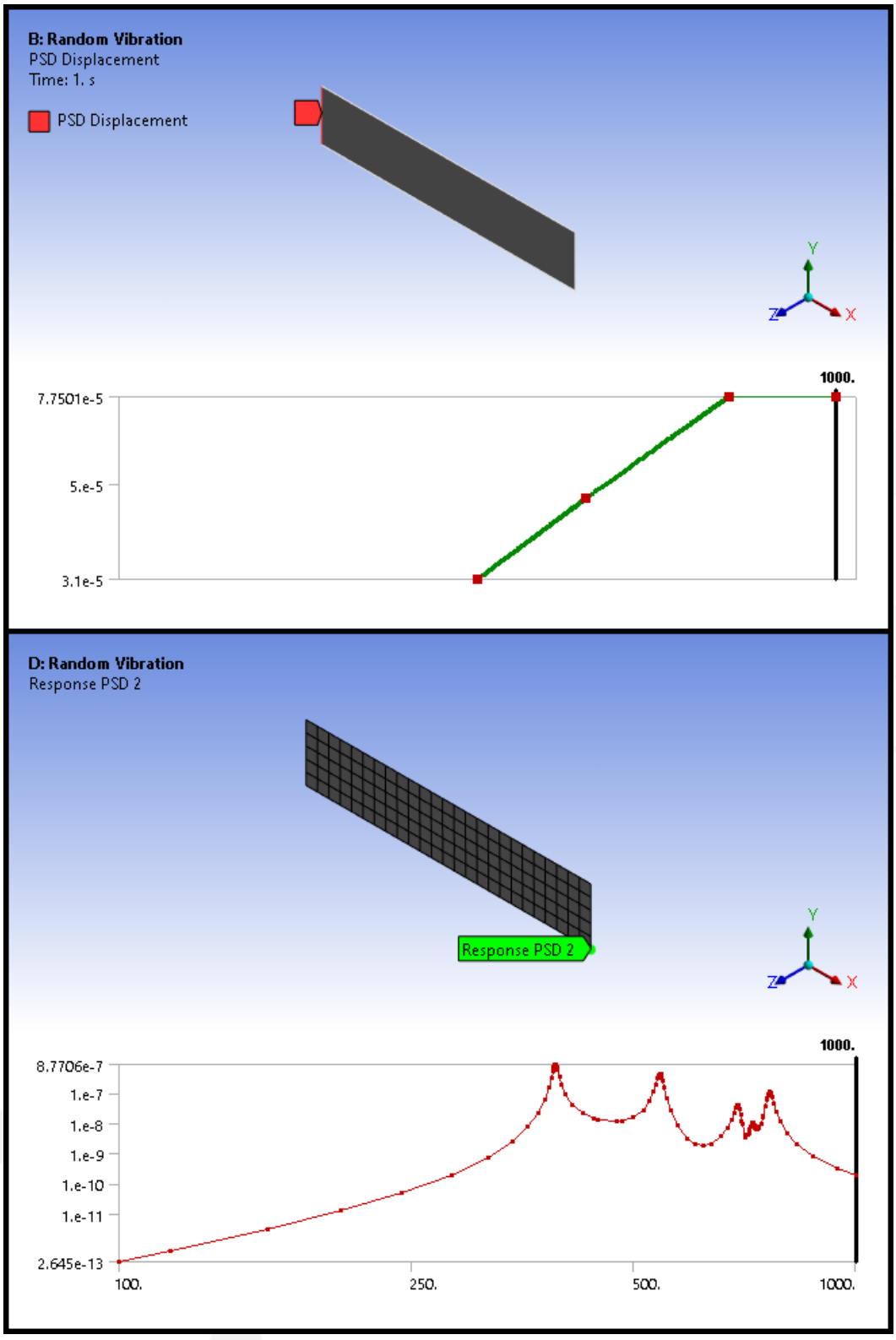


Append Graph Window Data

In addition, the dialog contains the check box **Append Graph**. As shown here, the application adds the content of the **Graph** window in the lower left-hand corner of the image. If there is no corresponding data in the **Graph** when this option is selected, the captured image is appended with a graph that includes no data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Accelerated Graphics

If you are using a Windows system that supports OpenGL 4.3, the application automatically utilizes the **Accelerated Graphics** feature. This feature uses graphical display technology that produces a faster display of mesh and result objects when you first select the object, with even greater gains

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

when you select the object a second/third time. In addition to faster graphical display times, this feature provides more accurate color contours, a higher fidelity vector display, and a Smoothed Particle Hydrodynamics (SPH) display.

Important:

Although released, additional development is being made to the feature. Some application features and displays are not yet compatible with **Accelerated Graphics** and will automatically revert to previous display technology. Support in this regard is ongoing.

Note:

Once active, the label "Accelerated Display" is placed below the Ansys logo in the Geometry window.

Go to a section topic:

- Application (p. 290)
- Associated Preferences (p. 290)
- Cautions and Limitations (p. 291)

Application

Activate the feature using the **Accelerated Graphics for Mesh/Result Display** property in the Options (p. 217) dialog (Graphics (p. 228) preferences). Property options include:

- **Program Controlled** (default): The application selects the graphical mode based on application settings and support status, such as analysis settings, the specified result, view options, etc. This is the recommended setting.
- **Yes, if possible**: This setting forces the use of **Accelerated Graphics** wherever possible, even if only partially supported. However, this setting may produce graphical display issues for other features and displays options that are not yet supported by the feature. For large, complicated models, the benefits of the enhanced display may outweigh these display issues.
- **No**: This setting disables the feature.

Note:

You can also activate this feature from the Graphics Toolbar (p. 97).

Associated Preferences

This feature has the following associated preferences, included in the **Performance** category of the Graphics (p. 228) preferences of the Options dialog.

- **Show Accelerated Display Message in Graphics Window**: Turn the “Accelerated Display” label on and off. Options include No (default) and Yes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Show Why Accelerated Graphics is Not Used**: Display an Information message for why Accelerated Graphics is not used for the mesh and results when it is not possible. Options include **No** (default) and **Yes**.
- **Use Faceted Lighting for Accelerated Graphics/Animation (Beta)**: This beta property enables you to modify the lighting. Options include **No** (default) and **Yes**. The **No** option specifies the standard lighting mode. This setting can cause issues with large deformation but maintains smooth lighting and is suitable for most models. The **Yes** setting uses an alternate faceted lighting mode. Lighting is done on a per-triangle basis resulting in a more jagged rendering of some models, but the lighting should be correct for any deformation.

Cautions and Limitations

Note the following when this feature is active:

- You may see minor graphical differences. For example, there could be slight differences in how the application displays translucency and higher resolution SPH symbol displays. These differences should not affect the accuracy of the display.
- The application uses an alternative system for rendering text. As a result:
  - You could see slight differences in text sizes and text placement compared to the graphical viewport in the legend, ruler, triad, etc.
  - You could see Unicode text (such as Japanese text) display limitations. For example, if the system name or a result name includes Unicode, or there are user labels that include Unicode text, this text may not render completely.
- The application does not yet support certain features, such as Section Planes and result probes. Processing contingencies for these features exist, however, you may see lag for larger models while using these features.

Controlling Graphs and Charts

The following controls are available for Graphs/Charts for Using Adaptive Convergence (p. 1852), and Fatigue Results (p. 2165) result items.

Feature	Control
Pan	Right Mouse Button
Zoom	Middle Mouse Button
Box Zoom	**Alt**+Left Mouse Button
Rotate (3D only)	Left Mouse Button
Perspective Angle (3D only)	**Shift**+Left Mouse Button
Display Coordinates (2D only)	**Ctrl**+Left Mouse Button along graph line

Tips for working with graphs and charts:

- Some features are not available for certain graphs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Zoom will zoom to or away from the center of the graph. Pan so that your intended point of focus is in the center prior to zooming.
- If the graph has a Pan/Zoom control box, this can be used to zoom (shrink box) or pan (drag box).
- Double-clicking the Pan/Zoom control box will return it to its maximum size.

Managing Graphical View Settings

Using the **Manage View** option of the **Tools** group (p. 45) on the **Home** tab, you can save graphical views and return to a specific view at any time. This option displays an independent window that you use to work with the feature. The feature enables you to create a list of desired views. You can export your view list and import it into different projects and maintain a consistent model view between multiple projects.

This section discusses the following topics:

Creating a View
Applying a View
Renaming a View
Deleting a View
Replacing a Saved View
Exporting a Saved View List
Importing a Saved View List
Copying a View to Mechanical APDL

Creating a View

To save the current graphical view:

1. Click the **Create a View** option in the **Manage Views** window. A new entry with the naming convention of "View #" is created.
2. As desired, enter a new name for the view.

You can now return to this view at any time using this view entry.

Note:

You must save the project to save your created views in the **Manage Views** window.

Applying a View

Saved graphical views are listed in the **Manage Views** window. You can return to a saved view at any time.

To return to a saved graphical view:

1. In the **Manage Views** window, select the view.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Apply View** button.

The **Geometry** window reflects the saved graphical view.

Renaming a View

To rename a saved graphical view:

1. In the **Manage Views** window, select the view you want to rename.
2. Click the **Rename** button, or press **F2**.
3. Enter the new view name.
4. Click the **Apply** button.

Deleting a View

To delete a saved graphical view:

1. In the **Manage Views** window, select the view you want to delete.
2. Click the **Delete** button.

Replacing a Saved View

To replace a saved view with the current graphical view:

1. In the **Manage Views** window, select the view you want to update.
2. Click the **Replace saved view based on current graphics** button.

Exporting a Saved View List

You can export a saved graphical view list to an XML file. This file can then be imported into other projects.

To export a saved view list:

1. In the **Manage Views** window, click the **Export** button.

   The **Save As** window appears.
2. Navigate to the file directory where you want to store the XML file and enter the desired file name.
3. Click **Save**.

Importing a Saved View List

Saved view lists can be exported to XML files. You can then import a saved view list from an XML file to a different project.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To import a saved graphical view list:

1. In the **Manage Views** window, click the **Import** button.

   The **Open** window appears.

2. Select the file you want to import.

3. Click **Open**.

Copying a View to Mechanical APDL

You can copy a saved graphical view as a Mechanical APDL command and insert the command into a Mechanical APDL file. The view in Mechanical APDL will then be consistent with the selected graphical view.

To copy a graphical view to Mechanical APDL:

1. In the **Manage Views** window, right-click a view and select **Copy as MAPDL Command**.

2. Create or open an existing Commands (APDL) file.

3. Paste the new Mechanical APDL command into the file. The settings structure is:

   ```
   /FOC
   /VIEW
   /ANG
   /DIST
   ```

4. Select the **Solve** button, and the new view is available in the Commands (APDL) file.

Creating Section Planes

The **Section Plane** feature creates cuts or slices on your model so that you can view internal geometry, mesh, and/or result displays. The graphical display and operation of the feature varies depending upon whether you are displaying your model as geometry or if you are displaying the mesh or a result. You can create as many as six active Section Planes for a model. Once this maximum is met, you can add additional planes, but you cannot activate (or view) them until you have deactivated (unchecked) an existing plane and then activated the desired plane.

Tip:

Activate the Consider Section Planes When Mesh Picking (p. 228) preference in order to select newly displayed finite element entities (nodes, elements, and element faces) from geometries within the model that were hidden before adding the specified Section Plane. Review the Selecting Mesh Entities Displayed by the Section Plane (p. 301) topic at the end of the section to make sure that you understand the requirements and limitations of this feature.

Go to a section topic:

- Application (p. 295)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

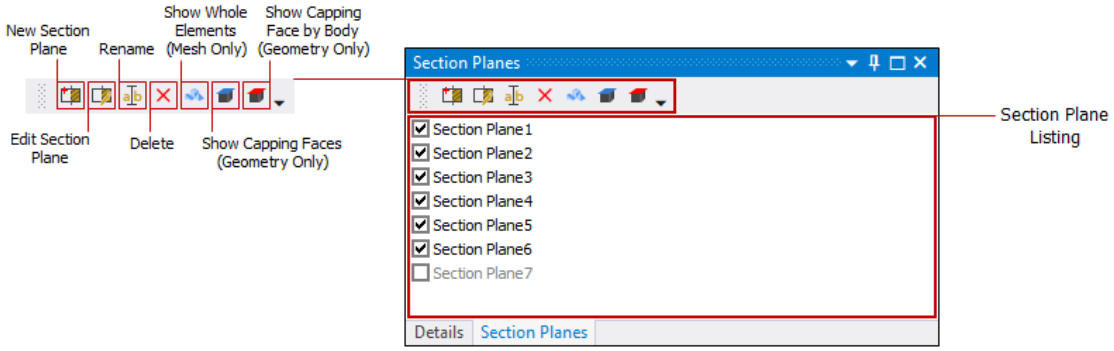
Published: 2023-10-23

- Section Plane Tools (p. 295)
  - New Section Plane (p. 296)
  - Edit Section Plane (p. 296)
  - Delete Section Plane (p. 298)
  - Rename Section Plane (p. 298)
  - Show Elements (p. 298)
  - Show Capping Faces (p. 299)
  - Show Capping Faces by Body Color (p. 300)
- Selecting Mesh Entities Displayed by the Section Plane (p. 301)

See the next two sections, Understanding Section Plane Display Differences (p. 303) and Working with Section Plane Results (p. 306), for information about display differences for section planes as well as display characteristics for when you apply a Section Plane to a result.

Application

Select the **Section Plane** option from the **Insert** group of the Home tab (p. 40) to open the **Section Planes** window illustrated below. The window displays a list of existing section planes (once created) and also provides the tools used to add, modify, or delete you section planes.



Note:

The Section Planes tool supports up to six (6) defined planes. Once you exceed six, unchecking an existing plane enables you to activate any defined planes greater than six.

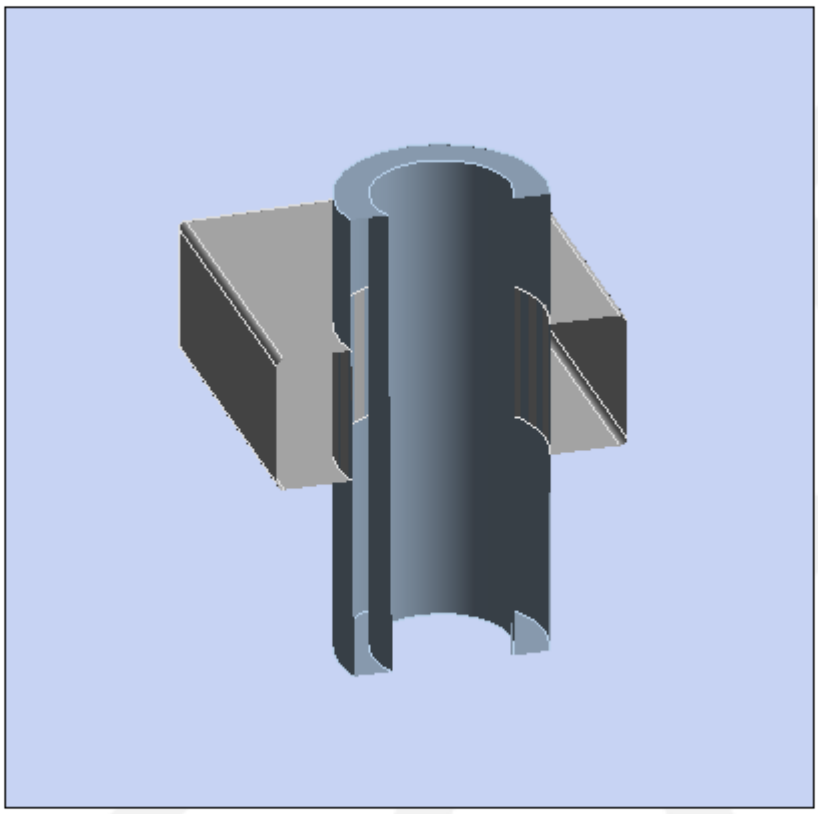
Section Plane Tools

The **Section Planes** window provides the following tools. You toggle these tools on and off by selecting the button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

New Section Plane

Select this option and create a new Section Plane in the Geometry window. Drag the mouse pointer across the geometry where you want to create a section plane. The new section plane automatically displays in the Section Planes window with a default name of "**Section Plane #**." The checkmark next to the plane's name indicates it is an active section plane. You can construct additional Section Planes by clicking the New Section Plane button and dragging additional lines across the model. Note that activating multiple planes displays multiple sections.



Important:

Note that for incidences, such as very large models, where the accessible memory is exhausted, the New Section Plane tool reverts to a Hardware Slice Mode that prohibits the visualization of the mesh on the cut-plane.

Edit Section Plane

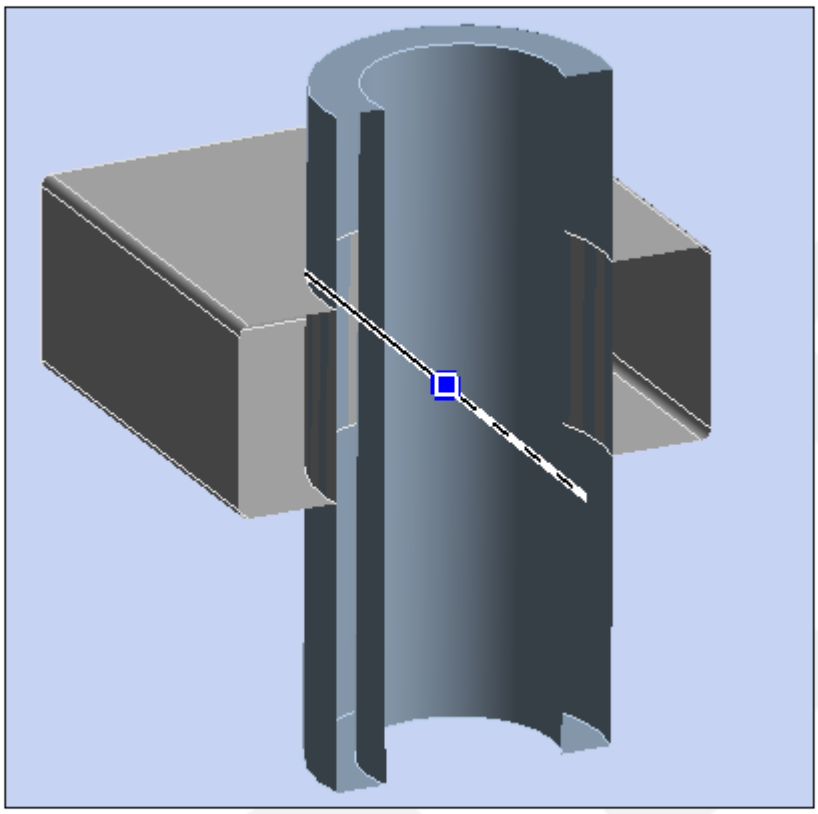
Highlight one of the Section Planes available in the window listing and then select this option to edit the highlighted section plane.

To edit a section plane:

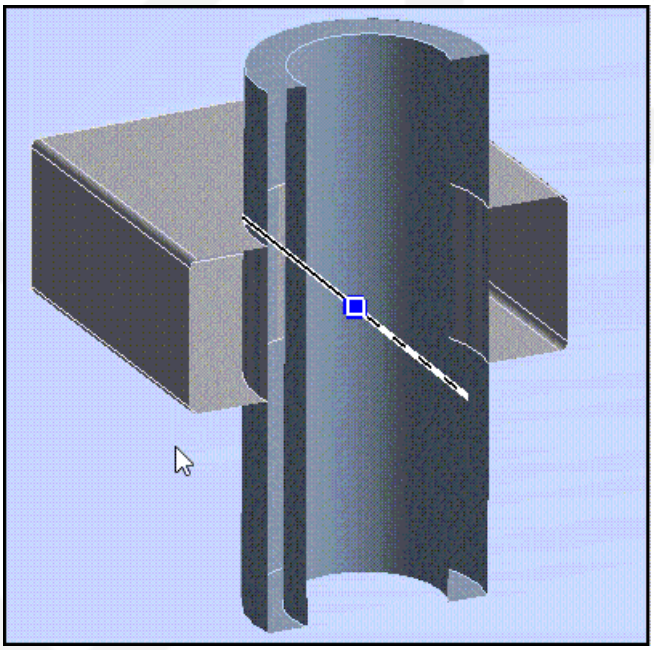
1. In the **Section Planes** window, select the plane you want to edit.
2. Click the **Edit Section Plane** button. The section plane's anchor appears.
3. Drag the Section Plane or Capping Plane anchor to change the position of the plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can click the line on either side of the anchor to view the exterior on that side of the plane. The anchor displays a solid line on the side where the exterior is being displayed. Clicking the same side a second time toggles between solid line and dotted line, that is, exterior display back to section display.



This animation shows the result of dragging the anchor (not visible for PDF versions of the Help).



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Delete Section Plane

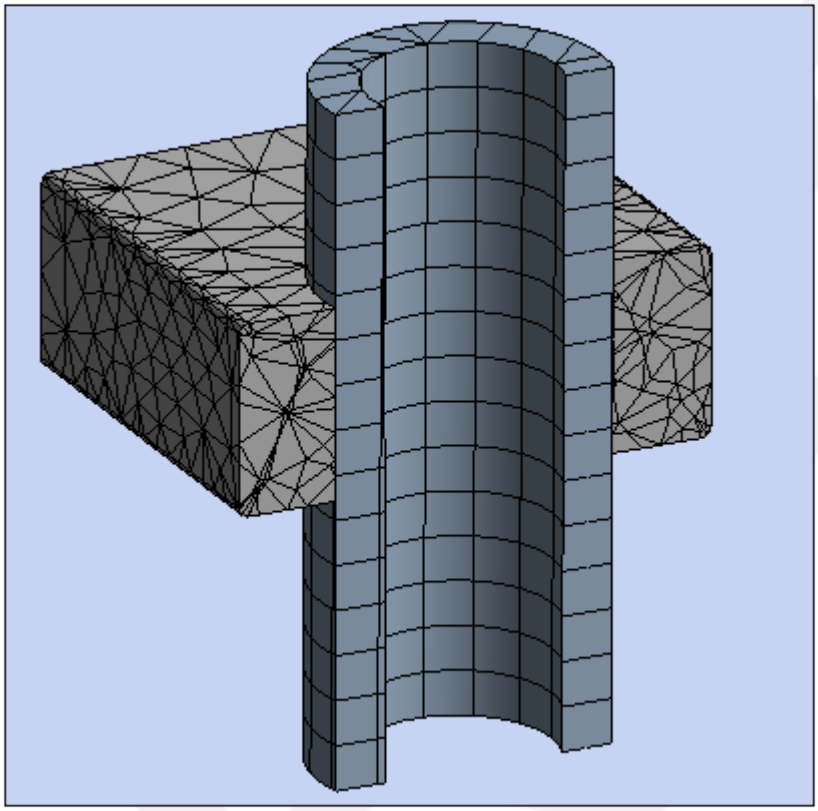
This option deletes a selected Section Plane from the listing.

Rename Section Plane

Select this option to rename a Section Plane.

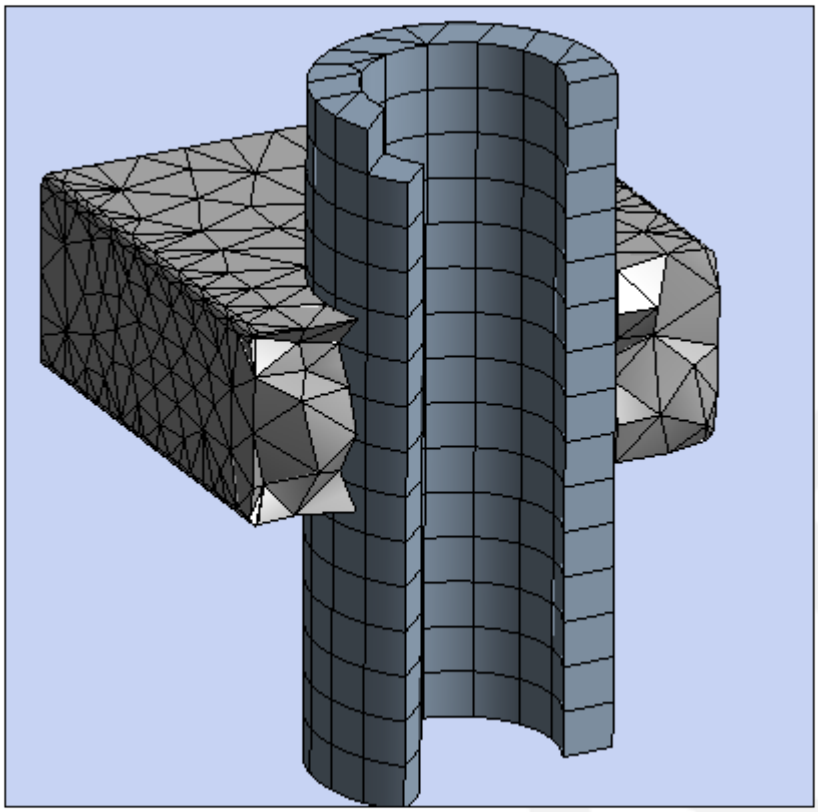
Show Elements

When you have the **Mesh** object selected or you have the **Show Mesh** feature activated, this selection causes any partially sliced elements to display entirely.



When you are viewing a Mesh display, you can use the **Show Whole Elements** button to display the adjacent elements to the section plane which may be desirable in some cases.

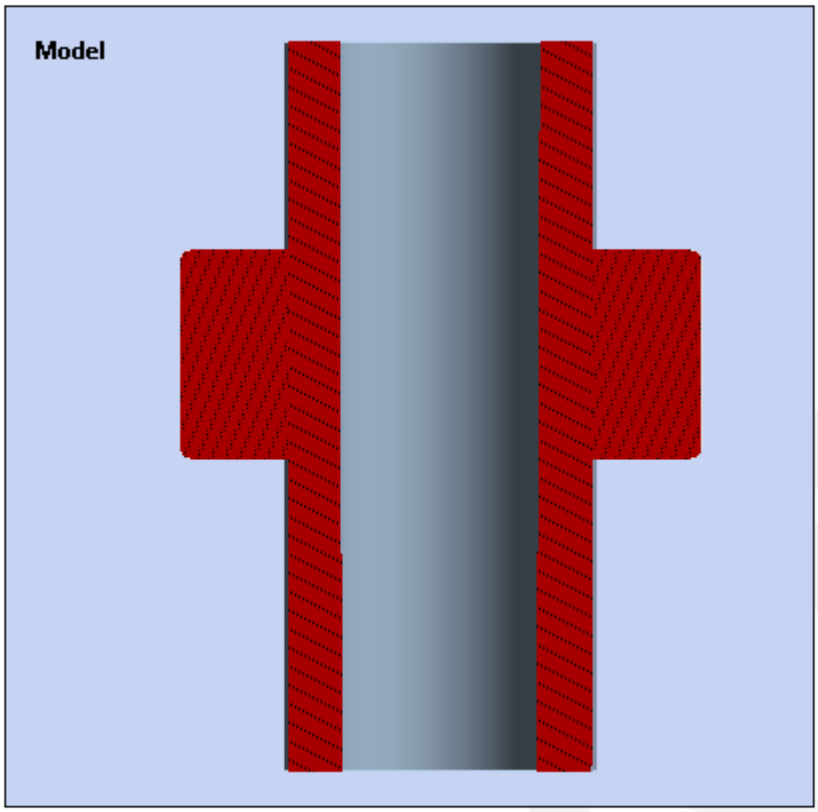
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.



Show Capping Faces

When *only one* Section Plane is contained in the window, by default, the slice is not capped and you can see the interior of the geometry. Selecting this option caps the geometry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

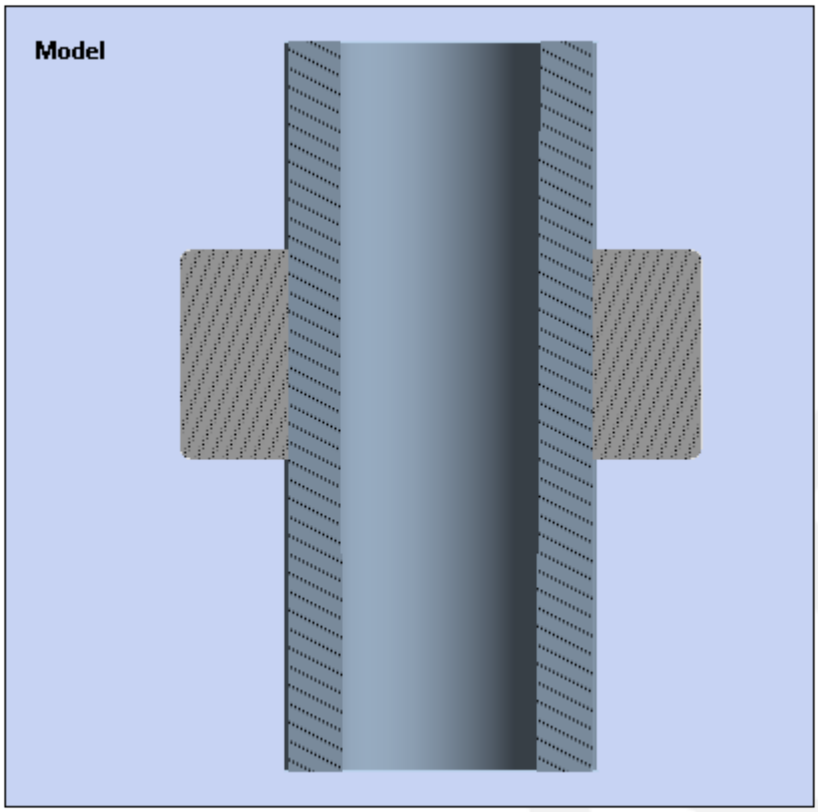


Show Capping Faces by Body Color

This selection works in tandem with the **Show Capping Faces** option. Selecting this option changes the color of the capped geometry surface to match the body color of the geometry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

Certain objects that display geometry annotations may not correctly display capping. Examples include contact objects, joints, and objects displaying spatially varying loads.

Caution:

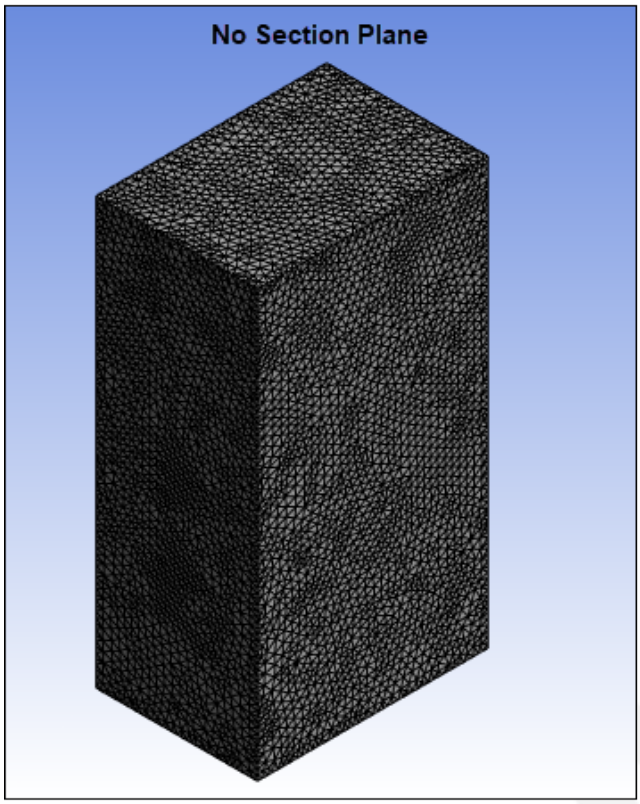
When using the **Section Planes** feature with shell bodies, make sure that the **Thick Shells and Beams** view option is turned on. Turning this view feature off changes the graphical display. The coloring for the top and bottom surfaces can degrade. So much so that the application could display both sides of a section plane simultaneously and as a result, the application could display inaccurate results.

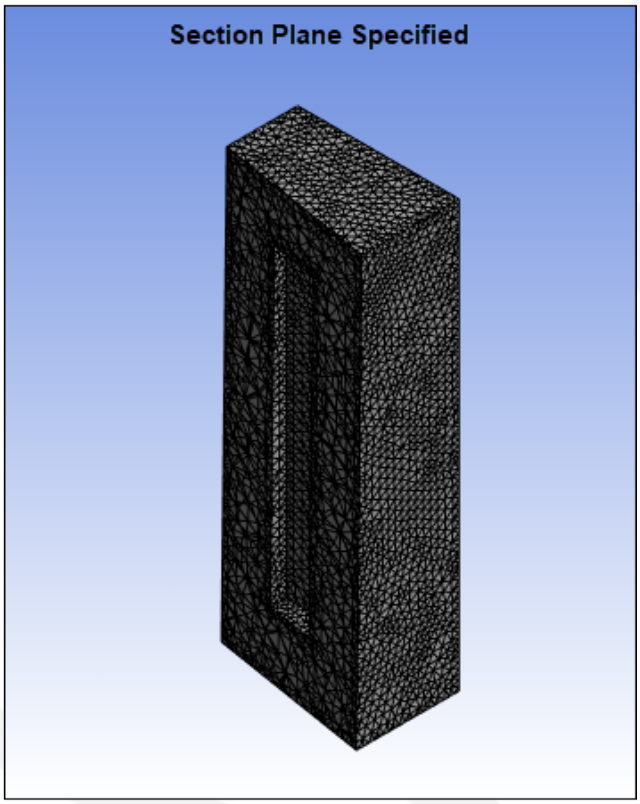
Selecting Mesh Entities Displayed by the Section Plane

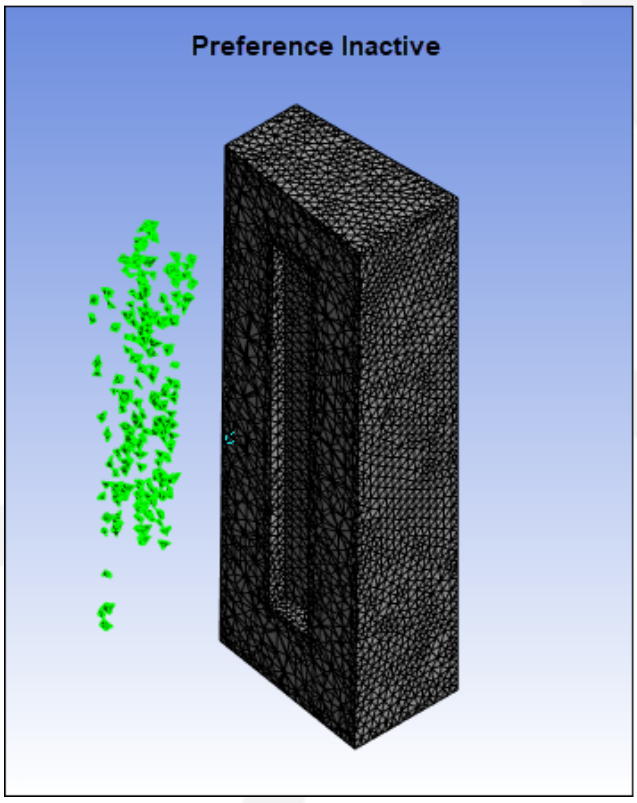
`

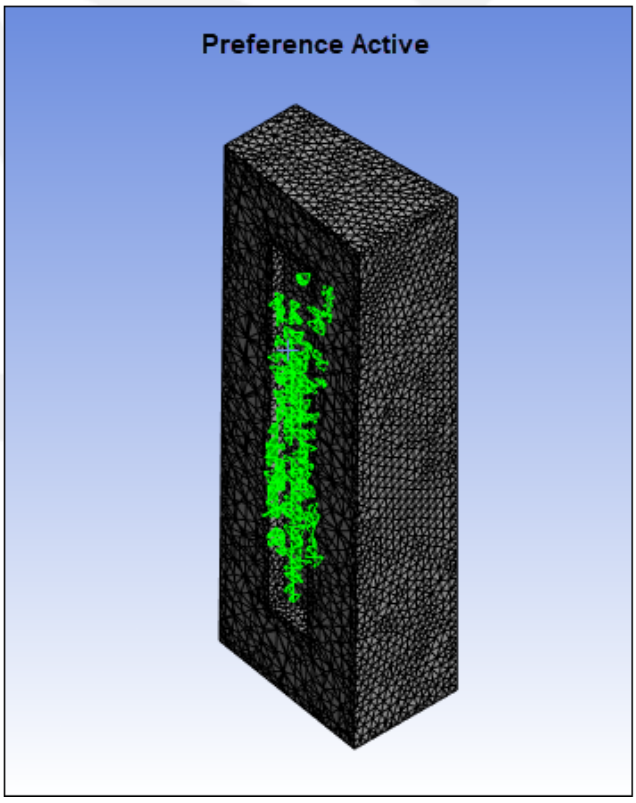
When you activate the Consider Section Planes When Mesh Picking (p. 228) preference, the application displays finite element entities (nodes, elements, and element faces) from geometries within the model that were not visible before adding the specified Section Plane. As illustrated below, in previous versions of the application, the application unexpectedly picked finite element entities on the hidden geometries (of the Section Plane). The application essentially did not recognize a Section Plane existed. This preference addresses the graphical selection limitation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23









Note:

This preference:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

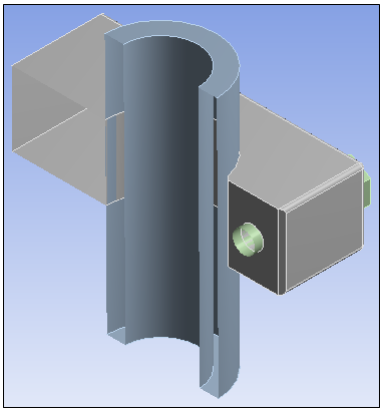
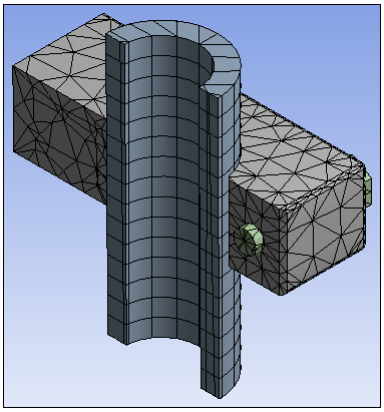
- Is supported for all active section planes.
- Does not support capped faces.
- Supports all selection modes: single, box, lasso and volume.
- Supports direct mesh selection on results.

Understanding Section Plane Display Differences

The Section Plane acts differently depending if you are viewing a result, mesh, or geometry. When viewing a result or a mesh, the cut is performed by a software algorithm. When viewing geometry, the cut is performed using a hardware clipping method. This hardware clipping cuts away the model in a subtractive method. The software algorithm cuts away the model but always starts with the whole model. Examples of these methods are illustrated below.

Note:

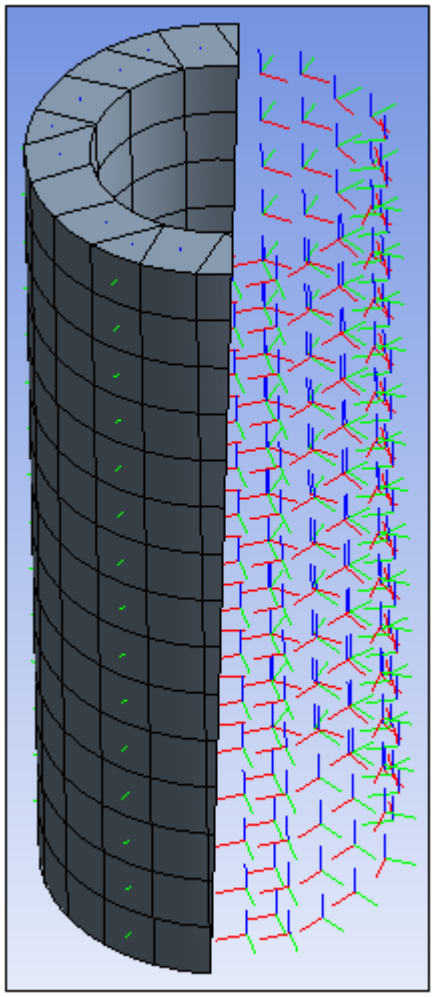
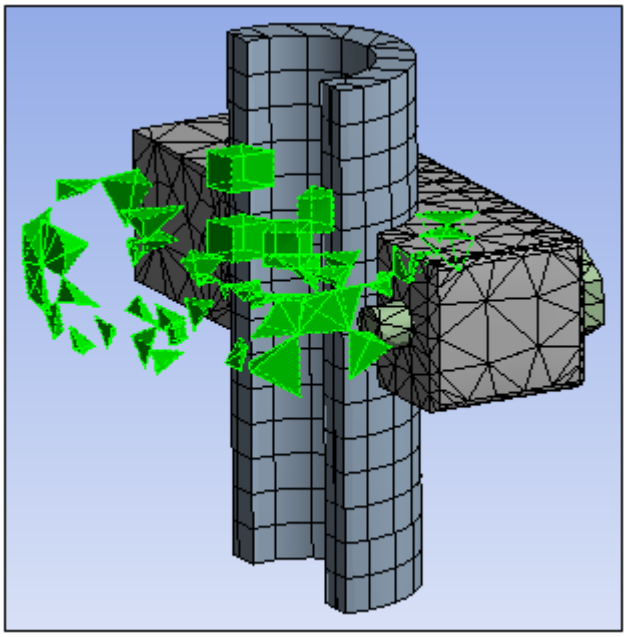
The software algorithm always caps the surfaces created by the section plane as opposed to the hardware clipping method that may or may not cap the surface depending on the display options you have selected. See the Creating Section Planes (p. 294) section for the capping display options. When capping, the software algorithm creates a visible surface at the intersection of the object and the section plane.

Geometry Display	Mesh Display
	

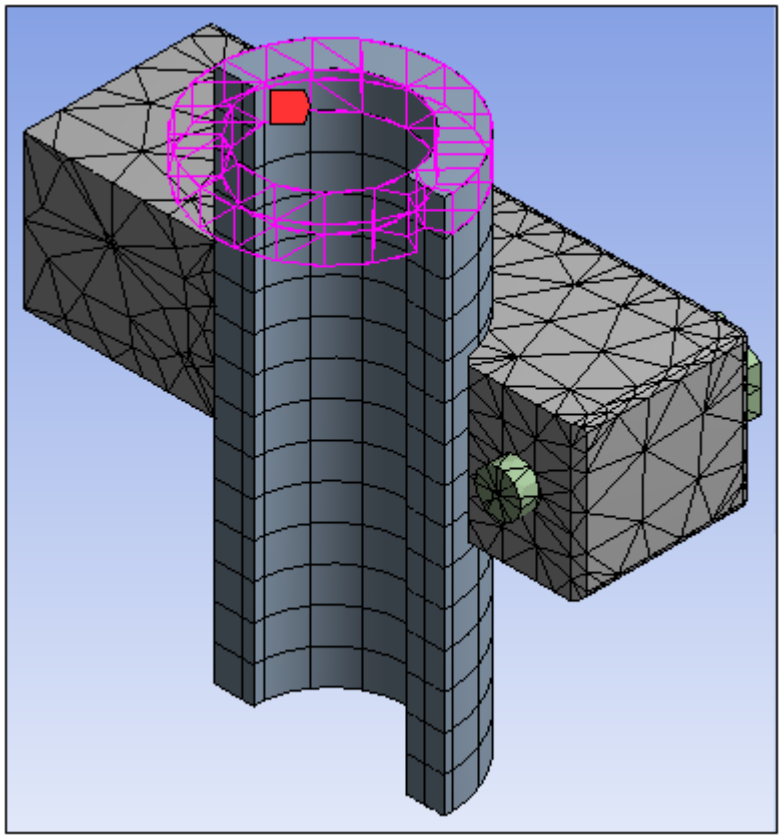
In addition, and as illustrated in the examples below, **Section Planes** do not cut the orientation or element displays if you employ:

- Element Orientation (p. 1008) feature
- Element-based, element face-based, or node-based selections
- Named Sections scoped to elements, element faces, or nodes
- Hit Point selections

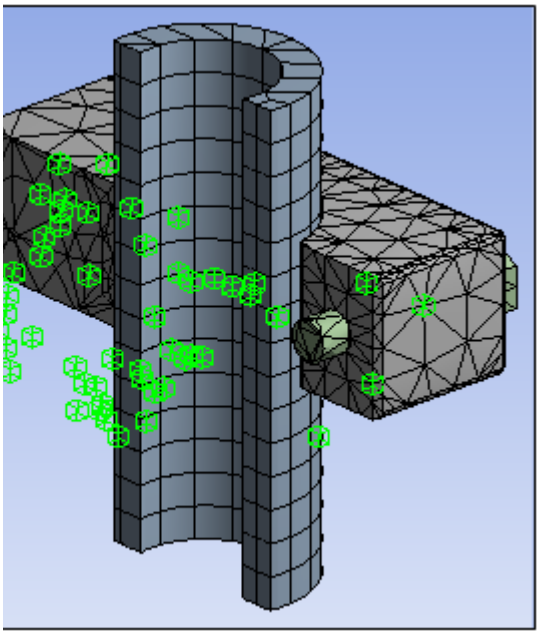
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Element Orientation Display	Element Selections
	
Element-Based Named Selection Display	**Node Selections**

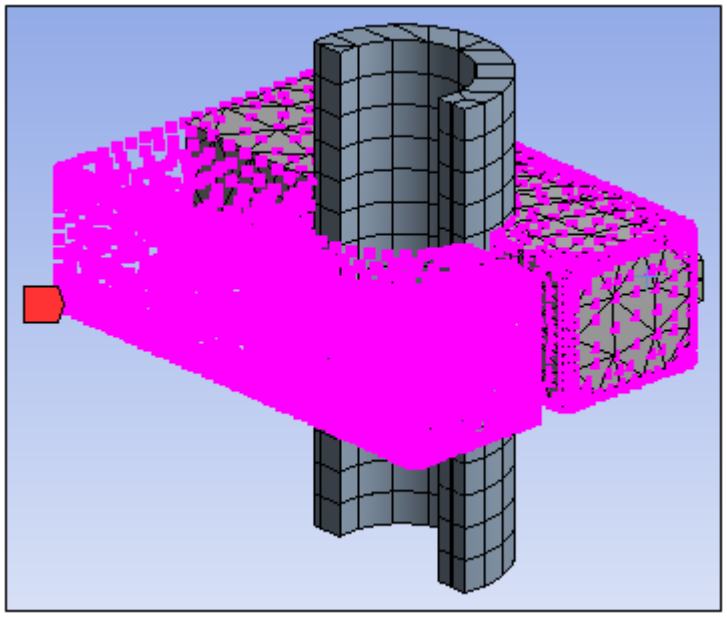
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

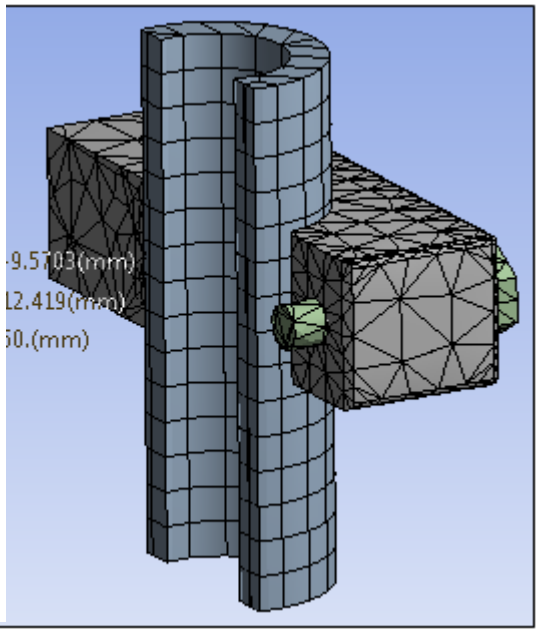


Node-Based Named Selection Display



Hit Point Display





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Working with Section Plane Results

View Options

When creating a Section Plane on a result, if the Section Plane feature is active, you can use the following options from the **Edges** drop-down menu on the Result Context Tab (p. 62):

- **Show Undeformed WireFrame**: displays the wireframe with the deformations added to the nodes. This is intended to help you interpret the image when you drag the anchor across smaller portions of the model.
- **Show Undeformed Model**: displays the portion of model behind the plane as a deformed gray scale image. In order to not clutter the graphics display in this situation, the application does **not** currently display undeformed bodies.

Deformation Scaling

When you create a Section Plane, the slice it creates is flat. If you create a Section Plane on a deformed shape and then change the scaling (p. 63) of the result, the Section Plane deforms accordingly and the plane may no longer be flat. Furthermore, if you select a different object in the tree and then return to the result that includes the Section Plane, the Section Plane re-plots as originally defined and creates new flat surface on the new deformation scale. As a result, the display of the result changes. This change can be significant.

Important:

When you are actively working with a **Section Plane**, the application disables back-face culling. Back-face culling is a computer graphics technique that discards polygons facing away from the camera and assumes the polygons to be occluded. Back-face culling is disabled to make sure that the application displays the cut section of the model correctly. Furthermore, this display necessity may cause color bleeding for coplanar polygons on shell models because the application draws both sides of the shell.

Viewing Annotations

Annotations provide the following visual information:

- Boundary of the scope region by coloring the geometry for edges, faces or vertices.
- An explicit vertex within the scope.
- A 3D arrow to indicate direction, if applicable.
- Text description or a value.
- A color cue (structural vs. thermal, etc.).

Note:

Custom annotations that you create using the Label (p. 308) feature remain visible even when you suppress the body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

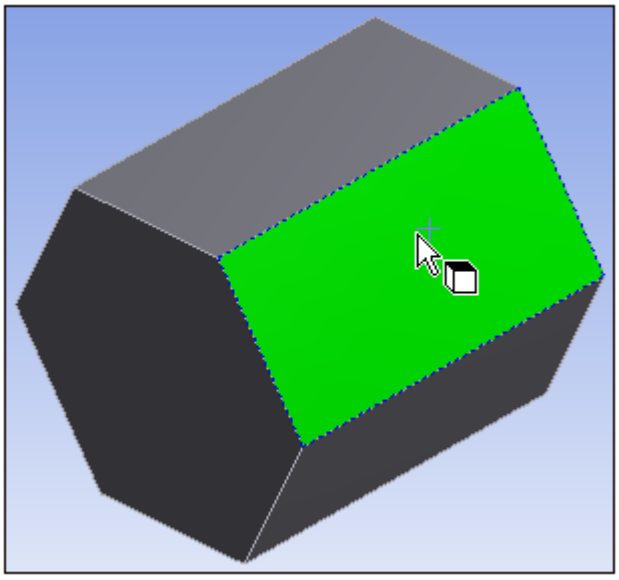
This section addresses the following types of annotations:

Highlight and Select Graphics (p. 307)
Scope Graphics (p. 307)
Annotation Graphics and Positioning (p. 308)
Annotations of Multiple Objects (p. 309)
Rescaling Annotations (p. 309)

In addition, you can also specify preferences for your annotations. For more information, see Specifying Annotation Preferences (p. 310).

Highlight and Select Graphics

You can interactively highlight and select topology, such as the face illustrated below. The topology selection highlights when you click it.



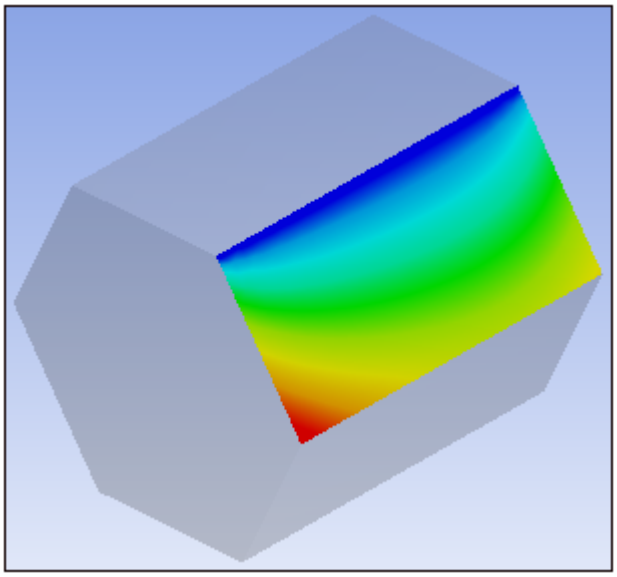
See Selecting Geometry (p. 258) for details on highlighting and selection.

Scope Graphics

In general, selecting an object in the Outline (p. 105) displays its Scope by painting the geometry and displays text annotations and symbols as appropriate. The display of scope via annotation is carried over into the Report Preview (p. 174) if you generate a figure.

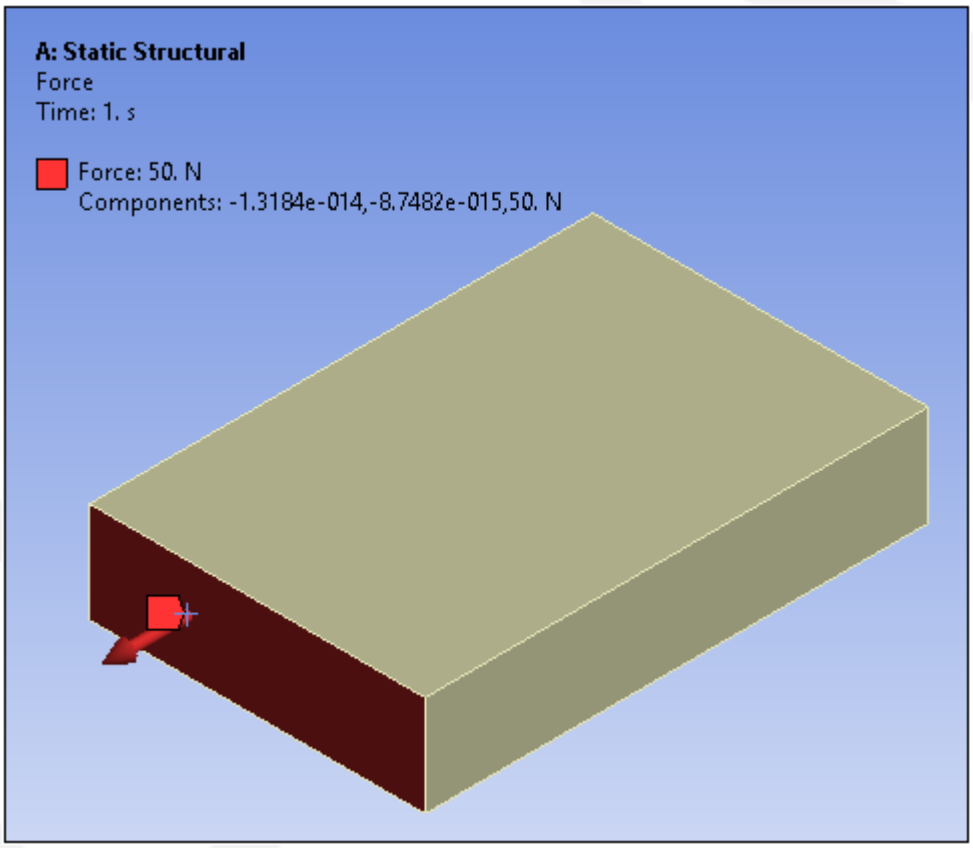
Contours are painted for results on the scoped geometry. No boundary is drawn.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Annotation Graphics and Positioning

A *label* consists of a block arrow cross-referenced to a color-coded legend. For vector annotations, a 3D arrow originates from the tip of the label to visualize direction relative to the geometry.



Use the pointer after selecting the **Label** option on the Graphics Toolbar (p. 97) for managing annotations and to drag the annotation to a different location within the scoping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

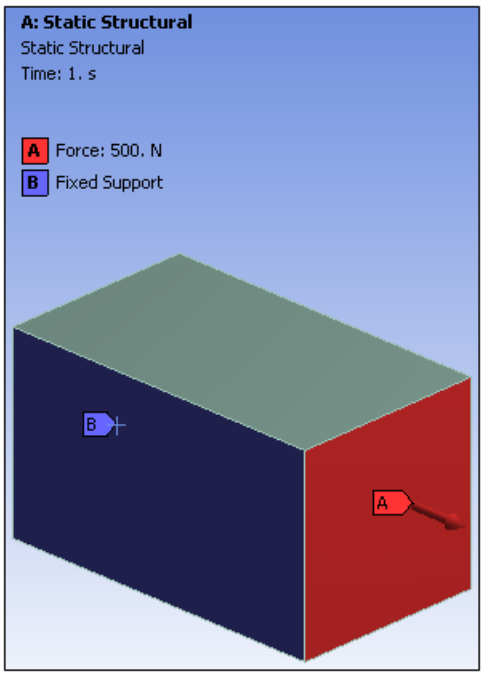
- If other geometry hides the 3D point, for example, the point lies on a back face, the block arrow is unfilled (transparent).
- The initial placement of an annotation is at the pick point. You can then move it by using the **Label** toolbar button for managing annotations.
- Drag the label to adjust the placement of an annotation. During the drag operation the annotation moves only if the tip lies within the scope. If the pointer moves outside the scope, the annotation stops at the boundary.

Annotations of Multiple Objects

When multiple individual objects or a folder (such as environment, contact, or named selections) are selected in the Outline (p. 105), an annotation for each one appears on the geometry. The default number of annotations shown is 10, but you can change it to any value from 0 to 50 using the **Max Number of Annotations to Show** property in the **Graphics** options (p. 228) of the **Options** dialog. For more information, see the Annotations (p. 79) topic in the *Selection Tab* section.

Note:

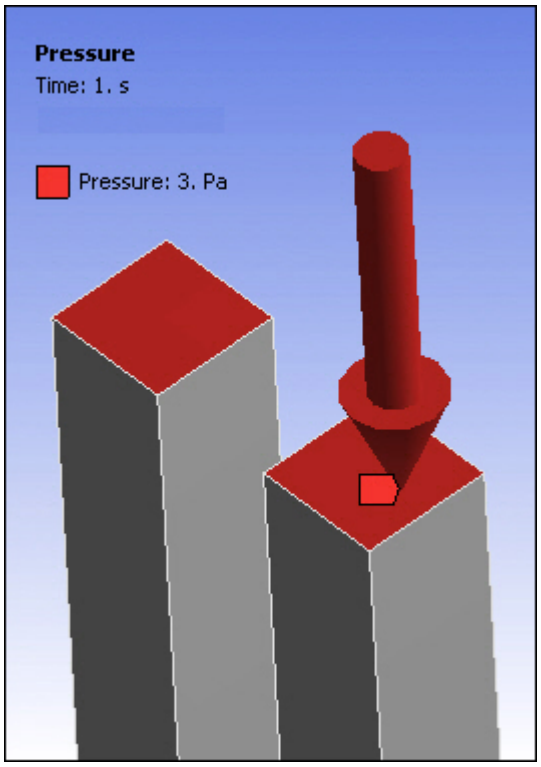
If you have a large number of objects, you may want to display each object as a different color. See the **Random** option of the Annotations (p. 79) group.

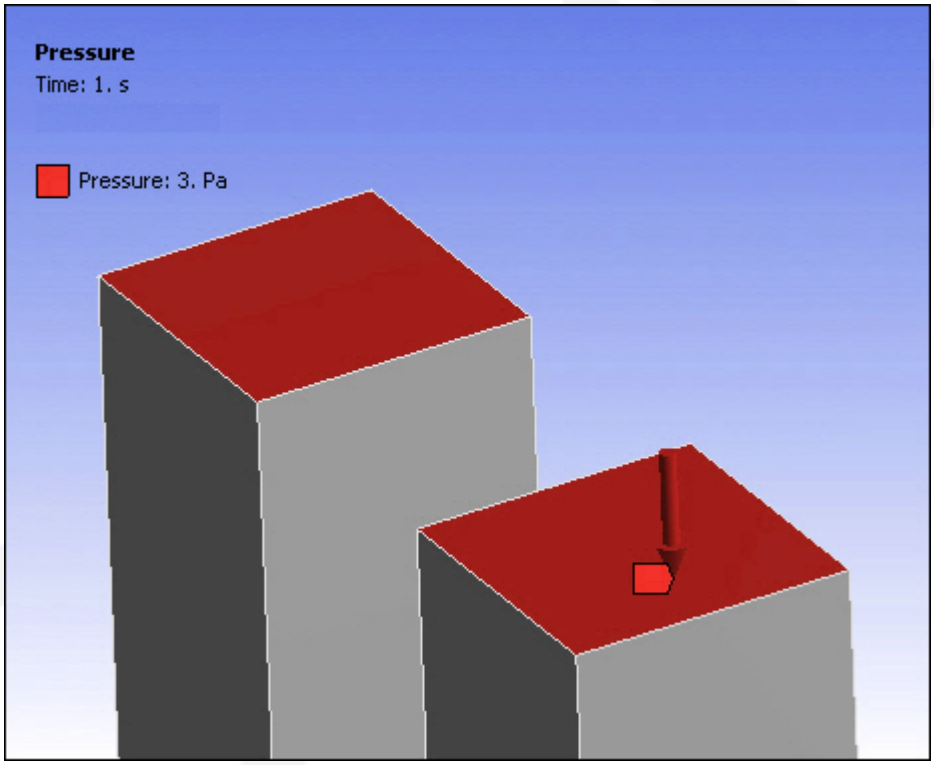


Rescaling Annotations

This feature modifies the size of annotation symbols, such as load direction arrows, displayed in the Mechanical application. For example, and as illustrated below, you can reduce the size of the pressure direction arrow when zooming in on a geometry selection. To change the size of an annotation, select the **Rescale** option in the **Annotation** group (p. 79) on the **Display** tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Specifying Annotation Preferences

The **Annotation Preferences** dialog box controls the visibility of all annotations, such as custom annotations and annotation labels, annotations on objects such as cracks, point masses, and springs.

To set your annotation preferences:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

1. Click the **Preferences** option in the **Annotation** (p. 79) group on the **Display** tab.

   The **Annotation Preferences** dialog box appears. By default, all annotations are selected, and thus set to visible.

2. Under **Basic Annotations**, select or clear the check boxes for the following options:

   - **View Annotations**: Toggles the visibility of annotations in the Geometry window.
   - **View User Defined Graphics Annotations**: Toggles the visibility of custom user annotation in the Geometry window.
   - **View Annotation Labels**: Toggles the visibility of annotation labels (p. 308) in the Geometry window.

3. Under **Remote Boundary Conditions**, select or clear the check boxes for the following options:

   - **Point Masses**: Toggles the visibility of annotations for point masses.
   - **Springs**: Toggles the visibility of annotations for springs.
   - **Beam Connections**: Toggles the visibility of annotations for beam connections.
   - **Bearings**: Toggles the visibility of annotations for bearings.

   Note:

   The size range for Point Masses and Springs is from 0.2-2 (Small-0.2, Default-1, Large-2).

4. Under **Remote Boundary Conditions**, slide the indicator to specify the size of the annotations for **Point Masses** and **Springs**.

5. Under **Additional Display Preferences**, select or clear the check boxes for the following options:

   - **Crack Annotations**: Toggles the visibility of annotations on crack objects.
   - **Individual Force Arrows on Surface Reactions**: Toggles the visibility of individual force arrows on surface reactions.
   - **Body Scoping Annotations**: Toggles the visibility of annotations on body scoping.

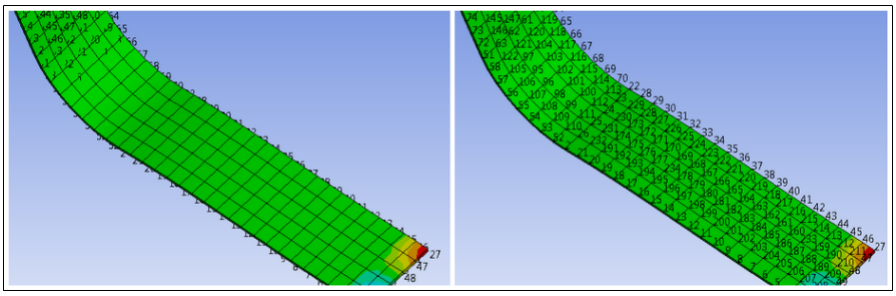
6. Under **Mesh Display**, select or clear the check boxes for the following options:

   - **Mesh Annotations**: Toggles the visibility of mesh node and mesh element annotations in Named Selection displays.
   - **Node Numbers**: Toggles the visibility of mesh node numbers in Named Selection, Mesh, and Result displays. This selection also provides options to specify a numerical range of which nodes to display. For example, display the nodes **1** (**Min**) through **200** (**Max**). An increment (**Inc**) property enables you to further define the range so that it selects

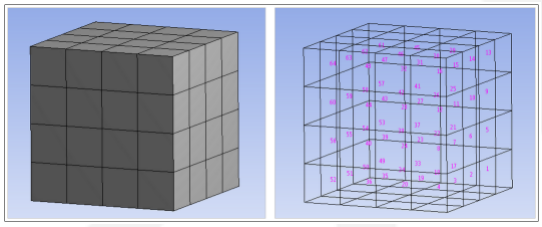
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

only every *N*th value (for example, every 5th node). The default Minimum value is **1** and the default Maximum value is **100000**.

Depending upon the number of nodes that you are displaying as well as how you have positioned your model in the Geometry window, **Node Numbers** may not fully display, as illustrated below. The **Rescale Annotation** option, available in the Graphics Toolbar (p. 97), adjusts the size of annotation symbols, as such, this option may improve the display issue.



- **Element Numbers**: Toggles the visibility of mesh element numbers in Named Selection, Mesh, and Result displays. This selection also provides options to specify a numerical range of which elements to display. Because Element Numbers are displayed at the centroid of the elements, Wireframe mode is required to properly display all Element Numbers.



- **Plot Elements Attached to Named Selections**: Toggles the visibility of elements for all items in the Named Selections group. For nodal Named Selections, this option shows the full elements, while for face or body Named Selections this option shows just the element faces. This option does not affect Line Bodies. You must have the **Show Mesh** button toggled off to see the elements in the Named Selection.

7. When you are finished specifying your annotation preferences, click **Apply Changes** to apply your preferences and leave the dialog box open, or click **OK** to apply and close.

Controlling Lighting

The **Details** view properties of the Model object provide lighting controls that affect the display in the **Geometry** window.

Note:

The application saves **Lighting** property settings with your analysis. Lighting changes propagate throughout the features of the application and are used when you export images.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Inserting Comments, Images, and Figures

You can insert Comment objects, Image objects, or Figure objects under various parent objects in the Mechanical tree to add text or graphical information that pertain specifically to those parent objects. Refer to their individual objects reference pages for descriptions. The use of a **Comment** and/or an **Image**, is essentially intuitive. However, for the **Image Plane from File** and the **Figure** options, there are additional capabilities and characteristics, as discussed below.

Image Plane

The **Image Plane from File** option enables you to import an image into Mechanical and place it on or around you model based on a coordinate system setting. The option is primarily used with the Construction Line (p. 895) feature to overlay and accurately sketch one or more line segments. However, you can use it any way you want. For example, you can place logos on your model, specify a background, set up physical rulers to help measure deformations. You may also wish to use it with an ACT extensions to generate images or 2D graphics that you can then programmatically/automatically render when you import a model into Mechanical.

Figure

Figures allow you to:

- Preserve different ways of viewing an object (viewpoints and settings).
- Define illustrations and captions for a report.
- Capture result contours, mesh previews, environment annotations etc., for later display in Report (p. 174).

Clicking the **Figure** button in the Home Tab (p. 40) creates a new **Figure** object inside the selected object in the Outline (p. 105). Any object that displays 3D graphics may contain figures. The new figure object copies all current view settings, including those defined in the Options (p. 217), such as the **Font Magnification Factor**, and gets focus in the **Outline** automatically.

View settings maintained by a figure include:

- Camera settings
- Result settings
- Legend configuration

A figure's view settings are fully independent from the global view settings. Global view settings are maintained independently of figures.

Figure Behaviors

- If you select a figure after selecting its parent in the Outline, the Geometry window transforms to the figure's stored view settings automatically (e.g. the graphics may automatically pan/zoom/rotate).
- If you change the view while a figure is selected in the Outline, the figure's view settings are updated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you reselect the figure's parent in the Outline, the Geometry window resumes the global view settings. That is, figure view settings override but do not change global view settings.
- Figures always display the data of their parent object. For example, following a geometry Update and Solve, a result and its figures display different information but reuse the existing view and graphics options. Figures may be moved or copied among objects in the Outline to display different information from the same view with the same settings.
- You may delete a figure without affecting its parent object. Deleting a parent object deletes all figures (and other children).
- In the Outline (p. 105), the name of a figure defaults to simply **Figure** appended by a number as needed.
- You may enter a caption for a figure as a string in the figure's details. It is your responsibility to maintain custom captions when copying figures.
- For a result object that includes one or more **Figure** objects, if you clear (**Clear Generated Data** option) the parent object's data or re-solve the analysis, the application also clears any result settings of the child **Figure** objects. The application does maintain Camera settings and legend configurations, as noted above, such as the last viewing setting.

Hotkeys and Keyboard Shortcuts

Mechanical supports a variety of hotkeys and hotkey combinations that enable you to quickly execute actions. For example, the popular hotkey "N" is used to create Named Selections once you select a geometric or mesh entity.

Hotkey and keyboard actions are described in the following topics:

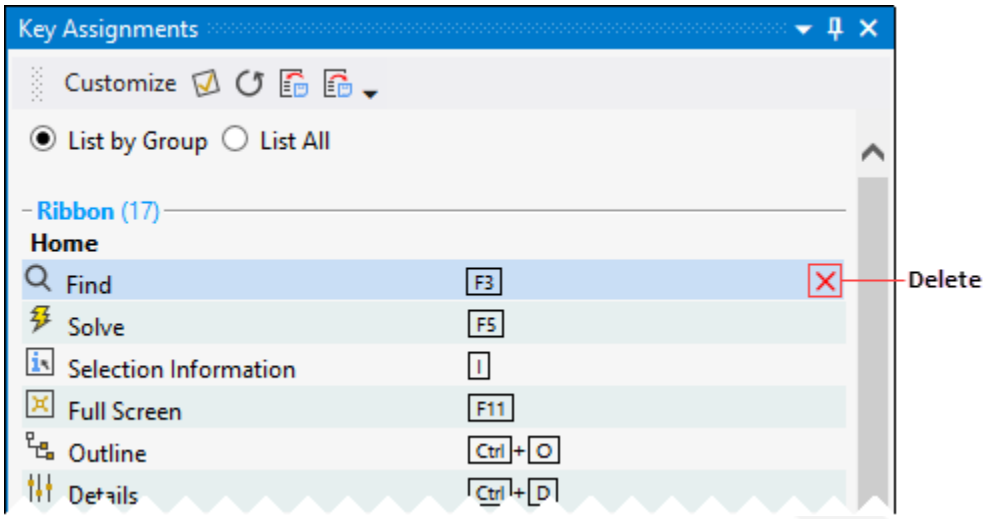
- Key Assignment Pane (p. 314)
- Ribbon Shortcuts (p. 315)
- Hotkeys and Hotkey Combination Descriptions (p. 316)
- Keyboard Number Pad Shortcuts (p. 320)

Key Assignment Pane

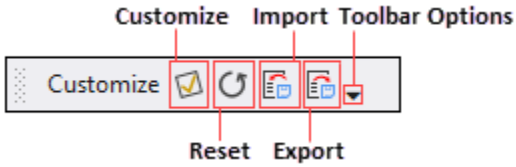
You use the **Key Assignments** pane, illustrated below, to view application assigned hotkeys. The option to display this window is available in the **Tools** (p. 45) group of the **Home** tab. Hotkeys are listed either by groups or you can list all available key assignments.

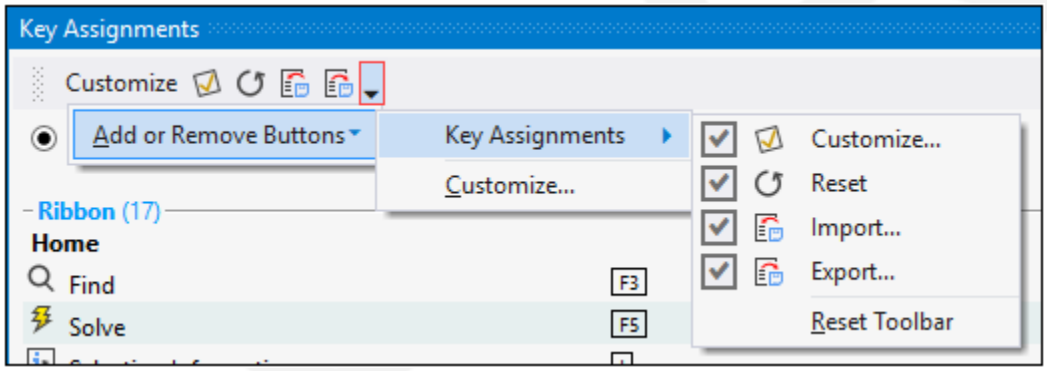
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



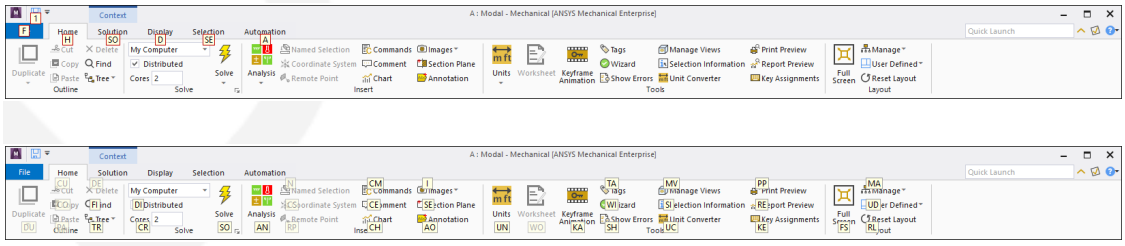
This window also provides a toolbar that includes 1) an option that opens the **Customize** (p. 143) dialog that enables you to modify hotkeys, 2) a reset option, 3) options to Import and Export user-defined key assignments, 4) a drop-down menu you can use to display or hide default options.





Ribbon Shortcuts

In addition to the options of the **Key Assignments** window, selecting the **[Alt]** key displays keyboard selection options. As illustrated below, when you select the **[Alt]** key, letters for each tab display. When you select one of these letters, additional options also display for the given tab.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Hotkeys and Hotkey Combination Descriptions

Application default hotkeys and their actions are described below.

Selection Filters

Ctrl+**P**: Activate **Vertex** selection.

Ctrl+**E**: Activate **Edge** selection.

Ctrl+**F**: Activate **Face** selection.

Ctrl+**B**: Activate **Body** selection.

Ctrl+**M**: Activate option to merge virtual topologies. Perform this key combination once you have selected one or more edges in order to merge adjacent faces into one or more virtual face(s) or select one or more faces to merge the faces into one or more virtual face(s).

Ctrl+**N**: Activate **Node** selection.

Ctrl+**K**: Activate **Element Face** selection.

Graphical Actions

G: When the **Geometry** window has focus, this key displays the content of the Graph (p. 194) window in the **Geometry** window. This feature is also available from the Graphics Toolbar (p. 97) and a context (right-click) menu.

H: When the **Geometry** window has focus, and you have modified the view of your model, for example, examining a part up-close, this key returns the view to the application default, Isometric View.

I: When the **Geometry** window has focus, this key toggles the display of the Selection Information (p. 180) Window on/off.

K: Create a local probe at the largest result value.

L: Create a local probe at the smallest result value.

M: When the **Geometry** window has focus, this key opens the Select Mesh By ID (p. 279) dialog box.

N: When the **Geometry** window has focus, this key opens the Named Selection (p. 1057) dialog window, enabling you to create a named Selection for the selected geometric or mesh entity.

Z: When the **Geometry** window has focus, this key zooms in on the selected entity (body, face, edge, vertex). If no geometric entity is selected in the **Geometry** window, this key acts the same as the **F7** (Zoom to Fit) option.

Ctrl+**A**: Selects all entities based on the active selection filter (bodies, faces, edges, vertices, nodes).

Ctrl+**C**: Copy the content of the graphics window into the clipboard. The content is available to paste (**Ctrl** + **V**) into a compatible application. Default image resolution settings can be changed using the **Graphics Resolution** option under the **Graphics** (p. 228) preference.

Ctrl+**D**: Display/hide the Details pane.

F4: This key enables you to move a vertex or a node to a new location. When you are using the Node Move feature, hold this key to reposition the desired node if you want to relax the node and move the node away from the geometry it is associated with.

F6: Toggle between the **Shaded Exterior and Edges**, **Shaded Exterior**, and **Wireframe** views.

F7: Executes the **Zoom to Fit** option (also available on the Graphics Toolbar (p. 97)).

F8: Hide selected faces.

F9: Hide selected bodies in **Geometry** window and hide/show selected bodies from the tree view.

F10: Maximize/Minimize the Ribbon.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Ctrl+**L**: Activate **Element** selection.

Ctrl+**Q**: This option adds your current clipboard (p. 104) geometry or mesh selection to the existing geometry or mesh selection(s).

Ctrl+**R**: This option clears all clipboard (p. 104) selections

Ctrl+**W**: This option removes your current clipboard (p. 104) geometry or mesh selection from the existing geometry or mesh selection(s).

Shift+**F1**: Activate **Extend to Adjacent** selection.

Shift+**F2**: Activate **Extend to Limits** selection.

Shift+**F3**: Activate

F11: When the **Geometry** window has focus, this key toggles the full screen display on/off.

Shift+**F8**: Show Hidden Face(s).

Ctrl+**F9**: Hide all other bodies.

Shift+**F9**: Show all bodies.

Arrow Keys: Pan your model in the Geometry (p. 129) window. The window needs to be the active interface element.

Ctrl+**Arrow Keys**: Rotate the model about the vertical or horizontal axis of the model's orientation in the **Geometry** window.

Shift+**[1]**: Turn the **Anti-Aliasing (MSAA)** preference **Off**.

Shift+**[2]**: Set the **Anti-Aliasing (MSAA)** preference to **Low (2x)**.

Shift+**[3]**: Set the **Anti-Aliasing (MSAA)** preference to **Medium (4x)**.

Shift+**[4]**: Set the **Anti-Aliasing (MSAA)** preference to **High (8x)** (default).

Shift+**W**: Increases the number of layers displayed on a mesh that includes poor quality elements. See the Check Mesh Quality section for more information.

Shift+**S**: Decreases the number of layers displayed on a mesh that includes poor quality elements. See the Check Mesh Quality section for more information.

Outline Pane Actions

These hotkeys require an object in the Outline pane to be selected or that the pane has focus:

F1: Open the Mechanical User's Guide to a section associated with the selected object.

F2: Rename a selected object.

F3: Open the search (Find In Tree) dialog box.

Ctrl+**G**: Group objects.

Shift+**Ctrl**+**G**: Ungroup objects.

Ctrl+**O**: Display/hide the Outline pane.

Ctrl+**S**: Save the project.

Solution/Retrieve Results

The **F5** key starts the solution process for a selected Environment or if you have an object selected in the tree that is above all of the properly defined environments, then all branches will be solved. This action is also available when the **Graphics** window has focus.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Extend to Connection selection.

Shift+**F4**: Activate **Extend to Instances** selection.

Ctrl+**U**: Retrieve current result data when you are using Result Plot Trackers.

Go To Options

Ctrl+**1**: Highlight the corresponding Bodies in tree.

Ctrl+**2**. Highlight the contacts for the selected bodies.

Ctrl+**3**: Highlight the contacts common to the selected Bodies

Ctrl+**4**: Highlight the corresponding Named Selections object in the tree.

Ctrl+**5**: Highlight all Named Selections common the to given Named Selection.

Ctrl+**6**: Highlight the Mesh Controls in the tree that are associated with the selected bodies.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Animation Actions

The following hotkeys are available for the animation feature when the **Geometry** window has focus.

P: Plays and pauses the animation.

S: Stops the animation.

Shift+**Right/Left Arrow Key**: Reverse or advance the animation frame by frame.

Note:

For the **Standard Animation (SA)** setting, you can abort the animation creation process using the **ESC** key.

Use a Macro

Ctrl+**F7**: Display a Open dialog to

DRAFT

select and then run a macro file.

Note:

When you place your cursor within the **Geometry** window, and given a generated mesh, and the **Node** or **Element** selection option is active, holding the right mouse button and then clicking the left mouse button scrolls through the available **Select Mode** options (single section, box selection, box volume, lasso, lasso volume).

Keyboard Number Pad Shortcuts

Certain display functionalities are also available via the number pad on your keyboard provided the **NumLock** key is enabled. The numbers correlate to the following functionality:

0 = View Isometric

1 = +Z Front

2 = -Y Bottom

3 =+X Right

4 = Previous View

5 = Default Isometric

6 = Next View

7 = -X Left

8 = +Y Top

9 = -Z Back

. (dot) = Set Isometric

7 -X Left	8 +Y Top	9 -Z Back
4 ← Previous	5 Default Isometric	6 Next →
1 +Z Front	2 -Y Bottom	3 +X Right
0 View Isometric		. Set Isometric

Wizards

Wizards provide a layer of assistance above the standard user interface. They are made up of tasks or steps that help you interpret and work with simulations. Conceptually, the wizards act as an agent between you and the standard user interface.

Wizards include the following features:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- An interactive checklist for accomplishing a specific goal
- A reality check of the current simulation
- A list of a variety of high-level tasks, and guidance in performing the tasks
- Links to useful resources
- A series of Callout windows which provide guidance for each step

Note:

Callouts close automatically, or you may click inside a Callout to close it.

Wizards use hyperlinks (versus command buttons) because they generally represent links to locations within the standard user interface, to content in the help system, or to a location accessible by a standard HTML hyperlink. The status of each step is taken in context of the currently selected Outline (p. 105) object. Status is continually refreshed based on the Outline state (not on an internal wizard state). As a result you may:

- Freely move about the Outline (p. 105) (including between branches).
- Make arbitrary edits without going through the wizards.
- Show or hide the wizards at any time.

Wizards are docked to the right side of the standard user interface for two reasons:

- The Outline (p. 105) sets the context for status determination. That is, the wizards interpret the Outline rather than control it. (The user interface uses a top-down left-right convention for expressing dependencies.)
- Visual symmetry is maintained.

To close wizards, click the ×. To show/hide tasks or steps, click the section header. Options for wizards are set in the Wizard (p. 243) section of the **Options** dialog box (p. 217) under the Mechanical application.

The Mechanical Wizard (p. 321) is available for your use in the Mechanical application.

Mechanical Wizard

Display the Mechanical Wizard by selecting the wizard button in the **Tools** (p. 45) group on the **Home** (p. 40) tab. You can close the wizard at any time by clicking the close button at the top of the panel. To show or hide the sections of steps in the wizard, click the section header.

Note:

The Mechanical Wizard is not supported on the Linux platform.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Features of the Mechanical Wizard

The Mechanical Wizard works like a web page consisting of collapsible groups and tasks. Click a group title to expand or collapse the group; click a task to activate the task.

When activated, a task navigates to a particular location in the user interface and displays a callout with a message about the status of the task and information on how to proceed. Activating a task may change your tab selection, cursor mode, and Outline (p. 105) selection as needed to set the proper context for proceeding with the task.

You may freely click tasks to explore the Mechanical application. Standard tasks WILL NOT change any information in your simulation.

Callouts close automatically based on your actions in the software. Click inside a callout to close it manually.

Most tasks indicate a status via the icon to the left of the task name. Rest your mouse on a task for a description of the status. Each task updates its status and behavior based on the current Outline (p. 105) selection and software status.

Tasks are *optional*. If you already know how to perform an operation, you don't need to activate the task.

Click the **Choose Wizard** task at the top of the Mechanical Wizard to change the wizard goal. For example, you may change the goal from **Find safety factors** to **Find fatigue life**. Changing the wizard goal *does not* modify your simulation.

At your discretion, simulations may include any available feature not covered under **Required Steps** for a wizard. The Mechanical Wizard does not restrict your use of the Mechanical application.

You may use the Mechanical Wizard with databases from previous versions of the Mechanical application.

Types of the Mechanical Wizards

There are wizards that guide you through the following simulations:

- Safety factors, stresses and deformation
- Fatigue life and safety factor
- Natural frequencies and mode shapes
- Optimizing the shape of a part
- Heat transfer and temperatures
- Magnetostatic results
- Contact region type and formulation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Steps for Using the Mechanical Application

This section describes the overall workflow involved when performing any analysis in the Mechanical application. The following workflow steps are described:

- Create Analysis System
- Define Materials
- Attach Geometry/Mesh
- Define Part Behavior
- Create a Simulation Template
- Create a Geometry in Mechanical
- Define Substructures
- Define Connections
- Apply Mesh Controls and Preview Mesh
- Establish Analysis Settings
- Define Initial Conditions
- Apply Pre-Stress Effects for Implicit Analysis
- Apply Loads and Supports
- Perform Solution
- Review Results
- Create Report (optional)

Create Analysis System

There are multiple ways to open Ansys Mechanical and specify a desired analysis system.

Important

Unit System Behavior: Whenever you open a project from a previous session, the unit system specified in that session automatically becomes the active system in the new Mechanical session. A good habit is to verify the active unit system whenever you open a project.

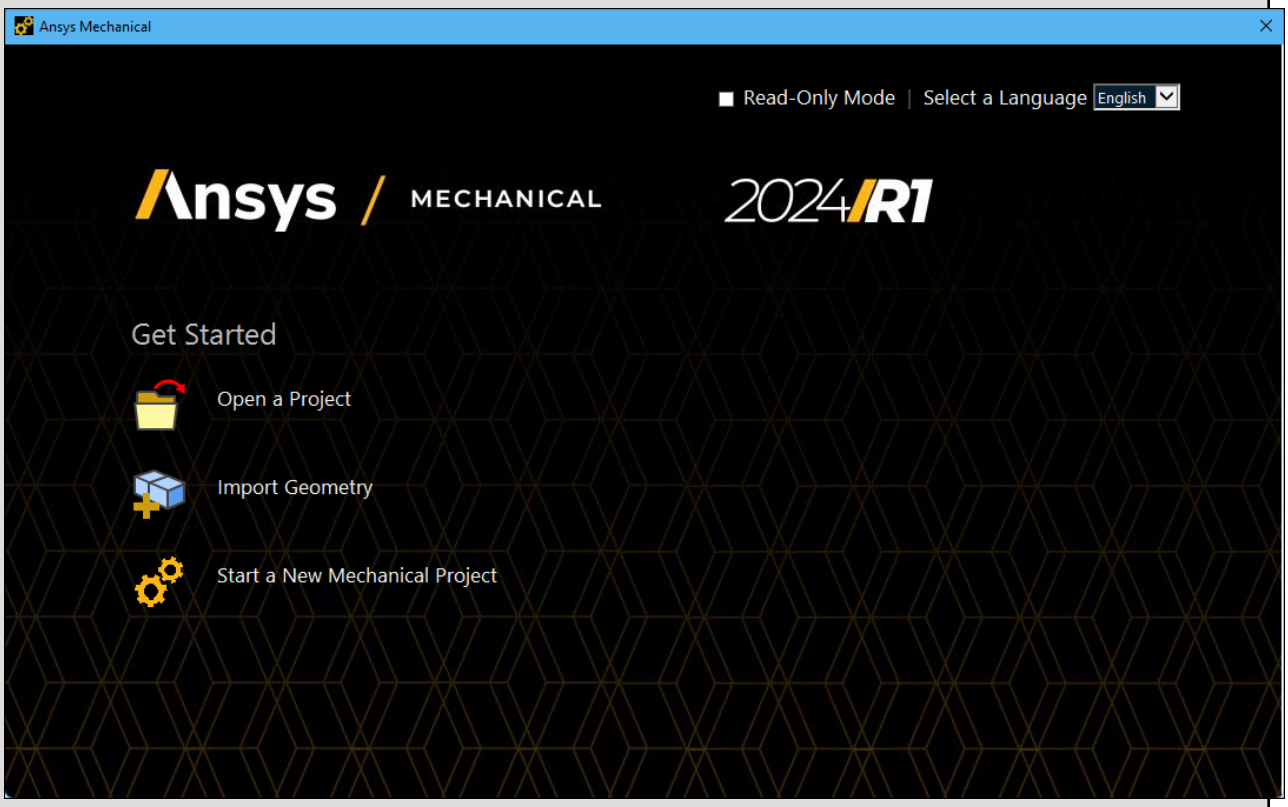
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Opening Mechanical from the Start Menu

To open Mechanical independently of Workbench and create an analysis system from within the application:

1. Open the **Ansys 2024 R1** menu of the **Start Menu** and select **Mechanical 2024 R1**.

2. Open an existing project, import a geometry, or open a new Mechanical project without a geometry. Supported project file types include `*.mechdb`, `*.mechpz`, `*.dsdb`, and `*.mechdat`. When opening Mechanical independently, `*.mechdb` is the default file type and `*.mechpz` is the archived file for this type.



Note:

The above dialog is the first instance of the pane for a newly installed version of the application. The color scheme of this dialog can be changed using the Options (p. 217) dialog preferences of the Style (p. 244) category of the **UI Options** group, you can change the color theme, from Dark to Light, or Classic.

3. If you do not open an existing project, select a desired analysis type from the **Analysis** drop-down menu of the Home (p. 42) Context tab (**Insert** group) for your new project.

Important:

Following a new installation of the product, if you open Mechanical independently, preferences from your previous installation are not automatically migrated. More im-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

portantly, when you subsequently open the Workbench application, preferences from your previous installation are automatically migrated. As a result, the application overwrites any options or preferences that you applied when previously opening Mechanical independently.

Opening Mechanical from Ansys Workbench

See the Working Through a System section of the Workbench User's Guide for the steps and background to create an analysis system in Workbench and open Mechanical. That section outlines the necessary workflow and supported analysis types.

Geometry Not Required

When configuring your analysis in Workbench, it is not necessary that you include geometry. You can simply skip the step of specifying a geometry and open Mechanical by double-clicking a supported cell, such as the **Model** cell, or right-clicking on a supported cell and selecting the **Edit** option. This can be done to create a template (p. 346) for use with multiple geometries.

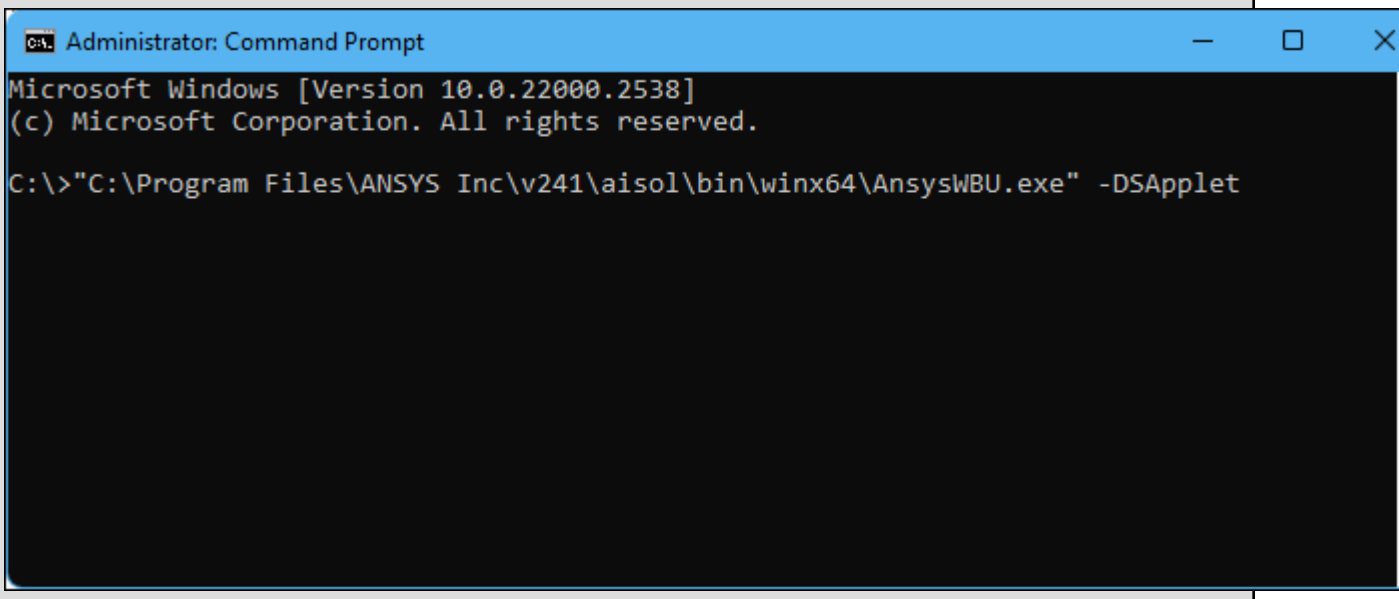
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Opening Mechanical from the Command Line on Windows

1. Open the **Command Prompt** window from the Windows **Start** menu.

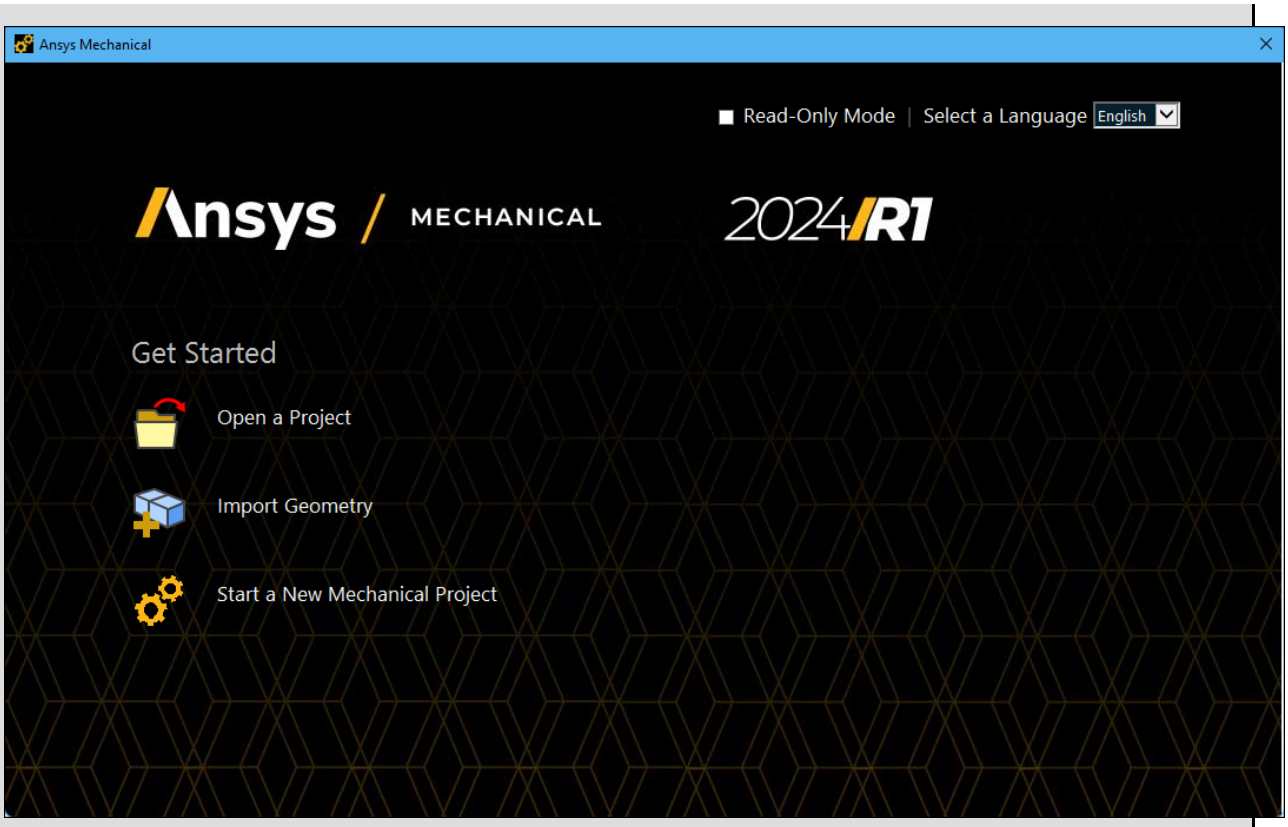
2. Enter (copy and paste) the following in the **Command Prompt** pane and press Enter to run the command. The example command has a file path of a typical installation location. Change the details of the file path to the specifics of your installation directory and add arguments as desired.

```
"C:\Program Files\ANSYS Inc\v241\aisol\bin\winx64\AnsysWBU.exe" -DSApplet
```



3. Open an existing project, import a geometry, or open a new Mechanical project without a geometry. Supported project file types include `*.mechdb`, `*.mechpz`, `*.dsdb`, and `*.mechdat`. When opening Mechanical independently, `*.mechdb` is the default file type and `*.mechpz` is the archived file for this type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

- Standalone supports the following project file types: `*.mechdb`, `*.dsdb`, and `*.mechdat`. The `*.mechdb` file type is new and unique to the standalone application.
- Once you select an option, this window disappears.

4. If you do not open an existing project, select a desired analysis type from the **Analysis** drop-down menu of the Home (p. 42) Context tab (**Insert** group) for your new project.

Command Line Arguments

Supported Command Line arguments include the following. These arguments are case **Insensitive**.

```
-AppModeMech                        to skip Start Page
-script "script_filename"           to run a script from a file
-file "filename"                    to open a Mechanical database or import a geometry file (see examples)
-b                                  to run script in batch mode
-x                                  to run script with auto exit
-ScriptArgs                         to pass arguments from the Command Prompt to the script
-readonly                           to open Mechanical in read-only mode
-language fr                        to specify the displaying language as French. Similarly, set it to en-us fo
                                        ja for Japanese (Windows platform only), and zh for Chinese (Windows pl
```

Script Examples

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech -script "D:\script_example1.py"
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech -b -script "D:\script_example1.py"
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech –x -script "D:\script_example1.py""
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech -file "D:\script_example1.mechdb"
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech -file "D:\Example1.agdb" -script "D:\scrip
"%AWP_ROOT241%\aisol\bin\winx64\AnsysWBU.exe" -DSApplet -AppModeMech -script "D:\script_example1.py -ScriptArgs
  for this example, the script file script_example1.py can use the ScriptArgs by referencing args.
```

Opening Mechanical from the Command on Linux

Execute the following command in terminal from home directory to launch a Linux session:

```
./ansys_inc/v241/aisol/.workbench -DSApplet
```

Define Materials

Go to a section topic:

- Define Materials (p. 328)
- Overview of Part Behavior Based on Material Properties (p. 328)
- Material Dependent Damping Definition (Engineering Data and Mechanical) (p. 329)

Define Materials

You must use the Engineering Data Workspace to define and modify material properties as well as import material libraries. Typically, you access this workspace from the **Engineering Data** cell of a Mechanical analysis system included in the Project Schematic of Workbench.

Import Materials

For the Windows platform, when you open Mechanical, either from Workbench or independently, you can import materials or material libraries that are available in Engineering Data using the Assignment (p. 345) property of part and body objects.

In addition, when you open Mechanical independently, the Materials object provides the context (right-click) menu option **Import Materials** that enables you to import material files in `.xml` format.

Note:

Any changes you make to the local material libraries in Engineering Data, including material property changes, library additions, etc., will be reflected in Mechanical, whether opened through Workbench or independently once you have closed and reopened the applications.

Overview of Part Behavior Based on Material Properties

A part's response is determined by the material properties assigned to the part.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Depending on the application, material properties can be linear or nonlinear, as well as temperature-dependent.
- *Linear* material properties can be constant or temperature-dependent, and isotropic or orthotropic.
- Nonlinear material properties are usually tabular data, such as plasticity data (stress-strain curves for different hardening laws), Hyperelastic material data.
- To define temperature-dependent material properties, you must input data to define a property-versus-temperature graph.
- To define material-based damping properties, you must specify data in the Material Depending Damping property group. Mechanical supports material-based damping in addition to damping specified in the application. See below for a listing of the analysis types (p. 329) that support material-based damping.
- Although you can define material properties separately for each analysis, you have the option of adding your materials to a material library by using the **Engineering Data** tab. This enables quick access to and re-use of material data in multiple analyses.
- For all orthotropic material properties, by default, the Global Coordinate System is used when you apply properties to a part in the Mechanical application. If desired, you can also apply a local coordinate system (p. 1191) to the part.

Material Dependent Damping Definition (Engineering Data and Mechanical)

In the Engineering Data workspace, you can specify Material Depending Damping using the **Damping Ratio** and the **Constant Structural Damping Coefficient** properties.

These **Material Dependent Damping** properties have the same name as the properties in the Damping Controls (p. 1462) of Mechanical. The **Damping Ratio** property in Engineering Data generates the command **MP**,**DMPR**. In Mechanical, the **Damping Ratio** property generates the command **DMPRAT**. Similarly, the **Constant Structural Damping Coefficient** property defined in Engineering Data generates the command **MP**,**DMPS** and if defined in Mechanical, the property generates the command **DMPSTR**. The solver supports the use of these commands in combination or individually.

The following tables list the analysis types that support material-based damping defined in Engineering Data. Note that some analyses require specific settings or conditions in order to support material damping definitions. The Yes entries below indicate which command is written to the input file or whether both commands are written to the input file, based on certain settings/conditions.

	Modal (without Damping)	**Modal (Full Damped)**	**Modal (Reduced Damped)**
Damping Ratio (**MP**,**DMPR**)	Yes	No	No
Constant Structural Damping Coefficient (**MP**,**DMPS**)	No	Yes	Yes

	Harmonic Response (MSUP)	**Transient (MSUP)**
Damping Ratio (**MP**,**DMPR**)	Yes[a]	Yes[a]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	Harmonic Response (MSUP)	Transient (MSUP)
Constant Structural Damping Coefficient (**MP**,**DMPS**)	Yes[b]	Yes[b]

[a] This analysis type requires that you set the **Damped** property to **No** in the **Analysis Settings** of the upstream Modal Analysis. The **MP**,**DMPR** command is not sent again in the linked MSUP Harmonic/MSUP Transient analysis, as the application defines **Damping Ratio** (**MP**,**DMPR**) in the upstream Modal analysis. For a stand-alone MSUP Harmonic Response analysis, you define the **Damping Ratio** accordingly in the analysis.

[b] This analysis type requires that you set the **Damped** property to **Yes** in the **Analysis Settings** of the upstream Modal Analysis. The **MP**,**DMPS** command is not sent again in the linked MSUP Harmonic/MSUP Transient analysis, as the application defines the **Constant Structural Damping Coefficient** (**MP**,**DMPS**) in the upstream Modal analysis. For a stand-alone Full Harmonic Response analysis, you define the Constant Structural Damping Coefficient (MP,DMPS) accordingly in the analysis.

	Harmonic Response (Full)	Transient (Full)
Damping Ratio (**MP**,**DMPR**)	No	No
Constant Structural Damping Coefficient (**MP**,**DMPS**)	Yes[b]	Yes[b]

	Response Spectrum	Random Vibration
Damping Ratio (**MP**,**DMPR**)	Yes[a]	Yes[a]
Constant Structural Damping Coefficient (**MP**,**DMPS**)	No	No

[a] The application defines **Damping Ratio** (**MP**,**DMPR**) in the upstream Modal analysis. Therefore, there is no need redefine in this downstream analysis.

Attach Geometry/Mesh

There are several methods to open a geometry or mesh in the Mechanical application. Generally, you specify your geometry when you specify your analysis system on the Workbench **Project Schematic**. However, Mechanical also enables you to open the application *without* first specifying a geometry.

The following sections describe geometry and mesh import options as well as CAD related options and terminology:

Importing Geometry or Mesh from Workbench

Importing Geometries or Mesh from Mechanical

Updating Geometry from Within the Mechanical Application

CAD Interface Terminology

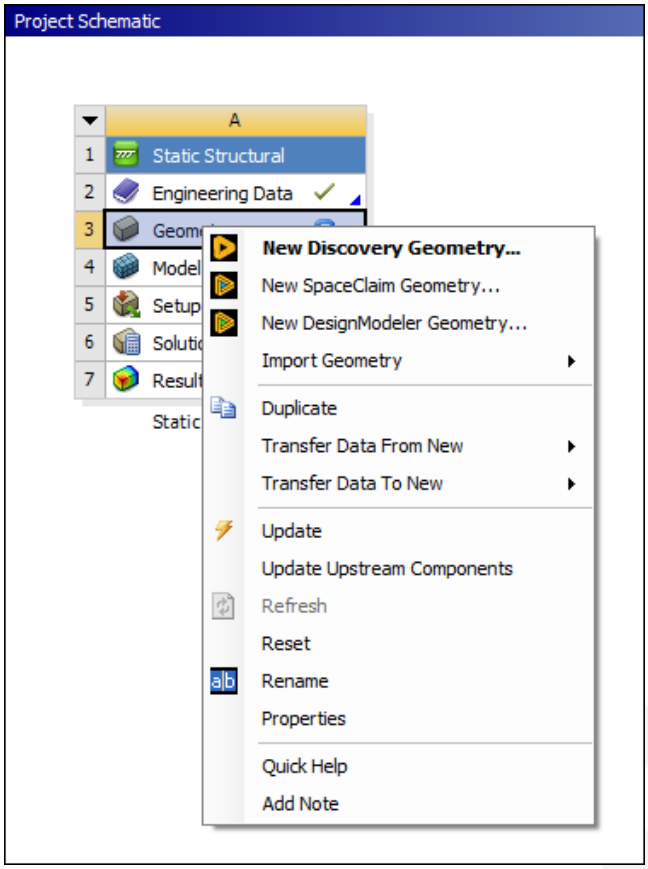
Importing Geometry or Mesh from Workbench

From Workbench, you open a geometry or mesh file as illustrated here:

Importing Geometry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

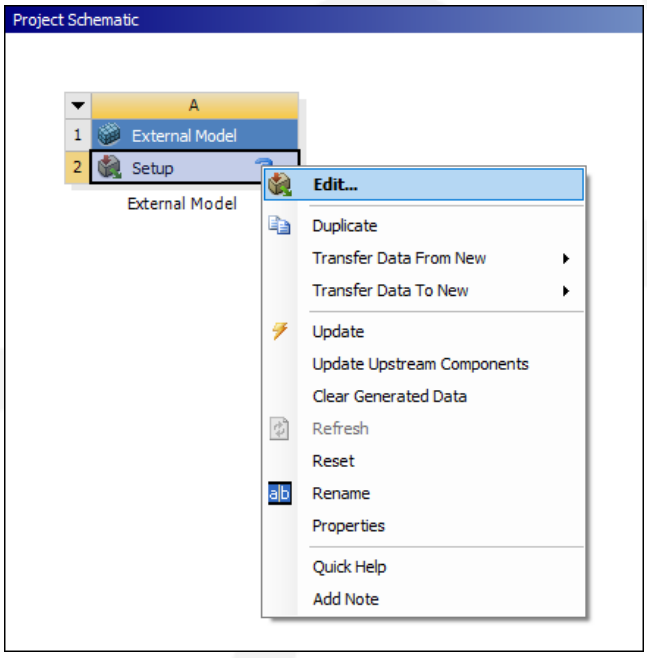
Published: 2023-10-23



Once you have inserted a Mechanical Model system or an analysis system, right-click the **Geometry** cell. The menu options enable you to:

- Open the Discovery application and specify your model.
- Open the SpaceClaim application and specify your model.
- Open the DesignModeler application and specify your model.
- Import a geometry from a supported CAD System (p. 2351) file using the **Import Geometry** option.

Importing Mesh

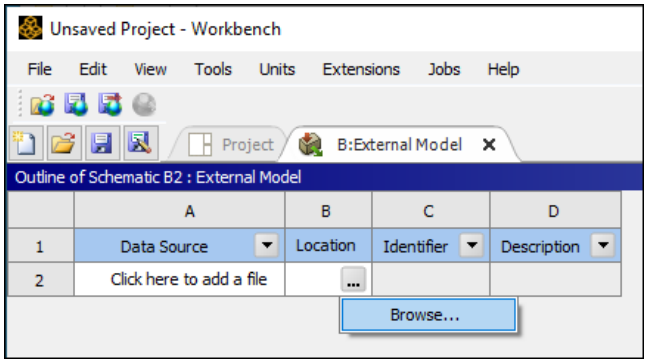


You can import mesh files using the **External Model** Component System.

1. Highlight the **Setup** cell and select **Edit**.
2. On the **External Model** tab, browse to your mesh file.

This feature imports Mechanical APDL common database (.cdb), Abaqus Input (.inp), NASTRAN Bulk Data (.bdf, .dat, .nas), Fluent Input (.msh, .cas), and ICEM CFD Input (.uns) files. For more

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



information, see the Importing Mesh-Based Geometry (p. 908) section.

Tip:

You can drag and drop a supported geometry or mesh file directly onto the **Project Schematic**. Based on the file type, the application automatically creates a **Geometry** system or an **External Model** system linked to a **Mechanical Model** system.

Note:

- If you create a link between the **Solution** cell of an analysis system and the **Model** cell of a downstream system, the application automatically transfers the resulting (deformed) geometry from the upstream analysis. See the Geometry from Deformation Results (p. 1022) section of the Help for more information.
- The Electronic Computer-Aided Design (ECAD) files, `Ansys EDB`, `ODB++`, and `IPC2851`, can be opened using the **Geometry** cell of the analysis system.

Importing Geometries or Mesh from Mechanical

The steps you use to import geometry or external model (mesh) files depend upon how you have opened the application.

Go to a section topic:

- Mechanical Opened Independently
  - Import Geometry (p. 333)
  - Import External Model (Mesh) File (p. 334)
- Mechanical Opened from Workbench
  - Import Primary Source (p. 335)
  - Import Secondary Source (p. 336)

Material Assignment

By default, when you first import your model into Mechanical, any bodies that do not include material assignment are assigned the application's default material. If you subsequently update your geometry from the source application, Mechanical does not assign

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

default materials to new bodies. If the geometry update includes a new body without an assigned material, the body becomes underdefined and requires you to specify a material.

Mechanical Opened Independently

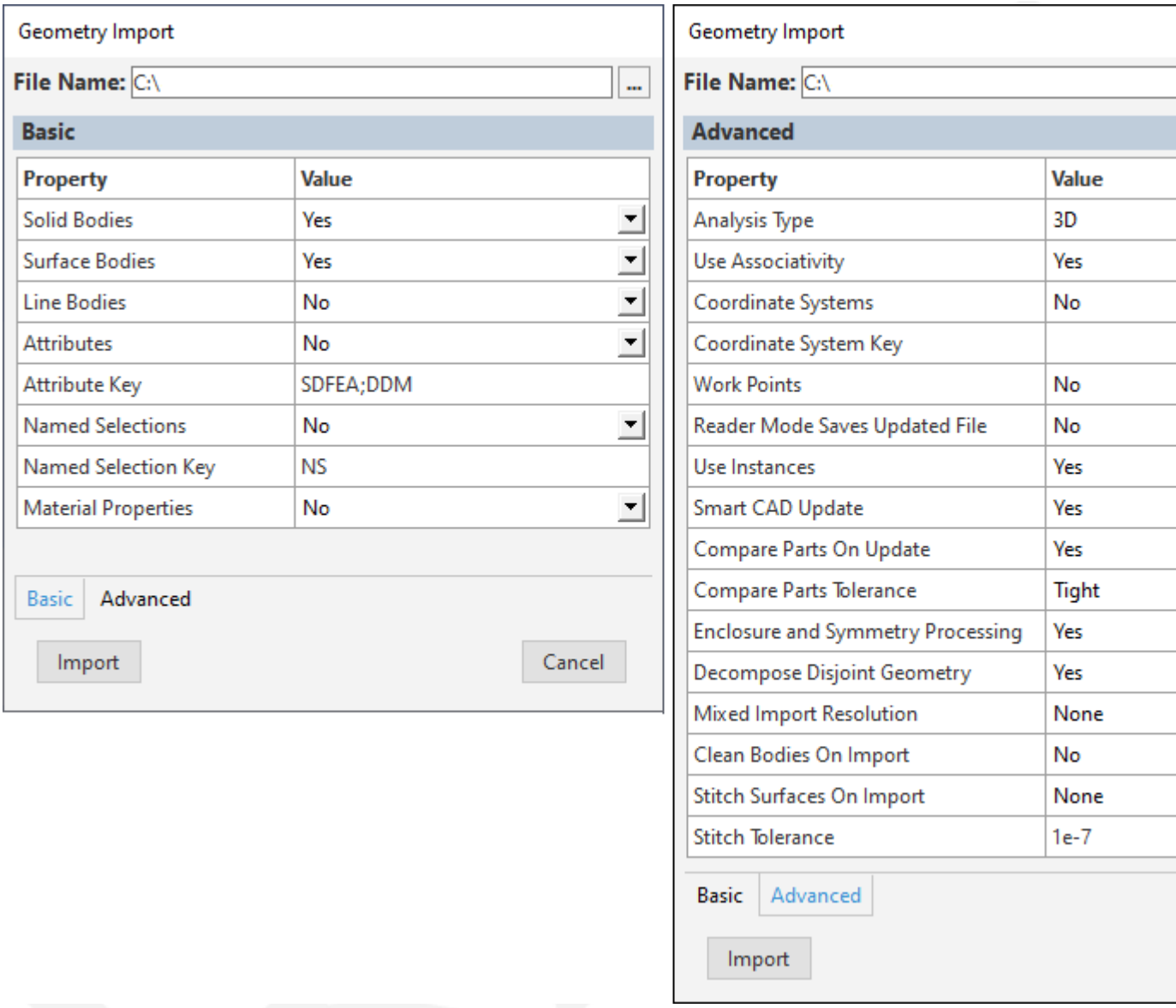
When you open Mechanical independently, you can import either geometry or an (external) model file.

Import Geometry

1. Select the **Geometry Imports** object.
2. Select the **Add Geometry Import** option from the **Geometry Imports** Context tab. The **Add Geometry Import** option includes a drop-down menu that lists recently imported files as well as the option **From File** that opens a dialog to select a file.
3. From the dialog, select a desired geometry file from the dialog.
4. The **Import Geometry** dialog displays automatically. This dialog includes two tabs: **Basic** and **Advanced**. Change property settings as needed and select the Import button.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The application automatically imports the selected geometry and adds a Geometry Import object to the Outline. The application also updates the Geometry object with the imported parts/bodies.

Import External Model (Mesh) File

1. Select the **Geometry Imports** object.
2. Select the **Add Model Import** option from the **Geometry Imports** Context tab. The **Add Geometry Import** option includes a drop-down menu that lists recently imported files as well as the option **From File** that opens a dialog to select a file.
3. From the dialog, select a desired Mechanical APDL common database (`.cdb`) file.

Important:

This is the only supported file type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Specify Details pane properties as needed and then select the **Generate** option from the context tab. See the External Model section in the *Workbench User's Guide* for descriptions of the Details properties.

Once generated, the application updates the **Geometry** object with the imported parts/bodies as well as any additional finite element data. See the Importing Mesh-Based Geometry (p. 908) for more information about imported data types.

Mechanical Opened from Workbench

When you open Mechanical without a geometry, use the Geometry Imports folder and the child **Geometry Import** object to import geometries into Mechanical.

Import Primary Source

The "primary source" is the geometry imported from Workbench. For convenience, it can now be attached directly from Mechanical. The primary source uses the current **Basic**/**Advanced Geometry** Preference settings for the **Geometry** cell in Workbench.

Note:

If you have already imported geometry, a secondary source geometry is added instead. See the description below.

Select the **Geometry Imports** folder, and, using the Context tab (p. 55) **Geometry Imports**, either:

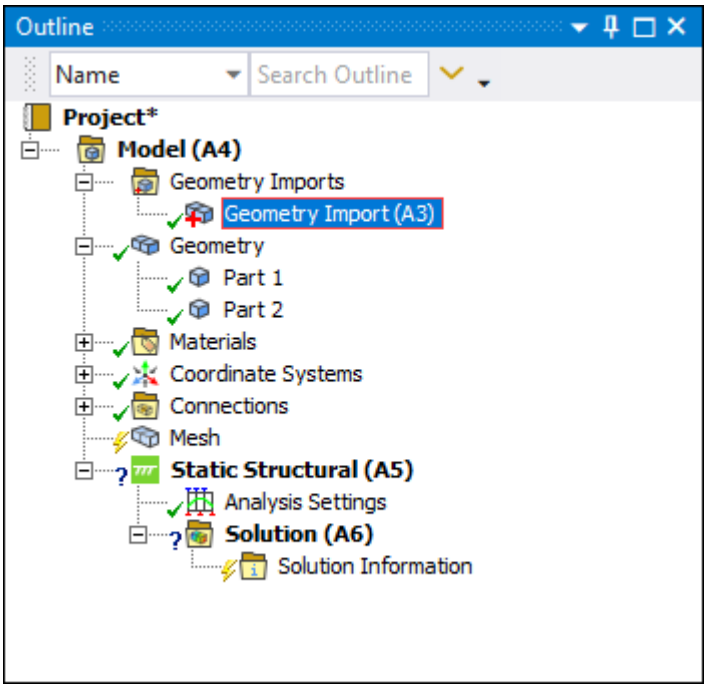
1. Click the **Import Geometry** option. A file selection dialog displays that enables you to select the desired file.

   Or...

2. Expand the drop-down list for the option to select a model from the list of recently imported geometries. For convenience, a **From File** option is also available.

If no other geometries are currently imported, the application automatically imports the selected geometry and updates the **Geometry Import (A3)** object. The application adds parts/bodies (for example, Part 1 and Part 2 shown below) as children to the **Geometry** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Import Secondary Sources

Non-primary sources are referred to as "secondary" geometries/models. These are imported into Mechanical separately. Separate preferences can be specified for each import.

Once you have a model imported into Mechanical, you can add additional models using the **Geometry Imports** features. Use the steps below to add models to your analysis.

1. Select the **Geometry Imports** object, and using the Context tab (p. 55) **Add Geometry Import** option, either:

   a. Select the option. A dialog displays that enables you to navigate to the desired file.

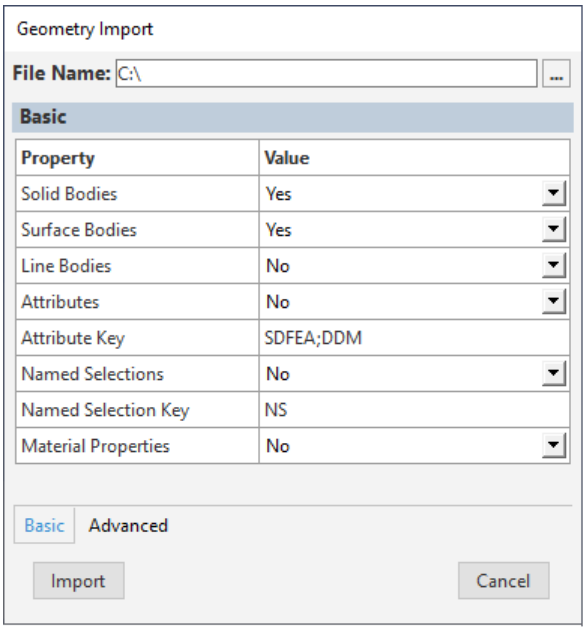
   Or...

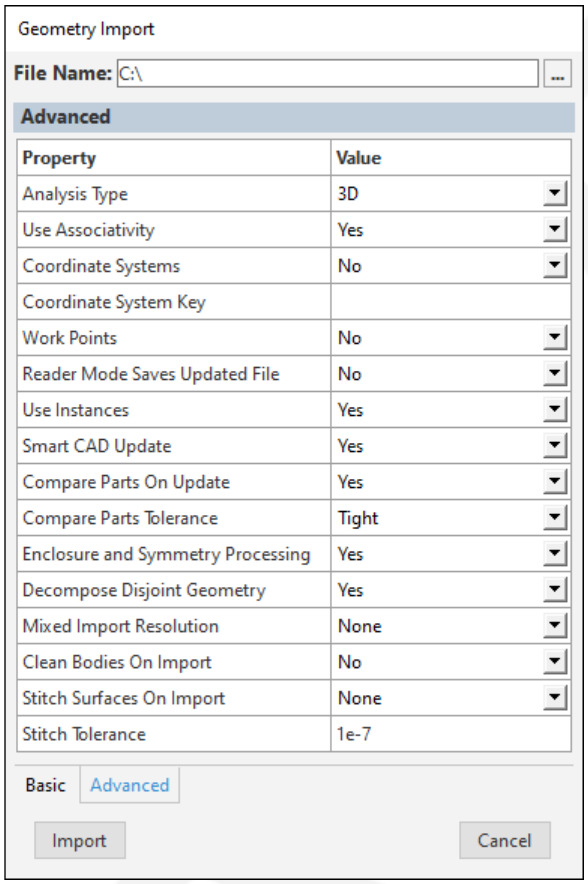
   b. Expand the drop-down list for the option to select a model from the list of recently imported geometries. For convenience, a **From File** option is also available.

2. As illustrated below, change the **Basic** and/or **Advanced** properties of the **Geometry Import** property pane. This property pane displays automatically.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

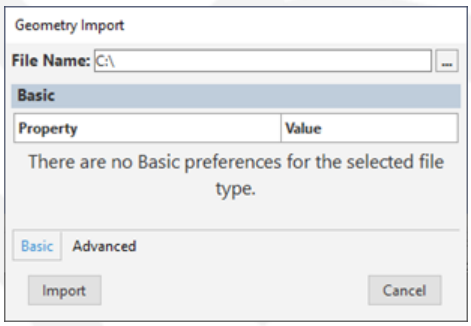
Published: 2023-10-23

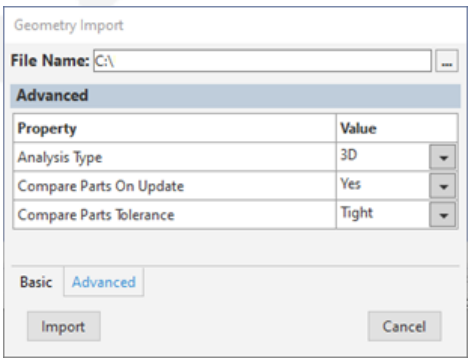




Note:

- Parameters are not supported for secondary imports.
- As illustrated, for DesignModeler and SpaceClaim files, the pane includes fewer properties.

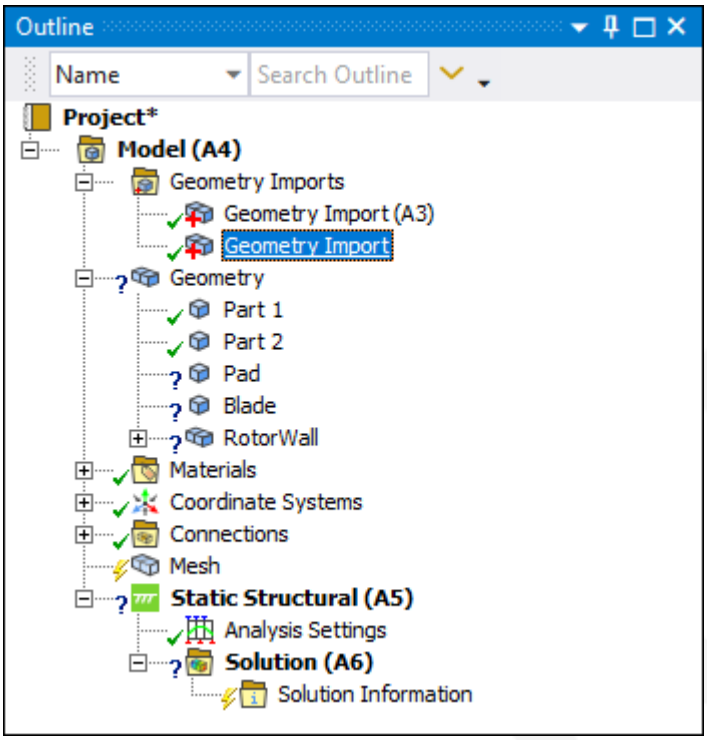




- In general, these are the same preferences that are available for the **Geometry** cell of an analysis system on the Workbench Project Schematic before you have specified a model to import. You should review the Geometry Preferences section of the *CAD Integration* documentation as well as the Geometry Import section of the *Workbench User's Guide* for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Select **Import**. A secondary **Geometry Import** object is automatically inserted into the outline, as illustrated below. The parts/bodies (shown below as Pad, Blade, and Rotor) are added as children to the **Geometry** object.



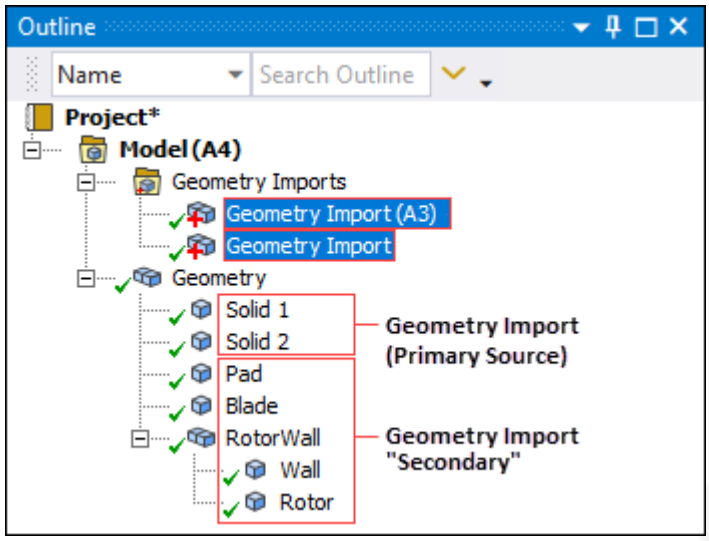
Important:

The application does not automatically assign materials to secondary imports. Note the "?" state for the above parts under the **Geometry** object. You need to manually specify a Material for the parts.

In the example below, you can see an example with parts and bodies under the **Geometry** object from multiple **Geometry Import** objects. Note that there is no separation or special treatment between parts/bodies from either Primary or Secondary sources.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Tip:

For convenience, you can display the context (right-click) menu on the **Geometry Import** objects and select the option **Go To** > **Corresponding Bodies in Tree**, to automatically select the parts/bodies for the import object. For CAD models, you can replace the geometry as desired using the Replace Geometry option on the Geometry Context tab.

Note:

Working with Model Assembly and Secondary Geometries

If you have an analysis that includes model assembly (p. 989), be sure to perform this operation before importing any secondary geometries. Existing secondary geometries are lost during the model assembly process.

Tip:

For Windows systems only, you can drag and drop a supported geometry or mesh file directly into the **Geometry** window. The application automatically attaches the model to the analysis system. For mesh files opened from Workbench, the application automatically inserts an upstream **External Model** system linked to your analysis system. Imported mesh files adopt the default unit system.

For CAD models, you can replace the primary and secondary source geometries using the **Reload** options.

Updating Geometry from Within the Mechanical Application

You can update all geometry by selecting the **Update Geometry from Source** context menu option, accessible by right-clicking on the **Geometry** tree object or anywhere in the **Geometry** window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The following update options are also available:

- Selective Update (p. 340)
- Smart CAD Update (p. 341)

Note:

The **Update Geometry from Source** option is not available if your analysis includes a secondary geometry.

Selective Update

Using the Geometry object right-click menu option **Update Selected Parts**>**Update: Use Geometry Parameter Values**, you can selectively update individual parts and synchronize the Mechanical model to the CAD model. This option reads the latest geometry and processes any other data (parameters, attributes, etc.) based on the current user preferences for that model.

Note:

Changes to either the number of turns or the thickness properties associated with a body do not update the CAD model.

This update feature only applies to part(s) that you select and other instances of the same part(s) that were previously imported. It does not import new parts added in the CAD system following the original import or last complete update. Assembly parameter values are always updated.

In addition, this feature is not a tool for removing parts from the Mechanical application tree, however; it will remove parts which have been selected for update in WB, but that no longer exist in the CAD model if an update is successful (if at least one valid part is updated).

The Update Selected Parts feature supports the associative geometry interfaces for:

- DesignModeler
- Autodesk Inventor
- CATIA V5
- Creo Elements/Direct Modeling
- Creo Parametric (formerly Pro/ENGINEER)
- Solid Edge
- NX
- SOLIDWORKS

With the exception of AutoCAD, executing the selective update feature on any unsupported interface will complete a full update of the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Smart CAD Update

Using the Geometry Preferences, you enable the **Smart CAD Update**. Note that Geometry Preferences are supported by a limited number of CAD packages. See the **Project Schematic Advanced Geometry Options** table for details.

CAD Interface Terminology

The CAD interfaces can be run in either plug-in mode or in reader mode.

- **Attaching geometry in plug-in mode:** requires that the CAD system be running.
- **Attaching geometry in reader mode:** does not require that the CAD system be running.

Additional Geometry Import Procedures

The following table describes additional options for specifying and importing geometry into the application based on certain conditions.

Procedure	Condition	Procedural Steps
Specifying geometry options.	Optional task that can be done before attaching geometry.	1. In an analysis system schematic, perform either of the following: • Right-click the **Geometry** cell and choose **Properties** OR • Select the **Geometry** cell for your analysis and select **View** > **Properties**. 2. Specify Geometry properties as desired.
Attaching SpaceClaim geometry and/or mesh to the Mechanical application.	You have created and generated a geometry and/or mesh in SpaceClaim.	Double-click the **Model** cell. Mechanical opens and displays the geometry and/or mesh.
	SpaceClaim is not running. Your geometry and/or mesh is stored in an `.scdoc` file.	1. Select the **Geometry** cell in an analysis system schematic. 2. Browse to the `.scdoc` file by right-clicking the **Geometry** cell and selecting **Import Geometry** > **Browse**. 3. Double-click the **Model** cell. Mechanical opens and displays the geometry and/or mesh.
Attaching DesignModeler geometry to the	You have created and generated a geometry in DesignModeler.	Double-click the **Model** cell. Mechanical opens and displays the geometry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Procedure	Condition	Procedural Steps
Mechanical application.	DesignModeler is not running. Your geometry is stored in an `.agdb` file.	1. Select the **Geometry** cell in an analysis system schematic. 2. Browse to the `.agdb` file by right-clicking the **Geometry** cell and selecting **Import Geometry** > **Browse**. 3. Double-click the **Model** cell. Mechanical opens and displays the geometry.
Attaching CAD geometry to the Mechanical application.	CAD system is running.	1. Select the **Geometry** cell in an analysis system schematic. 2. Right-click the **Geometry** cell listed to select geometry for import. 3. If required, set geometry options for import into the Mechanical application by highlighting the **Geometry** cell and choosing settings under **Preferences** in the Properties Panel. 4. Double-click the **Model** cell in the same analysis system schematic. The Mechanical application opens and displays the geometry.
	CAD system is not running. Geometry is stored in a native CAD system file, or in a CAD "neutral" file such as Parasolid or IGES.	1. Select the **Geometry** cell in an analysis system schematic. 2. Browse to the CAD file by right-clicking on the **Geometry** cell and selecting **Import Geometry**. 3. Double-click the **Model** cell in the **Project Schematic**. The Mechanical application opens and displays the geometry.
Open Mechanical without a geometry.	Specify the type of analysis you wish to perform.	Double-click the **Model** cell or right-click the cell and select **Edit**. Mechanical opens without a geometry. You can attach a geometry from within Mechanical as desired.

Define Part Behavior

After attaching geometry, you can access settings related to part behavior by right-clicking on the **Model** cell in the analysis system schematic and choosing **Edit**. The Mechanical application opens with the environment representing the analysis system displayed under the **Model** object in the tree.

An **Analysis Settings** object is added to the tree. See the Establish Analysis Settings (p. 349) overall step for details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

An **Initial Condition** object may also be added. See the Define Initial Conditions (p. 349) overall step for details.

The Mechanical application uses the specific analysis system as a basis for filtering or making available only components such as loads, supports and results that are compatible with the analysis. For example, a **Static Structural** analysis type will allow only structural loads and results to be available.

Presented below are various options provided in the Details view for parts and bodies following import.

Stiffness Behavior

In addition to making changes to the material properties of a part, you may designate a part's **Stiffness Behavior** as being flexible, rigid, as a gasket, and can specify a line body as a stiff beam, essentially making the body rigid.

- Setting a part's behavior as rigid essentially reduces the representation of the part to a single point mass thus significantly reducing the solution time.
- A rigid part will need only data about the density of the material to calculate mass characteristics. Note that if density is temperature dependent, density will be evaluated at the reference temperature (p. 344). For contact conditions, specify Young's modulus.
- Flexible and rigid behaviors are applicable only to static structural, transient structural, rigid dynamics, explicit dynamics, and modal analyses.
- Gaskets can be defined in one of two ways:
  1. By setting the **Stiffness Behavior** as **Gasket**. In this case, a Gasket Mesh Control will be added as a child of the gasket body in the model tree. You need to define the source face of the gasket in the Gasket Mesh Control to define the gasket material orientation.
  2. By setting the **Stiffness Behavior** as **Flexible**. In this case, you need to define a Gasket Mesh Control in the mesh folder. The gasket mesh control in the mesh folder can be applied to multiple bodies, so if there are many gasket bodies this option may be a more convenient approach to setting up the gaskets.
- Gasket Bodies (p. 853) are only applicable to static structural analyses. The **Material Assignment** of gasket bodies should reference an appropriate gasket material.

Flexible is the default **Stiffness Behavior**. To change, simply select **Rigid**, **Gasket**, or **Stiff Beam** (for a line body only) from the **Stiffness Behavior** drop-down menu. Also see the Rigid Bodies (p. 852), Gasket Bodies (p. 853), and/or Stiff Beam (p. 856) sections.

Note:

Rigid behavior is not available for the Samcef or ABAQUS solver.

Coordinate Systems

The **Coordinate Systems** object and its child object, **Global Coordinate System**, is automatically placed in the tree with a default location of 0, 0, 0, when a model is imported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For solid parts and bodies: by default, a part and any associated bodies use the Global Coordinate System. If desired, you can apply a apply a local coordinate system (p. 1191) to the part or body. When a local coordinate system is assigned to a Part, by default, the bodies also assume this coordinate system but you may modify the system on the bodies individually as desired.

For surface bodies, solid shell bodies, and line bodies: by default, these types of geometries generate coordinates systems on a per element type basis. It is necessary for you to create a local coordinate system and associated it with the parts and/or bodies using the Coordinate System setting in the Details view for the part/body if you wish to orient those elements in a specific direction.

Reference Temperature

The default reference temperature is taken from the environment (**By Environment**), which occurs when solving. This necessarily means that the reference temperature can change for different solutions. The reference temperature can also be specified for a body and will be constant for each solution (**By Body**). Selecting **By Body** will cause the **Reference Temperature Value** field to specify the reference temperature for the body. It is important to recognize that any value set **By Body** will only set the reference temperature of the body and not actually cause the body to exist at that temperature (unlike the **Environment Temperature** entry on an environment object, which does set the body's temperature).

Note:

Selecting **By Environment** can cause the body to exist at that temperature during the analysis but selecting **By Body** will only ever effect reference temperature. So if the environment temperature and the body have a different specification, thermal expansion effects can occur even if no other thermal loads are applied.

Note:

If the material density is temperature dependent, the mass that is displayed in the Details view will either be computed at the body temperature, or at 22°C. Therefore, the mass computed during solution can be different from the value shown, if the **Reference Temperature** is the **Environment**.

Note:

When nonlinear material effects are turned off, values for thermal conductivity, specific heat, and thermal expansion are retrieved at the reference temperature of the body when creating the Mechanical APDL solver input.

Reference Frame

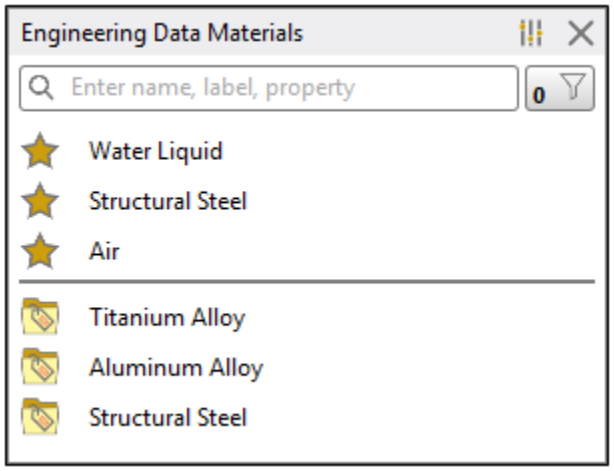
The Reference Frame determines the analysis treatment perspective of the body for an Explicit Dynamics analysis. The Reference Frame property is available for solid bodies when an Explicit Dynamics system is part of the solution. The valid values are **Langrangian** (default) and **Eulerian (Virtual)**. Eulerian is not a valid selection if **Stiffness Behavior** is set to **Rigid**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Material Assignment

Once you have attached your geometry, you can change the material assigned to the parts and bodies of your model. When you select a Part or Body object in the Outline, there is an **Assignment** property available in the Details view for each. This property provides a selectable fly-out menu that opens that opens the following **Engineering Data Materials** window. By default, this window lists the materials included in the Engineering Data favorites, as symbolized with the star icon as well as any other materials that you made available from the Engineering Data workspace, such as titanium and aluminum alloy show below. Selecting a material from this window assigns it to the currently selected part or body.



When you edit the currently assigned material, create a material, or import a material, you work in the Material Data tab. Once you have completed any of those operations, you must refresh the **Model** cell in the Project Schematic to bring new data into Mechanical.

Note:

- The **Assignment** property can be designated as a parameter.
- To model a gasket, the material assignment should reference a valid Gasket Material Model.

Nonlinear Material Effects

You can also choose to ignore any nonlinear effects from the material properties.

- By default the program will use all applicable material properties including nonlinear properties such as stress-strain curve data.
- Setting **Nonlinear Effects** to **No** will ignore any nonlinear properties only for that part.
- This option will allow you to assign the same material to two different parts but treat one of the parts as linear.
- This option is applicable only for static structural, transient structural, steady state thermal and transient thermal analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Thermal Strain Effects

For structural analyses, you can choose to have Workbench calculate a **Thermal Strain** (p. 2032) result by setting **Thermal Strain Effects** to **Yes**. Choosing this option enables the coefficient of thermal expansion to be sent to the solver.

Cross Section

When a line body is imported into the Mechanical application, the Details view displays the **Cross Section** field and associated cross section data. These read-only fields display the name and data assigned to the geometry in DesignModeler or the supported CAD system, if one was defined. See Line Bodies (p. 872) for further information.

Model Dimensions

When you attach your geometry or model, the model dimensions display in the Details Pane (p. 119) in the Bounding Box sections of the Geometry or Part objects. Dimensions have the following characteristics:

- Units are created in your CAD system.
- ACIS model units, if available during import and/or update, are used.
- Other geometry units are automatically detected and set.
- Assemblies must have all parts dimensioned in the same units.

Create a Simulation Template

You can open the Mechanical application without importing a geometry and specify any number of environmental conditions. To do so, select your desired analysis type in Workbench, select the **Model** cell, and open Mechanical. After you have defined all the desired aspects of your analysis, you can then save your simulation scenario. All without a geometry. This gives you the flexibility to simulate your scenario against different geometries.

The ability to define Worksheet-based Named Selections (p. 1061) in this situation has the most strategic benefit. Specifically, once you specify criterion-based Named Selections, you can scope a wide range of objects, such as, Remote Points, Coordinate Systems, loading conditions, support conditions, results, etc. Once you save your project, you can use with various models.

Limitations

Note the following conditions when importing a system that includes criteria-based Named Selections. For an existing template, if you import a system that contains a:

- Remote Point scoped to a criteria-based Named Selection, the Remote Point does may not update properly to include the location of the Named Selection. You can correct this condition by specifying an appropriate Coordinate System for the Remote Point instead of using the default Global Coordinate System setting.

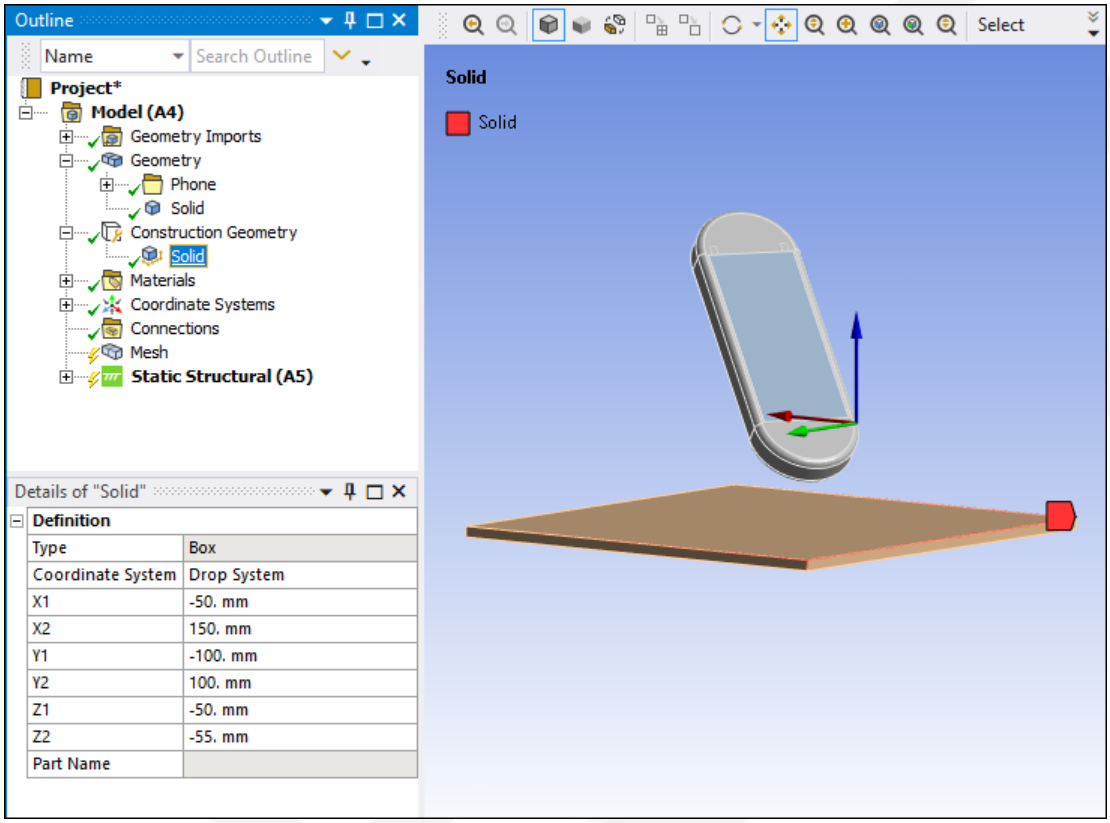
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- A loading condition scoped to criteria-based Named Selection and whose direction is defined by a **Vector**, the loading direction may not be updated after attaching the geometry. You can correct this condition by specifying the load by **Components**.

Create a Geometry in Mechanical

Using the **Construction Geometry** feature (**Solid** option), you can create and add a solid geometry within Mechanical. An example is illustrated below. The cellular telephone model is being dropped onto a solid body that was created in Mechanical. For more information, see the specifying a Solid (p. 892) section of the Help.



Define Substructures

Substructuring is a procedure that condenses a group of finite elements into one element represented as a matrix. The single-matrix element is called a superelement. You can use a superelement in an analysis as you would any other element type. By summarizing the mechanical behavior of the bodies, they can be assembled to synthesize the response of a more complex structure, or to model flexible bodies in the Rigid Dynamics solver.

See the Substructure Analysis (p. 781) section for more information about the use of this feature.

Supported Analysis Types

Substructuring currently supports the following analysis types:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Harmonic Response (p. 496) (MSUP only)
- Modal (p. 515)
- Random Vibration (p. 524)
- Response Spectrum (p. 533)
- Rigid Dynamics (p. 550)

Define Connections

Once you have addressed the material properties and part behavior of your model, you may need to apply connections to the bodies in the model so that they are connected as a unit in sustaining the applied loads for analysis. Available connection features are:

- **Contacts** (p. 1222): defines where two bodies are in contact or a user manually defines contact between two bodies.
- **Joints** (p. 1292): a contact condition in the application that is defined by a junction where bodies are joined together that has rotational and translational degrees of freedom.
- **Mesh Connections**: used to join the meshes of topologically disconnected surface bodies that reside in different parts.
- **Springs** (p. 1395): defines as an elastic element that connects two bodies or a body to "ground" that maintains its original shape once the specified forces are removed.
- **Bearings** (p. 1413): are used to confine relative motion and rotation of a rotating machinery part.
- **Beam Connections** (p. 1404): used to establish body to body or body to ground connections.
- **End Releases** (p. 1409) are used to release degrees of freedoms at a vertex shared by two or more edges of one or more line bodies.
- **Spot Welds** (p. 1407): connects individual surface body parts together to form surface body model assemblies.

Given the complex nature of bodies coming into contact with one another, especially if the bodies are in motion, it is recommended that you review the Connections (p. 1207) section of the documentation.

Apply Mesh Controls and Preview Mesh

Meshing is the process in which your geometry is spatially discretized into elements and nodes. This mesh along with material properties is used to mathematically represent the stiffness and mass distribution of your structure.

Your model is automatically meshed at solve time. The default element size is determined based on a number of factors including the overall model size, the proximity of other topologies, body curvature, and the complexity of the feature. If necessary, the fineness of the mesh is adjusted up to four times (eight times for an assembly) to achieve a successful mesh.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If desired, you can preview the mesh before solving. Mesh controls are available to assist you in fine tuning the mesh to your analysis. Refer to the Meshing Help for further details.

To preview the mesh in the Mechanical Application:

See the Previewing Surface Mesh section.

To apply global mesh settings in the Mechanical Application:

See the Global Mesh Controls section.

To apply mesh control tools on specific geometry in the Mechanical Application:

See the Local Mesh Controls section.

To use virtual topology:

All virtual topology operations in the Mechanical application are described in the Virtual Topology section of the Meshing Help.

Establish Analysis Settings

Each analysis system includes an Analysis Settings object. You use this object to define a variety of solution options, such as step controls, solution type, large deflection, as well as specific output data to include in the result file. The properties available in the **Analysis Settings** Details pane vary based on the analysis type, and their settings can be critical to achieving a complete and accurate solution.

See the Analysis Settings for Most Analysis Types (p. 1417) section for a complete listing and description of all available properties.

Define Initial Conditions

Based on your analysis type, Mechanical enables you to begin your analysis with an initial specification using one of the object types described below. Initial specifications include Initial Conditions, links to an existing solved or associated environment, or an Initial Temperature. These objects are all default objects (included with) of the individual analysis types.

Analysis Type	Object	Description
Coupled Field Transient (p. 391) and Transient Structural (p. 665)	**Initial Conditions** folder	By default, a transient analysis is at rest. However, you can define velocity as an initial condition by inserting a **Velocity** object under the **Initial Conditions** folder.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Type	Object	Description
Explicit Dynamics	**Initial Conditions** folder: **Pre-Stress** object	Because an Explicit Dynamics analysis is better suited for short duration events, preceding it with an implicit analysis may produce a more efficient simulation especially for cases in which a generally slower (or rate-independent) phenomenon is followed by a much faster event, such as the collision of a pressurized container. For an **Explicit Dynamics** system, the **Initial Conditions** folder includes a **Pre-Stress** object to control the transfer of data from an implicit static or transient structural analysis to the explicit dynamics analysis. Transferable data include the displacements, or the more complete Material State (displacements, velocities, stresses, strains, and temperature). See Recommended Guidelines for Pre-Stress Explicit Dynamics for more information. An explicit dynamics analysis is at rest by default. However, for **Explicit Dynamics** and LS-DYNA systems, you can define velocity or angular velocity as initial conditions by inserting a **Velocity** object, a **Drop Height** object, or an **Angular Velocity** object under the **Initial Conditions** folder. If initial velocity, drop height, or angular velocity are defined in an LS-DYNA Restart, they override the velocities calculated at the end of the previous calculation.
Random Vibration (p. 524), Response Spectrum (p. 533), Harmonic Response MSUP (Mode-Superposition) linked (p. 509), or Transient (MSUP) linked (p. 674)	**Initial Conditions** folder: **Modal** object	A Random Vibration, Response Spectrum, Harmonic (Mode-Superposition - MSUP) linked or a Transient (MSUP) linked analysis *must* use the mode shapes derived in a Modal analysis.
Coupled Field Harmonic (p. 380) and Coupled Field Modal (p. 386)	**Pre-Stress** object	You can use the physics settings and results of an upstream Coupled Field Static analysis in downstream Coupled Field Harmonic and Coupled Field Modal analyses. Also see the Prestressed Coupled Field Harmonic Analysis (p. 382) and Prestressed Coupled Field Modal Analysis (p. 388) sections.
Eigenvalue Buckling (p. 487)	**Pre-Stress** object	An Eigenvalue Buckling analysis *must* use the stress-stiffening effects of a static structural analysis. See the Eigenvalue Buckling Analysis (p. 487) section for details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Type	Object	Description
Harmonic Response (p. 496) (Full)	**Pre-Stress** object	A Harmonic Response (p. 496) (**Full**) analysis linked to a Static Structural analysis can use the stress results to account for stress-stiffening effect. Also see the Harmonic Response (Full) Analysis Using Pre-Stressed Structural System (p. 506) section.
Modal (p. 515)	**Pre-Stress** object	A Modal analysis can use the stress results from a Static Structural analysis to account for stress-stiffening effect. See the Modal Analysis (p. 515) section for details.
Steady-State Thermal (p. 657)	**Initial Temperature** object	For a Steady-State Thermal analysis, you have the ability to specify an initial temperature.
Substructure Generation (p. 543)	**Pre-Stress** object	For a Substructure Generation analysis, an upstream Static Structural analysis (only) can be used to create pre-stress effects.
Transient Thermal (p. 680)	**Initial Temperature** object	For a Transient Thermal analysis, the initial temperature distribution should be specified.

Note:

When available in the **Outline**, you can apply temperatures from a Steady-State Thermal or a Transient Thermal analysis to a Static Structural, Transient Structural, or Electric analysis as a **Imported Body Temperature** load using the context (right-click) menu option **Import Load** that is available on the **Environment** object of the structural or electric analysis. The **Import Load** option provides a menu you can use to select the desired analysis you wish to link to. You can also perform this linking by dragging and dropping the Solution object of the Steady-State Thermal or a Transient Thermal analysis onto the Environment object of the structural or electric analysis. You can also apply **Heat Generation** from an Electric analysis to a Steady-State Thermal or Transient Thermal analysis using these methods.

Requirements/Usage

Depending upon the analysis type, an appropriate object is automatically added to the **Outline** enabling you to set an initial analysis specification:

- For a **Transient Structural** analysis, use the **Initial Conditions** object to insert **Velocity**. For an **Explicit Dynamics** analysis, use the **Initial Conditions** object to insert **Velocity**, **Angular Velocity**, and **Drop Height**. These values can be scoped to specific parts of the geometry.
- For a **Harmonic Response**, **Modal**, **Eigenvalue Buckling**, **Substructure Generation**, or **Explicit Dynamics** analysis, use the Details view of the **Pre-Stress** object to define the associated **Pre-Stress Environment** (p. 352). For an **Explicit Dynamics** analysis, use the Details view of this object to select either **Material State** (displacements, velocities, strains and stresses) or **Displacements** only modes, as well as the analysis time from the implicit analysis which to obtain the initial condition. For **Displacements** only, a **Time Step Factor** may be specified to convert nodal DOF displacements in the implicit solution into constant velocities for the explicit analysis according to the following expression:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Velocity = Implicit displacement/(Initial explicit time step x time step factor)

Note:

The **Displacements** only mode is applicable only to results from a linear, static structural analysis.

- For a **Random Vibration** or **Response Spectrum** analysis, you must point to a modal analysis using the drop-down list of the **Modal Environment** field in the Details view.
- For the **Steady-State** and **Transient Thermal** analyses, use the Details of the **Initial Temperature** object to scope the initial temperature value. For a Transient Thermal analysis that has a non-uniform temperature, you need to define an associated Initial Temperature Environment.
- The Details view of the Modal (Initial Conditions) object for linked Mode-Superposition Harmonic and Mode-Superposition Transient analyses displays the name of the pre-stress analysis system in the **Pre-Stress Environment** field, otherwise the field indicates **None** or **None Available**.

Apply Pre-Stress Effects for Implicit Analysis

Mechanical leverages the power of linear perturbation technology for all pre-stress analyses performed within Mechanical. This includes pre-stress Modal (p. 515) analyses, Full Harmonic Response analysis using a Pre-Stressed Structural System (p. 506) analyses, as well as Eigenvalue Buckling (p. 487) analyses.

The following features are available that are based on this technology:

- Large deflection static analysis followed by pre-stress modal analysis. Thus the static analysis can be linear or nonlinear including large deflection effects.

Note:

– If performing a pre-stress modal analysis, it is recommended that you always include large deflection effects to produce accurate results in the modal analysis.

– Pre-stress results should always originate from the same version of the application as that of the modal solution.

– Although the modal results (including displacements, stresses, and strains) will be correctly calculated in the modal analysis, the deformed shape picture inside Mechanical will be based on the initial geometry, not the deformed geometry from the static analysis. If you desire to see the mode shapes based on the deformed geometry, you can take the result file into Mechanical APDL.

- True contact status as calculated at the time in the static analysis from which the eigen analysis is based.
- Support for cyclic analysis.
- Support for multiple result sets in the static analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For a pre-stressed eigen analysis, you can insert a **Commands** object (p. 2303) beneath the **Pre-Stress** initial conditions object. The commands in this object will be executed just before the first solve for the pre-stressed modal analysis.

Pressure Load Stiffness

If the static analysis has a pressure load applied "normal to" faces (3D) or edges (2-D), this could result in an additional stiffness contribution called the "pressure load stiffness" effect. This effect plays a significant role in follow-on Modal analyses, Eigenvalue Buckling analyses, and in Harmonic Response (Full) analyses, however, the effect can be more prominent in an Eigenvalue Buckling analysis.

Different buckling loads may be predicted from seemingly equivalent pressure and force loads in a buckling analysis because in the Mechanical application a force and a pressure are not treated the same. As with any numerical analysis, we recommend that you use the type of loading which best models the in-service component. For more information, see the *Mechanical APDL Theory Reference*, under *Structures with Geometric Nonlinearities> Stress Stiffening> Pressure Load Stiffness*.

Restarts from Multiple Result Sets

A property called **Pre-Stress Define By** is available in the Details view of the **Pre-Stress** object in the eigen analysis. It is set to **Program Controlled** by default which means that it uses the last solve point available in the parent static structural analysis as the basis for the eigen analysis. There are three more read only properties defined in the Details view of the **Pre-Stress** object – **Reported Loadstep**, **Reported Substep** and **Reported Time** which are set to **Last**, **Last**, and **End Time** or **None Available** by default depending on whether or not there are any restart points available in the parent static structural analysis. These read only properties show the actual load step, sub step and time used as the basis for the eigen analysis.

You can change **Pre-Stress Define By** to **Load Step**, and then another property called **Pre-Stress Loadstep** will appear in the Details view. **Pre-Stress Loadstep** gives you an option to start from any load step in the static structural analysis. If you use this property, then Mechanical will always pick the last substep available in that load step. You can see the actual reported substep and time as read only properties. The input value of load step should be less than or equal to the number of load steps in the parent static structural analysis. Loadstep **0** stands for the last load step available.

You can change **Pre-Stress Define By** to **Time**, and then another property called **Pre-Stress Time** will appear in the Details view. **Pre-Stress Time** gives you an option to start from any time in the static structural analysis. If there is no restart point available at the time of your input, then Mechanical will pick the closest restart point available in the static structural analysis. You can see the actual reported load step, sub step and time as read only properties. The input value of time should be non-negative and it should be less than the end time of parent static structural analysis. Time **0** stands for end time of the parent analysis. If there is no restart point available in the input loadstep and the number of restart points in the parent analysis is not equal to zero, then the following error message appears:

"There is no restart point available at the requested loadstep. Change the restart controls in the parent static structural analysis to use the requested loadstep."

Note:

If you use **Pre-Stress Time**, then Mechanical will pick the closest restart point available. It may not be the last sub step of a load step; and if it is some intermediate substep in a load

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

step, then the result may not be reproducible if you make any changes in the parent static structural analysis or you solve it again.

If there is no restart point available in the parent static structural analysis, then **Reported Loadstep**, **Reported Substep** and **Reported Time** are set to **None Available** regardless of the user input of **Load Step/Time** but these will be updated to correct values once the analysis is solved with the correct restart controls for the parent structural analysis.

Contact Status

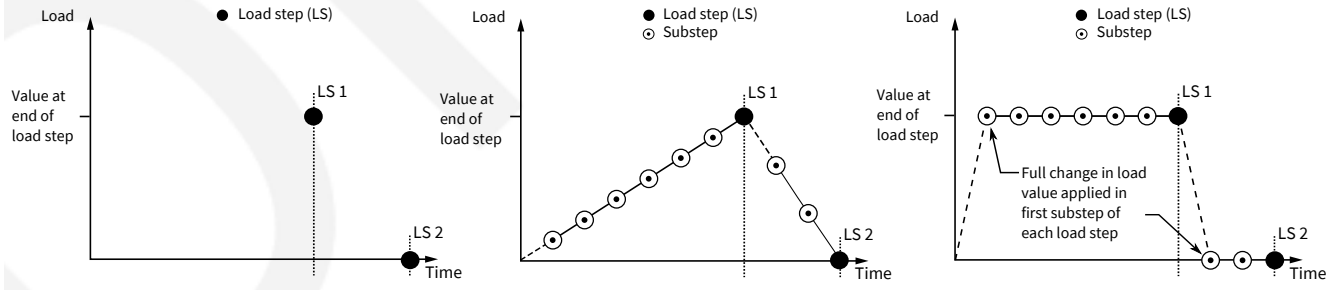
You may choose contact status for the pre-stressed eigen analysis to be true contact status, force sticking, or force bonded. A property called **Contact Status** is available in the Details view of the **Pre-Stress** object in the eigen analysis. This property controls the CONTKEY field of the Mechanical APDL **PERTURB** command.

- **Use True Status** (default): Uses the current contact status from the restart snapshot. If the previous run for parent static structural is nonlinear, then the nonlinear contact status at the point of restart is frozen and used throughout the linear perturbation analysis.
- **Force Sticking**: Uses sticking contact stiffness for the frictional contact pairs, even when the status is sliding (that is, the no sliding status is allowed). This option only applies to contact pairs whose frictional coefficient is greater than zero.
- **Force Bonding**: Uses bonded contact stiffness and status for contact pairs that are in the closed (sticking/sliding) state.

Apply Loads and Supports

You apply loads and support types based on the type of analysis. For example, a stress analysis may involve pressures and forces for loads, and displacements for supports, while a thermal analysis may involve convections and temperatures.

Loads applied to static structural, transient structural, rigid dynamics, steady-state thermal, transient thermal, magnetostatic, electric, and thermal-electric analyses default to either step-applied or ramped. That is, the values applied at the first substep stay constant for the rest of the analysis or they increase gradually at each substep.



(a) Load as specified in two load steps

(b) Load as applied - ramped (KBC,0)

(c) Load as applied - stepped (KBC,1)

You can edit the table of load vs. time and modify this behavior as needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

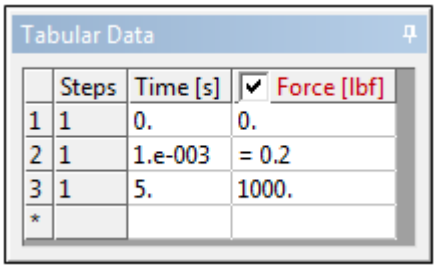
By default you have one step. However you may introduce multiple steps at time points where you want to change the analysis settings such as the time step size or when you want to activate or deactivate a load. An example is to delete a specified displacement at a point along the time history.

You do not need multiple steps simply to define a variation of load with respect to time. You can use tables or functions to define such variation within a single step. You need steps only if you want to guide the analysis settings or boundary conditions at specific time points.

When you add loads or supports in a static or transient analysis, the **Tabular Data** and **Graph** windows appear. You can enter the load history, that is, Time vs Load tabular data in the tabular data grid. Another option is to apply loads as functions of time. In this case you will enter the equation of how the load varies with respect to time. The procedures for applying tabular or function loads are outlined under the Defining Boundary Condition Magnitude (p. 1771) section.

Note:

- You can also import or export load histories from or to any pre-existing libraries.
- If you have multiple steps (p. 1492) in your analysis, the end times of each of these steps will always appear in the load history table. However you need not necessarily enter data for these time points. These time points are always displayed so that you can activate or deactivate the load over each of the steps. Similarly the value at time = 0 is also always displayed.
- If you did not enter data at a time point then the value will be either a.) a linearly interpolated value if the load is a tabular load or b.) an exact value determined from the function that defines the load. An "=" sign is appended to such interpolated data so you can differentiate between the data that you entered and the data calculated by the program as shown in the example below. Here the user entered data at Time = 0 and Time = 5. The value at Time = 1e-3, the end time of step 1, is interpolated.



To apply loads or supports in the Mechanical Application:

See the Setting Up Boundary Conditions (p. 1497) section.

Perform Solution

Mechanical uses the same solver kernels that Ansys Mechanical APDL (MAPDL) uses. At the Solve step, Mechanical passes its data to the appropriate MAPDL solver kernel, based on the type of analysis to be performed. That kernel then passes the solution data back to Mechanical, where you are able to look

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

at the results. Because the same solver kernels are used, you will obtain the same results from Mechanical that you would if doing the same analysis in MAPDL.

Based on the analysis type, the following solvers are available in Mechanical:

- **Mechanical Ansys Parametric Design Language (MAPDL) Solver**.
- **Ansys Rigid Dynamics Solver**: only available for Rigid Dynamics Analysis.
- **LS-DYNA Solver**: only available for Explicit Dynamics analysis.
- **Explicit Dynamics Solver**: only available for Explicit Dynamics analysis.
- **Samcef Solver**: only available for Static Structural, Transient Structural, Steady-State Thermal, Transient Thermal, Modal, and Eigenvalue Buckling analyses.
- **ABAQUS Solver**: only available for Static Structural, Transient Structural, Steady-State Thermal, Transient Thermal, and Modal analyses.

You can execute the solution process on your local machine or on a remote machine such as a powerful server you might have access to. The Remote Solve Manager (RSM) feature allows you to perform solutions on a remote machine. Once completed, results are transferred to your local machine for post processing. Refer to the Solve Modes and Recommended Usage (p. 1803) section for more details.

Solution Progress

Because nonlinear or transient solutions can take significant time to complete, a progress pane is displayed in the status bar (p. 135) to indicate the overall progress of solution. More detailed information on solution status can be obtained from the **Solution Information** object (p. 1833) contained under the **Solution** folder. The **Solution Information** object enables you to:

- View solver output.
- Monitor output such as convergence criteria for nonlinear problems.
- Diagnose possible reasons for convergence difficulties by plotting Newton-Raphson residuals.
- Monitor certain result items, such as displacement or temperature.

Solve References for the Mechanical Application

See the Understanding Solving (p. 1791) section for details on the above and other topics related to solving.

Review Results

The analysis type determines the results available for you to examine after solution. For example, in a structural analysis, you may be interested in equivalent stress results or maximum shear results, while in a thermal analysis, you may be interested in temperature or total heat flux. The Using Results (p. 1927) section lists various results available to you for postprocessing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To add result objects in the Mechanical application:

1. Highlight a **Solution** object in the tree.
2. Select the appropriate result from the Solution Context Tab (p. 62) or use the right-mouse click option.

To review results in the Mechanical application:

1. Click a result object in the tree.
2. After the solution has been calculated, you can review and interpret the results in the following ways:
   - Contour results (p. 62) - Displays a contour plot of a result such as stress over geometry.
   - Vector Plots (p. 74) - Displays certain results in the form of vectors (arrows).
   - Probes (p. 1946) - Displays a result at a single time point, or as a variation over time, using a graph and a table.
   - Charts (p. 1929) - Displays different results over time, or displays one result against another result, for example, force vs. displacement.
   - Animation (p. 2260) - Animates the variation of results over geometry including the deformation of the structure.
   - Stress Tool (p. 2058) - to evaluate a design using various failure theories.
   - Fatigue Tool (p. 2066) - to perform advanced life prediction calculations.
   - Contact Tool (p. 2079) - to review contact region behavior in complex assemblies.
   - Beam Tool (p. 2087) - to evaluate stresses in line body representations.

Note:

Displacements of rigid bodies are shown correctly in transient structural and rigid dynamics analyses. If rigid bodies are used in other analyses such as static structural or modal analyses, the results are correct, but the graphics will not show the deformed configuration of the rigid bodies in either the result plots or animation.

Note:

If you resume a Mechanical model from a project or an archive that does **not** contain result files, then results in the **Solution** tree can display contours but restrictions apply:

- The result object cannot show a deformed shape; that is, the node-based displacements are not available to deform the mesh.
- The result object cannot animate.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Contours are **not** available for harmonic results that depend upon both real and imaginary result sets.

See the Using Results (p. 1927) section for more references on results.

Create Report (optional)

Workbench includes a provision for automatically creating a report based on your entire analysis. The documents generated by the report are in HTML. The report generates documents containing content and structure and uses an external Cascading Style Sheet (CSS) to provide virtually all of the formatting information.

Report References for the Mechanical Application

See the Report Preview (p. 174) section.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Types

You can perform analyses in the Mechanical application using pre-configured analysis systems (as listed below). Each analysis description assumes that you are familiar with the analysis type as well as with the information presented in Steps for Using the Mechanical Application (p. 323) section, specifically the Create Analysis System (p. 323) topic. The availability of features for an analysis may differ based on the solver you select.

Also note that it is not necessary to specify a geometry or mesh, or even an analysis type to initiate a simulation. Mechanical provides options to perform these actions *after* you have opened the application.

Adaptivity Analyses

Coupled Field Analysis Types

Electric Analysis

Explicit Dynamics Analysis

Fracture Analysis

Linear Dynamics Analysis Types

Magnetostatic Analysis

Rigid Dynamics Analysis

Static Structural Analysis

Steady-State Thermal Analysis

Thermal-Electric Analysis

Transient Structural Analysis

Transient Structural Analysis Using Linked Modal Analysis System

Transient Thermal Analysis

Special Analysis Topics

In addition, you can enhance your analysis and add capabilities using the **Commands** Object (p. 2303) to execute Mechanical APDL commands in the Mechanical application.

Adaptivity Analyses

Mechanical enables you to perform nonlinear adaptive analyses using the **Nonlinear Adaptive Region** and **Geometry Based Adaptivity** options found in the Conditions (p. 1697) drop-down menu of the **Environment** context tab.

Nonlinear adaptivity is a capability of the solution process that enables the application to adapt to changing conditions during a nonlinear analysis. The solution process uses a feedback mechanism to discretely and continuously adjust internal parameters automatically so that you can obtain an accurate and convergent solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You use these conditions to either change the mesh during the solution process or to model the boundary geometry accurately during mesh adaptivity:

Nonlinear Adaptive Region

Geometry Based Adaptivity

Nonlinear Adaptive Region

The **Nonlinear Adaptive Region** condition enables you to change the mesh during the solution phase to improve precision without incurring a great deal of computational penalties. The Nonlinear Adaptive Region feature is completely automatic. It does not require any user input during the solution phase. It acts as a remesh controller based on certain criteria. The criteria determine whether or not the mesh requires modification and, if so, which parts need to be modified. This feature is based on load stepping, requiring you to define a number of steps for your analysis, while also allowing you to activate and/or deactivate the feature on a per step basis.

This condition may be useful for nonlinear problems that experience convergence difficulties or accuracy issues because of elemental distortions. Large deformation problems are best suited to the use of the condition. However, it is also useful for cases where large deformation is small but requires mesh adaptation to better capture the physics and give a more accurate solution. Review the Nonlinear Mesh Adaptivity Usage Considerations section of the *Nonlinear Adaptivity Analysis Guide* for more information about analysis cases when the feature can be useful.

Requirements

The **Nonlinear Adaptive Region** condition requires the **Store Results At** property to be set to **All Time Points** in the **Output Controls** category of the **Analysis Settings**.

Preprocessing Support Limitations

Note the following preprocessing limitations for this condition.

- When there is a hexahedral mesh for a multibody part, the Nonlinear Adaptive Region can only be scoped to a single body.
- The Nonlinear Adaptive Region is not supported if your model is a multibody part or multibody assembly with a mixture of linear and quadratic elements.
- It is not supported for Convergence.
- Cannot be used in combination with the following features/conditions on the same part:
  - Cyclic Symmetry
  - Beam Contact Formulation
  - Contact Behaviors: Auto Asymmetric
  - Point Mass, Beam Connection, Joints, Spring, and Bearing
  - Spatially varying boundary conditions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Not supported if the analysis includes a Pressure or an Imported Pressure that have the following combination of settings:
  - **Applied By** property set to **Direct** option.
  - **Loaded Area** property set to **Initial**.
  - **Large Deflection** property (Analysis Settings > Solver Controls) set to **On**.
- Cannot be used in combination with the following boundary conditions:
  - Coupling
  - Constraint Equation
  - Remote Displacement, Remote Force, and Moment specified with the **Behavior** property set to **Beam**.
- Cannot be used in combination with Weak Springs (p. 1433) (COMBIN14 element).
- The following materials properties are not supported:
  - Cast Iron
  - Concrete
  - Cohesive Zone
  - Damage Initiation Criteria and Damage Evolution Law
  - Microplane
  - Shape Memory Alloy
  - Swelling
- When linking analyses, you cannot apply the solution phase modified mesh to the linked system.
- When using the **Nonlinear Adaptive Region** during the restart of an analysis, the **Nonlinear Adaptive Region** object does not support Named Selections if your model contains a mesh change prior to the restart point.
- If your analysis failed to converge and you are adding a new **Nonlinear Adaptive Region** object, it is necessary that the contact object property, **Behavior**, was set to either **Symmetric** or **Asymmetric** for the initial solution that was processed.
- When the **Mesh** option is specified for the **Criterion** property:
  - does not support high order elements for 2D analyses.
  - does not support self-contact for 3D analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Post Processing Support Limitations

Because this condition causes mesh changes during the course of the solution process, there are result scoping limitations.

- Only **Body** scoping is permitted (for bodies whose meshes will change). Therefore, if you scope any result or probe on a vertex, edge, or face of a body that experiences a mesh change, the analysis will not solve. This limitation is a result of the base mesh of the body being represented by nodes only. This limitation also applies to probes scoped to boundary conditions (via **Location Method** property).
- Element selection on a result is not supported. However, if you 1) have an element-based named selection and 2) activate the **Preserve During Solve** property, you can specify results for the named selection using the **Solver Component Names** option of the Solution Quantities and Result Summary (p. 1961) page of the **Worksheet** (accessed via the Solution object).
- If you have a result object selected, certain mesh-based features are not available, including mesh selection filters (nodes (p. 269), elements, and element faces (p. 276)), as well as the ability to display a Node ID in the probe label (p. 206), and Selecting Nodes and Elements by ID (p. 279).
- Does not support the multiple result set options (p. 1990) of the **By** property: **Maximum Over Time**/**Time of Maximum** or **Minimum Over Time**/**Time of Minimum**.
- Penetration plot following remesh may show the curve discontinuity.
- Is not supported when transferring the deformed geometry (p. 1022) and mesh of a **Deformation** result.
- When using the Deformation result tracker (p. 1847) to graph displacement, note there is a display limitation for the graph. The tracker reads and displays data contained in the `jobname.nlh` file. This file contains incremental displacement data collected after re-meshing occurs. That is, the re-meshed model is considered as a new reference.

Analysis Types

Nonlinear Adaptive Region is available for Static Structural (p. 652) analyses.

For additive manufacturing Inherent Strain simulations, a special case of nonlinear mesh adaptivity is available in which simulation time is reduced by *coarsening* elements in previously deposited layers, thereby reducing the total number of elements. See Using Adaptive Meshing (AM Octree) for details.

Common Characteristics

The following section outlines the common characteristics that include application requirements of the condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Nonlinear Adaptive Region** condition include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Nonlinear Adaptive Region**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Elements: *Supported* for element-based Named Selections only.

Note:

– Elements must be of the same element type, material, nodal orientation, and element orientation.

– If two regions with different element or material attributes require re-meshing, you must impose nonlinear adaptive regions separately.

– The application does not support mixed order Tetrahedral mesh elements defined on one region or when used with multiple regions.

Condition Application

To apply a **Nonlinear Adaptive Region**:

1. On the **Environment** Context tab: click **Conditions**>**Nonlinear Adaptive Region**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Nonlinear Adaptive Region**.
2. Specify the **Scoping Method**.

Note:

You can scope multiple Nonlinear Adaptive Regions to the same entity to give yourself more control on multiple load step settings that are local to the Nonlinear Adaptive Region condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

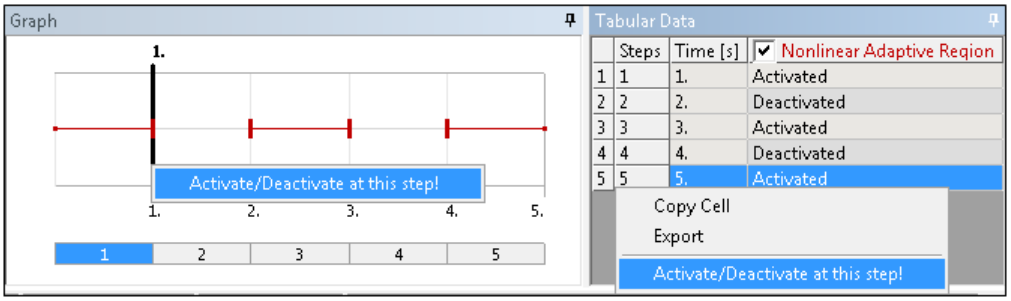
3. Specify the **Criterion** property: options include **Energy**, **Box**, and **Mesh**. The **Mesh** option is the recommended setting.

   Following your selection, properly specify the associated properties.

   Note:

   Once you select an option for the Criteria property, the **Analysis Settings** category Nonlinear Adaptivity Remeshing Controls (p. 1440) becomes available and you may modify the available properties as needed.

4. When the condition is defined, the **Graph** and **Tabular Data** windows provide a right-mouse click option to **Deactivate** (or **Activate**) the condition for a desired load step. No remeshing will occur at the deactivated load step as the NLADAPTIVE command is set to OFF. The default setting is **Activated**. For a restart analysis, the application sets the newly added **Nonlinear Adaptive Region** to **Deactivated**.



   Note:

   You may wish to review the **Activation/Deactivation of Loads** topic in the Step Controls (p. 1418) section of the Help. The **Nonlinear Adaptive Region** condition is displayed in the graph for the **Analysis Settings** object.

Details View Properties

The selections available in the Details view are described below.

Category	Properties/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Displays the type of geometry (Body) and the number of geometric entities (1 Body, 2 Bodies) to which the condition has been applied using the selection tools.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Options/Description
	• **Named Selection**: Indicates that the geometry selection is defined by a **Named Selection**. – **Named Selection**: Visible when the **Scoping Method** property is set to **Named Selection**. This field provides a drop-down list of available user-defined element-based (only) Named Selections. You may wish to review the **Mechanical APDL References and Notes** at the bottom of the page for specific command execution information regarding these selections.
Definition	**Criterion**: Options included **Energy**, **Box**, or **Mesh**. **Energy** If **Energy** is selected, you need to enter an **Energy Coefficient**. **Energy Coefficient**: This criterion checks the strain energy of any element that is part of the defined component using the following calculation. A component is defined as the assembly of all scoped elements. $E_e \geq c_1 \cdot E_{Total} / NUME$ where: • E_e = strain energy of single target element. • c_1 = user input energy coefficient. • E_{total} = total strain energy of the component. • NUME = number of elements of the component. If this criterion is satisfied at an element, the program refines the element. A smaller **Energy Coefficient** improves the potential to trigger the mesh regeneration for the same loads. **Box** If **Box** is selected, the following additional properties require entries: • **Coordinate System** (default is **Global Coordinate System**): defines the minimum values of the box. • **Length X**: defines the diagonal length on global X axis. • **Length Y**: defines the diagonal length on global Y axis. • **Length Z** (for 3D): defines the diagonal length on global Z axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Options/Description
	Mesh (3D Analysis) For 3D analyses, if you select the **Mesh** option, the **Tetrahedral** and **Hex-Dominant** categories display. **Tetrahedral** The **Options** property displays. Available options include **Skewness**, **Jacobian Ratio**, or **Skewness and Jacobian Ratio** (default. If the **Options** property is set to either **Skewness** or **Skewness and Jacobian Ratio**, you need to enter a **Skewness Value** for the mesh elements. The element skewness is defined as: Skewness = $(V_{reg} - V_{el}) / V_{reg}$ Where: V_{el} is the volume of the element under calculation and V_{reg} is the volume of the standard tetrahedral linear element located in the same sphere as the element under calculation. When the element under calculation has an ideal shape (that of a standard tetrahedral element), $V_{reg} = V_{el}$ and **Skewness Value** = 0. When the element under calculation has the least ideal shape (that of a flat element), $V_{el} = 0$ and **Skewness Value** = 1. Therefore, for the linear tetrahedral element, **Skewness Value** is always between 0 and 1, with 1 being a flat element. The default value is **0.9**. The recommended **Skewness Value** is approximately **0.85** to **0.9**. When the **Option** property is set to **Jacobian Ratio** or **Skewness and Jacobian Ratio**, you need to enter a **Jacobian Ratio Value** for the mesh elements. This value defines the maximum Jacobian ratio at element integration points. The default value is **`0.1`**. **Hex-Dominant** If the body selected in the scoping includes hexahedral mesh, then this category and its properties are applicable to solution processing. The application uses an initial hexahedral-dominant mesh which is converted to a tetrahedral mesh during the solution process once the category's criterion is met. For a tetrahedral only mesh, application ignores this category and its properties. For this category: • The **Options** property is read-only and the application sets it to **Shape**. • The **Maximum Corner Angle** property has the default setting of **`155°`**. The entry range for this property is **`0°`** - **`180°`**. An

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Description
	element is re-meshed when any of its corner angles reach this specified value. **Mesh (2D Analysis)** For 2D analyses: • The **Options** property is read-only and the application sets it to **Shape**. • The **Maximum Corner Angle** property has the default setting of **160°** (recommended). The entry range for this property is **0°** - **180°**. An element is re-meshed when any of its corner angles reach this specified value. All the following common criteria properties are available for all **Criterion** property options as well as 2D analyses: • **Check At**: Options included **Equally Spaced Points** or **Specified Recurrence Rate**. • **Value**: This property further defines the **Check At** property. The default value is **1**. • **Time Range**: options included **Entire Load Step** or **Manual**. When specified as **Manual**, you need to enter a **Start Time** and **End Time**. The **End Time** value must be larger than the **Start Time** value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the condition.

Initial Mesh Load Specifications

You can apply the following boundary conditions directly to the initial mesh generated for each load step rather than on the newly generated mesh only by setting the object's **Apply To Initial Mesh** property to **Yes**.

- Imported Displacement (p. 1746)
- Imported Body Temperature (p. 1740)
- Imported Pressure (p. 1752)

For more information, see the Initial-Mesh Loading and Constraint section in the *Nonlinear Adaptivity Analysis Guide* of the Mechanical APDL documentation.

View Changed Mesh Results

Following the solution process, to determine if the mesh was changed:

- Select the **Solution** object or a **Result** object, the **Tabular Data** window displays the substeps with a changed mesh (**Changed Mesh** column = **Yes**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Select the **Solution Information** object and set the **Solution Output** property to **Force Convergence**. A chart displays. Remesh Points are shown by solid orange vertical lines.
- Create a **User Defined Result** (p. 2194) (using the **PNUMELEM** Expression) to view the new elements that have relatively larger element identities than the original element identities. You can duplicate this result and specify a **Result Set** (**By** property) for a result prior to a remesh and one at a remesh point, and using the **Viewports** (p. 284) feature, directly compare the (before and after) elements in the graphics window.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this condition.

- The **Nonlinear Adaptive Region** is applied with the **NLADAPTIVE** command.
- When the **Scoping** property is defined as **Geometry**, the **Nonlinear Adaptive Region** condition uses the **CM** command to create the **Nonlinear Adaptive Region** component.
- When the **Scoping** property is defined as a **Named Selection**, the **Nonlinear Adaptive Region** condition uses the **CMBLOCK** command to create the **Nonlinear Adaptive Region** component.
- The **CMSEL**,**ALL** command and the **ESEL**,**ALL** command are issued at beginning of the **NLADAPTIVE** command.
- During a Structural Analysis, the **Nonlinear Adaptive Region** is applied using the PLANE182 (2-D Low Order), PLANE183 (2-D High Order), SOLID285 (3-D Linear Tetrahedral), and SOLID187 (3-D Quadratic Tetrahedral) element types.
- When a **Nonlinear Adaptive Region** is scoped to a body/element, the associated part is meshed with SOLID285 element type if they are linear tetrahedral or SOLID187 element type if they are quadratic tetrahedral.
- When a **Nonlinear Adaptive Region** is deactivated for certain steps, the **NLADAPTIVE** command is set to be **OFF** in the corresponding load steps. Relatively, an activated Nonlinear Adaptive Region sets the **NLADAPTIVE** command to be **ON**.
- When a **Nonlinear Adaptive Region** is applied, the **ETCONTROL ,SET** command is **not** issued.

Note:

For additional guidance about how to best use this feature, see the Mesh Nonlinear Adaptivity Hints and Recommendations section in the *Mechanical APDL Advanced Analysis Guide*.

Nonlinear Adaptive Region Solving Limitations

The purpose of nonlinear adaptive region is to repair a distorted mesh in order to overcome convergence problems caused by the distortion. It is effective only when the mesh distortion is caused by a large, nonuniform deformation. Nonlinear adaptive region cannot help if divergence occurs for any other reason such as unstable material, unstable structures, or numerical instabilities.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unstable Material

Most nonlinear material models, especially those employing hyperelastic materials, have their own applicable ranges. When a deformation is too large or a stress state exceeds the applicable range, the material may become unstable. The instability can manifest itself as a mesh distortion, but nonlinear adaptive region cannot help in such cases. While it is sometimes difficult to determine when material is unstable, you can check the strain values, stress states, and convergence patterns. A sudden convergence difficulty could mean that material is no longer stable. The program also issues a warning at the beginning of the solution indicating when hyperelastic material could be unstable, although such a warning is very preliminary and applies only to cases involving simple stress states.

Unstable Structures

For some geometries and loads, a deformation may cause a "snap-through," or local buckling. Such behavior can also manifest itself as a mesh distortion, but one that nonlinear adaptive region cannot repair. The effect is usually easy to detect by closely checking the deformed region or the load-versus-time (displacement) curve.

Numerical Instabilities

A condition of numerical instability can occur when a problem is nearly overconstrained. The constraints can include kinematic constraints such as applied displacements, couplings, and constraint equations, and volumetric constraints introduced by fully incompressible material in mixed u-P elements. In many cases, numerical instability is apparent even in the early stages of an analysis.

Geometry Based Adaptivity

The **Geometry Based Adaptivity** condition enables you to model the boundary geometry accurately during mesh adaptivity by placing the new nodes and elements on the model boundary geometry during remeshing. This is different from conventional mesh adaptivity applied through the Nonlinear Adaptive Region (p. 360) object, where the new nodes and elements on the boundary are placed on the old mesh facets during remeshing. The use of this object cannot be combined with the use of the **Nonlinear Adaptive Region** feature. The application generates an error if both objects are present in an analysis.

Note:

Review the Nonlinear Mesh Adaptivity for Geometry-Preserving Adaptive Analysis (NLAD-GPAD) documentation in the *Mechanical APDL Nonlinear Adaptivity Analysis Guide* for more information.

Go to a section topic:

- Requirements (p. 370)
- Modelling Limitations (p. 370)
- Postprocessing Limitations (p. 371)
- Analysis Types (p. 372)
- Common Characteristics (p. 372)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Condition Application (p. 373)
- Details Properties (p. 374)

Requirements

The **Geometry Based Adaptivity** condition requires the **Large Deflection** property to be set to **Off** in the **Solver Controls** category of the **Analysis Settings**. As a result, the application specifies this condition as described in the **NLMESH** section of the *Mechanical APDL Command Reference*. Furthermore, you cannot create **Command (ADPL)** object entries that activate the **Large Deflection** property.

Refinement via general remeshing is supported with energy-based (**NLADAPTIVE,,,ENERGY**), or position-based (**NLADAPTIVE,,,BOX**) criteria only.

Note:

Refinement through splitting is not supported in **Geometry Based Adaptivity**.

The **Geometry Based Adaptivity** condition enables you to change the mesh during the solution phase to improve precision without incurring a great deal of computational penalties. The feature is completely automatic. It does not require any user input during the solution phase. It acts as a remesh controller based on certain criteria. The criteria determine whether or not the mesh requires modification and, if so, which parts need to be modified. This feature is based on load stepping, requiring you to define a number of steps for your analysis, while also allowing you to activate and/or deactivate the feature on a per step basis.

Review the Nonlinear Mesh Adaptivity for Geometry-Preserving Adaptive Analysis (NLAD-GPAD) section of the *Nonlinear Adaptivity Analysis Guide* for more information about analysis cases when the feature can be useful.

Modelling Limitations

Note the following preprocessing limitations for this condition.

- This condition only supports linear and quadratic tetrahedral elements (SOLID187 and SOLID285).
- The geometry-based adaptivity is not supported if your model is a multibody part or multibody assembly with a mixture of linear and quadratic elements.
- It does not supported the use of the Convergence object.
- Cannot be used in combination with the following features/conditions on the same part:
  - Cyclic Symmetry
  - Beam Contact Formulation
  - Contact Behaviors: Auto Asymmetric
  - Point Mass, Beam Connection, Joints, Spring, and Bearing
  - Spatially varying boundary conditions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Not supported if the analysis includes a Pressure or an Imported Pressure that have the following combination of settings:
  - **Applied By** property set to **Direct** option.
  - **Loaded Area** property set to **Initial**.
  - **Large Deflection** property (Analysis Settings > Solver Controls) set to **On**.
- Cannot be used in combination with the following boundary conditions:
  - Coupling
  - Constraint Equation
  - Remote Displacement, Remote Force, and Moment specified with the **Behavior** property set to **Beam**.
- Cannot be used in combination with Weak Springs (p. 1433) (COMBIN14 element).
- The following materials properties are not supported:
  - Cast Iron
  - Concrete
  - Cohesive Zone
  - Damage Initiation Criteria and Damage Evolution Law
  - Microplane
  - Shape Memory Alloy
  - Swelling
- When linking analyses, you cannot apply the solution phase modified mesh to the linked system.
- When using the **Geometry Based Adaptivity** during the restart of an analysis, the **Geometry Based Adaptivity** object does not support Named Selections if your model contains a mesh change prior to the restart point.
- If your analysis failed to converge and you are adding a new **Geometry Based Adaptivity** object, it is necessary that the contact object property, **Behavior**, was set to either **Symmetric** or **Asymmetric** for the initial solution that was processed.

Postprocessing Limitations

Because this condition causes mesh changes during the course of the solution process, there are result scoping limitations.

- Only **Body** scoping is permitted (for bodies whose meshes will change). Therefore, the evaluation of those results or probes will fail which will be scoped on a vertex, edge, or face of a

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

body that experiences a mesh change. This limitation also applies to probes scoped to boundary conditions (via **Location Method** property).

- Element selection on a result is not supported. However, if you 1) have an element-based named selection and 2) activate the **Preserve During Solve** property, you can specify results for the named selection using the **Solver Component Names** option of the Solution Quantities and Result Summary (p. 1961) page of the **Worksheet** (accessed via the Solution object).
- If you have a result object selected, certain mesh-based features are not available, including mesh selection filters (nodes (p. 269), elements, and element faces (p. 276)), as well as the ability to display a Node ID in the probe label (p. 206), and Selecting Nodes and Elements by ID (p. 279).
- Does not support the multiple result set options (p. 1990) of the **By** property: **Maximum Over Time**/**Time of Maximum** or **Minimum Over Time**/**Time of Minimum**.
- Penetration plot following remesh may show the curve discontinuity.
- Is not supported when transferring the deformed geometry (p. 1022) and mesh of a **Deformation** result.

Analysis Types

Geometry Based Adaptivity is available for Static Structural (p. 652) analyses.

Common Characteristics

The following section outlines the common characteristics that include application requirements of the condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Geometry Based Adaptivity** condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Geometry Based Adaptivity**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Element Face: **Not Supported**.
- Elements: **Not Supported**.

Note:

- Elements must be of the same element type, material, nodal orientation, and element orientation.
- If two regions with different element or material attributes require re-meshing, you must impose geometry-based adaptivity regions separately.
- The application does not support mixed order Tetrahedral mesh elements defined on one region or when used with multiple regions.

Condition Application

To apply a **Geometry Based Adaptivity**:

1. On the **Environment** Context tab: click **Conditions**>**Geometry Based Adaptivity**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Geometry Based Adaptivity**.
2. Specify the **Scoping Method**.

   Note:

   You can scope multiple Geometry Based Adaptivity objects to the same entity to give yourself more control on multiple load step settings that are local to the Nonlinear Adaptive Region condition.

3. Specify the **Criterion** property: options include **Energy** and **Box**.

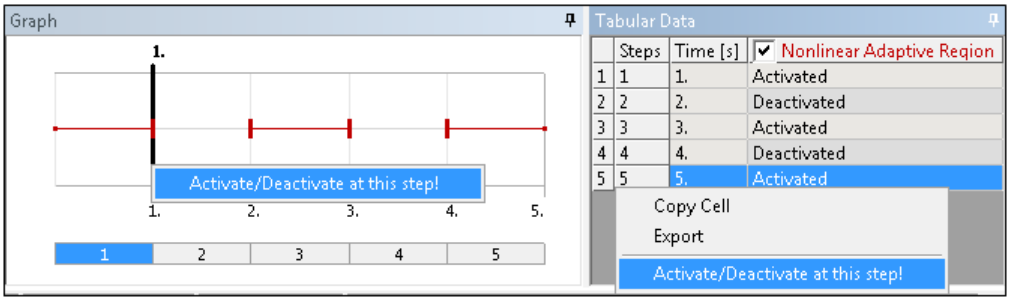
   Following your selection, properly specify the associated properties.

   Note:

   Once you select an option for the **Criteria** property, the **Analysis Settings** category Adaptivity Remeshing Controls (p. 1440) becomes available and you may modify the available properties as needed.

4. When the condition is defined, the **Graph** and **Tabular Data** windows provide a right-mouse click option to **Deactivate** (or **Activate**) the condition for a desired load step. No remeshing will occur at the deactivated load step as the NLADAPTIVE command is set to OFF. The default setting is **Activated**. For a restart analysis, the application sets the newly added **Geometry Based Adaptivity** to **Deactivated**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

You may wish to review the **Activation/Deactivation of Loads** topic in the Step Controls (p. 1418) section of the Help. The **Geometry Based Adaptivity** condition is displayed in the graph for the **Analysis Settings** object.

Details Properties

The selections available in the Details are described below.

Category	Properties/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Displays the type of geometry (Body) and the number of geometric entities (1 Body, 2 Bodies) to which the condition has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a **Named Selection**. – **Named Selection**: Visible when the **Scoping Method** property is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. You may wish to review the **Mechanical APDL References and Notes** at the bottom of the page for specific command execution information regarding these selections.
Definition	**Criterion**: Options included **Energy** and **Box**. **Energy** If **Energy** is selected, you need to enter an **Energy Coefficient**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Options/Description
	Energy Coefficient: This criterion checks the strain energy of any element that is part of the defined component using the following calculation. A component is defined as the assembly of all scoped elements. $E_e \geq c_1 \cdot E_{Total} / NUME$ where: • E_e = strain energy of single target element. • c_1 = user input energy coefficient. • E_{total} = total strain energy of the component. • NUME = number of elements of the component. If this criterion is satisfied at an element, the program refines the element. A smaller **Energy Coefficient** improves the potential to trigger the mesh regeneration for the same loads. **Box** If **Box** is selected, the following additional properties require entries: • **Coordinate System** (default is **Global Coordinate System**): defines the minimum values of the box. • **Length X**: defines the diagonal length on global X axis. • **Length Y**: defines the diagonal length on global Y axis. • **Length Z** (for 3D): defines the diagonal length on global Z axis. All the following common criteria properties are available for all **Criterion** property options as well as 2D analyses: • **Check At**: Options included **Equally Spaced Points** or **Specified Recurrence Rate**. • **Value**: This property further defines the **Check At** property. The default value is **1**. • **Time Range**: options included **Entire Load Step** or **Manual**. When specified as **Manual**, you need to enter a **Start Time** and **End Time**. The **End Time** value must be larger than the **Start Time** value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the condition.

Mechanical APDL References and Notes

For additional guidance for how to use this feature, see the NLAD-GPAD Best Practices section of the *Mechanical APDL Nonlinear Adaptivity Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Initial Mesh Load Specifications

You can apply the following boundary conditions directly to the initial mesh generated for each load step rather than on the newly generated mesh only by setting the object's **Apply To Initial Mesh** property to **Yes**.

- Imported Displacement (p. 1746)
- Imported Body Temperature (p. 1740)
- Imported Pressure (p. 1752)

For more information, see the Initial-Mesh Loading and Constraint section in the *Nonlinear Adaptivity Analysis Guide* of the Mechanical APDL documentation.

View Changed Mesh Results

Following the solution process, to determine if the mesh was changed:

- Select the **Solution** object or a **Result** object, the **Tabular Data** window displays the substeps with a changed mesh (**Changed Mesh** column = **Yes**).
- Select the **Solution Information** object and set the **Solution Output** property to **Force Convergence**. A chart displays. Remesh Points are shown by solid orange vertical lines.
- Create a **User Defined Result** (p. 2194) (using the **PNUMELEM** Expression) to view the new elements that have relatively larger element identities than the original element identities. You can duplicate this result and specify a **Result Set** (**By** property) for a result prior to a remesh and one at a remesh point, and using the **Viewports** (p. 284) feature, directly compare the (before and after) elements in the graphics window.

Coupled Field Analysis Types

Introduction

The Coupled Field analyses in Mechanical enable you to simulate interaction between multiple physics types. The availability of analysis settings, boundary conditions, results, etc. is based on the specified physics as well as the analysis type you select. For example, if you specify a Coupled Field Static analysis, all common features are available for structural physics, such as Force, Deformation, etc. Supported physics configurations include:

	Coupled Field Harmonic	Coupled Field Modal	Coupled Field Static	Coupled Field Transient
Structural	✓	✓	✓	✓
Acoustics	✓	✓	✓	✓
Electric	✓[a]	✓[a]	✓[b]	✓[b]
Thermal			✓	✓

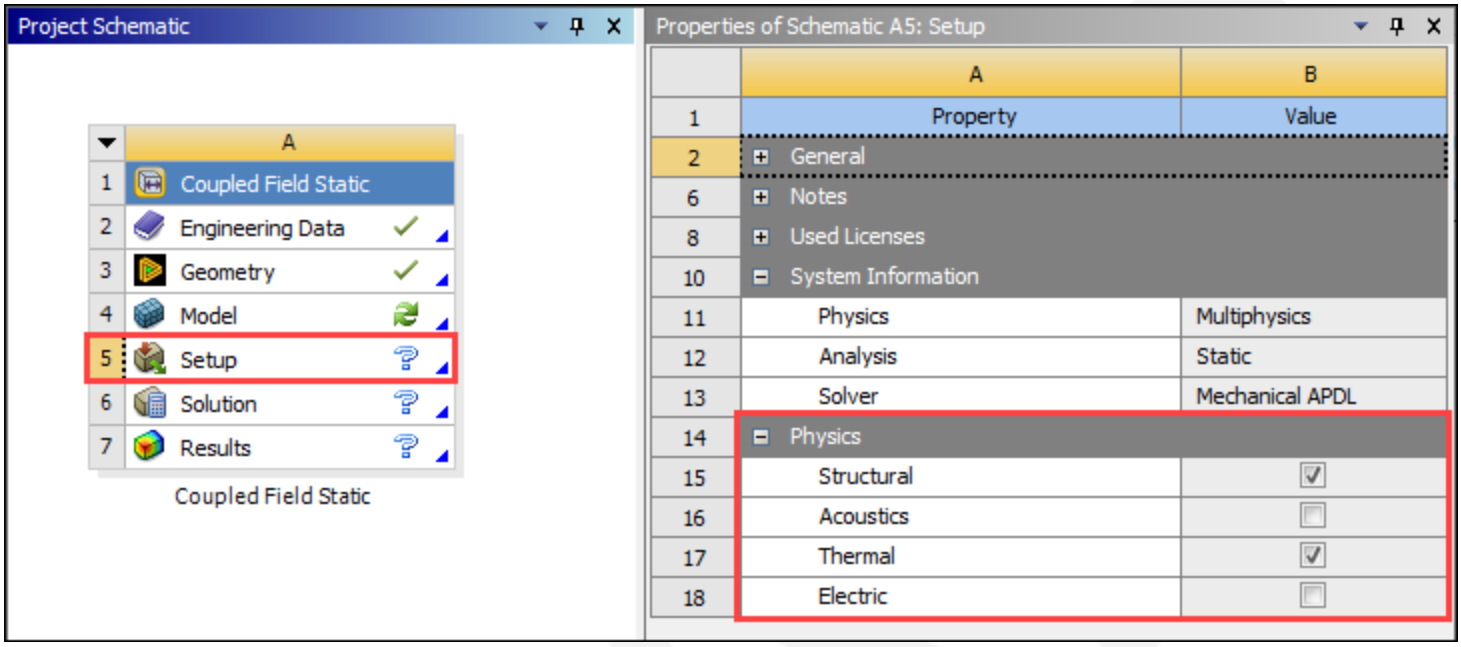
[a] Supports charge-based formulation only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

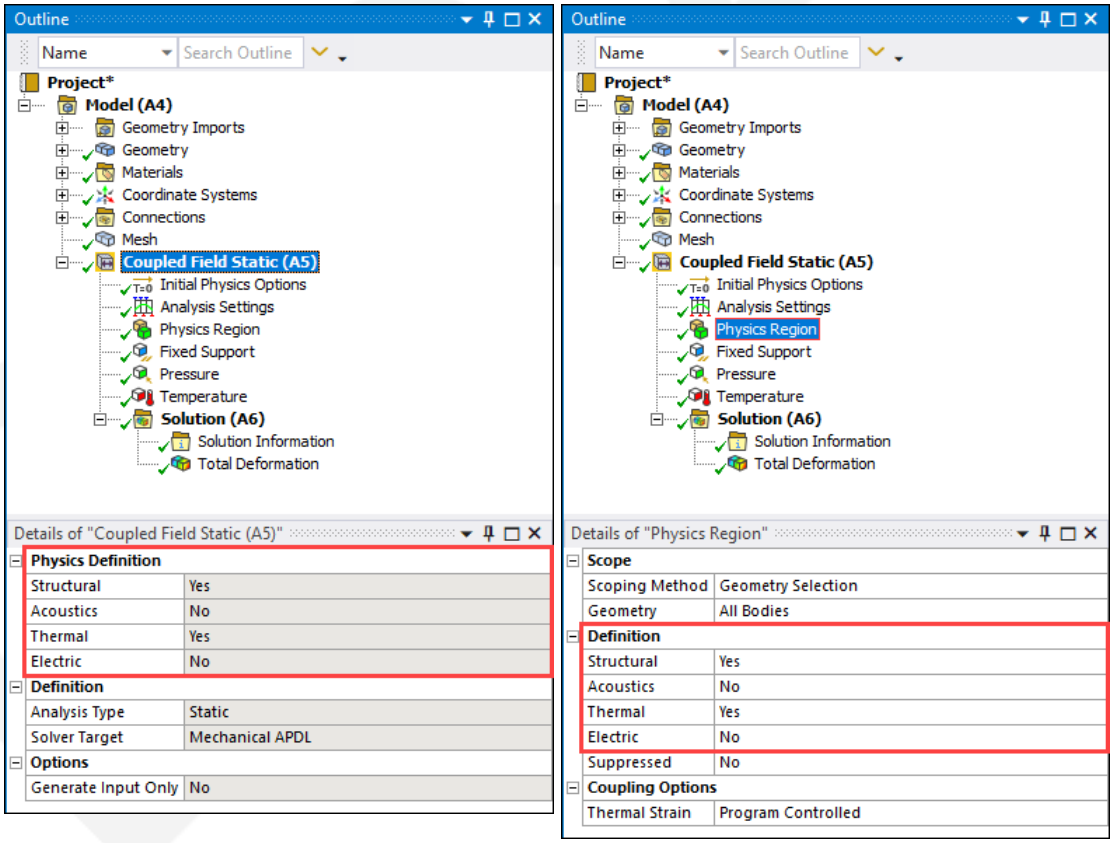
Published: 2023-10-23

[b] Supports charge-based and current-based formulation.

You can view the physics specified for your Coupled Field analysis from the properties of the **Setup** cell in Workbench.



And, in Mechanical, you can view the selected physics from the Environment object, and you can view and change the physics definition using the Physics Region object.



Coupling

The following table shows the interaction between the types of physics available in the application and the analysis types. You specify the interaction between the physics types and the analyses using either the 1) **Physics Region**, such as Thermal Strain, or through 2) coupling conditions, such as Plastic heating, or 3) the application always includes Piezoelectric coupling via Structural-Electric charge interaction.

Physics	Coupled Field Harmonic	Coupled Field Modal	Coupled Field Static	Coupled Field Transient
Structural Thermal	Not Supported		Thermal strain Thermoplasticity Thermoviscoelasticity	Thermal strain Thermoelastic Damping Thermoplasticity Thermoviscoelasticity
Electrostatic Structural	Electrostatic Force[a]	Electrostatic Force[a]	Electrostatic Force	Electrostatic Force
Structural-Acoustics	Fluid Solid Interface	Fluid Solid Interface	Fluid Solid Interface	Fluid Solid Interface
Structural Electric (Charge)	Piezoelectric	Piezoelectric	Piezoelectric	Piezoelectric
Thermal Electric (Conduction)			Joule Heating Seebeck Peltier	Joule Heating Seebeck Peltier

[a] Requires prestress Coupled Field Static.

Important:

The **Element Control** property of the Geometry object is, by default, set to **Program Controlled**. This setting allows the application to choose the best Mechanical APDL element options (KEYOPTS). For example, if your coupled field analysis includes nonlinearities, the application could automatically (and without your knowledge):

- Change the structural-thermal coupling from **Strong** to **Weak**.
- Specify that all coupling types use a uniform reduced integration scheme instead of Full Integration (as defined by the Brick Integration Scheme (p. 857) property of Part/Body objects).

As a result, to make sure that the application does not automatically change the element options, use the **Manual** setting for the **Element Control** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

See the Automatic Selection of Element Technologies and Formulations section of the Mechanical APDL Element Reference for more information.

Note:

- The **Physics Region** supports the activation of more than two physics types and therefore, coupling. In this case, there is a cumulative effect for each individual physics interaction with the other supported physics.
- If a body has more than two physics associated with its scoping, the interaction of the physics types is a combination of two different physics interactions.
- You can set the **Thermal Strain** property, of **Physics Region** object, to **Strong** (Matrix) coupling or **Weak** (Load Vector) coupling. Review the Coupled Effects section of the Coupling documentation in the *Mechanical APDL Theory Reference* for more background information.
- You can specify Thermoplasticity and Thermoviscoelasticity using the Plastic Heating (p. 1715) and Viscoelastic Heating (p. 1717) coupling conditions. Review the Thermoplasticity and Thermoviscoelasticity sections of the Coupling documentation in the *Mechanical APDL Theory Reference* for more background information.
- Joule Heating is considered as a load vector. Review the Thermoelectrics section of the Coupling documentation in the *Mechanical APDL Theory Reference* for more background information.
- For Piezoelectric coupling, the Piezoelectric matrix must be defined in the Engineering Data Workspace. You may want to review the Piezoelectrics section of the Coupling documentation in the *Mechanical APDL Theory Reference* for more background information.
- Currently, linear elements generated by the application use element SOLID226 (a high-order element) with the mid-side nodes dropped, **except** for structural-thermal analyses. For structural-thermal analyses, the application uses element SOLID225.

Coupled Field Simulations

The following sections discuss the steps and requirements to perform, processing limitations of, and industry-based applications for, coupled field simulations.

Coupled Field Harmonic Analysis

Prestressed Coupled Field Harmonic Analysis

Coupled Field Modal Analysis

Prestressed Coupled Field Modal Analysis

Coupled Field Static Analysis

Coupled Field Transient Analysis

Limitations

Application Examples and Background

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Coupled Field Harmonic Analysis

This analysis enables you to simulate the following physics types, independently or in combination, in a harmonic environment:

- Stand-alone Acoustics physics
- Piezoelectric Coupling
- Coupling of Structural and Acoustics physics
- Piezoelectric coupling with Acoustics physics

Note:

Piezoelectric analyses couple structural and electric physics with materials that have natural properties, such as quartz and ceramics.

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Points to Remember

- You can use an upstream Coupled Field Static (p. 391) analysis to prestress a Coupled Field Harmonic (p. 382) analysis.
- When beginning the analysis, you need to properly define the Physics Region object(s). The **Physics Region** object(s):
  - Is automatically included.
  - Requires each body of the model to be specified by a physics.
  - Defines the physics of the entire system/analysis.
  - Specifies physics type per body as needed.
  - Needs to be scoped to at least one body with more than one physics type or to an acoustic body.

This section assumes that you have an understanding of the general workflow for performing a simulation. As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview of the workflow.

Define Physics Region(s)

For this analysis, the physics types available for the **Physics Region** object include:

- Structural
- Acoustics

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Electric (Charge)

When **Structural** property is set to **Yes** and **Electric** property is set to **Charge**, the **Coupling Option** category, that includes the **Piezoelectric** property displays and indicates that the region is a piezoelectric region (read-only setting of **Yes**).

You can add **Physics Region** objects as desired by:

1. Highlighting the **Environment** object and selecting the **Physics Region** option on the Environment Context Tab (p. 61) or right-click the Environment object or within the **Geometry** window and select **Insert** > **Physics Region**.

2. Define all of the properties for the new object.

For additional information, see the Physics Region object reference section.

Specify Analysis Settings

The analysis type supports the following Analysis Settings (p. 1417):

- **Options** (p. 1447)
- **Advanced** (p. 1461)
- **Output Controls** (p. 1473)
- **Analysis Data Management** (p. 1484)

Apply Loads and Supports

The Environment Context tab (p. 61) provides the various groups of loads, supports, and conditions. In addition, the following **Electric** loads and boundary conditions are specific to **Coupled Field Harmonic** analysis:

- **Electric Charge** (p. 1625)
- **Voltage** (p. 1627) (Supports Phase Angle using Beta Options)
- **Voltage (Ground)** (p. 1633)
- **Voltage Coupling** (p. 1700)

See the Boundary Conditions (p. 1500) section for additional information.

Results

Note that for many results, the default setting for geometry selection is either **All Structural Bodies**, **All Acoustic Bodies**, or **All Electric Bodies**, depending on the given result type.

When you select the Solution (p. 62) Context tab, it provides the common result options as well as **Electric** (p. 2162) results. The Electric result drop-down menu contains the following results that are specific to this analysis:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Total Electric Flux Density**
- **Directional Electric Flux Density**

In addition, the Probes (p. 2163) drop-down menu includes **Charge Reaction** and **Impedance** probes and the Frequency Response (p. 2164) drop-down menu includes **Voltage**, **Charge Reaction**, and **Impedance** chart options. These result options are unique to this analysis type.

See the Using Results (p. 1927) section for more information.

Prestressed Coupled Field Harmonic Analysis

Mechanical enables you to specify an upstream Coupled Field Static (p. 391) analysis whose effects are used to perform a downstream Coupled Field Harmonic analysis. The supported physics combinations include:

- Coupling of Structural and Acoustics physics
- Piezoelectric Coupling
- Piezoelectric coupling with Acoustics physics
- Coupling of Electrostatic and Structural physics
- Electrostatic Structural coupling with Acoustics physics
- Electrostatic Structural coupling with Piezoelectric coupling

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Points to Remember

- To perform a prestressed Coupled Field Harmonic analysis you need to first perform a Coupled Field Static (p. 391) analysis, and properly link it to the downstream coupled field analysis.
- When beginning the analysis, you need to properly define the Physics Region object(s). The **Physics Region** object(s):
  - Is automatically included.
  - Requires each body of the model to be specified by a physics.
  - Defines the physics of the entire system/analysis.
  - Specifies physics type per body as needed.
  - Needs to be scoped to at least one body with more than one physics type or to an acoustic body.
- Any change to the physics settings in the upstream analysis will be automatically reflected in the downstream analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The **Thermal** physics type is not supported. If this physics type is active in the upstream analysis, the downstream analysis will be invalid.

This section assumes that you have an understanding of the general workflow for performing a simulation. As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview of the workflow.

Create Analysis System

Because this analysis is linked to (and based on) structural responses, a **Coupled Field Static** analysis is a prerequisite. This setup enables the two analysis systems to share resources, such as Engineering Data, Geometry, and the boundary condition type definitions that are defined the in the structural analysis.

From the **Toolbox**, drag a **Coupled Field Static** template to the Project Schematic. Then, drag a **Coupled Field Harmonic** template directly onto the **Solution** cell of the structural template.

Tip:

You can create a pre-stress environment in a **Coupled Field Harmonic** system that is already open in Mechanical by:

1. Selecting the **Coupled Field Static** option from the **Analysis** drop-down menu on the **Home** (p. 40) (or displayed) tab.
2. Setting the **Pre-Stress Environment** property (of the **Pre-Stress** object) to the **Coupled Field Static** system.

Specify Analysis Settings

The downstream Coupled Field Harmonic analysis supports the following Analysis Settings (p. 1417):

- **Options** (p. 1447)
- **Rotordynamics Controls** (p. 1490)
- **Advanced** (p. 1461)
- **Output Controls** (p. 1473)
- **Damping Controls** (p. 1462)
- **Analysis Data Management** (p. 1484)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Define Initial Conditions

The Pre-Stress object of the **Coupled Field Harmonic** analysis must point to the linked **Coupled Field Static** analysis.

Note:

- All structural loads, including Inertial (p. 1500) loads, such as Acceleration and Rotational Velocity, are deleted from the Harmonic analysis portion of the simulation once the loads are applied as initial conditions (p. 349). This is a result of the **Load Control** property of the Pre-Stress object being set (by default) to the **Keep All Displacements as Zero**. You can modify this to property to control load generation. See the description of the **Load Control** property in the Pre-Stress object reference for more information. Also see the Mechanical APDL command **PERTURB,HARM,,,DZEROKEEP** for more details.
- For **Pressure** boundary conditions in the Static Structural analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different results in the follow-on Full-Harmonic Analysis. See the Pressure Load Stiffness (p. 353) topic in the Applying Pre-Stress Effects for Implicit Analysis (p. 352) Help Section for more information about using a pre-stressed environment.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement**, or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Harmonic solution. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution. If you define a **Nodal Displacement** in the Harmonic analysis at the same location and in the same direction as in the Structural Static analysis, it overwrites the previous loading condition and/or boundary condition in the Harmonic solution.
- If a voltage is defined in the Coupled Field Static analysis, all non-zero voltages are set to zero for the Harmonic solution. If you define voltage at the same location in the Harmonic solution, the previous loading is overwritten.

Apply Loads and Supports

The Environment Context tab (p. 61) provides the following for the prestressed Coupled Field Harmonic analysis:

Inertial	Acoustic Excitations
Acceleration (p. 1501) (Phase Angle is not supported.)	Mass Source Surface Velocity

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Acoustic Loads

- Temperature
- Impedance Sheet
- Static Pressure

Acoustic Boundary Conditions

- Pressure
- Impedance Boundary
- Absorption Surface
- Radiation Boundary
- Absorption Element
- Free Surface
- Thermo-Viscous BLI Boundary
- Rigid Wall
- Symmetry Plane
- Port
- Far-field Radiation Surface

Electric

- Voltage (p. 1627) (can be scoped to **Voltage Coupling** defined in upstream Coupled Field Static[a])
- Voltage (Ground) (p. 1633)
- Voltage Coupling (p. 1700)

- Diffuse Sound Field
- Incident Wave Source
- Port in Duct

Note:

Incident Wave Source and Port in Duct must use a Port defined in the Harmonic Acoustics analysis.

Direct FE

- Nodal Force (p. 1721)
- Nodal Pressure (p. 1724) (Phase Angle is not supported.)
- Nodal Displacement (p. 1728) - At least one non-zero Component is required for the boundary condition to be fully defined.

These load can only be scoped to node-based Named Selection and only support constant loading.

Note:

Any other boundary conditions must be defined in the prerequisite (parent) static Analysis, such as Support Type boundary conditions.

[a] If you apply a **Voltage** load, without the use of **Voltage Coupling** scoping option, to any of the geometries included in the **Voltage Coupling** condition defined in the upstream system, the entities will no longer be coupled and will be overwritten.

Results

See the Using Results (p. 1927) section for descriptions of all supported result types.

All results generally default to the corresponding physics setting, that is, **All Acoustic Bodies**, **All Structural Bodies**, or **All Electric Bodies**. You can individually scope most results to mesh or geometric entities on bodies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Coupled Field Modal Analysis

This analysis enables you to simulate the following physics types, independently or in combination, in a modal environment:

- Stand-alone Acoustics physics
- Coupling of Structural and Acoustics physics
- Piezoelectric Coupling
- Piezoelectric coupling with Acoustics physics

Note:

Piezoelectric analyses couple structural and electric physics with materials that have natural properties, such as quartz and ceramics.

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Points to Remember

- You can use an upstream Coupled Field Static (p. 391) analysis to prestress a Coupled Field Modal (p. 388) analysis.
- When beginning the analysis, you need to properly define the Physics Region object(s). The **Physics Region** object(s):
  - Is automatically included.
  - Requires each body of the model to be specified by a physics.
  - Defines the physics of the entire system/analysis.
  - Specifies physics type per body as needed.
  - Needs to be scoped to at least one body with more than one physics type or to an acoustic body.

This section assumes that you have an understanding of the general workflow for performing a simulation. As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview of the workflow.

Define Physics Region(s)

For this analysis, the physics types available for the **Physics Region** object include:

- Structural
- Acoustics

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Electric (Charge)

When **Structural** property is set to **Yes** and **Electric** property is set to **Charge**, the **Coupling Option** category, that includes the **Piezoelectric** property displays and indicates that the region is a piezoelectric region (read-only setting of **Yes**).

You can add **Physics Region** objects as desired by:

1. Highlighting the **Environment** object and selecting the **Physics Region** option on the Environment Context Tab (p. 61) or right-click the Environment object or within the **Geometry** window and select **Insert** > **Physics Region**.
2. Define all of the properties for the new object.

For additional information, see the Physics Region object reference section.

Specify Analysis Settings

The analysis type supports the following Analysis Settings (p. 1417):

- **Solver Controls** (p. 1428)
- **Rotordynamics Controls** (p. 1490)
- **Options** (p. 1447)
- **Advanced** (p. 1461)
- **Damping Controls** (p. 1462)
- **Output Controls** (p. 1473)
- **Analysis Data Management** (p. 1484)

Apply Loads and Supports

The Environment Context tab (p. 61) provides the various groups of loads, supports, and conditions. In addition, the following **Electric** loads and boundary conditions are specific to **Coupled Field Modal** analyses:

- **Voltage (Ground)** (p. 1633)
- **Voltage Coupling** (p. 1700)

See the Boundary Conditions (p. 1500) section for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Results

Note that for many results, the default setting for geometry selection is either **All Structural Bodies**, **All Acoustic Bodies**, or **All Electric Bodies**, depending on the given result type.

Important:

To evaluate the Electric Field Intensity or Electric Flux Density results, be sure that you set the properties that compute the elemental results (Stress, Strain, etc.) in the **Output Controls** of the **Analysis Settings** to **Yes**.

See the Using Results (p. 1927) section for more information.

Prestressed Coupled Field Modal Analysis

Mechanical enables you to specify an upstream Coupled Field Static (p. 391) analysis whose effects are to be used to perform a downstream Coupled Field Modal analysis. The supported physics combinations include:

- Coupling of Structural and Acoustics physics
- Piezoelectric Coupling
- Piezoelectric coupling with Acoustics physics
- Coupling of Electrostatic and Structural physics
- Electrostatic Structural coupling with Acoustics physics
- Electrostatic Structural coupling with Piezoelectric coupling

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Points to Remember

- To perform a prestressed **Coupled Field Modal** analysis you need to first perform a **Coupled Field Static** analysis, and properly link it to the downstream coupled field analysis.
- When beginning the analysis, you need to properly define the Physics Region object(s). The **Physics Region** object(s):
  - Is automatically included.
  - Requires each body of the model to be specified by a physics.
  - Defines the physics of the entire system/analysis.
  - Specifies physics type per body as needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Needs to be scoped to at least one body with more than one physics type or to an acoustic body.

- Any change to the physics settings in the upstream analysis will be automatically reflected in the downstream analysis
- The **Thermal** physics type is not supported. If this physics type is active in the upstream analysis, the downstream analysis will be invalid.

This section assumes that you have an understanding of the general workflow for performing a simulation. As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview of the workflow.

Create Analysis System

Because this analysis is linked to (and based on) structural responses, a **Coupled Field Static** analysis is a prerequisite. This setup enables the two analysis systems to share resources, such as Engineering Data, Geometry, and the boundary condition type definitions that are defined the in the structural analysis.

From the **Toolbox**, drag a **Coupled Field Static** template to the Project Schematic. Then, drag a **Coupled Field Modal** template directly onto the **Solution** cell of the structural template.

Tip:

You can create a pre-stress environment in a **Coupled Field Modal** system that is already open in Mechanical by:

1. Selecting the **Coupled Field Static** option from the **Analysis** drop-down menu on the **Home** (p. 40) (or displayed) tab.
2. Setting the **Pre-Stress Environment** property (of the **Pre-Stress** object) to the **Coupled Field Static** system.

Specify Analysis Settings

The downstream Coupled Field Modal analysis supports the following Analysis Settings (p. 1417):

- **Options** (p. 1447)
- **Solver Controls** (p. 1428)
- **Rotordynamics Controls** (p. 1490)
- **Advanced** (p. 1461)
- **Output Controls** (p. 1473)
- **Analysis Data Management** (p. 1484)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Define Initial Conditions

The Pre-Stress object of the Coupled Field Modal analysis must point to the upstream Coupled Field Static analysis.

Note:

- When you perform a pre-stressed Modal analysis, the support conditions from the static analysis are used in the Modal analysis. You cannot apply any new supports in the Modal analysis portion of a pre-stressed modal analysis. When you link your Modal analysis to a Structural analysis, all structural loading conditions, including Inertial loads, such as Acceleration and Rotational Velocity, are deleted from the Modal portion of the simulation. This is a result of the **Load Control** property of the Pre-Stress object being set (by default) to the **Keep All Displacements as Zero**. You can modify this to property to control load generation. See the description of the **Load Control** property in the Pre-Stress object reference for more information. Also see the Mechanical APDL command **PERTURB,MODAL,,,DZEROKEEP** for more details.
- For a Pressure (p. 1521) load in the Coupled Field Static analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure to act as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For the same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different modal results in the follow-on Modal Analysis. See the Pressure Load Stiffness (p. 353) topic in the Applying Pre-Stress Effects for Implicit Analysis (p. 352) section for more information about using a pre-stressed environment.
- If displacement loading is defined with either the Displacement (p. 1666), Remote Displacement (p. 1674), Nodal Displacement (p. 1728), or Bolt Pretension (p. 1566) (specified as a Lock, Adjustment, or Increment) in the Static Structural analysis, these loads become fixed boundary conditions for the Modal solution. If the Modal solution is followed by a Harmonic solution, these displacement loads become fixed boundary conditions for the Harmonic solution as well. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution.

Apply Loads and Supports

The Environment Context tab (p. 61) provides the following for the prestressed Coupled Field Modal analysis:

Acoustic Loads	Acoustic Boundary Conditions
Temperature	Pressure
Impedance Sheet	Impedance Boundary
	Absorption Surface

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Static Pressure	Radiation Boundary
	Absorption Element
	Free Surface
	Thermo-Viscous BLI Boundary
	Rigid Wall
	Symmetry Plane
	Port
	Far-field Radiation Surface

Any structural and/or electric support used in the Coupled Field Static analysis persists. Therefore, you are not allowed to add new support in the pre-stressed Coupled Field Modal analysis.

Results

See the Using Results (p. 1927) section for descriptions of all supported result types.

All results generally default to the corresponding physics setting, that is, **All Acoustic Bodies**, **All Structural Bodies**, or **All Electric Bodies**. You can individually scope most of the results to mesh or geometric entities on bodies.

Coupled Field Static Analysis

The Coupled Field Static analysis simulates the steady state interaction of different physics types (see below). This section assumes that you have an understanding of the general workflow for performing a simulation. See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-thermal solutions as well as some examples.

This analysis enables you to simulate the following physics types, independently or in combination, in a static environment:

- Coupling of Structural and Thermal physics
- Coupling of Structural and Acoustics physics
- Piezoelectric (charge-based) coupling
- Piezoelectric (charge-based) coupling with Acoustics physics
- Coupling of Thermal and Electric conduction physics
- Coupling of Structural and Thermoelectric conduction
- Stand-alone Electrostatic
- Coupling of Electrostatic and Structural physics
- Electrostatic Structural coupling with Acoustics physics

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Electrostatic Structural coupling with Piezoelectric coupling

Note:

Piezoelectric (charge-based) analyses couple structural and electric physics with materials that have natural properties, such as quartz and ceramics.

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Points to Remember

When beginning the analysis, you need to properly define the:

- The application automatically inserts an Initial Physics Options object in order to specify an initial and reference temperature for the analysis.
- To simulate the thermoviscoelasticity coupling effect, the **Viscoelastic Heating** condition (p. 1717) must be scoped to a body whose material assignment includes the Viscoelastic material properties **Prony Shear Relaxation** and **Prony Volumetric Relaxation**, as defined in Engineering Data.
- To simulate the thermoplasticity coupling effect, the **Plastic Heating** (p. 1715) condition object can be added and must be scoped to bodies whose material properties has the Plasticity effects
- When performing an Electrostatic Structural analysis:
  - Set the **Large Deflection** property (**Analysis Settings** > Solver Controls (p. 1428)) to **On**.
  - Specify your mesh using a single layer of low-order elements (no mid-side nodes) to avoid air mesh distortion. A quadrilateral mesh that collapses uniaxially typically works best.
- When you specify a **Physics Region** using only the **Electric** property, set to **Charge** (electrostatic elements), make sure that the material assigned to the scoped bodies includes relative permittivity definition.
- When your analysis includes electrostatic elements, the application automatically sets the electric charge reaction sign to negative. See the SOLID122 section (or PLANE121 and SOLID123) of the *Mechanical APDL Element Reference* for this KEYOPT setting.
- When performing a standalone Electrostatic analysis, the application automatically specifies the **Contact Region** object property, Small Sliding (p. 1244) as **No**. This applies to all contact conditions.

As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Recommendation for Contact Stiffness

When your analysis couples electrostatic and structural physics and a contact condition is defined, using the default setting (**Program Controlled**) for the **Formulation** property, and the bodies in contact have higher stiffness differences between them, Ansys recommends that you manually specify the **Normal Stiffness** property using the **Absolute Value** option. When selected, the **Normal Stiffness Value** property displays. You define the entry as follows:

Absolute Value = SMAX/(SC x D)

Where:

SMAX (Electrostatic Force per unit area) = (x V^2)/(2D^2)

V = Potential difference across the air gap.

D = Thickness of the air gap (minimum value).

= The absolute permittivity of the air gap.

SC (scale factor) = 1e-4 (refine for higher accuracy).

Review the description of the Normal Stiffness (p. 1249) property for additional information.

Define Initial Physics Options

Specify the temperature settings and values of the **Initial Physics Options** object. You use the **Initial Physics Options** object to specify the initial temperature and reference temperature of the parts/bodies specified as either Thermal or Structural (using the Physics Region object) during a Coupled Field Static (p. 391) analysis. For the Structural Settings, you specify a Reference Temperature. Typically for most other analysis types in Mechanical, you define a Reference Temperature from the Environment object.

Define Physics Region(s)

During a Coupled Field analysis, a **Physics Region** object is automatically included. All of the bodies of the model must have a physics type specified by a **Physics Region** object. You use this object to specify the geometry bodies that belong to the supported physics types. By default, the **Structural** and **Thermal** properties are set to **Yes**. The **Coupled Field Static** analysis provides the following physics types.

- **Structural**
- **Acoustics**
- **Thermal**: Note that when the **Structural** and **Thermal** properties are set to **Yes** (default settings), the **Coupling Options** category displays. This category includes the **Thermal Strain** property. You use this property to specify the thermoelasticity coupled effects included through the **Thermal Strain**. Options include **Program Controlled**, **Strong**, and **Weak**.
- **Electric**: The options for this property include **No**, **Charge**, and **Conduction**. Review the Physics Region object reference page for property descriptions.

You can add **Physics Region** objects as desired by:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Highlighting the **Environment** object and selecting the **Physics Region** option on the Environment Context Tab (p. 61) or right-click the Environment object or within the **Geometry** window and select **Insert** > **Physics Region**.
2. Define all of the properties for the new object.

For additional information, see the Physics Region object reference section.

Specify Analysis Settings

The analysis type supports the following Analysis Settings:

- Step Controls (p. 1418)
- Solver Controls (p. 1428)
- Rotordynamics Controls (p. 1490)
- Restart Controls (p. 1438)
- Restart Analysis (p. 1437)
- Radiosity Controls (p. 1446)
- Nonlinear Controls (p. 1468)

> **Note:**
>
> To improve convergence for thermal-electric coupling, set the **Nonlinear Controls** property, **Line Search**, to the **On** setting.

- Output Controls (p. 1473)
- Analysis Data Management (p. 1484)
- Visibility (p. 1491)

Apply Loads and Supports

The Environment Context tab (p. 61) provides the various groups of loads, supports, and conditions, including various Acoustic loads and boundary conditions (p. 1520) and the following **Electric** loads and boundary conditions are specific to **Coupled Field Static** analyses:

- **Electric Charge** (p. 1625)
- **Voltage** (p. 1627) (Supports Phase Angle)
- **Voltage (Ground)** (p. 1633)
- **Current** (p. 1630)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

In addition, and depending upon physics definitions, the following Conditions (p. 1697) are specific to Coupled Field Static analyses:

- **Voltage Coupling** (p. 1700)
- **Plastic Heating** (p. 1715)
- **Viscoelastic Heating** (p. 1717)

See the Boundary Conditions (p. 1500) section for additional information.

Results

The Solution Context tab (p. 62) provides the various groups of result options. The analysis supports Structural (p. 2092), Thermal (p. 2155), and Electric (p. 2163) Probes. For many result objects, the default setting for geometry is either **All Structural**, **All Electric**, **All Acoustics** or **All Thermal Bodies**, depending on the given result type.

See the Using Results (p. 1927) section for more information.

Coupled Field Transient Analysis

This analysis enables you to simulate the following physics types, independently or in combination, in a transient environment:

- Coupling of Structural and Thermal physics
- Coupling of Structural and Acoustics physics
- Coupling of Thermal and Electric Conduction physics
- Coupling of Structural and Thermoelectric Conduction
- Stand-alone Acoustics physics
- Piezoelectric (charge-based) Coupling
- Piezoelectric coupling (charge-based) with Acoustics physics
- Coupling of Electrostatic and Structural physics
- Electrostatic Structural coupling with Acoustics physics
- Electrostatic Structural coupling with Piezoelectric coupling

Note:

Piezoelectric analyses couple structural and electric physics with materials that have natural properties, such as quartz and ceramics.

See the Application Examples and Background (p. 400) section for an overview of types of problems that use coupled structural-electric solutions as well as some examples. Also see the Acoustics Analysis Overview section for more detailed information about performing an acoustics analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Points to Remember

- The application automatically inserts an Initial Physics Options object for this analysis type.
- The **Physics Region** object(s):
  - Is automatically included.
  - Requires each body of the model to be specified by a physics.
  - Defines the physics of the entire system/analysis.
  - Specifies physics type per body as needed.
  - Needs to be scoped to at least one body with more than one physics type or to an acoustic body.
- To simulate the thermoviscoelasticity coupling effect, the **Viscoelastic Heating** condition (p. 1717) must be scoped to a body whose material assignment includes the Viscoelastic material properties **Prony Shear Relaxation** and **Prony Volumetric Relaxation**, as defined in Engineering Data.
- To simulate the thermoplasticity coupling effect, the **Plastic Heating** (p. 1715) condition object can be added and must be scoped to bodies whose material properties has the Plasticity effects
- When performing an Electrostatic Structural analysis:
  - Set the **Large Deflection** property (**Analysis Settings** > Solver Controls (p. 1428)) to **On**.
  - Specify your mesh using a single layer of low-order elements (no mid-side nodes) to avoid air mesh distortion. A quadrilateral mesh that collapses uniaxially typically works best.

As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the of general analysis workflow.

Recommendation for Contact Stiffness

When your analysis couples electrostatic and structural physics and a contact condition is defined, using the default setting (**Program Controlled**) for the **Formulation** property, and the bodies in contact have higher stiffness differences between them, Ansys recommends that you manually specify the **Normal Stiffness** property using the **Absolute Value** option. When selected, the **Normal Stiffness Value** property displays. You define the entry as follows:

Absolute Value = SMAX/(SC x D)

Where:

SMAX (Electrostatic Force per unit area) = (x V^2)/($2D^2$)

V = Potential difference across the air gap.

D = Thickness of the air gap (minimum value).

= The absolute permittivity of the air gap.

SC (scale factor) = 1e-4 (refine for higher accuracy).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Review the description of the Normal Stiffness (p. 1249) property for additional information.

Define Initial Physics Options

Specify the temperature settings and values of the **Initial Physics Options** object. You use the **Initial Physics Options** object to specify the initial temperature and reference temperature of the parts/bodies specified as either Thermal or Structural (using the Physics Region object) during a Coupled Field Transient (p. 395) analysis. For the thermal field, you specify an Initial Temperature as either Uniform or Non-Uniform (Transient only). For the Structural Setting, you specify a Reference Temperature. Typically for most other analysis types in Mechanical, you define a Reference Temperature from the Environment object.

Important:

Currently, the Coupled Field Transient analysis only supports the **Uniform Temperature** option for the **Initial Temperature** property. However, the **Non-Uniform Temperature** setting is available when Beta Options are active.

Specify Analysis Settings

The analysis type supports the following Analysis Settings:

- Step Controls (p. 1418) (see below)
- Solver Controls (p. 1428)
- Restart Controls (p. 1438)
- Radiosity Controls (p. 1446)
- Nonlinear Controls (p. 1468)

Note:

To improve convergence for thermal-electric coupling, set the **Nonlinear Controls** property, **Line Search**, to the **On** setting.

- Output Controls (p. 1473)
- Damping Controls (p. 1462)
- Analysis Data Management (p. 1484)
- Visibility (p. 1491)

For a **Coupled Field Transient** analysis when the **Time Integration** property is set to **On** (default), based on the active physics of the environment, the following additional properties display and enable you to specify whether to turn a physics field on or off:

- **Structural Only**: Options include **Off** and **On** (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Thermal Only**: Options include **Off** and **On** (default) .
- **Electric Only**: Options include **Off** and **On** (default). Supported only for **Electric (Conduction)** physics only.

Note:

For the **Acoustics** and **Electric (Charge)** physics properties, **Time Integration** property is automatically set to **On**.

Note:

For thermal-electric coupling and stand-alone acoustics physics, the application uses the Newmark Time integration method. For all other coupling and physics combinations, application uses the HHT time integration method. See the Transient analysis section of the Mechanical APDL Theory Reference for more information.

Define Physics Region(s)

During a Coupled Field analysis, a **Physics Region** object is automatically included. All of the bodies of the model must have a physics type specified by a **Physics Region** object. You use this object to specify the geometry bodies that belong to the supported physics types. By default, the **Structural** and **Thermal** properties are set to **Yes**.

The **Coupled Field Transient** analysis provides the following physics types.

- **Structural**
- **Acoustics**
- **Thermal**: Note that when the **Structural** and **Thermal** properties are set to **Yes** (default settings), the **Coupling Options** category displays. This category includes the following properties:
  - **Thermal Strain**. You use this property to specify the thermoelasticity coupled effects included through the thermal strain. Options include **Program Controlled** (default), **Strong**, and **Weak**.
  - **Thermoelastic Damping**: Either **On** or **Off** (default).
- **Electric**: The options for this property include **No**, **Charge**, and **Conduction**. Review the Physics Region object reference page for property descriptions.

You can add **Physics Region** objects as desired by:

1. Highlighting the **Environment** object and selecting the **Physics Region** option on the Environment Context Tab (p. 61) or right-click the Environment object or within the **Geometry** window and select **Insert** > **Physics Region**.
2. Define all of the properties for the new object.

For additional information, see the Physics Region object reference section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Apply Loads and Supports

The Environment Context tab (p. 61) provides the various groups of loads, supports, and conditions, including various Acoustic loads and boundary conditions (p. 1520) and the following **Electric** loads and boundary conditions:

- **Electric Charge** (p. 1625)
- **Voltage** (p. 1627)
- **Voltage (Ground)** (p. 1633)
- **Current** (p. 1630)

In addition, and depending upon physics definitions, the following Conditions (p. 1697) are available:

- **Voltage Coupling** (p. 1700) (Electric)
- **Plastic Heating** (p. 1715) (Structural-Thermal)
- **Viscoelastic Heating** (p. 1717) (Structural-Thermal)

As needed, see the Boundary Conditions (p. 1497) section for additional information.

Results

The Solution Context tab (p. 62) provides the various groups of result options. The analysis supports Structural (p. 2092), Thermal (p. 2155), and Electric (p. 2163) Probes. For many result objects, the default setting for geometry is either **All Structural**, **All Electric**, **All Acoustics** or **All Thermal Bodies**, depending on the given result type.

See the Using Results (p. 1927) section for more information.

Limitations

General Limitations

Currently, Coupled Field analyses do not support the following features/capabilities:

- Fracture
- Solution Combination
- Fatigue Combination
- Fatigue Tool
- Condensed Parts
- General Axisymmetric Symmetry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Type Limitations

Coupled Field Harmonic

The **Coupled Field Harmonic** analysis does not support the **MSUP** setting for the **Solution Method** property and cannot include an upstream pre-stress system.

Piezoelectric Analysis (Harmonic or Modal)

For a Piezoelectric analysis (Harmonic or Modal), any specifications in the Damping Controls (p. 1462) (**Analysis Settings**) do not contribute to the damping energy or associated losses. Instead, you need to specify damping in the Engineering Data workspace by defining the properties of the Material Dependent Damping group and/or **Damping Factors**.

Coupled Field Transient

The **Coupled Field Transient** analysis does not support the Mode-Superposition (MSUP) solution method. As a result, it cannot be linked to an upstream Coupled Field Modal analysis.

External Model Limitations

Note the following limitations associated with importing data from an **External Model** system:

- You can import structural and thermal data for **Coupled Field Static** and **Coupled Field Transient** analyses only.
- The lower-order 2D elements is not supported.

Boundary Condition Limitations

Boundary conditions cannot be imported from a Harmonic Response analysis to a Coupled Field Harmonic analysis.

And, when importing data from an External Data (p. 736) system, only the following imported loads (p. 1736) are supported:

- Imported Heat Flux
- Imported Temperature
- Imported Body Temperature (Imported Thermal Condition)

Application Examples and Background

Coupled Piezoelectric

You can employ piezoelectric solutions for problems such as:

- Direct piezoelectric effects for sensing technology and the converse effects for actuation technology.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Active noise control (ANC) and active vibration control (AVC) to minimize sound energy radiated by structures by using smart piezoelectric materials.
- Piezoelectric energy harvesting (PEH) to transform the kinetic energy of vibration and pressure into electric power.
- Non-destructive evaluation of structures looking at wave signatures, for example, diagnostic signs of construction defects by using piezoelectric materials as sensors and transducers.

Specific coupled structural-electrical engineering applications include:

- **Ultrasound Imaging**: Piezoelectric transducers are used in ultrasound imaging as a transmitter or receiver.
- **Oil and Gas Logging**: Piezoelectric transmitters and receivers are used extensively (in addition to acoustic-structural modeling) to ping the well casing and understand well integrity.
- **Underwater Sonar Application**: Piezoelectric material are used for wave generation and for receiving and interpreting the signals.
- **BAW/SAW Waveguides**: Piezoelectric waveguides are used to filter signals exploiting the resonance/antiresonance with applications in 5G technology.
- **Touchscreen Sensors**: Piezoelectric layers can act as pressure and force sensors. It can provide accurate, high-frequency, and rapid response. And it is widely used in industrial and aerospace applications.
- **Piezoelectric MEMS Microphones**: Piezoelectric microphones can provide large capacitance and it does not require bias voltage or backplate. It is also useful for prototyping microphones with unconventional geometries.
- **Piezoelectric Mass Sensor**: Piezoelectric devices are used for highly sensitive mass sensing by observing the shift in resonant frequencies.
- **Piezoelectric Gyroscopes**: Piezoelectric material induced vibrations in MEMS gyroscopes can be used for orientation measurements due to added Coriolis effect during rotation.
- **Piezoelectric Motors**: Ultra-sensitive piezoelectric linear and rotary motors can be used for nanometer scale precision on positioning with applications in various dynamic control applications.
- **Micro-Electro-Mechanical System (MEMS) Gyroscope**: As a part rotates, the Coriolis force will create an electrical current. MEMS gyroscopes are ultra-small, ultra-lightweight, and quick response.

Coupled Structural-Thermal

You can employ coupled structural-thermal solutions for problems such as:

- Large mechanical deformations where contact is established between surfaces late in the solution. These contact conditions form new heat flow pathways.
- Internal heat generation because of mechanical deformations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Heat generation due to relative sliding between contacting surfaces.
- Thermal properties problems where materials are dependent on the mechanical solution and vice-versa.
- Fracture or delamination problems where the material or structure undergoes deformations that modify heat flow pathways.
- Nonlinear thermal boundary conditions where the non-linearity is dependent on the Mechanical solution.
- Pressure and gap cases that depend on contact thermal properties.

Specific coupled structural-thermal engineering applications include:

- **Brake Pad Heating**: Relative sliding between the disc and brake pads cause significant frictional heat generation.
- **Plastic Seals**: Large plastic deformation of seals cause temperatures to rise because of plastic heating. This may lead to relaxation in contact pressure. In addition, when subjected to cyclic pressure loads, the contact surfaces may generate frictional heating.
- **Arc Welding**: High temperature material deposition (through element birth) and subsequent cooling may lead to distortions in the final geometrical shape because of thermal expansion/contraction.
- **Friction Stir Welding**: The process relies on frictional heat generation between the tool and the workpiece, this necessitates using coupled thermal-structural analysis.
- **Cancerous Tissue Ablation**: RF waves are used for internal heat generation in cancerous cells leading to ablation. Coupled thermal-structural analysis may be utilized in addition to model this effect (in addition to element death).
- **Metal Forming**: Plastic heat generation in regions undergoing large plastic deformations may result in contraction/expansion leading to distortion of the final part.
- **Vibration Isolation Pads**: For high frequency applications there may be an increase in temperature due to viscoelastic heating in vibration isolation pads leading to change in material response and reduced fatigue life.
- **Threaded Connectors**: For high temperature applications local plastic heating near the threads and frictional heating can lead to increase increased temperature, causing reduced fatigue life because of thermomechanical fatigue.
- **High-frequency Resonators**: Thermoelastic damping may affect the harmonic response of the resonators, coupled field thermal-structural solutions allow for including this effect.
- **Hyperelastic Seal Fatigue**: For high frequency loading, viscoelastic heating may lead to changes in material behavior and also reduce fatigue life, coupled thermal-structural solutions allow for including this effect.
- **Thermal Barrier/Coating Ablation**: Surface heat generation at the coating surfaces (such as in ceramic thermal protection systems in space shuttle) causes the surface to ablate. Coupled

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

thermal-structural analysis may be utilized in addition to model this effect (in addition to element death).

Electric Analysis

Introduction

An electric analysis supports Steady-State Electric Conduction. Primarily, this analysis type is used to determine the electric potential in a conducting body created by the external application of voltage or current loads. From the solution, other results items are computed such as conduction currents, electric field, and joule heating.

An Electric Analysis supports single and multibody parts. Contact conditions are automatically established between parts. In addition, an analysis can be scoped as a single step or in multiple steps.

An Electric analysis computes Joule Heating (p. 2162) from the electric resistance and current in the conductor. This joule heating may be passed as a load to a Thermal analysis (p. 657) simulation using an Imported Load (p. 1736) if the Electric analysis Solution data is to be transferred to Thermal analysis. Similarly, an electric analysis can accept a Thermal Condition (p. 1740) from a thermal analysis to specify temperatures in the body for material property evaluation of temperature-dependent materials.

Points to Remember

A steady-state electric analysis may be either linear (constant material properties) or nonlinear (temperature dependent material properties). Additional details for scoping nonlinearities are described in the Nonlinear Controls (p. 1468) section.

Once an Electric Analysis is created, Voltage (p. 1627) and Current (p. 1630) loads can be applied to any conducting body. For material properties that are temperature dependent, a temperature distribution can be imported using the Thermal Condition (p. 1740) option.

In addition, equipotential surfaces can be created using the Coupling Condition (p. 1698) load option.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag the **Electric** template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When using an Ansys license that includes the Emag license feature, only the following material properties are allowed: Isotropic Resistivity, Orthotropic Resistivity, Relative Permeability, Relative Permeability (Orthotropic), Coercive Force & Residual Induction, B-H Curve, B-H Curve (Orthotropic), Demagnetization B-H Curve. You may have to turn the filter off in the **Engineering Data** tab to suppress or delete those material properties/models that are not supported for the license.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

Note that 3D shell bodies and line bodies are not supported in an electric analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

Mechanical does not support Rigid Bodies in electric analyses. For more information, see the *Stiffness Behavior* documentation for Rigid Bodies (p. 852).

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

In an electric analysis, only bonded, face-face contact is valid. Any joints or springs are ignored. For perfect conduction across parts, use the MPC formulation. To model contact resistance, use Augmented Lagrange or Pure Penalty with a defined Electric Conductance (p. 1254).

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Only higher order elements are allowed for an electric analysis.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For an electric analysis, the basic Analysis Settings (p. 1417) include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Step Controls (p. 1418)

These properties are used to specify the end time of a step in a single or multiple step analysis.

Multiple steps are needed if you want to change load values, the solution settings, or the solution output frequency over specific steps. Typically you do not need to change the default values.

Output Controls (p. 1473)

These properties allow you to specify the time points at which results should be available for postprocessing. A multi-step analysis involves calculating solutions at several time points in the load history. However you may not be interested in all of the possible results items and writing all the results can make the result file size unwieldy. You can restrict the amount of output by requesting results only at certain time points or limit the results that go onto the results file at each time point.

Analysis Data Management (p. 1484)

Common **Analysis Data Management** properties are available for this analysis type.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

There is no initial condition specification for an Electric analysis.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are supported in a Steady-State Electric analysis:

- Voltage (p. 1627)
- Current (p. 1630)
- Coupling Condition (p. 1698) (Electric)
- Thermal Condition (p. 1594)

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

The **Solution Information** (p. 1833) object provides some tools to monitor solution progress.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the model during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Applicable results are all electric result types (p. 2162).

Once a solution is available, you can contour the results (p. 62) or animate the results (p. 2260) to review the responses of the model.

For the results of a multi-step analysis that has a solution at several time points, you can use probes (p. 1946) to display variations of a result item over the steps.

You may also wish to use the Charts (p. 1929) feature to plot multiple result quantities against time (steps). For example, you could compare current and joule heating. Charts can also be useful when comparing the results between two analysis branches of the same model.

Explicit Dynamics Analysis

Ansys Explicit Dynamics is a transient explicit dynamics Workbench application that can perform a variety of engineering simulations, including the modeling of nonlinear dynamic behavior of solids, fluids, gases and their interaction. Additionally, the LS-DYNA ACT extension is available to analyze a model using the LS-DYNA solver. Detailed information for running an Explicit Dynamics analysis can be found in the Explicit Dynamics Analysis Guide.

Fracture Analysis

Fracture analysis deals with the computation of fracture parameters that help you design within the limits of catastrophic failure of a structure. Fracture analysis assumes the presence of a crack in the structure. The fracture parameters computed are Stress Intensity Factors (SIFS), J-Integral (JINT), Energy Release Rates, Material Force, T-Stress and C*-Integral.

When a crack is associated with a SMART Crack Growth (p. 468) you can also compute Equivalent SIFS Range fracture results and time history results (Fracture Probes (p. 2127)) on any crack front node.

A Fracture analysis requires that you define a crack using an available crack definition. Since fracture parameter calculation requires knowledge of the mesh characteristics around the crack, the mesh must be generated before solving for fracture parameters. Fracture parameter computation is only applicable to Static Structural and Transient Structural analyses.

The following sections further describe the aspects of a Fracture Analysis as well as additional features available in the Fracture object of the application.

Fracture Analysis Workflows

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Limitations of Fracture Analysis

Fracture Meshing

Cracks

SMART Crack Growth

Interface Delamination and Contact Debonding

Multi-Point Constraint (MPC) Contact for Fracture

Solving a Fracture Analysis

Extracting Fracture Results from a Result File

Additional topics include:

Fracture Results (p. 2066)

Fracture Probes (p. 2127) (Supported when a crack is associated with a SMART Crack Growth.)

Mechanical APDL References

See the *Fracture Analysis Guide* in the Mechanical APDL documentation for additional information about fracture analyses. You may also wish to review the Fracture Analysis Benchmarks section. This section provides a set of benchmark examples that you can use to evaluate fracture-analysis capabilities. The benchmark results are compared with results from reference calculations, handbook solutions, and experimental testing.

Fracture Analysis Workflows

A Fracture analysis can be performed in two stages:

1. Defining a crack and computing fracture parameters.

Fracture parameters help to design engineering structures within limits of catastrophic failure. You can define a crack and compute the needed fracture parameters using the following methods:

- Define Crack Location using the Semi-Elliptical Crack, Elliptical Crack, or Ring Crack Object (p. 408).
- Define an Arbitrary Crack using a Surface Body (p. 410).
- Imported Crack Mesh (Pre-Meshed Crack) (p. 412).

2. Analyze/study the crack growth.

You can study the crack growth phenomenon using the SMART Crack Growth (p. 468) object. This object provides the following crack growth options:

- **Fatigue**: Use this option to model structures subjected to cyclic loading. Fatigue crack growth model used is Paris' law.
- **Static**: For this option, crack growth modeling is based on selected fracture parameters (SIFS or J-Integral) and criteria.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For additional technical information, refer to the Understanding Crack-Growth Mechanics section in the *Mechanical APDL Fracture Analysis Guide*.

This section describes the typical workflow for computing fracture parameters in the static structural analysis that contains cracks. The typical workflows are shown below:

Note:

For all workflows, the static structural analysis supports imported thermal loads from both steady-state thermal or transient thermal analysis by linking the set up cell of the static structural analysis to the upstream steady-state thermal or transient thermal analysis.

Define Crack Location using the Semi-Elliptical Crack/Elliptical Crack/Ring Crack Object

The steps shown below describe setting up the fracture analysis when using a Semi-Elliptical Crack, Elliptical Crack, or Ring Crack object.

Note:

You can also use these steps to compute fracture parameters in a Transient Structural analysis that contains cracks.

1. In Ansys Workbench, insert a Static Structural analysis in the project schematic.
2. Select a geometry.
3. Launch Mechanical.
4. Define a coordinate system location for the crack you will define. Note the following:
   - For semi-elliptical cracks, the specified coordinate system must be located on one of the surfaces of a solid body.
   - For elliptical and ring cracks, the specified coordinate system must be located inside of a solid body.
5. Align the axes of the coordinate system of the crack such that the direction of the specified coordinate system's Y-Axis is normal to the crack surface

   For Semi-Elliptical cracks lying on curved surfaces, ensure that the coordinate system's X-Axis is normal to the surface of the body at the coordinate system location. See Creating a Coordinate System Based on a Surface Normal (p. 1196) for details on how to orient such a coordinate system on a curved surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

6. Insert a **Fracture** folder in the **Outline**.

> **Note:**
>
> If you have an import base mesh that is hex-dominant, set **Re-mesh Hex-dominant to Tetrahedral** property of the **Fracture** folder to **On**. This setting automatically re-meshes an imported hex-dominant base mesh, on the solid body of a **Semi-Elliptical Crack** object, to a tetrahedral mesh. The resulting mesh is used to generate the initial crack mesh for the fracture parameters calculation and **SMART Crack Growth**.

7. Insert a **Semi-Elliptical Crack**, or an **Elliptical Crack**, or a **Ring Crack** object under the **Fracture** folder.
8. Specify the properties of the specified crack.
9. Generate the mesh by right-clicking the **Fracture** folder and selecting **Generate All Crack Meshes**.
10. Apply loads and boundary conditions. As needed, apply pressure on the crack face using a Nodal Pressure (p. 1724). Nodal pressure can be scoped using the automatically generated crack face Named Selection created under the crack object.
11. Ensure the **Fracture** setting under **Fracture Controls** (p. 1445) in the Analysis Settings is turned **On**.
12. Solve the simulation.
13. Insert the Fracture Tool and specify the desired crack using the **Crack Selection** property.
14. Insert desired fracture results under the **Fracture Tool**.
15. Post-process the fracture results.
16. Export to Excel or copy/paste from the chart if necessary.

Supplementary Steps to Define Semi-Elliptical Crack/Elliptical Crack/Ring Crack Object on Imported Base Mesh

Use the following steps to configure your simulation on the **Project Schematic** in order to specify an **Semi-Elliptical Crack**/**Elliptical Crack**/**Ring Crack** object on a mesh imported from External Model or Mechanical Model (p. 989).

External Model

1. From Ansys Workbench, insert an **External Model** system into the **Project Schematic**.
2. Open the **Setup** cell of the **External Model** tab and select the desired .cdb-based mesh that contains a solid body with tetrahedron/hex-dominant mesh. Return to the Project page and update the system.
3. Insert a **Static Structural** analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Connect the fully-defined **External Model** upstream system to the downstream **Static Structural** analysis.

5. Launch Mechanical from the **Static Structural** system and then follow the steps from the above **Define Crack Location using the Semi-Elliptical Crack/Elliptical Crack/Ring Crack Object** topic, beginning with Step 4.

Mechanical Model

1. From Ansys Workbench, insert a **Mechanical Model** system into the **Project Schematic**.

2. Using the **Geometry** cell, specify solid body/bodies needed to define the crack.

3. Open the **Model** cell of the **Mechanical Model** system and mesh the solid bodies with either tetrahedron/hex-dominant mesh.

4. Return to the **Project** page and update the **Mechanical Model** system.

5. Insert a **Static Structural** analysis.

6. Connect the fully-defined **Mechanical Model** upstream system to the downstream **Static Structural** analysis.

7. Launch Mechanical from the **Static Structural** system and then follow the steps from the above **Define Crack Location using the Semi-Elliptical Crack/Elliptical Crack/Ring Crack Object** topic, beginning with Step 4.

Define an Arbitrary Crack using a Surface Body

The steps shown below describe how to set up a fracture analysis using the **Arbitrary Crack** object during a Static Structural analysis. You can follow same steps to set up fracture analysis for a Transient Structural Analysis.

1. From Ansys Workbench, insert a Static Structural analysis into the **Project Schematic**.

2. Input the geometry, which has a surface body (and will represent crack surface).

3. Launch Mechanical.

4. Create a Coordinate System. The Y axis must be directed towards the normal of the crack's top face and the X axis helps to determine the crack extension direction.

5. Insert a **Fracture** folder in to the tree **Outline**.

Note:

If you have an import base mesh that is hex-dominant, set **Re-mesh Hex-dominant to Tetrahedral** property of the **Fracture** folder to **On**. This setting automatically re-meshes an imported hex-dominant base mesh, on the solid body of a **Arbitrary Crack** object, to a tetrahedral mesh. The resulting mesh is used to generate the initial crack mesh for the fracture parameters calculation and **SMART Crack Growth**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

6. Using the Fracture Context Tab (p. 52) or the context menu (right-click the folder), insert an **Arbitrary Crack** object into the **Fracture** folder.
7. Specify the properties of the Arbitrary Crack object.
8. Right-click the **Fracture** folder and select **Generate All Crack Meshes** to generate the mesh.
9. Apply loads and boundary conditions. As needed, apply pressure on the crack face using the Nodal Pressure (p. 1724) boundary condition. You can scope this boundary condition using the automatically generated crack-face Named Selection created under the object.
10. Make sure that the **Fracture** property under the **Fracture Controls** (p. 1445) of the Analysis Settings is turned to **On**.
11. Solve.
12. Add the Fracture Tool and scope it to the **Arbitrary Crack** object and add desired fracture results.
13. Post process the fracture results.
14. Export to Excel or copy/paste from the chart if necessary.

Supplementary Steps to Define Arbitrary Crack on Imported Base Mesh

Use the following steps to configure your simulation on the **Project Schematic** in order to specify an **Arbitrary Crack** on a mesh imported from **External Model**.

1. From Ansys Workbench, insert an External Model system into the **Project Schematic**.
2. Open the **External Model** tab and select the desired .cdb-based mesh that contains a solid body (that may also contain a surface body to define the crack shape) with tetrahedron mesh. Return to the Project page and update the system.

   Note:

   If the imported .cdb- based mesh file contains a surface body that can be used to define the crack shape, you can jump to Step 6.

3. Insert a Mechanical Model system.

   Note:

   If the imported .cdb-based mesh doesn't contain a surface body, then the **Mechanical Model** system is required to import the surface body (including its mesh) individually for later use when defining the Arbitrary Crack surface.

4. Using the **Geometry** cell of the Mechanical Model system, specify a surface body. This surface body represents crack surface. The crack surface must intersect with at least one face of the solid body specified in the External Model system and must not be embedded inside of the solid body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

5. Update the Mechanical Model system
6. Insert a **Static Structural** analysis.
7. Connect the fully-defined External Model and Mechanical Model (if defined above) upstream systems to the downstream **Static Structural** analysis.

Note:

Review Assembling External Models and Mechanical Models (p. 989) section for additional information.

---

8. Launch Mechanical from the **Static Structural** system and then follow the steps from the above **Define an Arbitrary Crack using a Surface Body** topic, beginning with Step 4.

Important:

- Arbitrary Crack mesh generated on an imported base mesh could affect the existing mesh (nodal and element) based Named Selection objects.
- When you have an Arbitrary Crack specified on an imported base mesh, the application of imported loads or constraints is not supported because the Arbitrary Crack mesh generation could affect the existing mesh (nodes) during remeshing.

Imported Crack Mesh (Pre-Meshed Crack)

This workflow describes using the Pre-Meshed Crack object for the computation of fracture parameters in 2D and 3D analysis using imported crack mesh. You can follow same steps to set up a Fracture analysis for a Transient Structural Analysis.

1. From Ansys Workbench, insert an External Model system into the Project Schematic.
2. Open the **External Model** tab and select the desired .cdb-based mesh that contains the crack mesh and its definition. Return to the Project page and update the system.
3. Insert a **Static Structural** analysis in the Project Schematic.
4. Connect the **Setup** cell of the External Model system to **Model** cell of the Static Structural system.
5. Launch Mechanical.
6. Create a Coordinate System with a Y axis perpendicular to the crack faces.
7. Insert a **Fracture** folder in the **Outline**.
8. Insert a **Pre-Meshed Crack** object under the **Fracture** folder.
9. Specify the **Pre-Meshed Crack** object details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

10. Associate the **Pre-Meshed Crack** object with the newly created Coordinate System.
11. Apply load and boundary conditions.

> **Note:**
>
> You cannot apply Pressure loads to the crack face. You can only apply Nodal Pressures (p. 1724) on a structured mesh via node-based Named Selections for the nodes defining the crack face.

12. Ensure the **Fracture** setting under **Fracture Controls** (p. 1445) in the Analysis Settings is turned **On**.
13. Solve.
14. Add the Fracture Tool and desired fracture results.
15. Post process the fracture results.
16. Export to Excel or copy/paste from the chart if necessary.

> **Note:**
>
> In 2D, you can draw the crack in the same model using DesignModeler and generate the crack mesh using the mesh connection feature.

Limitations of Fracture Analysis

This section describes the limitations for the generation of a crack mesh on the supported crack types (semi-elliptical, elliptical, ring, or arbitrary). In addition, it describes the limitations in the computation of fracture parameters for these crack types, including the Pre-Meshed Crack object.

1. A Fracture analysis does not support adaptive mesh refinement.
2. For **Arbitrary Crack**, **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack Corner Crack**, **Edge Crack**, **Through Crack**, and **Cylindrical Crack** objects, crack meshing requires that the base mesh is a quadratic tetrahedron mesh. Linear elements may exist farther away from the buffer zone on the same body to which these cracks are scoped.
3. Only 3D analyses support semi-elliptical, elliptical, ring, and arbitrary cracks.
4. You can scope a **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, **Corner Crack**, **Edge Crack**, **Through Crack**, or **Cylindrical Crack** to one solid body only.
5. You should define Semi-Elliptical cracks on the surfaces of a solid body only, and the crack cannot span more than one face.
6. When the crack dimensions of an **Elliptical Crack** or a **Ring Crack** intersect with the free surfaces of the solid body, the application only supports the **Tetrahedrons** option of the **Mesh Method** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

7. The **Stiffness Behavior** property of the geometry scoped to a **Arbitrary Crack**, **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, **Corner Crack**, **Edge Crack**, **Through Crack**, or **Cylindrical Crack** must be set to **Flexible**.

8. You can scope an **Arbitrary Crack** to one solid body only. And, you can only scope the **Crack Surface** property of the crack to a single surface body.

9. The scoped crack front nodal selection of the **Pre-Meshed Crack** object must exist in geometries with a flexible stiffness behavior definition.

10. It is not recommended that you specify a Part Transformation (p. 1014) on a body that includes an **Arbitrary Crack**. This could lead to the application detecting an incorrect extension direction for the crack front nodes.

11. Arbitrary Cracks can only be meshed with the Tetrahedrons Mesh Method (**Mesh Method** set to **Tetrahedrons**).

12. Fracture parameter computations based on the VCCT technique are only supported for lower order crack mesh. Hence, VCCT based fracture parameter computations are only supported for **Pre-Meshed Crack** object.

13. In order to retain restart points when the **Fracture** property (**Analysis Settings** > Fracture Controls (p. 1445)), is set to **On**, you must set the **Generate Restart Points** property (**Analysis Settings** > Restart Controls (p. 1438) to **Manual**.

14. Solution restarts are not supported if you are computing fracture parameters for **SMART Crack Growth**.

15. The semi-elliptical, elliptical, ring, or arbitrary crack top and bottom face nodes are not connected through any constraint equation. Therefore, the nodes of the top face can penetrate the bottom face or vice versa based on the applied loads and constraints. In these scenarios, you may need to create a constraint equation between crack faces for the solution using the Commands (APDL) object.

16. The graphical view of a **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, **Corner Crack**, **Edge Crack**, **Through Crack**, or **Cylindrical Crack** may differ from the generated mesh. For more information, see the section on Cracks (p. 424).

17. The **Arbitrary Crack**, **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, **Corner Crack**, **Edge Crack**, **Through Crack**, or **Cylindrical Crack** objects are not supported in combination with the following features:

    - Cyclic Region
    - Structural Linear Periodic Symmetry Region
    - Interface Delamination

18. Interpolated displacements for the facets in a surface construction object may fail to demonstrate the proper deformation of a Semi-Elliptical crack. For more information, see Surface Displays and Fracture (p. 1961).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

19. In order to use the Fracture Tool to extract the fracture results and probe results from the results file of another analysis using the **Read Result Files** option, it is necessary that the meshes from both systems, including the crack mesh, match.

20. Fracture parameter calculations based on domain integrations such as SIFs, J-integral, or Material Force are not supported when contact elements exist inside the domain. The calculations may become path-dependent unless the contact pressure is negligible.

21. If you have a Hyperelastic material assigned in the analysis, only Material Force fracture results are supported.

Fracture Meshing

The Meshing application supports fracture meshing capabilities enabling you to insert multiple surface cracks and/or multiple embedded cracks (Elliptical and Ring) into the body mesh. This fracture mesh can then be used to analyze crack fronts in Static Structural and Transient structural analyses. Fracture meshing uses a Fracture object that can contain multiple Arbitrary Crack, Semi-Elliptical, Elliptical Crack, and/or Ring Crack objects.

Go to a section topic:

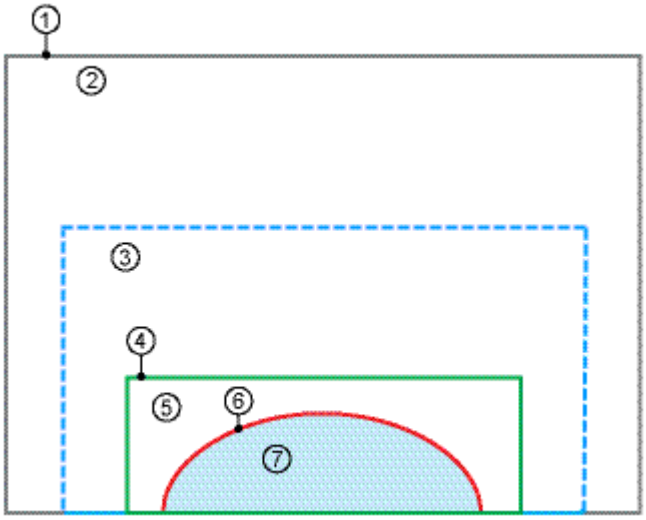
- Overview (p. 415)
- Characteristics and Limitations of Fracture Meshing (p. 416)
  - General (p. 417)
  - For Semi-Elliptical Cracks (p. 419)
  - For Arbitrary Cracks (p. 421)

Overview

The following figure illustrates many of the components of fracture meshing that are discussed throughout this section. This figure is a sliced top view of a semi-elliptical crack.

Overview of Fracture Meshing Components

DRAFT



1. Body to which the semi-elliptical crack is scoped.
2. Base mesh, which always consists of quadratic tetrahedron elements. For the generation of fracture mesh, the base mesh inside and surrounding the region of the buffer zone (3) must be a quadratic tetrahedron mesh. However, non-tetrahedron/linear mesh may exist farther away from the buffer zone on the same body to which the semi-elliptical crack is scoped.
3. Buffer zone, which is always filled with quadratic tetrahedron elements.
4. Interface between the buffer zone (3) and the fracture affected zone (5), illustrated by the green line. Contact pair 1 is defined at the interface between the elements in the buffer zone and the elements in the fracture affected zone.
5. Fracture affected zone, which is filled with quadratic elements (hex and wedge).
6. Crack front, located along the crack shape and illustrated by the red line. The crack shape is semi-elliptical.
7. Discontinuity planes, illustrated by the light blue shading. Consist of two planes at the same location (the crack front plane).

For semi-elliptical, elliptical, or ring cracks, you use the **Mesh Method** property to specify the mesh as either **Hex Dominant** (default) or **Tetrahedrons**.

Note:

When you specify the **Mesh Method** property as **Tetrahedrons** to mesh a **Semi-Elliptical Crack**, **Elliptical Crack**, or **Ring Crack**, the application does not create the **Fracture Affected Zone** (Step 5) and as a result the Interface (Step 4) does not exist. All other components remain the same.

Characteristics and Limitations of Fracture Meshing

Note the following information when using fracture meshing:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

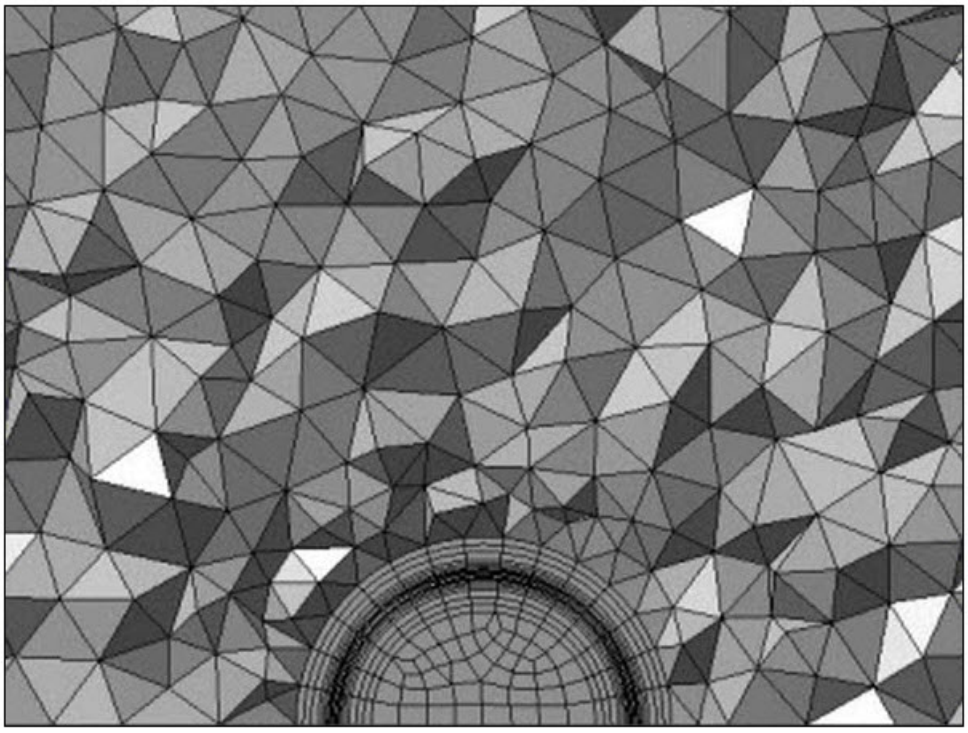
General

- Fracture meshing is a post mesh process which occurs during a separate step after the base mesh is generated. If you do not generate a base mesh before you invoke **Generate All Crack Meshes**, the base mesh is generated first and the crack meshing occurs in a separate step after base meshing is complete.
- When the **Mesh Method** property is set to **Tetrahedrons**, then meshing a Semi-Elliptical, Elliptical crack, Ring crack, and/or Arbitrary crack does not generate contact pairs in the interface region of the crack mesh and base mesh.

Note:

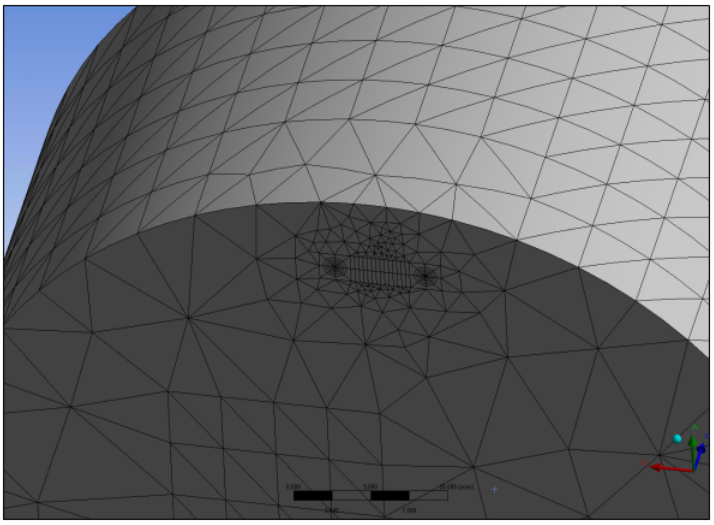
When the free surfaces of a solid body intersect with either an **Elliptical Crack** or a **Ring Crack**, the **Hex Dominant** setting for the **Mesh Method** property is not supported.

- For Semi-Elliptical, Elliptical, or Ring cracks with a Tetrahedron mesh, the **Largest Contour Radius** and **Mesh Contours** properties affect the crack mesh only when the **Front Element Size** property is set to **Default**. These properties do not have an effect on the crack mesh when the **Front Element Size** property is user defined. For Arbitrary cracks the **Largest Contour Radius** and **Mesh Contour** properties do not have an effect on the crack mesh.
- For the base mesh inside and surrounding the region of the buffer zone, fracture meshing supports quadratic tetrahedron elements only. Linear elements may exist farther away from the buffer zone on the same body to which the semi-elliptical, elliptical, ring, or arbitrary crack is scoped.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Fracture meshing is a part-based meshing operation. It is supported for all part/body-based tetrahedron mesh methods.
- Fracture meshing does not support insertion of a crack that spans multiple bodies.
- The fracture mesh overrides the settings of the base mesh. You can insert a semi-elliptical crack and/or arbitrary crack on topology to which a sizing control, match control, or mapped Face Meshing control is already applied. However, fracture meshing does not respect sizing controls. It will also eliminate or disable a match or mapped Face Meshing control. For example and as illustrated below, fracture meshing has overwritten a mapped Face Meshing control.



- Fracture meshing does not support mesh refinement.
- Fracture meshing is supported for static structural and transient structural analyses only.
- Crack insertion does not alter the geometry/topology of a model.
- For Error Limits, fracture meshing supports the **Standard Mechanical** option only.
- Once inserted, the Fracture folder object:
  - Cannot be suppressed.
  - Can only be deleted if the folder is empty.
- **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, and **Arbitrary Crack** objects can be suppressed, deleted, or duplicated. When a crack definition changes after meshing, only the Fracture object in the tree **Outline** is invalidated.
- The plane for semi-elliptical, elliptical, and ring cracks, always lies in the X-Z plane of the specified coordinate system.
- The major radius of the semi-elliptical crack/elliptical crack/ring crack grows in the Z direction. The minor radius of the elliptical crack/ring crack grows in the X direction of the specified coordinate system, and the minor radius grows of the semi-elliptical crack in the positive X direction of the specified coordinate system.

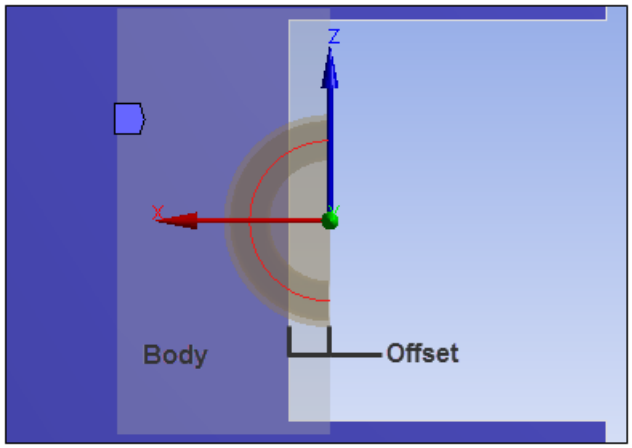
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

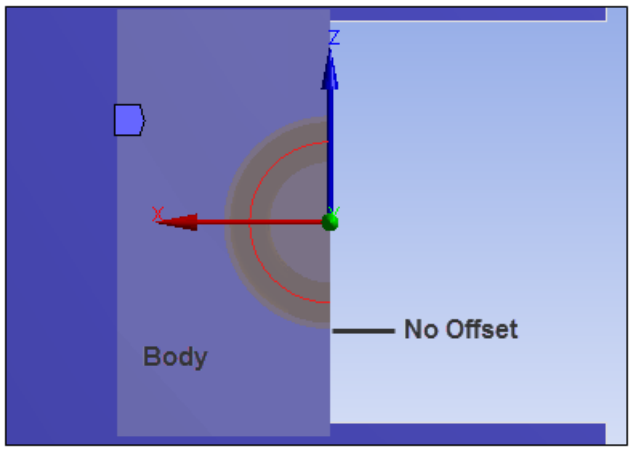
- When you use the Part Transform (p. 1014) feature on a meshed model that has a crack specified on a part, each time you change the Part Transform, the application clears the mesh on the body with cracks and requires re-mesh of the crack-based body.

For Semi-Elliptical Cracks

- Fracture meshing supports semi-elliptical crack insertion on planar surfaces and curved surfaces. For curved surfaces, you can insert semi-elliptical cracks on convex or concave surfaces. If you try to insert a single crack that extends across a surface that has both types of curves, crack insertion may fail.
- Fracture meshing does not support semi-elliptical crack insertion at a corner.
- The specified coordinate system should lie on the surface of the body to which the semi-elliptical crack is scoped. If it does not, and the **Project to Nearest Surface** property is set to **Yes**, the software projects the coordinate system onto the body's surface and modifies the center of the ellipse, as shown below.

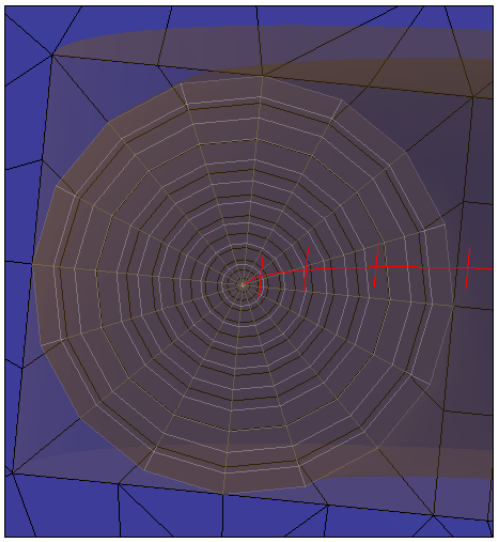


In this case, you will need to define the **Hit Point Normal** and orient the primary axis. The center of the ellipse is moved in the X direction so the major radius of the semi-elliptical crack grows in the Z direction, and the minor radius grows in the positive X direction of the specified coordinate system. The offset projects back to the surface of the body, making the offset close to 0. The illustration below shows the resultant mesh. For more information on creating a coordinate system aligned with a hit point, see the Creating a Coordinate System Based on a Surface Normal (p. 1196) section of the help.

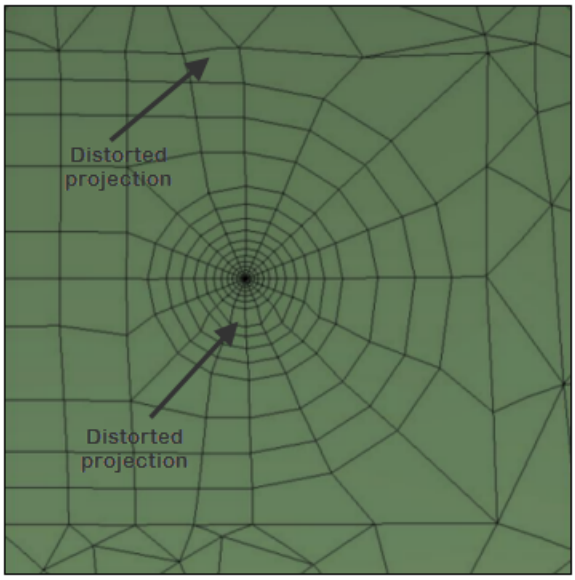


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- You can use the **Annotation Preferences** dialog box to toggle the visibility of annotations for cracks. For details, refer to the Probe, Maximum, and Minimum (p. 71) section of the help.
- As illustrated by the figure below, the mesh generated for the mesh contours (black lines) will not necessarily match the preview of the mesh contours (white lines).



- When cracks are inserted in curved surfaces, the projection of the crack template on the surface may be slightly distorted on the template, as shown in the following figure.

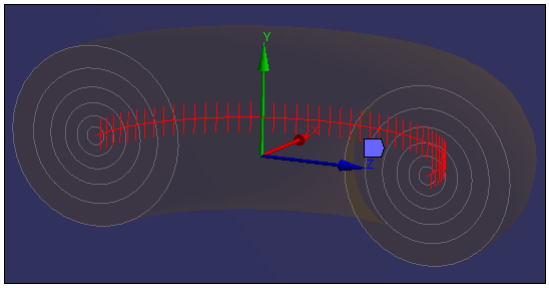


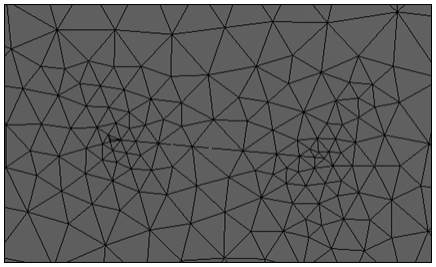
- Illustrated below is the graphics preview of a semi-elliptical crack that uses the Tetrahedrons Mesh Method. The graphics preview always shows the structured hex dominant shape irrespective of the mesh method selection. It does not accurately represent the generated crack mesh as compared to the generated crack mesh image. However, you can use the preview to examine the effect of the mesh parameters on the generated crack mesh.

Graphics Preview	**Generated Crack Mesh - Tetrahedrons Mesh Method**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

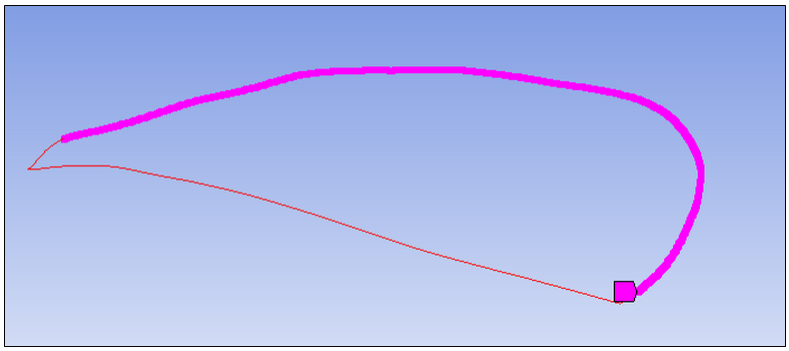
Published: 2023-10-23





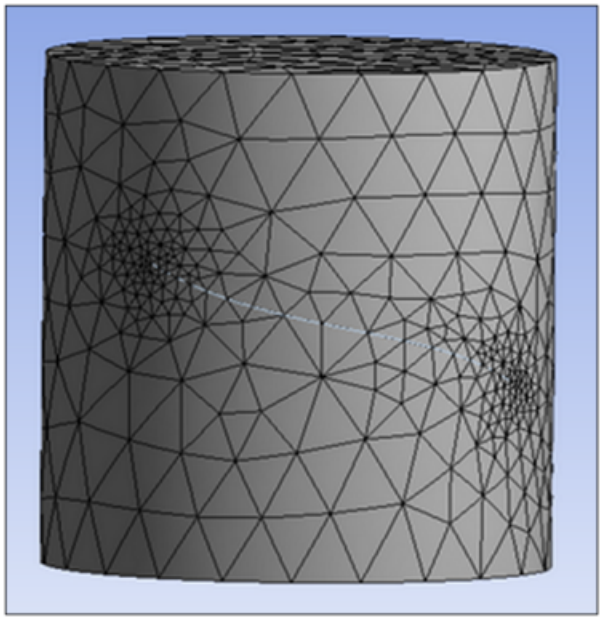
For Arbitrary Cracks

- Arbitrary cracks only support crack mesh generation using **Tetrahedrons** as the **Mesh Method**.
- Arbitrary crack supports both planar and non-planar crack.
- Arbitrary cracks support surface, embedded, through, and corner cracks.
- Fracture meshing of Arbitrary cracks automatically detects proper intersection points between the crack surface body and its scoped geometry selection to extract the crack front. The image shown below shows that even if the crack surface is extended beyond the surface of the scoped geometry selection, the crack front nodes only include the intersection points and the points that are located inside of the surface.



- When generating the mesh for an Arbitrary crack, it is required that the coordinate system is selected such that the crack surface is located on both sides of the Z axis and that the crack top face is located in the positive Y axis.
- You can define multiple cracks on the same body as long as their buffer zones do not intersect.
- The arbitrary crack mesh illustrated below displays the tetrahedrons as the mesh method used to generate non-planar crack mesh. The crack front nodes of the generated crack mesh are extracted from the intersection points of the crack surface to the cylinder body.

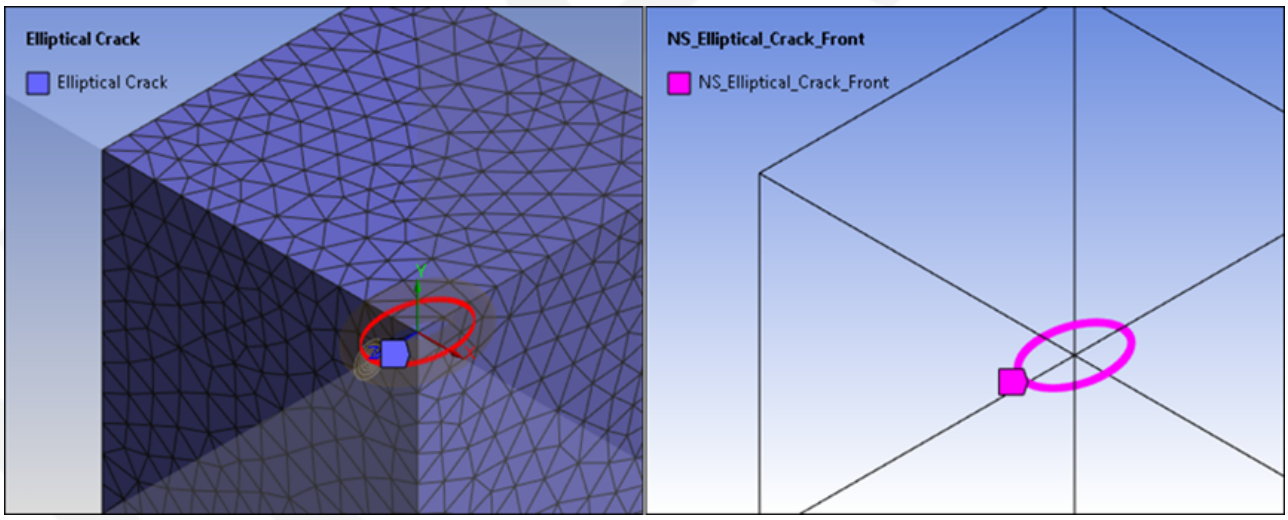
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Embedded and Intersecting Arbitrary Crack Mesh

You can also specify a crack surface body that is embedded inside a solid body or intersecting the free surfaces of a solid body. Crack mesh generation using these crack surfaces enables you to insert:

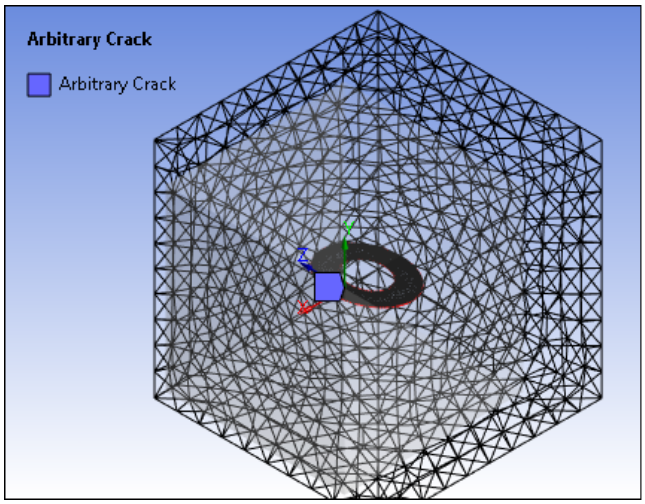
Arbitrary Elliptical Crack

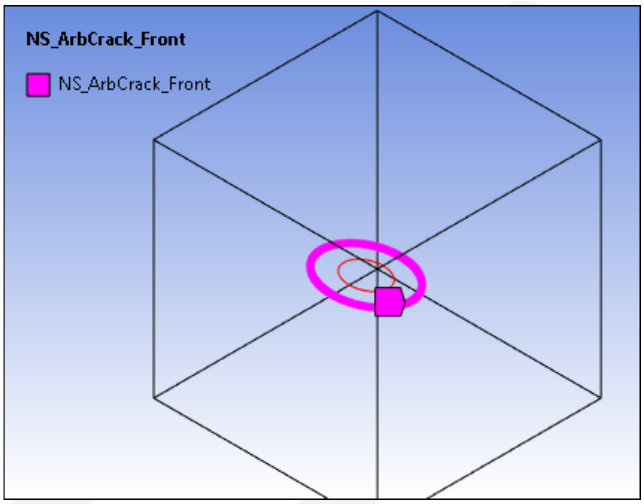


Arbitrary Ring Crack

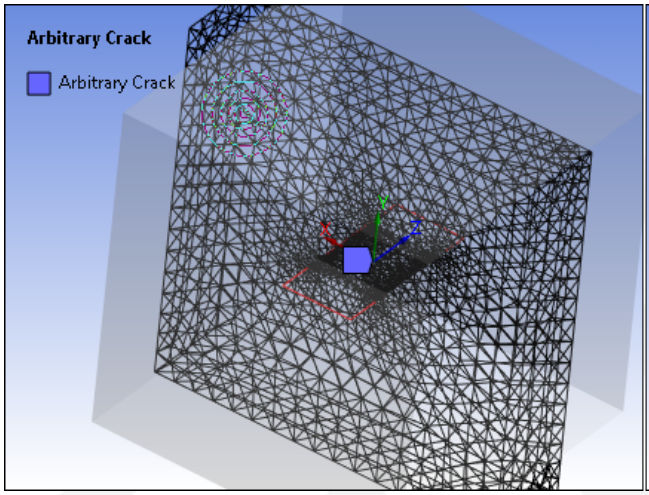
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

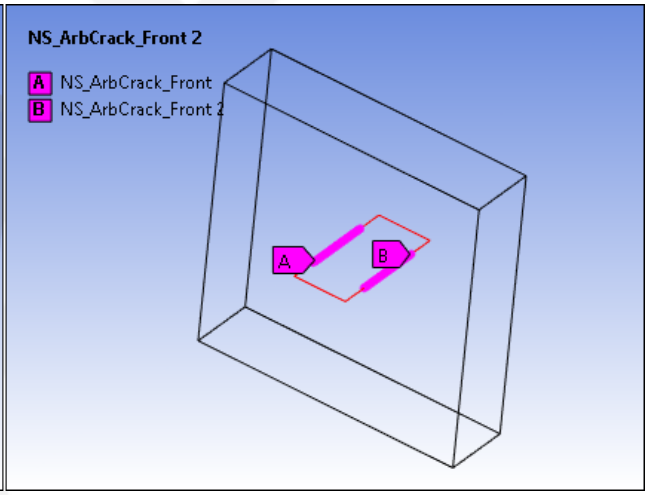
Published: 2023-10-23



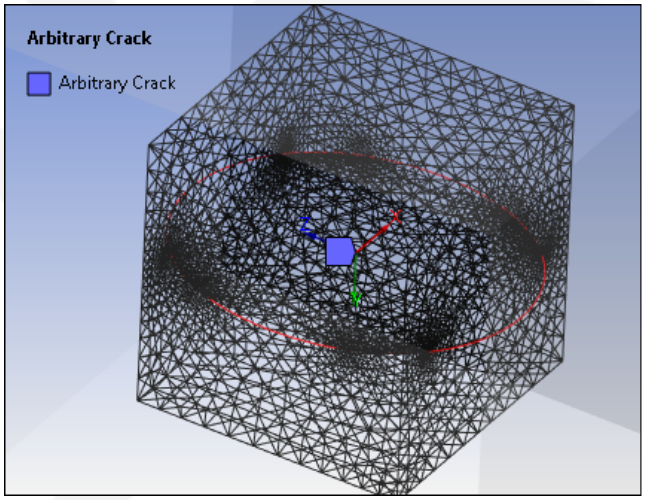


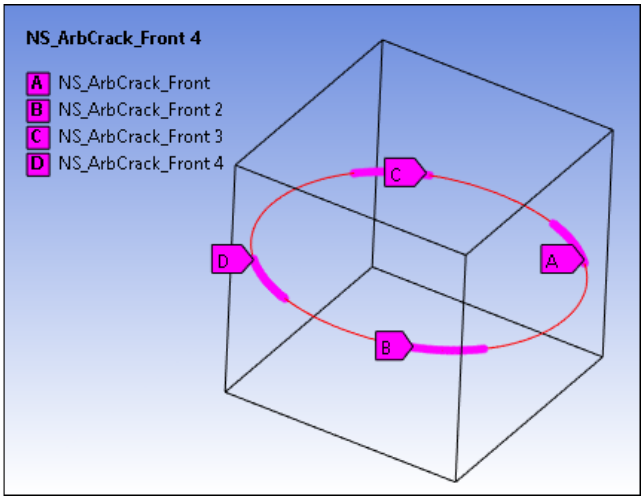
Arbitrary Through Crack





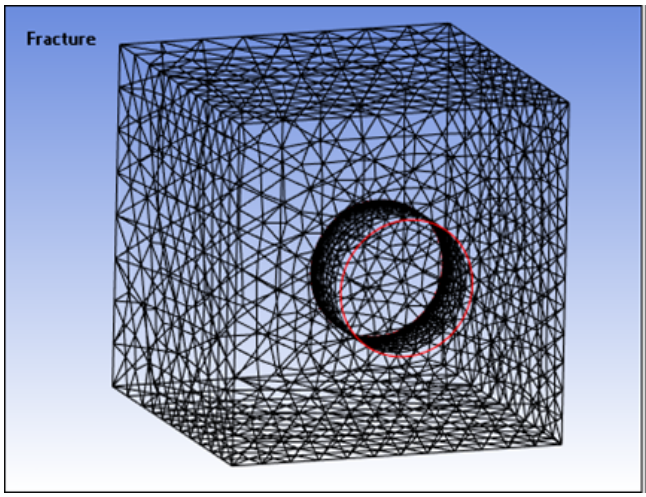
Arbitrary Crack Intersecting Multiple Corners

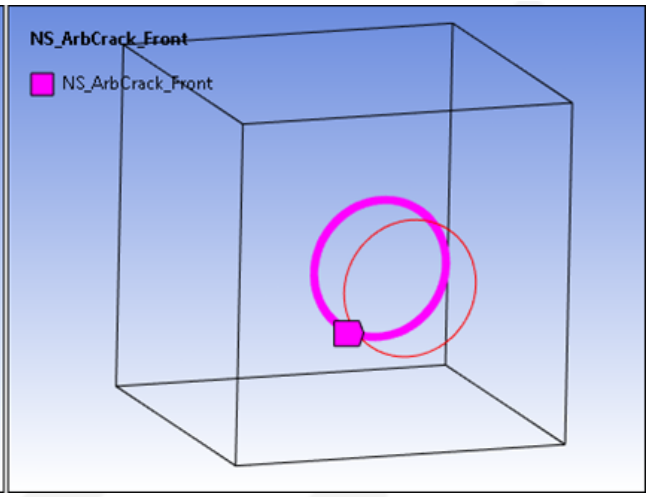




Arbitrary Cylinder Crack

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Cracks

The following sections describe the crack definitions available for the Fracture feature.

Crack Overview
Defining an Arbitrary Crack
Defining a Semi-Elliptical Crack
Defining an Elliptical Crack
Defining a Ring Crack
Defining a Corner Crack
Defining an Edge Crack
Defining a Through Crack
Defining a Cylindrical Crack
Defining a Pre-Meshed Crack
Special Handling of Named Selections for Crack Objects

Crack Overview

A crack is characterized by its shape, crack front/tip, crack discontinuity plane, crack normal, and crack direction. A crack front in three dimensional analyses represents the line of separation of the discontinuous crack surface. The same is represented by a crack tip in two dimensional analyses. A crack inside Ansys Mechanical can be defined using a **Semi-Elliptical Crack** object, an **Elliptical Crack** object, a **Ring Crack** object, or an **Arbitrary Crack** object, or a **Pre-Meshed Crack** object. These objects can be inserted under the **Fracture** folder.

Arbitrary Crack objects use a surface body selection to define an arbitrary crack shape in three dimensional analyses. The exterior edges of the surface body define the crack front and the surface itself defines the discontinuous crack plane. **Semi-Elliptical Crack**/**Elliptical Crack**/**Ring Crack** objects uses the geometric parameters to define the semi-elliptical/elliptical/ring crack shape and crack front in three dimensional analyses. These geometric inputs along with additional input parameters on the arbitrary crack and semi-elliptical/elliptical/ring crack object defines the region and shape of the generated crack mesh. Internally, the crack mesh generation is performed after the creation of the base mesh. By default, the crack mesh generation automatically creates a node-based named

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

selection for the crack front under the **Semi-Elliptical Crack**/**Elliptical Crack**/**Ring Crack**/**Arbitrary Crack** objects.

A **Pre-Meshed Crack** definition assumes that the crack meshes, representing the discontinuity or flaw in the structure, have already been generated. That is, the pre-meshed crack does not internally generate the crack mesh using Fracture Meshing (p. 415), as other crack objects do, but instead assumes that the crack mesh has been generated beforehand. A **Pre-Meshed Crack** object uses a node-based named selection to analyze crack front; this nodal named selection is required for the computation of fracture parameters. If a geometric edge represents a crack front, you must first convert it to a node-based named selection using the Worksheet criteria before it can be used by the Pre-Meshed Crack (p. 465) object. See the next section, **Defining a Pre-Meshed Crack (p. 465)**, for more information on the Pre-Meshed Crack.

The orientation of the crack plays a vital role in the fracture parameter calculations. **Semi-Elliptical Crack**/**Elliptical Crack**/**Ring Crack** and **Pre-Meshed Crack** objects supports planar cracks where the crack normal can be defined using a single coordinate system. The orientation of a **Semi-Elliptical Crack**/**Elliptical Crack**/**Ring Crack** object is reflected by its **Crack Coordinate System**, while the orientation of a **Pre-Meshed Crack** object is reflected by its Coordinate **System** property setting. The orientation must be defined such that the y-axis is normal to the crack surface while the x-axis helps align the crack extension direction. **Arbitrary Cracks** on the other hand supports both planar and non-planar cracks. The orientation of an **Arbitrary Crack** object is reflected by its **Coordinate System** property setting. A single coordinate system cannot define the crack normal for non-planar cracks which is varying along the crack front. Hence, the Y axis of the **Coordinate System** property setting is directed towards the crack top face normal and X axis helps align the crack extension direction.

Note:

When you insert a Commands (APDL) object under a crack, the object provides a **Crack Front Selection** property that enable you to execute commands specific crack front definitions. Property options include **All** (default), **First**, **Last**, and **By Number**. When you select the **By Number** option, an additional property displays: **Crack Front Number**. The default value for this property is **All**. You can use it to specify a desired value using the entry tool that displays when you select the field.

Tip:

To achieve coordinate system alignment to the face normal, create your coordinate system as described in Creating a Coordinate System Based on a Surface Normal (p. 1196) and assign the created coordinate system to the **Semi-Elliptical Crack** object. You can also set the **Align with Face Normal** property to **Yes** to generate a crack mesh whose coordinate system is aligned to the face normal and that is reflected by the **SECrack Coordinate System** object that is added under the **Semi-Elliptical Crack** object. Otherwise, the **Semi-Elliptical Crack** object's coordinate system can be at an inclination to the face normal direction. For the **Pre-Meshed Crack** and **Arbitrary Crack** objects, the origin of the coordinate system must be located on the open side of the crack.

Note: The graphical view of the semi-elliptical crack may differ from the mesh generated. Possible reasons include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- A crack definition unsuitable for valid mesh creation may result in some layers being "peeled off" to create a valid mesh.
- The crack contour may be shrunk to fit into the mesh domain.
- The crack coordinate system may be changed to align it to surface normal.
- The center of the crack may be changed to create the crack on the surface.
- The crack is meshed with gradation from the contour center to the outside results in difficulty distributing the crack mesh.
- The offset of the crack is not suitable for the crack contour, resulting in a contour that must be reduced to ensure all element contours fit into the template.

Defining an Arbitrary Crack

The steps for defining an Arbitrary crack are presented here.

1. Select the **Model** object in the Tree Outline.
2. Insert a Fracture object into the tree **Outline** by right-clicking on the **Model** object and selecting **Insert** > **Fracture** from the context menu. Alternatively, select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

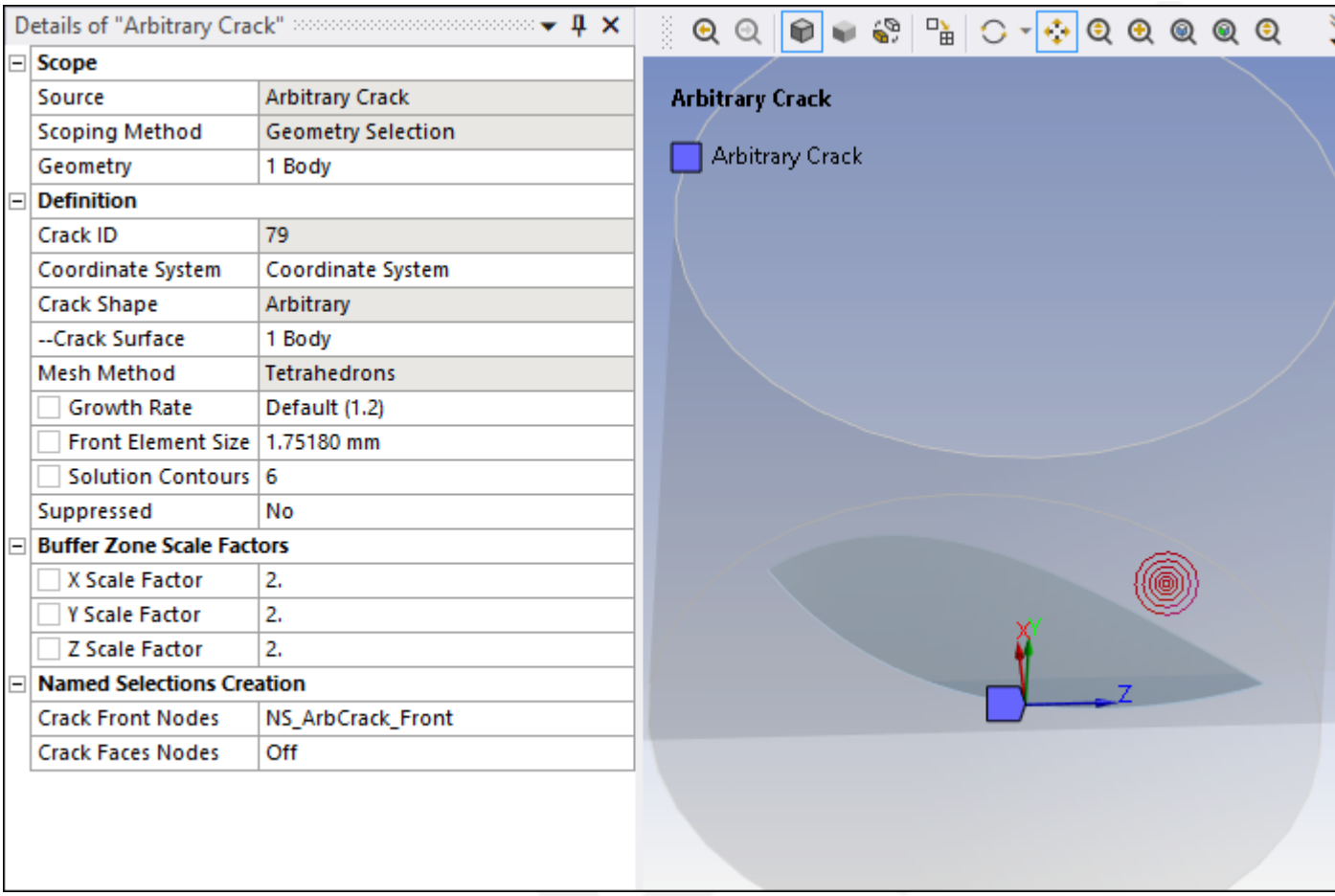
   Note:

   Only one **Fracture** object is valid per **Model**.

3. Insert an Arbitrary Crack object into the tree by right-clicking on the **Fracture** object and selecting **Insert** > **Arbitrary Crack** from the context menu. Alternatively, click the **Arbitrary Crack** option from the **Crack** group of the **Fracture Context Tab** (p. 52).
4. An Arbitrary Crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details View, and then click **Apply**.
5. To further define the **Arbitrary Crack**, use the following Details pane properties. As you specify values for the properties, the image in the **Geometry** window previews the entered data. An example of Arbitrary crack definition, including an image of the crack on the model, is illustrated below.

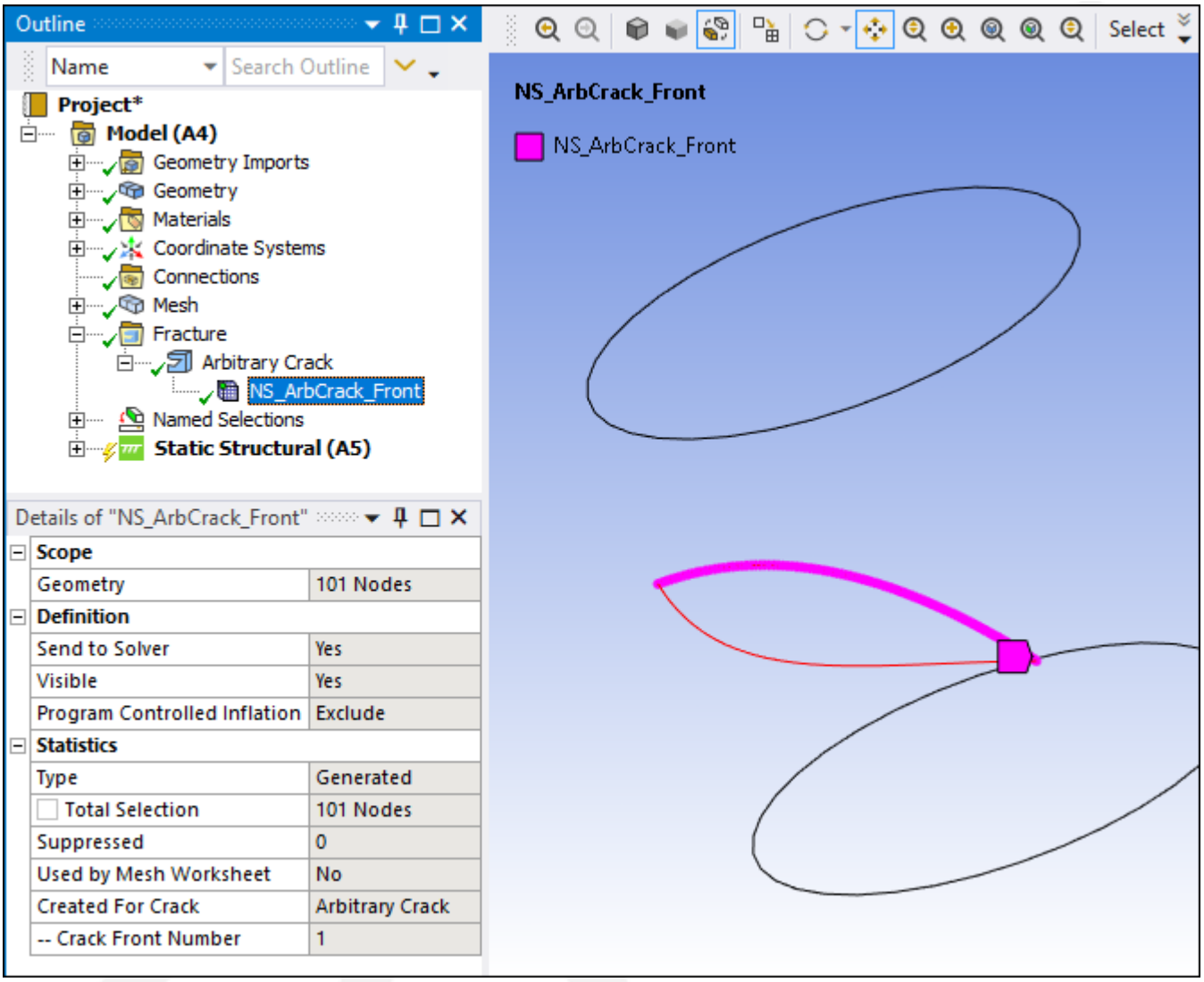
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



As defined by the application generated Named Selection, the portion of the curve located inside the cylinder, including the intersection points, defines the shape of the crack front.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- **Crack ID**: A read-only property that displays a unique system generated identification number ID for the crack object. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: This property specifies the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards the normal of the crack's top face.
- **Crack Shape**: Read-only property set to **Arbitrary**.
- **Crack Surface**: You use this property to specify the surface body to be used as the crack surface. This surface body can be on a surface or embedded within the solid body, or it can intersect with the free surfaces of the solid body. This generates an arbitrary crack that is a surface crack, shaped as an ellipse, a ring, a cylinder, it can be a through-crack that cuts through a solid body, or a corner crack that intersects at multiple corners of the solid body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

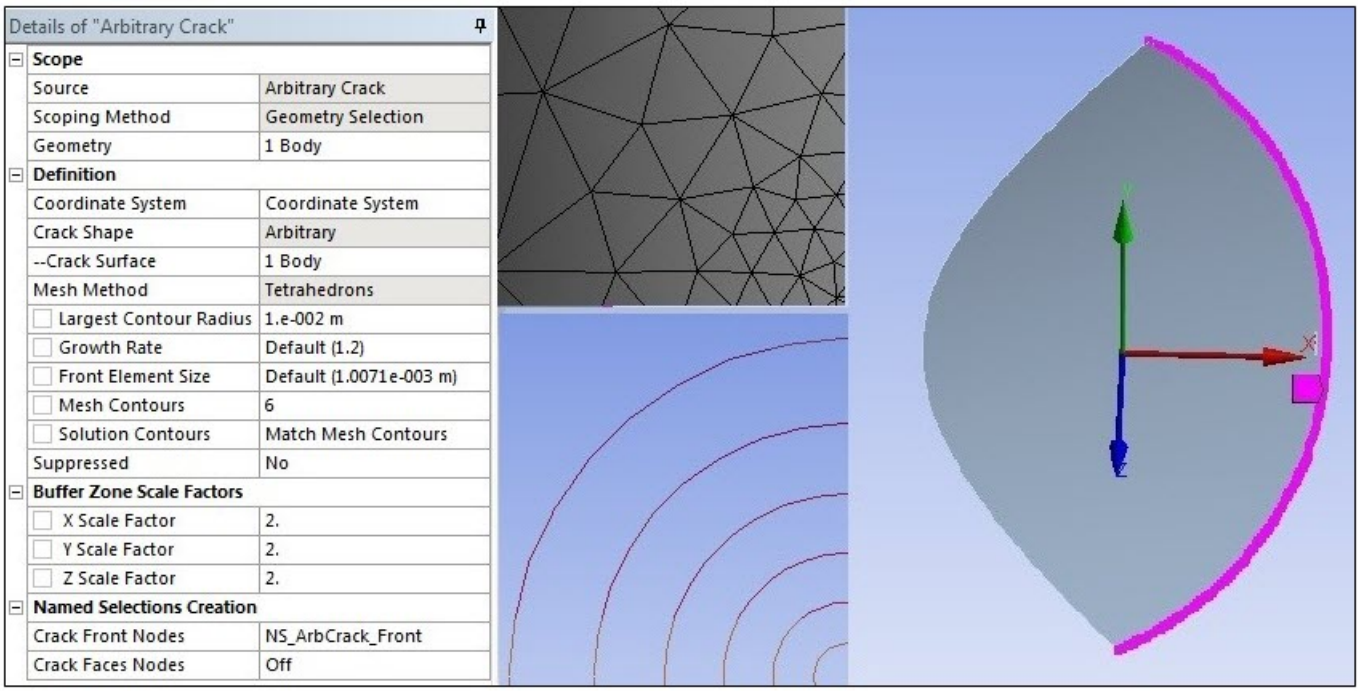
Published: 2023-10-23

- **Mesh Method**: Read-only property set to **Tetrahedrons**.

> **Note:**
>
> When this crack intersects with the free surfaces of a solid body, the **Hex Dominant** setting is not supported.

- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**. If you specify a **Front Element Size** value, this property is not applicable.
- **Growth Rate** Specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size** Specifies the element size for the crack front. The default value is computed using the values of **the Largest Contour Radius** property and the **Growth Rate** property.
- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. Your entry must be equal to or greater than **1**. The default value is **6**. If you specify a **Front Element Size** value, this property is not applicable.

  The **Geometry** window can display only a maximum of 100 mesh contours, but you can specify a higher value and fracture meshing will respect it.



> **Note:**
>
> You can use the graphics preview of an arbitrary crack to examine the relative effect of the mesh parameters on the generated crack mesh. The first mesh contour's

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

radius shown as a bull's-eye view in **Geometry** window is equal to specified **Front Element Size**. The mesh contours grow at the rate of the specified **Growth Rate** value as seen in the image. Also, the generated crack mesh is an unstructured tetrahedron mesh and may not accurately compare to all the mesh parameters seen in the **Geometry** window.

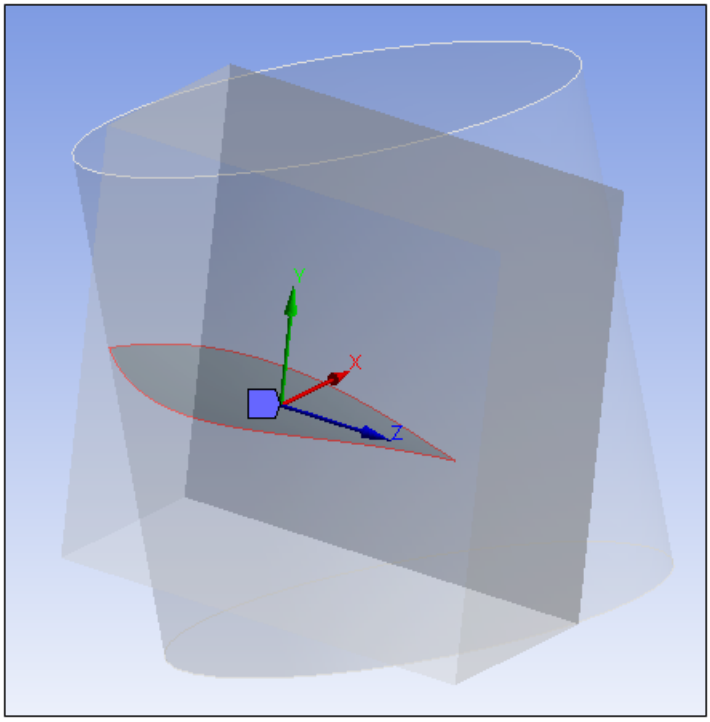
- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of the **Mesh Contours** property, and cannot be greater than **99**. By default, the value is set to **Match Mesh Contours**, indicating that the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the **Arbitrary Crack** object. The default is **No**.

  The **Arbitrary Crack** object is suppressed automatically if both the scoped body and scoped crack surface are suppressed.
- **Buffer Zone Scale Factors**: Control the size of the buffer zone in the X, Y, and Z directions, relative to the crack surface geometry dimensions. For each scaling parameter, use the slider to set a value from **2** to **50**. The default value is **2**. The maximum dimension among the three dimensions of the crack surface geometry is multiplied by the corresponding scale factors to create a buffer zone:
  - **X Scale Factor**
  - **Y Scale Factor**
  - **Z Scale Factor**

  As illustrated here, buffer zone scale factor annotations are not drawn in the negative X direction. That is, half of the cuboid is drawn to depict the buffer zone scale factors, but the half of the cuboid in the negative X direction is not drawn.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The remaining properties pertain to Named Selections that are created automatically when the fracture mesh is generated, as described in Step 6 below. To ensure associativity to the corresponding **Arbitrary Crack** object, the following default naming convention is used for these Named Selections.

The following naming convention is used for Named Selections for crack objects with the name **Arbitrary Crack**:

- **NS**_ *ArbCrack* _**Front**
- **NS**_ *ArbCrack* _**TopFace**
- **NS**_ *ArbCrack* _**BottomFace**

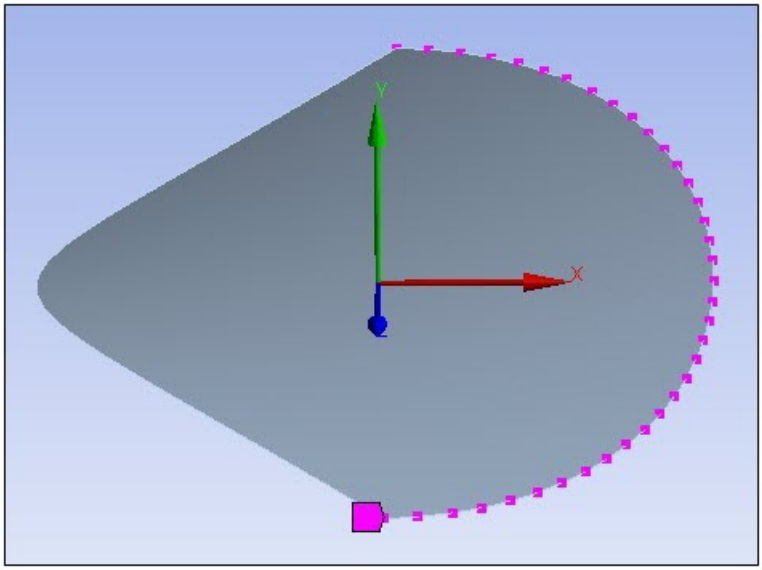
For example, for an **Arbitrary Crack** object named **Arbitrary Crack 4**, the default names are **NS_ArbCrack 4_Front**, **NS_ArbCrack 4_TopFace**, and **NS_ArbCrack 4_BottomFace**.

The Named Selections controls are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*ArbCrack*_**Front**). Contains nodes used for postprocessing of fracture parameter results.

  As shown, the X axis helps determine the extension direction of the crack front node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Important:

For cracks with multiple crack fronts, the **Crack Front Nodes** property, under **Named Selection Creation** category, only modifies the first crack front name if you have already generated the mesh. However, the application applies the property value to all crack fronts if you are generating the mesh for the first time or if the mesh is re-generated.

- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous.

  The default is **Off**. If **On**, the following additional fields appear:

  – **Top Face Nodes**: Identifies the Named Selection that is created automatically for the top face (**NS**_*ArbCrack*_**TopFace**). This face is discontinuity plane 1. Contains nodes used for applying a pressure to the top face.

  – **Bottom Face Nodes**: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*ArbCrack*_**BottomFace**). This face is discontinuity plane 2. Contains nodes used for applying a pressure to the bottom face.

6. Select the **Fracture** object or **Arbitrary Crack** object in the tree **Outline**, right-click, and select **Generate All Crack Meshes**.

   - When the fracture mesh is generated, the requested Named Selections are inserted into the tree Outline under the **Arbitrary Crack** object with which they are associated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Click **Show Mesh** on the Graphics Toolbar (p. 97) to display the fracture mesh.

Note:

Mesh near arbitrary crack is projected to facets if the geometry is faceted. Thus, if you see mesh being projected slightly away from the geometry, it might be because of the coarseness of geometry facets.

Defining a Semi-Elliptical Crack

The steps for defining a semi-elliptical crack are presented here.

1. Select the **Model** object in the Tree Outline.
2. Insert a Fracture object into the tree by right-clicking on the **Model** object and selecting **Insert** > **Fracture** from the context menu. Alternatively, select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

   Only one **Fracture** object is valid per **Model**.

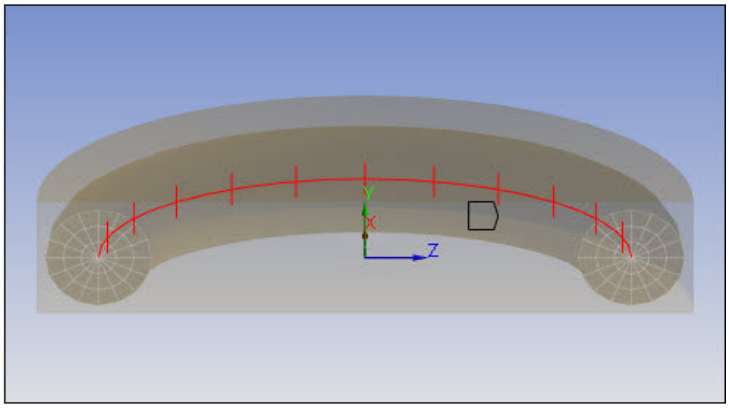
3. Insert a Semi-Elliptical Crack object into the Tree by right-clicking on the **Fracture** object and selecting **Insert** > **Semi-Elliptical Crack** from the context menu. Alternatively, click the **Semi-Elliptical Crack** from the **Crack** group of the **Fracture Context Tab** (p. 52).
4. A semi-elliptical crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details View, and then click **Apply**.
5. To further define the semi-elliptical crack, use the following controls. These controls appear in the Details View of the **Semi-Elliptical Crack** object. As you specify values for the controls, the image in the **Geometry** window previews the entered data.

   The following figure shows an example of a semi-elliptical crack definition and its corresponding image. The semi-elliptical curve defines the shape of the crack front, as shown by the red line.

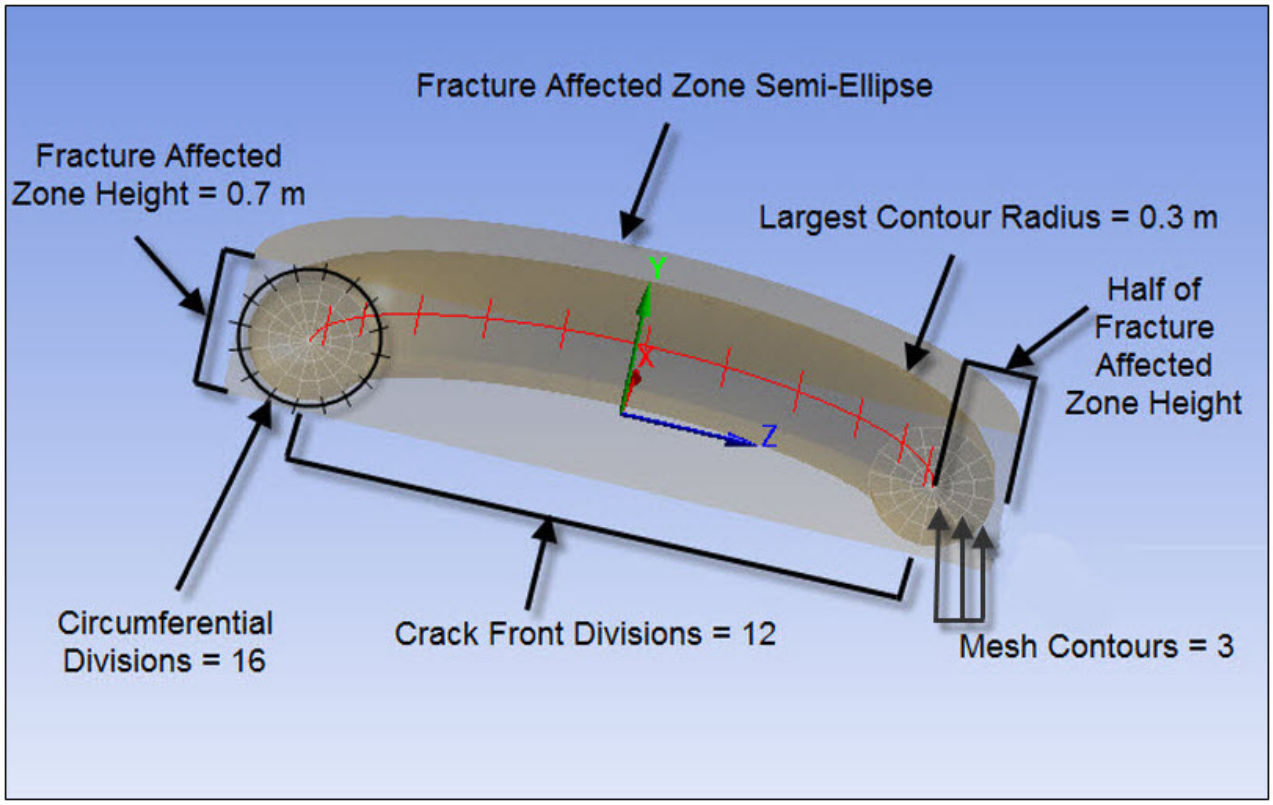
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Semi-Elliptical Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	86
Coordinate System	Plane11 2
Align with Face Normal	Yes
Project to Nearest Surface	Yes
Crack Shape	Semi-Elliptical
--Major Radius	1.5 m
--Minor Radius	1. m
Mesh Method	Hex Dominant
Largest Contour Radius	0.3 m
Crack Front Divisions	12
Fracture Affected Zone	Manual
Fracture Affected Zone Height	0.7 m
Circumferential Divisions	16
Mesh Contours	3
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
Named Selections Creation	



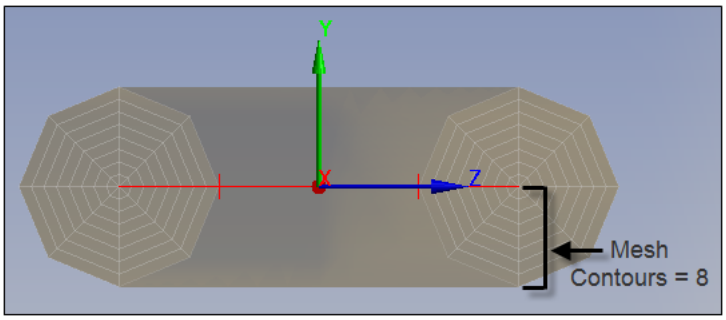
This figure provides a more detailed illustration of the fracture affected zone defined above. Notice that the values shown in the image below correspond to the Details View settings above.



This figure shows the detail of a crack for which **Mesh Contours** is set to **8**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- **Crack ID**: A read-only property that displays a unique system generated identification number ID for the crack object. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: This property specifies the coordinate system that defines (along with the **Align with Face Normal** and **Project to Nearest Surface** properties) the position and orientation of the crack. The Y axis of the specified coordinate system defines the crack plane normal. The coordinate system that you create must be a Cartesian coordinate system (**Type** property) and its origin cannot lie outside the bounding box of the body scoped to the crack.
- **Align with Face Normal**: This property defines the orientation of the **SECrack Coordinate System** object by aligning the primary axis of the coordinate system specified in the **Coordinate System** property to the normal of the nearest surface. The default setting is **Yes**. Setting this property to **No** excludes the capability of this property.
- **Project to Nearest Surface**: This property defines the origin of the **SECrack Coordinate System** by projecting the origin of the coordinate system specified in the **Coordinate System** property to the nearest surface. The default setting is **Yes**. Setting this property to **No** excludes the capability of this property.

Note:

The meshing process automatically creates the **SECrack Coordinate System** object as a child of the **Semi-Elliptical Crack** object from the inputs of the **Coordinate System**, the **Align with Face Normal**, and the **Project to Nearest Face** properties.

- **Crack Shape**: Read-only and set to **Semi-Elliptical**.
- **Major Radius**: Specifies the major radius, which defines the size of the crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.
- **Minor Radius**: Specifies the minor radius, which defines the size of the crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.
- **Mesh Method**: This property enables you to select the mesh method to be used to mesh the semi-elliptical crack. Options include **Hex Dominant** (default) and **Tetrahedrons**.
- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.

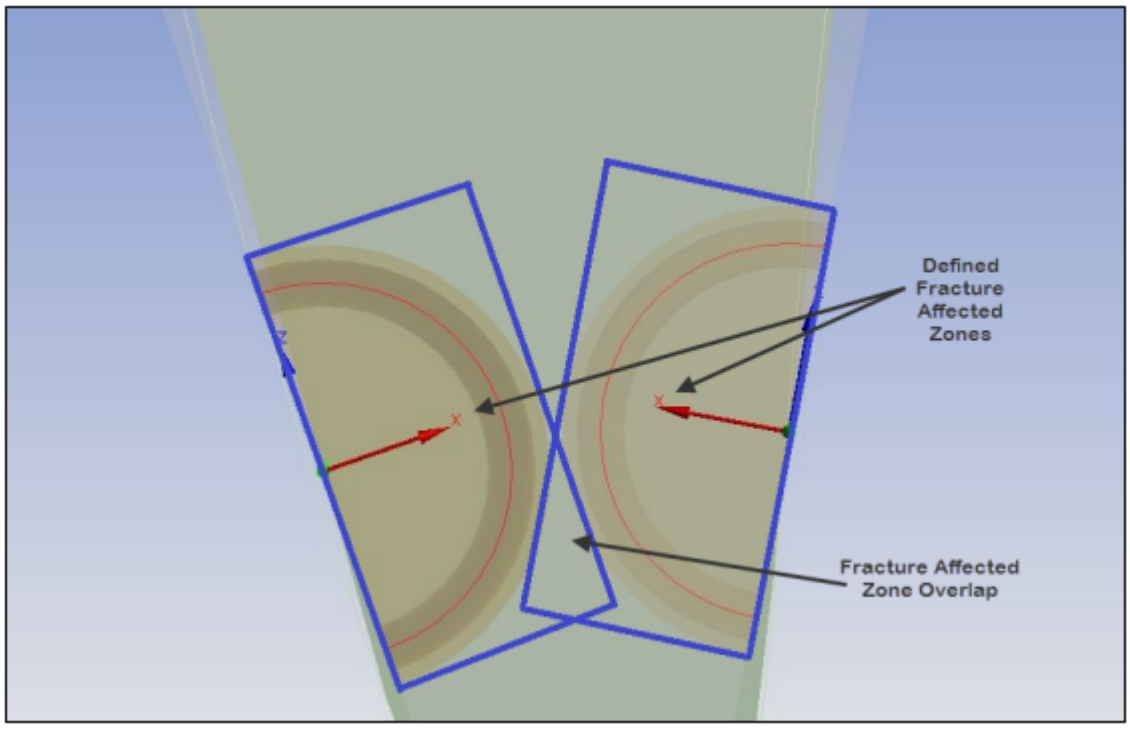
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Growth Rate** (**Mesh Method** set to **Tetrahedrons** only): Specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size** (**Mesh Method** set to **Tetrahedrons** only): Specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.
- **Crack Front Divisions** (**Mesh Method** set to **Hex Dominant** only): Specifies the number of divisions for the crack front. Your entry must be equal to or greater than **3**. The default is **15**.
  - The **Geometry** window can display only a maximum of 999 crack front divisions, but you can specify a higher value and fracture meshing will respect it.
- **Fracture Affected Zone** (**Mesh Method** set to **Hex Dominant** only): The fracture affected zone is the region that contains a crack. The **Fracture Affected Zone** control determines how the fracture affected zone height is defined:
  - **Program Controlled**: The software calculates the height, and **Fracture Affected Zone Height** is read-only. This is the default.
  - **Manual**: You enter the height in the **Fracture Affected Zone Height** field.
- **Fracture Affected Zone Height** (**Mesh Method** set to **Hex Dominant** only): This value specifies two things: 1) the height of the **Fracture Affected Zone**, which is in the Y direction of the crack coordinate system; and 2) the distance in totality by which the **Fracture Affected Zone** is extended in the positive and negative Z direction of the crack coordinate system from the crack front extremities.

  The shape of a Fracture Affected Zone is rectangular, regardless of the shape of the crack. Although buffer zones may overlap, care should be taken when defining multiple cracks that the zones do not overlap, as shown here, or the crack generation will fail.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- **Circumferential Divisions** (**Mesh Method** set to **Hex Dominant** only): Specifies the number of circumferential divisions for the crack shape. The value you enter must be a multiple of 8, and must be 8 or greater. The default is 8.

  – The **Geometry** window can display only a maximum of 360 circumferential divisions, but you can specify a higher value and fracture meshing will respect it.

- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be 1 or greater. The default is 6.

  – The **Geometry** window can display only a maximum of 100 mesh contours, but you can specify a higher value and fracture meshing will respect it.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.

- **Suppressed**: Toggles suppression of the **Semi-Elliptical Crack** object. The default is **No**.

  – The **Semi-Elliptical Crack** object is suppressed automatically if the scoped body is suppressed.

- **Buffer Zone Scale Factors**

  It controls the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

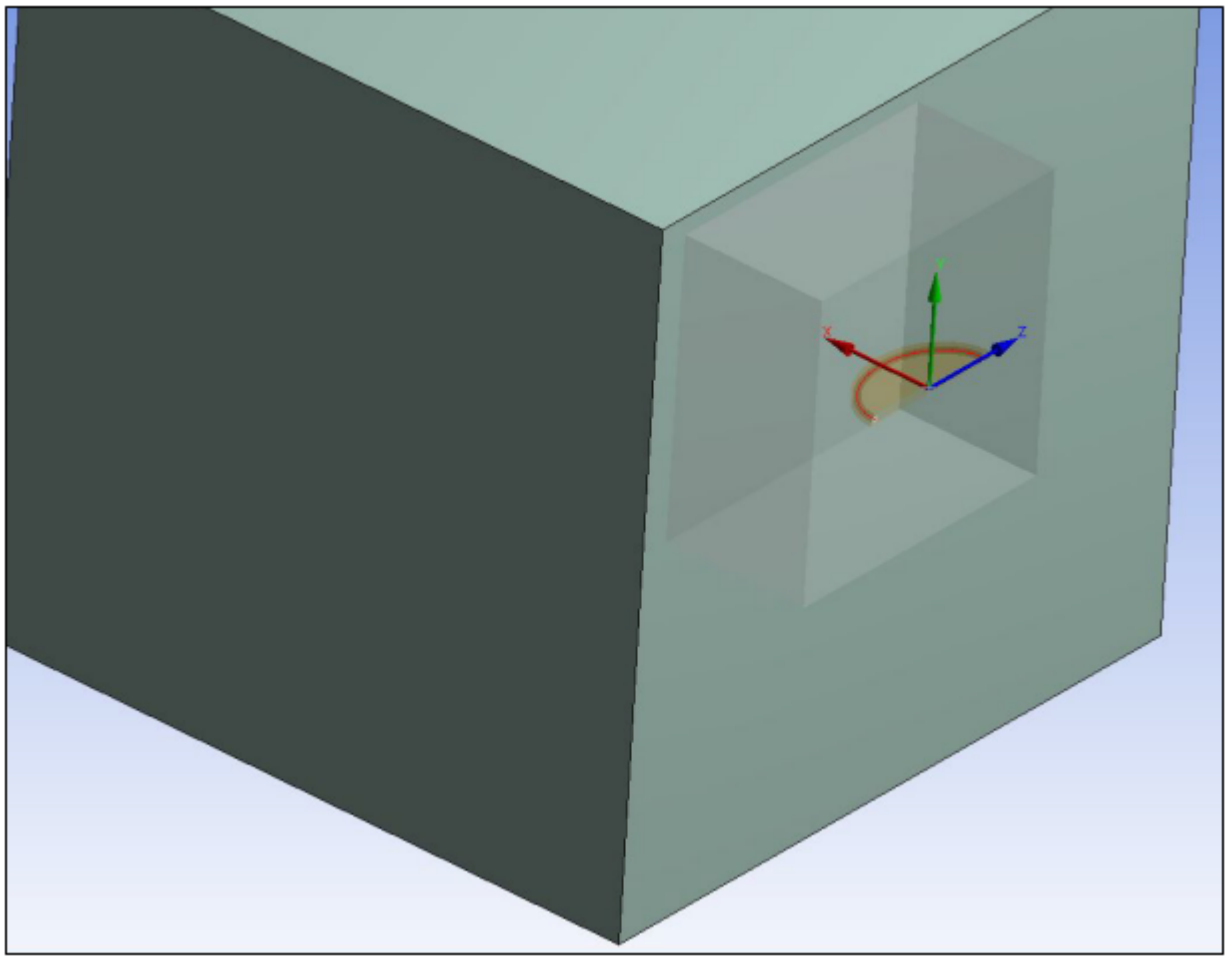
corresponding scale factors to create a buffer zone. When the **Mesh Method** is **Hex-Dominant**, the crack dimensions also include fracture-affected zones.

– **X Scale Factor**

– **Y Scale Factor**

– **Z Scale Factor**

As illustrated here, buffer zone scale factor annotations are not drawn in the negative X direction. That is, half of the cuboid is drawn to depict the buffer zone scale factors, but the half of the cuboid in the negative X direction is not drawn.



The remaining controls pertain to Named Selections that are created automatically when the fracture mesh is generated, as described in Step 6 below. To ensure associativity to the corresponding **Semi-Elliptical Crack** object, the following default naming convention is used for these Named Selections.

The following naming convention is used for Named Selections for crack objects with the name **Semi-Elliptical Crack**:

- **NS_** *SECrack* **_Front**
- **NS_** *SECrack* **_TopFace**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

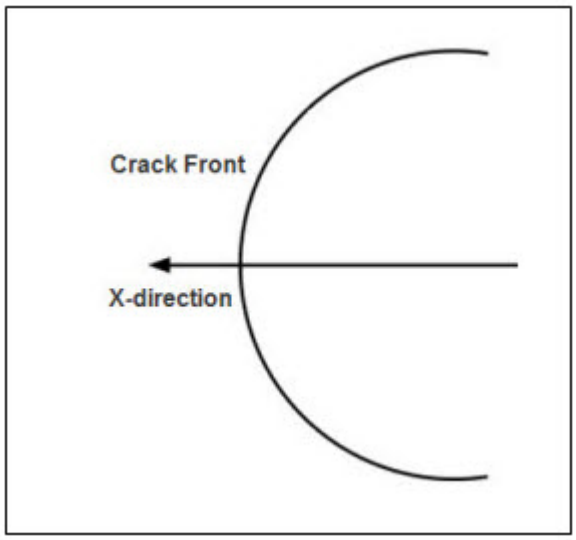
- **NS**_ *SECrack* _**BottomFace**
- **NS**_ *SECrack* _**Contact1**
- **NS**_ *SECrack* _**Target1**

For example, for a **Semi-Elliptical Crack** object named **Semi-Elliptical Crack 4**, the default names are **NS_SECrack 4_Front**, **NS_SECrack 4_TopFace**, **NS_SECrack 4_BottomFace**, **NS_SECrack 4_Contact1**, and **NS_SECrack 4_Target1**.

The Named Selections controls are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*SECrack*_**Front**). Contains nodes used for postprocessing of results.

  As shown, the crack front is always perpendicular to the X axis.



- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous.

  The default is **Off**. If **On**, the following additional fields appear:

  – **Top Face Nodes**: Identifies the Named Selection that is created automatically for the top face (**NS**_*SECrack*_**TopFace**). This face is discontinuity plane 1. Contains nodes used for applying a pressure to the top face.

  – **Bottom Face Nodes**: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*SECrack*_**BottomFace**). This face is discontinuity plane 2. Contains nodes used for applying a pressure to the bottom face.

- **Contact Pairs Nodes** (**Mesh Method** set to **Hex Dominant** only): Determines whether Named Selections are created automatically for the contact and target faces of the contact pair. The default is **Off**. If **On**, the additional fields listed below appear. Fracture meshing creates contact pair 1 between the fracture affected zone and the buffer zone.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Contact 1 Nodes**: Identifies the Named Selection that is created automatically for contact face 1 (**NS**_*SECrack*_**Contact1**). Contains nodes located on the contact face. The contact nodes are selected at the interface from the buffer zone of the base mesh.
- **Target 1 Nodes**: Identifies the Named Selection that is created automatically for target face 1 (**NS**_*SECrack*_**Target1**). Contains nodes located on the target face. The target nodes are selected at the interface from the fracture affected zone of the hex dominant mesh.

6. Select the **Fracture** object or **Semi-Elliptical Crack** object in the Tree Outline, right-click, and select **Generate All Crack Meshes**.
   - When the fracture mesh is generated, the requested Named Selections are inserted into the Tree Outline under the **Semi-Elliptical Crack** object with which they are associated.
   - Click **Show Mesh** on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining an Elliptical Crack

This section describes the steps to create an **Elliptical Crack**. You use elliptical cracks to insert embedded void cracks into a solid body.

Procedure

To define an **Elliptical Crack**:

1. Select the **Model** object.
2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

   Only one **Fracture** object is valid per **Model**.

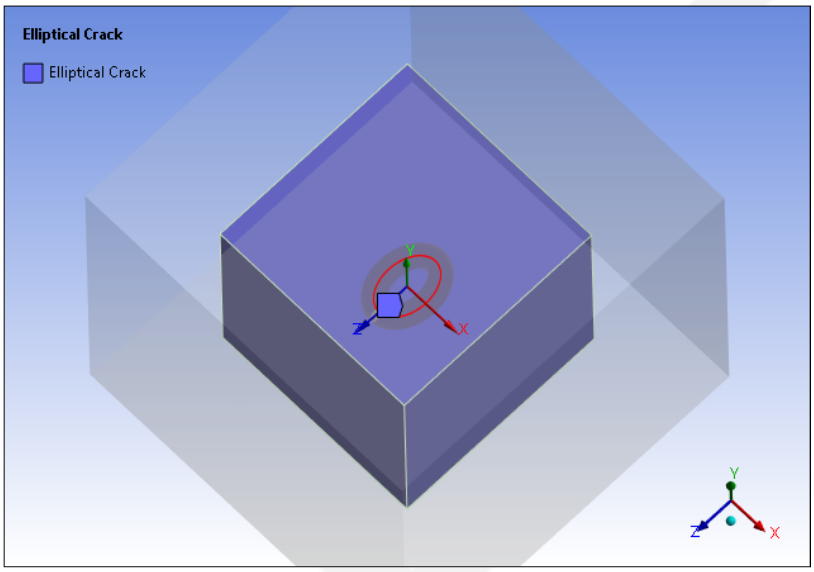
3. Insert a **Elliptical Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Elliptical Crack**. You can also select the **Elliptical Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.
4. A elliptical crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.
5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Examples entries are illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details of "Elliptical Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	88
Coordinate System	Centroid
Crack Shape	Elliptical
--Major Radius	3.e-003 m
--Minor Radius	2.e-003 m
Mesh Method	Tetrahedrons
Largest Contour Radius	1.e-003 m
Growth Rate	Default (1.2)
Front Element Size	Default (1.0071e-004 m)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_Elliptical Crack_Front
Crack Faces Nodes	On
--Top Face Nodes	NS_Elliptical Crack_TopFace
--Bottom Face Nodes	NS_Elliptical Crack_BottomFace



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards the normal of the crack plane and the crack plane always lies in the X-Z plane of the specified coordinate system.
- **Crack Shape**: Read-only property set to **Elliptical**.
- **Major Radius**: Specifies the major radius, which defines the size of the crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.
- **Minor Radius**: Specifies the minor radius, which defines the size of the crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.
- **Mesh Method**: This property enables you to select the mesh method to be used to mesh the crack. Options include **Hex Dominant** (default) and **Tetrahedrons**.

  Note:

  When this crack intersects with the free surfaces of a solid body, the **Hex Dominant** setting is not supported.

- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Growth Rate**: This property is only visible when the **Mesh Method** property is set to **Tetrahedrons**. It specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.

- **Front Element Size**: This property is only visible when the **Mesh Method** property is set to **Tetrahedrons**. It specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.

- **Crack Front Divisions**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. It specifies the number of divisions for the crack front. Your entry must be equal to or greater than **3**. The default value is **30**.

  Note:

  The maximum number of crack front divisions the **Geometry** window can display is 999. However, you can specify a higher value and the application will process it accordingly.

- **Fracture Affected Zone**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. This is the region that contains a crack. This property defines the height of the fracture affected zone. Property options include:

  – **Program Controlled** (default): The application calculates the height. When selected, the **Fracture Affected Zone Height** property is read-only.

  – **Manual**: You manually specify the height in the **Fracture Affected Zone Height** property when using this option.

- **Fracture Affected Zone Height**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. This value specifies two things: 1) the height of the **Fracture Affected Zone**, which is in the Y direction of the crack coordinate system; and 2) the distance in totality by which the **Fracture Affected Zone** is extended in the positive and negative Z direction of the crack coordinate system from the crack front extremities.

  The shape of a Fracture Affected Zone is rectangular, regardless of the shape of the crack. Although buffer zones may overlap, you should use care when defining multiple cracks so that the zones do not overlap.

  Important:

  The crack generation process will fail if buffer zones overlap.

- **Circumferential Divisions**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. It specifies the number of circumferential divisions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

for the crack shape. The value you enter must be a multiple of 8, and must be 8 or greater. The default is 8.

Note:

The maximum number of circumferential divisions the **Geometry** window can display is 360. However, you can specify a higher value and the application will process it accordingly.

- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than 1. The default setting is 6.

Note:

The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the **Elliptical Crack** object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Buffer Zone Scale Factors

This category includes the following properties:

- **X Scale Factor**
- **Y Scale Factor**
- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone. When the **Mesh Method** property is set to **Hex-Dominant**, the crack dimensions also include fracture-affected zones.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

below). To ensure the Named Selections accurately correspond to the **Elliptical Crack** object, the application uses the following naming convention:

- **NS**_*EllipticalCrack*_**Front**
- **NS**_*EllipticalCrack*_**TopFace**
- **NS**_*EllipticalCrack*_**BottomFace**
- **NS**_*EllipticalCrack*_**Contact1**
- **NS**_*EllipticalCrack*_**Target1**

For example, for a **Elliptical Crack** object named **Elliptical Crack 4**, the default names are **NS_EllipticalCrack 4_Front**, **NS_EllipticalCrack 4_TopFace**, **NS_EllipticalCrack 4_BottomFace**, **NS_EllipticalCrack 4_Contact1**, and **NS_EllipticalCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*EllipticalCrack*_**Front**). Contains nodes used for postprocessing of results.
- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*EllipticalCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*EllipticalCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

- **Contact Pairs Nodes**: This property displays when the **Mesh Method** property is set to **Hex Dominant**. It determines whether Named Selections are created automatically for the contact and target faces of the contact pair. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Contact 1 Nodes: Identifies the Named Selection that is created automatically for contact face 1 (**NS**_*EllipticalCrack*_**Contact1**). It contains nodes located on the contact face. The contact nodes are selected at the interface from the buffer zone of the base mesh.

  Target 1 Nodes: Identifies the Named Selection that is created automatically for target face 1 (**NS**_*EllipticalCrack*_**Target1**). It contains nodes located on the target

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

face. The target nodes are selected at the interface from the fracture affected zone of the hex dominant mesh.

Note:

Fracture meshing creates contact pair 1 between the fracture affected zone and the buffer zone.

6. Select the **Fracture** object or **Elliptical Crack** object, right-click, and select **Generate All Crack Meshes**.

   When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Elliptical Crack** object with which they are associated.

   Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining a Ring Crack

This section describes the steps to create an **Ring Crack**. You use ring cracks to insert embedded void cracks of a hallow disc shape into a solid body.

Procedure

To define a ring crack:

1. Select the **Model** object.
2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

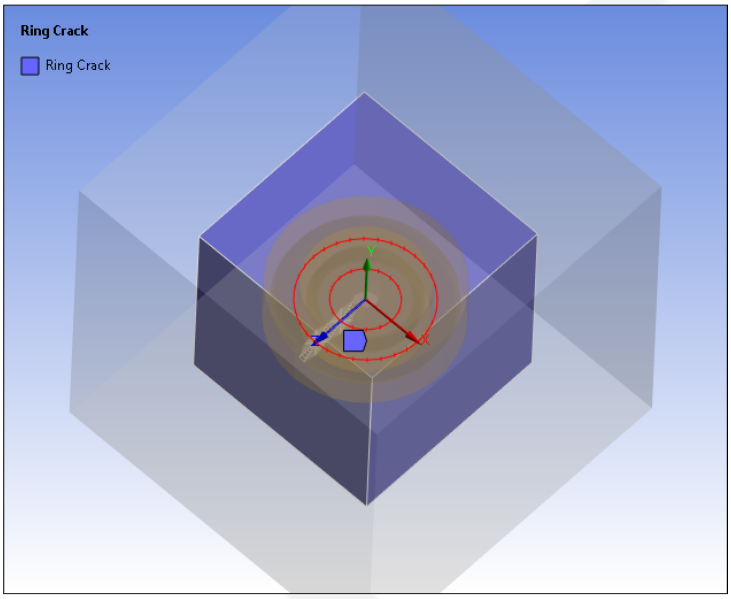
   Only one **Fracture** object is valid per **Model**.

3. Insert a **Ring Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Ring Crack**. You can also select the **Ring Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.
4. A ring crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.
5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Examples entries are illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Ring Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	87
Coordinate System	Coordinate System
Crack Shape	Ring
--Outer Major Radius	6. mm
--Outer Minor Radius	6. mm
--Inner Major Radius	3. mm
--Inner Minor Radius	3. mm
Mesh Method	Hex Dominant
Largest Contour Radius	2. mm
Outer Crack Front Divisions	32
Inner Crack Front Divisions	16
Fracture Affected Zone	Program Controlled
Fracture Affected Zone Height	5.2028 mm
Circumferential Divisions	8
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_RingCrack_Front
Crack Faces Nodes	On
--Top Face Nodes	NS_RingCrack_TopFace
--Bottom Face Nodes	NS_RingCrack_BottomFace
Contact Pairs Nodes	Off



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards the normal of the crack plane and the crack plane always lies in the X-Z plane of the specified coordinate system.
- **Crack Shape**: Read-only property set to **Ring**.
- **Outer Major Radius**: Specifies the outer major radius, which defines the size of the crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.
- **Outer Minor Radius**: Specifies the outer minor radius, which defines the size of the crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.
- **Inner Major Radius**: Specifies the inner major radius, which defines the size of the inner crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.
- **Inner Minor Radius**: Specifies the inner minor radius, which defines the size of the inner crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Mesh Method**: This property enables you to select the mesh method to be used to mesh the crack. Options include **Hex Dominant** (default) and **Tetrahedrons**.

  Note:

  When this crack intersects with the free surfaces of a solid body, the **Hex Dominant** setting is not supported.

- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.
- **Growth Rate**: This property is only visible when the **Mesh Method** property is set to **Tetrahedrons**. It specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Outer Front Element Size**: This property is only visible when the **Mesh Method** property is set to **Tetrahedrons**. It specifies the element size for the outer crack front. The default value is computed using crack length. Specify a value greater than 0.
- **Inner Front Element Size**: This property is only visible when the **Mesh Method** property is set to **Tetrahedrons**. It specifies the element size for the inner crack front. The default value is computed using crack length. Specify a value greater than 0.
- **Outer Crack Front Divisions**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. It specifies the number of divisions for the outer crack front. Your entry must be equal to or greater than 3. The default is 30.

  Note:

  The maximum number of crack front divisions the **Geometry** window can display is 999. However, you can specify a higher value and the application will process it accordingly.

- **Inner Crack Front Divisions**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. It specifies the number of divisions for the inner crack front. Your entry must be equal to or greater than 3. The default is 30.
- **Fracture Affected Zone**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. This is the region that contains a crack. This property defines the height of the fracture affected zone. Property options include:
  - **Program Controlled** (default): The application calculates the height. When selected, the **Fracture Affected Zone Height** property is read-only.
  - **Manual**: You manually specify the height in the **Fracture Affected Zone Height** property when using this option.
- **Fracture Affected Zone Height**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. This value specifies two things: 1) the height of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the **Fracture Affected Zone**, which is in the Y direction of the crack coordinate system; and 2) the distance in totality by which the **Fracture Affected Zone** is extended in the positive and negative Z direction of the crack coordinate system from the crack front extremities.

The shape of a Fracture Affected Zone is rectangular, regardless of the shape of the crack. Although buffer zones may overlap, you should use care when defining multiple cracks so that the zones do not overlap.

Important:

The crack generation process will fail if buffer zones overlap.

- **Circumferential Divisions**: This property is only visible when the **Mesh Method** property is set to **Hex Dominant**. It specifies the number of circumferential divisions for the crack shape. The value you enter must be a multiple of 8, and must be equal to or greater than 8. The default is 8.

  Note:

  The maximum number of circumferential divisions the **Geometry** window can display is 360. However, you can specify a higher value and the application will process it accordingly.

- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than 1. The default setting is 6.

  Note:

  The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.

- **Suppressed**: Toggles suppression of the **Ring Crack** object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Buffer Zone Scale Factors

This category includes the following properties:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **X Scale Factor**
- **Y Scale Factor**
- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone. When the **Mesh Method** property is set to **Hex-Dominant**, the crack dimensions also include fracture-affected zones.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6 below). To ensure the Named Selections accurately correspond to the **Ring Crack** object, the application uses the following naming convention:

- **NS**_*RingCrack*_**Front**
- **NS**_*RingCrack*_**Front2**
- **NS**_*RingCrack*_**TopFace**
- **NS**_*RingCrack*_**BottomFace**
- **NS**_*RingCrack*_**Contact1**
- **NS**_*RingCrack*_**Target1**

For example, for a **Ring Crack** object named **Ring Crack 4**, the default names are **NS_RingCrack 4_Front**, **NS_RingCrack 4_Front2**, **NS_RingCrack 4_TopFace**, **NS_RingCrack 4_BottomFace**, **NS_RingCrack 4_Contact1**, and **NS_RingCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack fronts (**NS**_*RingCrack*_**Front** and **NS_RingCrack 4_Front2**). Contains nodes used for postprocessing of results.

> **Important:**
>
> For cracks with multiple crack fronts, the **Crack Front Nodes** property, under **Named Selection Creation** category, only modifies the first crack front name if you have already generated the mesh. However, the application applies the property value to all crack fronts if you are generating the mesh for the first time or if the mesh is re-generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*RingCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*RingCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

- **Contact Pairs Nodes**: This property displays when the **Mesh Method** property is set to **Hex Dominant**. It determines whether Named Selections are created automatically for the contact and target faces of the contact pair. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Contact 1 Nodes: Identifies the Named Selection that is created automatically for contact face 1 (**NS**_*RingCrack*_**Contact1**). It contains nodes located on the contact face. The contact nodes are selected at the interface from the buffer zone of the base mesh.

  Target 1 Nodes: Identifies the Named Selection that is created automatically for target face 1 (**NS**_*RingCrack*_**Target1**). It contains nodes located on the target face. The target nodes are selected at the interface from the fracture affected zone of the hex dominant mesh.

Note:

Fracture meshing creates contact pair 1 between the fracture affected zone and the buffer zone.

---

6. Select the **Fracture** object or **Ring Crack** object, right-click, and select **Generate All Crack Meshes**.

   When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Ring Crack** object with which they are associated.

   Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining a Corner Crack

This section describes the steps to create a **Corner Crack**. You use this crack type to create a crack at any corner of a solid body. This crack must intersect at least two faces of the solid body.

Procedure

To define a **Corner Crack**:

1. Select the **Model** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

Note:

Only one **Fracture** object is valid per **Model**.

---

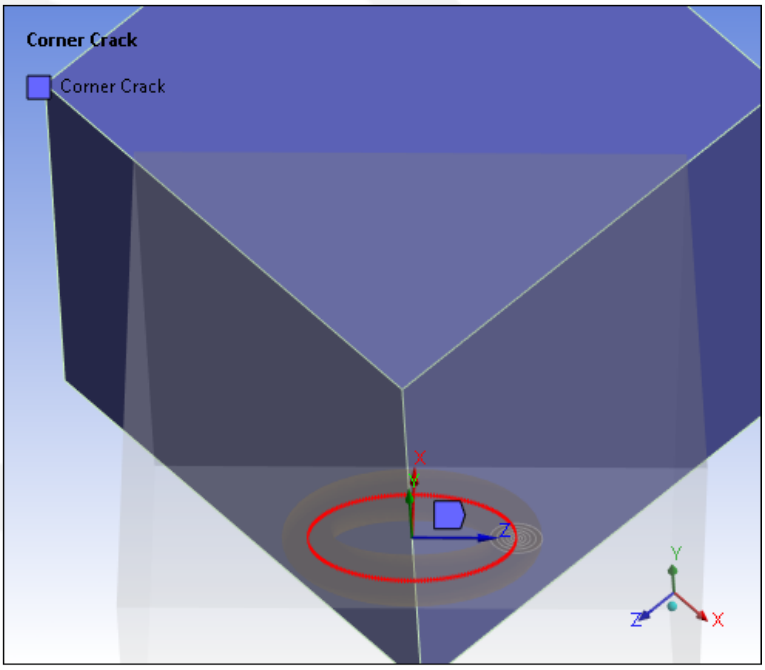
3. Insert a **Corner Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Corner Crack**. You can also select the **Corner Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.

4. A corner crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.

5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Example entries are illustrated below.

Details of "Corner Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	86
Coordinate System	Origin
Crack Shape	Corner
--Major Radius	4. mm
--Minor Radius	2. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	1. mm
Growth Rate	Default (1.2)
Front Element Size	Default (0.10071 mm)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_CornerCrack_Front
Crack Faces Nodes	Off



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the normal of the crack plane and the crack plane always lies in the X-Z plane of the specified coordinate system.

- **Crack Shape**: Read-only property set to **Corner**.
- **Major Radius**: Specifies the major radius, which defines the size of the crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.
- **Minor Radius**: Specifies the minor radius, which defines the size of the crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.
- **Mesh Method**: This is a read-only property set to **Tetrahedrons**, the only supported method.
- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.
- **Growth Rate**: This property specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size**: This property specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.
- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than 1. The default setting is **6**.

Note:

The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the crack object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Buffer Zone Scale Factors

This category includes the following properties:

- **X Scale Factor**
- **Y Scale Factor**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6 below). To ensure the Named Selections accurately correspond to the **Corner Crack** object, the application uses the following naming convention:

- **NS**_*CornerCrack*_**Front**
- **NS**_*CornerCrack*_**TopFace**
- **NS**_*CornerCrack*_**BottomFace**
- **NS**_*CornerCrack*_**Contact1**
- **NS**_*CornerCrack*_**Target1**

For example, for a **Corner Crack** object named **Corner Crack 4**, the default names are **NS_CornerCrack 4_Front**, **NS_CornerCrack 4_TopFace**, **NS_CornerCrack 4_BottomFace**, **NS_CornerCrack 4_Contact1**, and **NS_CornerCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*CornerCrack*_**Front**). Contains nodes used for postprocessing of results.
- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*CornerCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*CornerCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

6. Select the **Fracture** object or **Corner Crack** object, right-click, and select **Generate All Crack Meshes**.

When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Corner Crack** object with which they are associated.

Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Defining an Edge Crack

This section describes the steps to create an **Edge Crack**. You can define this crack type on any face of a solid in order to cut into the face of the solid body.

Procedure

To define an **Edge Crack**:

1. Select the **Model** object.
2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

   Only one **Fracture** object is valid per **Model**.

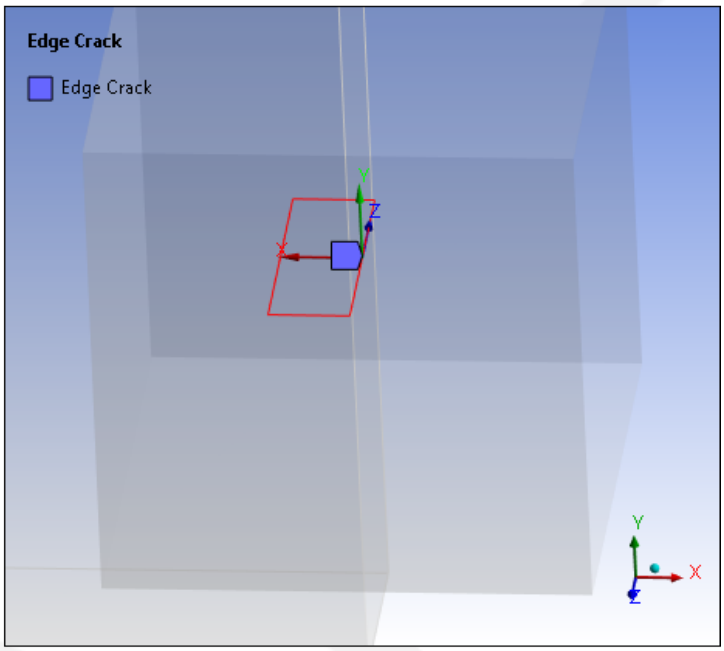
3. Insert a **Edge Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Edge Crack**. You can also select the **Edge Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.
4. A edge crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.
5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Example entries are illustrated below.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details of "Edge Crack"	
Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	110
Coordinate System	Edge Crack CS
Crack Shape	Edge
Start Width	2. mm
End Width	2. mm
Mid Width	2. mm
Depth	6. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	2. mm
Growth Rate	Default (1.2)
Front Element Size	Default (0.20141 mm)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_EdgeCrack_Front
Crack Faces Nodes	Off



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards the normal of the crack plane and the crack plane always lies in the X-Z plane of the specified coordinate system.
- **Crack Shape**: Read-only property set to **Edge**.
- **Mesh Method**: This is a read-only property set to **Tetrahedrons**, the only supported method.
- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.
- **Start Width**: Specifies the width at the start of the crack. Enter a value greater than **0**.
- **End Width**: Specifies the width at the end of the crack. Enter a value greater than **0**.
- **Mid Width**: Specifies the width at the centre of the start and end of the crack. Enter a value greater than **0**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Depth**: Specifies the depth of the crack. Enter a value greater than **0**.
- **Growth Rate**: This property specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size**: This property specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.
- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than 1. The default setting is **6**.

> **Note:**
>
> The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the crack object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Buffer Zone Scale Factors

This category includes the following properties:

- **X Scale Factor**
- **Y Scale Factor**
- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6 below). To ensure the Named Selections accurately correspond to the **Edge Crack** object, the application uses the following naming convention:

- **NS**_*EdgeCrack*_**Front**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **NS**_*EdgeCrack*_**TopFace**
- **NS**_*EdgeCrack*_**BottomFace**
- **NS**_*EdgeCrack*_**Contact1**
- **NS**_*EdgeCrack*_**Target1**

For example, for a **Edge Crack** object named **Edge Crack 4**, the default names are **NS_EdgeCrack 4_Front**, **NS_EdgeCrack 4_TopFace**, **NS_EdgeCrack 4_BottomFace**, **NS_EdgeCrack 4_Contact1**, and **NS_EdgeCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*EdgeCrack*_**Front**). Contains nodes used for postprocessing of results.
- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*EdgeCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*EdgeCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

6. Select the **Fracture** object or **Edge Crack** object, right-click, and select **Generate All Crack Meshes**.

   When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Edge Crack** object with which they are associated.

   Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining a Through Crack

This section describes the steps to create a **Through Crack**. You use this crack type to define a crack that cuts through a solid body from one surface to the other.

Procedure

To define a **Through Crack**:

1. Select the **Model** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

   Only one **Fracture** object is valid per **Model**.

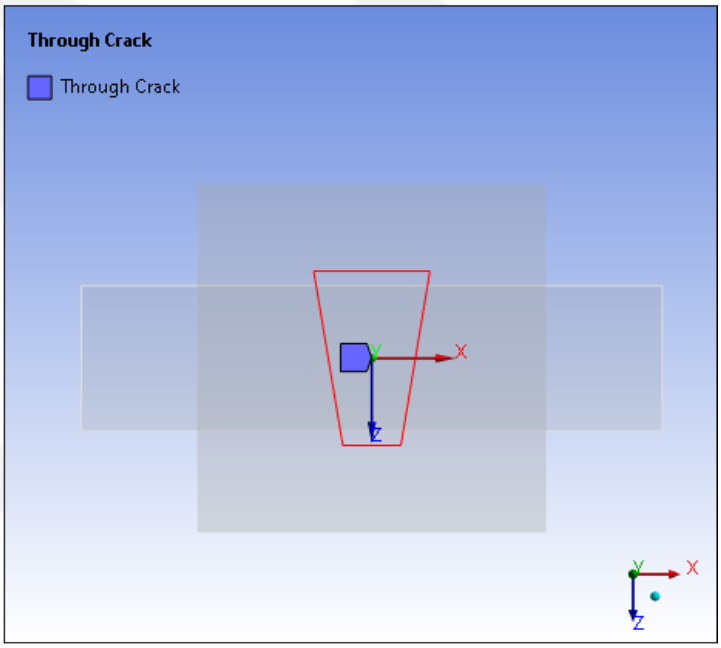
3. Insert a **Through Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Through Crack**. You can also select the **Through Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.

4. A through crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.

5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Example entries are illustrated below.

Details of "Through Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	102
Coordinate System	Coordinate System
Crack Shape	Through
Start Width	4. mm
End Width	2. mm
Mid Width	3. mm
Depth	6. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	2. mm
Growth Rate	Default (1.2)
Front Element Size	Default (0.20141 mm)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_ThroughCrack_Front
Crack Faces Nodes	Off



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The Y axis of the specified coordinate system must be directed towards the normal of the crack plane and the crack plane always lies in the X-Z plane of the specified coordinate system.
- **Crack Shape**: Read-only property set to **Through**.
- **Start Width**: Specifies the width at the start of the crack. Enter a value greater than **0**.
- **End Width**: Specifies the width at the end of the crack. Enter a value greater than **0**.
- **Mid Width**: Specifies the width at the centre of the start and end of the crack. Enter a value greater than **0**.
- **Depth**: Specifies the depth of the crack. Enter a value greater than **0**.
- **Mesh Method**: This is a read-only property set to **Tetrahedrons**, the only supported method.
- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.
- **Growth Rate**: This property specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size**: This property specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.
- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than 1. The default setting is **6**.

  Note:

  The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the crack object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Buffer Zone Scale Factors

This category includes the following properties:

- **X Scale Factor**
- **Y Scale Factor**
- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6 below). To ensure the Named Selections accurately correspond to the **Through Crack** object, the application uses the following naming convention:

- **NS**_*ThroughCrack*_**Front**
- **NS**_*ThroughCrack*_**TopFace**
- **NS**_*ThroughCrack*_**BottomFace**
- **NS**_*ThroughCrack*_**Contact1**
- **NS**_*ThroughCrack*_**Target1**

For example, for a **ThroughCrack** object named **ThroughCrack 4**, the default names are **NS_ThroughCrack 4_Front**, **NS_ThroughCrack 4_TopFace**, **NS_ThroughCrack 4_BottomFace**, **NS_ThroughCrack 4_Contact1**, and **NS_ThroughCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*ThroughCrack*_**Front**). Contains nodes used for postprocessing of results.
- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*ThroughCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*ThroughCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

6. Select the **Fracture** object or **Through Crack** object, right-click, and select **Generate All Crack Meshes**.

   When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Through Crack** object with which they are associated.

   Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining a Cylindrical Crack

This section describes the steps to create an **Cylindrical Crack**. You use the cylindrical crack option to insert a crack, with a cylindrical shape, into a solid body.

Procedure

To define an **Cylindrical Crack**:

1. Select the **Model** object.
2. Insert a Fracture object by right-clicking on the **Model** object and selecting **Insert** > **Fracture**. You can also select the **Fracture** option from the **Define** group on the Model Context Tab (p. 48).

   Note:

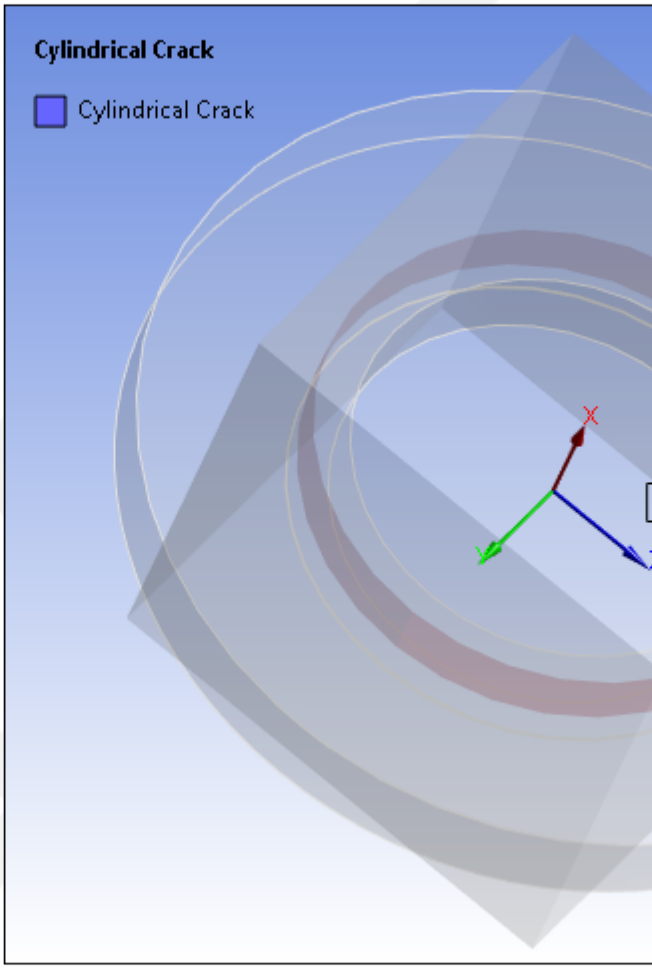
   Only one **Fracture** object is valid per **Model**.

3. Insert a **Cylindrical Crack** object by right-clicking on the **Fracture** object and selecting **Insert** > **Cylindrical Crack**. You can also select the **Cylindrical Crack** option from the **Crack** group of the **Fracture** (p. 52) Context Tab.
4. A cylindrical crack definition must always be scoped to a single solid body. Use the **Body** selection filter to pick a body in the **Geometry** window, click the **Geometry** field in the Details, and then click **Apply**.
5. Specify the properties of the crack. As you specify property values, the entries are reflected on the model in the **Geometry** window. Examples entries are illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Cylindrical Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	169
Coordinate System	Coordinate System
Crack Shape	Cylindrical
--Major Radius	0.6 m
--Minor Radius	0.6 m
--Height	0.15 m
Mesh Method	Tetrahedrons
Largest Contour Radius	2.5e-002 m
Growth Rate	Default (1.2)
Front Element Size	2.5e-002 m
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No
Buffer Zone Scale Factors	
X Scale Factor	2.
Y Scale Factor	2.
Z Scale Factor	2.
Named Selections Creation	
Crack Front Nodes	NS_CylindricalCrack_Front
Crack Faces Nodes	On
--Top Face Nodes	NS_CylindricalCrack_TopFace
--Bottom Face Nodes	NS_CylindricalCrack_BottomFace



Crack categories and properties include:

Definition

- **Crack ID**: A read-only property that displays a unique system generated identification value. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specify the coordinate system that defines the orientation of the crack. The X axis of the specified coordinate system must be directed towards the axis of the cylindrical crack and the first crack front always lies in the Y-Z plane of the specified coordinate system.
- **Crack Shape**: Read-only property set to **Cylindrical**.
- **Major Radius**: Specifies the major radius, which defines the size of the crack shape along the Y axis. Enter a value greater than **0**.
- **Minor Radius**: Specifies the minor radius, which defines the size of the crack shape along the Z axis. Enter a value greater than **0**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Height**: Specifies the height of the cylindrical crack to be inserted. The height is defined from the origin of the coordinate system (starting face of the cylinder) to the other end of the cylinder.

  Important:

  If the specified height is less than the gap between the origin of the coordinate system and the free surface of the solid geometry, the mesh generates two crack fronts and two crack surfaces. Otherwise, the mesh generates a single crack front and two crack surfaces.

- **Mesh Method**: This is a read-only property set to **Tetrahedrons**, the only supported method.
- **Largest Contour Radius**: Specifies the largest contour radius for the crack shape. Enter a value greater than **0**.
- **Growth Rate**: It specifies the factor with which the mesh layers will grow along the radius of the crack. Specify a value greater than **1**. The default value is **1.2**. The recommended value is equal to or greater than **1.1**.
- **Front Element Size**: It specifies the element size for the crack front. The default value is computed using crack length. Specify a value greater than **0**.
- **Mesh Contours**: Specifies the number of mesh contours for the crack shape. The value you enter must be equal to or greater than **1**. The default setting is **6**.

  Note:

  The maximum number of mesh contours the **Geometry** window can display is 100. However, you can specify a higher value and the application will process it accordingly.

- **Solution Contours**: Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of **Mesh Contours**, and cannot be greater than 99. By default, the value is **Match Mesh Contours**, indicating the number of **Solution Contours** is equal to the number of **Mesh Contours**. Entering 0 resets the value to **Match Mesh Contours**.
- **Suppressed**: Toggles suppression of the **Cylindrical Crack** object. The default is **No**. The application automatically suppresses the crack object if the scoped body becomes suppressed.

Buffer Zone Scale Factors

This category includes the following properties:

- **X Scale Factor**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Y Scale Factor**
- **Z Scale Factor**

These properties control the size of the buffer zone in the X, Y, and Z directions, relative to the dimensions of the crack. For each scaling parameter, use the slider to set a value from **2** to **50**. The default is **2**. The maximum dimension among the three directions of the crack is multiplied by the corresponding scale factors to create a buffer zone.

Named Selection Creation

The **Named Selection Creation** category of the **Details** displays Named Selections automatically created by the application following fracture mesh generation (Step 6 below). To ensure the Named Selections accurately correspond to the **Cylindrical Crack** object, the application uses the following naming convention:

- **NS**_*CylindricalCrack*_**Front**
- **NS**_*CylindricalCrack*_**TopFace**
- **NS**_*CylindricalCrack*_**BottomFace**
- **NS**_*CylindricalCrack*_**Contact1**
- **NS**_*CylindricalCrack*_**Target1**

For example, for a **Cylindrical Crack** object named **Cylindrical Crack 4**, the default names are **NS_CylindricalCrack 4_Front**, **NS_CylindricalCrack 4_TopFace**, **NS_CylindricalCrack 4_BottomFace**, **NS_CylindricalCrack 4_Contact1**, and **NS_CylindricalCrack 4_Target1**.

The properties of this category are defined as follows:

- **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (**NS**_*CylindricalCrack*_**Front**). Contains nodes used for postprocessing of results.
- **Crack Faces Nodes**: Determines whether Named Selections are created automatically for the crack's top face and bottom face. These faces are both located in the XZ plane and are discontinuous. Options include **Off** (default) and **On**. If set to **On**, the following additional properties appear:

  Top Face Nodes: Identifies the Named Selection that is created automatically for the top face (**NS**_*CylindricalCrack*_**TopFace**). This face is discontinuity plane 1. It contains nodes used for applying a pressure to the top face.

  Bottom Face Nodes: Identifies the Named Selection that is created automatically for the bottom face (**NS**_*CylindricalCrack*_**BottomFace**). This face is discontinuity plane 2. It contains nodes used for applying a pressure to the bottom face.

6. Select the **Fracture** object or **Cylindrical Crack** object, right-click, and select **Generate All Crack Meshes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When the fracture mesh is generated, the requested Named Selections are automatically inserted into the Outline under the **Cylindrical Crack** object with which they are associated.

Select the **Show Mesh** option on the Graphics Toolbar (p. 97) to display the fracture mesh.

Defining a Pre-Meshed Crack

A Pre-Meshed Crack is based on a previously-generated mesh and uses a node-based named selection to analyze crack fronts. In addition to cracks modeled in CAD and meshed manually in the Mechanical Application, this feature is also useful when you have a pre-existing mesh generated from an external source and imported to the existing database. The referenced named selection must contain references only to nodes.

Selecting the named selection is done through the Details view of the Pre-Meshed Crack object by selecting from the list of valid named selections in the Crack Front (Named Selection) property. Named selections that contain only nodes are offered as choices.

Important:

To properly define a **Pre-Meshed Crack** object, you need to create the necessary node-based Named Selections and generate all crack meshes in advance. For more information on named selections, see Working with Named Selections (p. 1057). As an alternative, a geometric based named selection can be converted into a node-based based named selection using the Worksheet. For more information, see Specifying Named Selections using Worksheet Criteria (p. 1061).

To define a pre-meshed crack:

1. Select the **Model** object in the Tree Outline.
2. Insert a **Fracture** object into the Tree by right-clicking the **Model** object and selecting **Insert** > **Fracture**.

   Note:

   Only one **Fracture** object is valid per **Model**.

   ---

3. Insert a **Pre-Meshed Crack** object into the Tree by right-clicking the **Fracture** object and selecting **Insert** > **Pre-Meshed Crack**. You can also select the **Pre-Meshed Crack** option from the **Crack** group of the **Fracture** Context Tab.
4. In the **Details View**:

   For 2D Analysis

   Specify the node-based named selection for the **Crack Tip (Named Selection)** property. This property defines the nodes to which the crack definition will be scoped.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For 3D Analysis

- Specify the node-based named selection for the **Crack Front (Named Selection)** property. This property defines the nodes to which the crack definition will be scoped.
- As desired, set the Crack Faces Nodes property to **On**. This property enables you to specify a crack top face and bottom face using node-based Named Selections. The default setting is **Off**. When set to **On**, the following additional properties display:
  - **Top Face Nodes**: Specify the top face (nodes) of the crack by selecting a valid node-based named selection from the drop-down list.
  - **Bottom Face Nodes**: Specify the bottom face (nodes) of the crack by selecting a valid node-based named selection from the drop-down list.

5. To further define the crack, use the following controls in the **Details View**.

- **Crack ID**: A read-only property that displays a unique system generated identification number ID for the crack object. The application uses this identifier when creating solution identifiers for fracture parameters.
- **Coordinate System**: Specifies the coordinate system that defines the position and orientation of the crack. The Y axis of the specified coordinate system defines the crack surface normal. The origin of the coordinate system represents the open side of the crack. You can select the default coordinate system or a local coordinate system that you have defined. The default is the **Global Coordinate System**. The valid coordinate system must be of type Cartesian.
- **Solution Contours**: Specifies the number of contours for which you want to compute the fracture result parameters.
- **Symmetry**: Specifies the crack symmetry about a line (in 2D analysis) or about a plane (in 3D analysis). The default is **No**.
- **Suppressed**: Toggles suppression of the **Pre-Meshed Crack** object. The default is **No**.

Note:

The **Pre-Meshed Crack** object is suppressed automatically if the scoped body is suppressed.

Special Handling of Named Selections for Crack Objects

Crack objects dynamically generate Named Selections during crack mesh generation. The Named Selections generated for **Arbitrary Crack**, **Semi-Elliptical Crack**, **Elliptical**, or **Ring** crack objects are a special type of Named Selection and have the following distinct properties:

- Named Selections for **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, or **Arbitrary Crack** objects are not inserted into the Named Selections branch of the Tree Outline. They appear in

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the Outline under the **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack** or **Arbitrary Crack** objects with which they are associated.

- You may rename a Named Selection by editing its name in the Details of the **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack** or **Arbitrary Crack** objects or by right-clicking the Named Selection in the tree Outline and selecting **Rename**. Use caution when renaming the Named Selection associated with the crack front (**NS**_*SECrack*_**Front**). If the name is not unique when compared to other Named Selections, the crack definition sent to the solver may contain the wrong set of crack front nodes.

  Important:

  For cracks with multiple crack fronts, the **Crack Front Nodes** property, under **Named Selection Creation** category, only modifies the first crack front name if you have already generated the mesh. However, the application applies the property value to all crack fronts if you are generating the mesh for the first time or if the mesh is re-generated.

- For a **Ring Crack**, two crack front named selections are created at the outer and inner intersection of the crack with the solid body.
- An **Arbitrary Crack** defined using an embedded or intersecting crack surface body could generate more than one crack front.
- You cannot insert, duplicate, copy, delete, or merge these Named Selections.
- You may scope nodal-based loads to these Named Selections.
- By default, nodal Named Selections show nodes attached to them in the **Geometry** window. To plot elements attached to these nodal Named Selections, refer to Specifying Annotation Preferences (p. 310) section of the help.
- If you delete **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, or **Arbitrary Crack** objects, all associated Named Selections are deleted.
- If you suppress **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, or **Arbitrary Crack** objects, the nodal selection of each associated Named Selection is cleared and the state of each Named Selections becomes suppressed.
- All Named Selections for **Semi-Elliptical Crack**, **Elliptical Crack**, **Ring Crack**, or **Arbitrary Crack** objects are sent to the solver. **Send to Solver** is always set to **Yes** and is read-only.
- If Named Selections for **Crack Faces Nodes** and/or **Contact Pairs Nodes** have been created and you subsequently set the fields to **Off**, the corresponding Named Selections are deleted when a new crack mesh is generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- See the Working with Named Selections (p. 1057) section of the Help for additional information.

Note:

For information about common fracture meshing problems and troubleshooting, see Fracture Meshing Problems (p. 2380).

SMART Crack Growth

You use the **SMART Crack Growth** object to simulate fatigue or static crack growth in engineering structures. SMART is a remeshing-based method for crack growth simulation. SMART stands for Separating, Morphing, Adaptive, and Remeshing Technology. SMART automatically uses a combination of morphing, adaptive, and remeshing methods to update mesh changes to simulate static or fatigue crack growth during the solution process. The mesh changes occur around the crack-front region only, leading to a computationally efficient solution of the crack-growth problem. See the following subsections for more information.

SMART Crack Growth Application

SMART Crack-Growth Assumptions and Limitations

Note:

When you define **SMART Crack Growth**, the application stores fracture results for time points based on the setting of the **Analysis Settings** > Output Controls (p. 1473) property **Store Results At** (p. 1481).

Important:

If the **Store Results At** property is not set to the **All Time Points** option, the **Crack Extension** and **Total Number of Cycles** probe results will be inaccurate. This is because these values are derived from the summation of results stored in the results file instead of the summation of results over all solved sub-steps. Therefore, Ansys recommends that you set the **Store Results At** property to **All Time Points** for accurate **Crack Extension** and **Total Number of Cycles** probe results. Furthermore, Ansys recommends that you use this property to specify time points, even if you issue additional **OUTRES** commands through the **Commands (APDL)** object.

Note:

- For additional technical information on this feature, refer to the SMART Method for Crack-Growth Simulation section in the *Mechanical APDL Fracture Analysis Guide*.
- For specific information about the properties of the object, see the SMART Crack Growth object reference page.
- For a video example of the feature, see the Fatigue Crack Growth Analysis using SMART Crack Growth Tutorial.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

SMART Crack Growth Application

The following procedure describes the steps to define a **SMART Crack Growth** object. The procedure assumes that you have already inserted a Fracture folder and defined the prerequisite crack. This crack can be any of the supported crack types. The feature supports the definition of multiple **SMART Crack Growth** objects when your model contains multiple cracks.

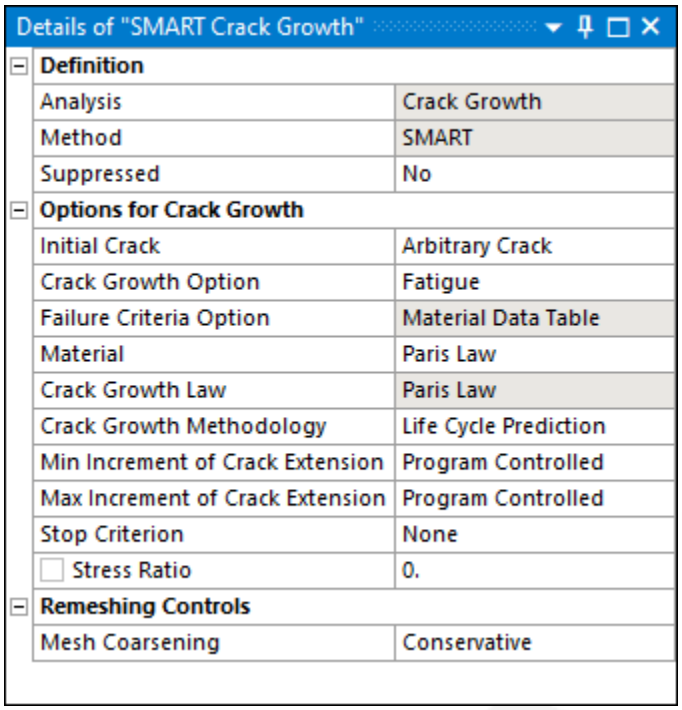
1. Insert a **SMART Crack Growth** object into the **Outline** by right-clicking on the **Fracture** object and selecting **Insert** > **SMART Crack Growth** from the context menu. Alternatively, you can select the **SMART Crack Growth** option on the Fracture Context Tab (p. 52) or right-click in the **Geometry** window and select **Insert** > **SMART Crack Growth**.

2. Select the new **SMART Crack Growth** object in the **Outline**. The **Initial Crack** and **Material** properties highlight. First, select the crack that you have already created for the **Initial Crack** property. An example is illustrated below using a **Pre-Meshed Crack**. Once selected, the application automatically selects the default **Material** of scoped crack body (specified by the **Initial Crack** property), in this case, Material1.

Details of "SMART Crack Growth"

Definition	
Analysis	Crack Growth
Method	SMART
Suppressed	No
Options for Crack Growth	
Initial Crack	
Crack Growth Option	Fatigue
Failure Criteria Option	Material Data Table
Material	
Crack Growth Law	Paris Law
Crack Growth Methodology	Life Cycle Prediction
Min Increment of Crack Extension	Program Controlled
Max Increment of Crack Extension	Program Controlled
Stop Criterion	None
Stress Ratio	0.
Remeshing Controls	
Mesh Coarsening	Conservative

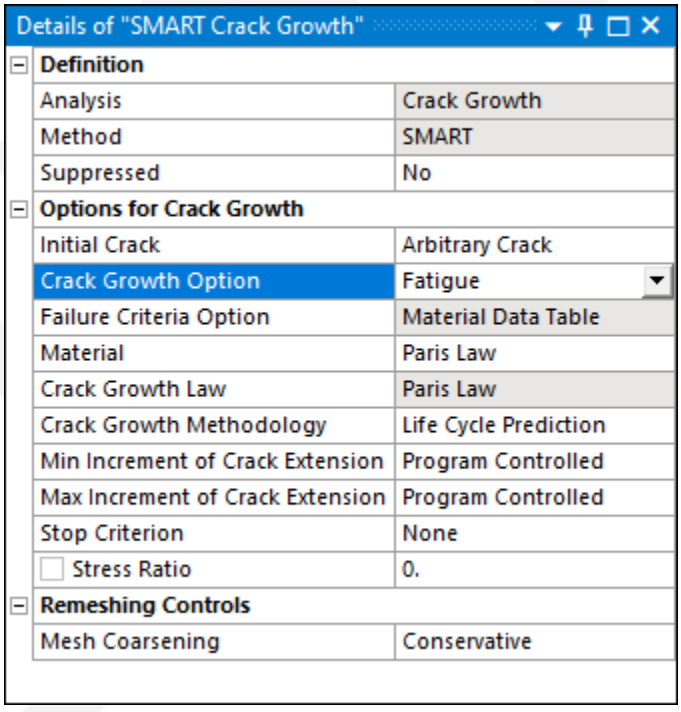
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Specify the **Crack Growth Option** property. The setting of this property is based on the desired type of crack growth propagation. Select **Fatigue** (default) or **Static**.

Fatigue

If you set the **Crack Growth Option** property to **Fatigue**, your structure is subject to constant amplitude cyclic load.



The following other properties are available in the category:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Failure Criteria Option**: This is a read-only property that is set to **Material Data Table**.
- **Material**: The application automatically selects the default material that is assigned to the crack specified that is specified in the **Initial Crack** property. You can change the material using the property's fly-out menu. Any material that you select must include the material property Paris' Law.
- **Crack Growth Law**: This is a read-only property that displays the given Crack Growth Law: **Paris' Law**.
- **Crack Growth Methodology**: Fatigue crack growth can be modeled using either **Life Cycle Prediction** (default) or **Cycle By Cycle** methodologies. If you specify **Cycle By Cycle**, the property **Incremental Number of Cycles** displays. Use this property to specify the incremental number of cycles during a substep. The default setting is **10**.

  The following additional properties are also available:

  – **Min Increment of Crack Extension**: This property specifies the minimum crack extension increment value. The options include **Program Controlled** (default) and **Manual**. The **Program Controlled** option uses the default minimum increment value. If you set the property to **Manual**, The **Min Increment Value** property displays and enables you to specify a value. The default value is **0**.

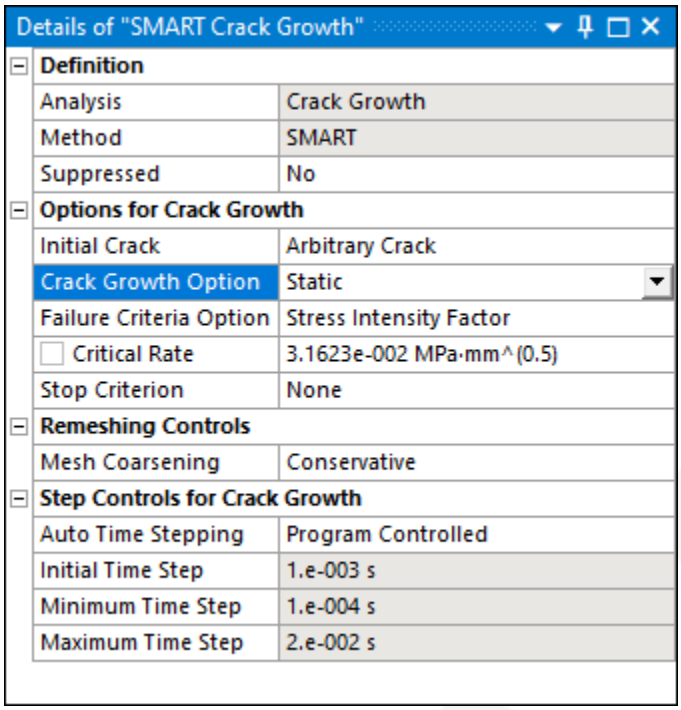
  – **Max Increment of Crack Extension**: This property specifies the maximum crack extension increment value. The options include **Program Controlled** (default) and **Manual**. The **Program Controlled** option uses the default maximum increment value. If you set the property to **Manual**, The **Max Increment Value** property displays and enables you to specify a value. The default value is **0**.

  – **Stress Ratio**: You use this property to specify the stress ratio. The default value is **0**. The entry range is less than **1**.

Static

If you set the **Crack Growth Option** property to **Static**, crack growth modeling is based on selected fracture parameters and criteria.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Using the **Failure Criteria Option** property, you can specify the failure criteria as either **Stress Intensity Factor** (default) or **J-Integral**. Each option requires you to specify a value in the dependent property **Critical Rate**. The default value for the Critical Rate property is **0** for either option. The unit system of the **Critical Rate** property varies based on your selection. The property can be parameterized.

Note:

Either the **Stress Intensity Factor** or **J-Integral** fracture parameter can be computed in one solution. Additionally, you can compute **Equivalent SIFS Range** fracture result and related time history results on any crack front node (Fracture Probes (p. 2127)).

Furthermore, when you set the **Crack Growth Option** property to **Static**, an additional category displays for the object: **Step Controls for Crack Growth**. The properties for this category include:

- **Auto Time Stepping**: Property options include **Program Controlled** (default) or **Manual**. Setting the property to Manual enables you to modify the following time step properties, otherwise they are read-only.
- **Initial Time Step**: Defines the initial time step to initiate crack growth.
- **Minimum Time Step**: Minimum time step for subsequent crack growth.
- **Maximum Time Step**: Maximum time step for subsequent crack growth.

4. As needed, specify the **Stop Criterion** property. Options include **None** (default), **Max Crack Extension**, **Free Boundary**, **Max Stress Intensity Factor**, and **Max Total Number of Cycles** (displayed when the **Crack Growth Option** property is set to **Fatigue**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When you select the **Max Crack Extension**, **Max Stress Intensity Factor**, or the **Max Total Number of Cycles**, the **--Stop Value** property displays. This property requires you to enter a value for the selected **Stop Criterion** option. Once the maximum stop criterion limit is reached, the application stops the solution process. In this instance, the solution is incomplete and the **Solution** folder will not be in solved state because the solution is not complete for all time points. If the maximum stop criterion limit is not reached during solution, then the solution process completes normally.

5. Specify the **Mesh Coarsening** property. You use this property to define a desired level of coarsening of the excessively fine mesh surrounding the crack front during the SMART crack growth remeshing processing. Options include:

   - **Conservative** (default): This option performs the least amount of mesh coarsening and creates a larger number of nodes and elements after a substantial amount of crack-growth. This options offers the greatest remeshing success rate.
   - **Moderate**: This option modestly reduces the number of nodes and elements and offers a satisfactory remeshing success rate.
   - **Aggressive**: This option quickly coarsens the mesh in the region behind crack front and offers the largest reduction in the number of nodes and elements following a substantial amount of crack-growth.

6. Repeat the steps above to define additional **SMART Crack Growth** objects when multiple cracks exist in the model.

SMART Crack-Growth Assumptions and Limitations

Go to a section topic:

- General Assumptions and Limitations (p. 473)
- Postprocessing Limitations (p. 475)
- Multiple SMART Crack Growth Objects (p. 475)

General Assumptions and Limitations

Review the following for the use of the **SMART Crack Growth** feature:

- Only supports Static Structural analyses.
- Supports 3-D crack growth only.
- Supports Mixed Mode Crack Growth for Mode I and II only. If using J-Integral as Crack Growth Criterion, then only straight (Mode I) crack growth is supported.
- Supports higher order tetrahedron mesh. To use it with a hex-dominant base mesh (for Arbitrary Crack and Semi-Elliptical Crack objects only) you need to set the Fracture object property **Re-mesh Hex-dominant to Tetrahedral** to **On**. This will re-mesh the bodies scoped to the **Arbitrary Crack** and **Semi-Elliptical Crack** objects to tetrahedrons when **Generate All Crack Meshes** (right-click) menu option is executed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Material behavior is assumed to be linear elastic isotropic.
- Fatigue crack growth is based on Paris' Law.
- Plasticity effects, nonlinear geometry effects, load-compression effects, and crack-tip-closure effects are not considered.
- Supports the Bonded, No-Separation, Frictionless, Rough, and Frictional contact Type (p. 1234) property options with CONTA174 and TARGE170 when they remain outside the remeshing zone during the simulation. SMART does not support any contact element inside the remeshing zone.
- Function based loads and tabular loads with time as independent variable are not supported for fatigue crack growth analysis.
- Node- and element-based components are not maintained after remeshing. Therefore, Ansys recommends that you use the **Tabular Data** option of the **Define By** property to define the load in a multi-step analysis.
- Does not currently support **Point Mass** or **Distributed Mass**.
- Only supports the following Imported Loads (p. 1736):
  - Imported Pressure
  - Imported Force
  - Imported Displacement
  - Imported Body Temperature
  - Imported Initial Stress
- Does not currently support Hydrostatic Pressure.
- Either SIFS or J-Integral fracture parameter can be computed in one solution.
- Restarting the analysis is currently not supported.

Note:

Activate beta options to make solution restarts available for **SMART Crack Growth**.

- When the crack grows to the point of breaking the structural component apart, all solution results are set to zero and no crack-front information is reported.
- **Graphics Limitation**: During your analysis, if your SMART crack grows from one body to another body, you need to scope the corresponding bodies to a Material Assignment object (p. 1049) and assign them with the same material using **Material Name** property. This in turn facilitates the proper display of the crack front contour result in the graphics window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Postprocessing Limitations

Note the following **SMART Crack Growth** postprocessing limitations:

- Probes scoped to the same body as a **SMART Crack Growth** object will **not** evaluate.
- The **Crack Extension** probe displays the sum of the crack extension increments from only those substeps in which crack extension increment information has been saved. That is, if crack results are not written for each specified Substep, as is the case when **Analysis Settings** > **Output Controls** property **Store Results At** is not set to **All Time Points** (or using the **OUTRES,CINT,Value** command snippet) in a Static Structural environment, the application does not display the entire crack extension for the **Crack Extension** probe. It is because these values are derived from the summation of results stored in the results file instead of the summation of results over all solved sub-steps. This same behavior is found with **Total Number of Cycles** probe result. Even if you issue additional **OUTRES** commands through the **Commands (APDL)** object, Ansys recommends that you issue **OUTRES,CINT,ALL** to specify all time points,

Multiple SMART Crack Growth Objects

Make sure of the following setting when you have specified multiple **SMART Crack Growth** objects:

- Each crack specified on the model must be associated with an unique **SMART Crack Growth** object (1:1 ratio).
- The **Crack Growth Option** property must be set to the same option (**Fatigue** or **Static**) for all **SMART Crack Growth** objects.
- The **Failure Criteria Option** property must be set to the same option (**SIFS** or **J-Integral**) when the **Crack Growth Option** property is set to **Static** for all **SMART Crack Growth** objects.
- The **Crack Growth Methodology** property must be set to the same option (**Life Cycle Prediction** or **Cycle By Cycle**) when the **Crack Growth Option** property is set to **Fatigue** for all **SMART Crack Growth** objects.

Note:

The **SMART Crack Growth** feature uses local re-meshing and adaptation techniques and therefore has restrictions for the crack extension size.

For example, the feature might modify your entries in the **Min Increment of Crack Extension** or **Max Increment of Crack Extension** properties for either **Crack Growth Methodology**, either **Life Cycle Prediction** or **Cycle By Cycle**.

When the **Crack Growth Methodology** property is set to **Life Cycle Prediction** and the **Max Increment of Crack Extension** manual entry is larger than 1.5 times of the element size at the crack front, the **Max Increment of Crack Extension** value is modified to 1.5 times of the element size. Similarly, if the manual entry for the **Min Increment of Crack Extension** is smaller than 0.25 times of the element size at the crack tip, the **Min Increment of Crack Extension** value is modified to 0.25 times of the element size.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When the **Crack Growth Methodology** property is set to **Cycle By Cycle**, and you enter a number of cycles that is too large or too small, then the algorithm automatically re-defines the number of cycles according to the crack front element size. However, the **Min Increment of Crack Extension**, the **Max Increment of Crack Extension**, and the **Increment Number of Cycles** property values may not be satisfied during the solution to ensure a robust mesh change.

Interface Delamination and Contact Debonding

Adhesives are commonly used to bond structural components into assemblies or to bond layers of material into composite laminates. Simulations often assume the bonding layer to be of infinite strength, but you may want to model the progressive separation of the adhesive as it reaches some known criteria, such as a stress limit. Of the existing theories that define these failure criteria, Mechanical supports the Cohesive-Zone Model (CZM) method and the Virtual Crack Closure Technique (VCCT) method. (See the Cohesive Zone Material (CZM) Model in the *Theory Reference* and VCCT-Based Crack-Growth Simulation in the *Fracture Analysis Guide* for more information about these methods.) In either case, the separation occurs along a predefined interface and cannot propagate in an arbitrary direction

Mechanical supports the following features for modeling interface delamination and debonding:

- **Interface Delamination** – utilizes Mechanical APDL interface elements (INTER202 through INTER205) and supports the CZM and VCCT methods. Neither method supports interfaces with lower order triangle faces. Specifically, a prism with a triangle face on the interface or a tetrahedral element with a face on the interface. And, the VCCT does not support higher order elements.
- **Contact Debonding** utilizes Mechanical APDL contact elements (CONTA171 through CONTA177) and supports the CZM method.

For additional technical information about Interface Delamination, see Modeling Interface Delamination with Interface Elements in the *Fracture Analysis Guide*. For more information about Contact Debonding, see Modeling Interface Delamination with Contact Elements (Debonding) in the *Fracture Analysis Guide*.

See the Interface Delamination Application (p. 477) and Contact Debonding Application (p. 480) sections for the steps to specify and configure these features. In addition, if you are using the Ansys Composite PrepPost (ACP) application in combination with the Interface Delamination feature, see the steps in the Interface Delamination and Ansys Composite PrepPost (ACP) (p. 481) section.

Analysis Type Requirements

Interface Delamination is supported by Static Structural and Transient Structural analyses only.

Any analysis type may contain a **Contact Debonding** object, but only the Static Structural and Transient Structural analyses support the progressive separation of an interface. **Contact Debonding** also supports linear perturbation, which allows you to simulate the vibration (Pre-stressed Modal) or stability (Eigenvalue Buckling) characteristics of a partially delaminated structure. You can also use the modes extracted in the Pre-stressed model to perform Mode Superposition analyses such as Harmonic Response, Response Spectrum, and Random Vibration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Interface Delamination Application

The **Interface Delamination** feature employs either the Virtual Crack Closure Technique (VCCT) method or the Cohesive-Zone Model (CZM) method for defining failure criteria. The VCCT method is a fracture mechanics based method and therefore requires an initial crack (in the form of a Pre-Meshed Crack) in the geometry. The CZM method uses relationships between the separations and tractions along the interface. Note that the CZM method is sensitive to mesh size and material parameters. The convergence of CZM models can generate issues, such as loading step size and stabilization.

Important:

The **Interface Delamination** feature is not supported with **Semi-Elliptical** or **Arbitrary** cracks.

To correctly insert the structural interface elements (INTER202 through INTER205) into the mesh, the Interface Delamination feature requires that the sides of the interface have identical element patterns.

Both the VCCT and CZM methods provide the option to use either the **Matched Meshing** or the **Node Matching** generation method. **Matched Meshing** requires that you create a Mesh **Match Control** at the delamination interface.

A **Match Control** requires that both faces referenced by the **Match Control** belong to the same part, so it is necessary that you create a multi-body part without shared topology. This can be accomplished in a CAD application, such as DesignModeler. **Matched Meshing** is the recommended **Generation Method** because it quickly obtains the matching node pairs from the mesh.

Caution:

The application will not respect mesh matching controls when one or more mesh Refinement controls exist. This may result in mismatched node pairs and element faces.

If using a **Match Control** is not an option and it is necessary to use the **Node Matching** method, you must ensure that node pairs and element faces match. Because it is necessary for Mechanical to search the scoped geometry for matching node pairs within the specified **Distance Tolerance**, this method can be slower and less robust than the **Matched Meshing** method.

Note:

The Interface Delamination feature does not support adaptive mesh refinement.

Also see the Interface Delamination Object Reference page for information about the properties of this feature.

Apply Interface Delamination

To specify **Interface Delamination**:

1. Insert a **Fracture** folder in the **Tree Outline**. The **Fracture** object becomes active by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. On the **Fracture** Context Tab (p. 52): click **Interface Delamination**. Or, right-click:

   - the **Fracture** tree object and select **Insert**>**Interface Delamination**.

     Or...

   - *in* the Geometry window and select **Insert**>**Interface Delamination**.

3. Select the desired **Method**: either **VCCT** (p. 478) (default) or **CZM** (p. 479). The properties vary based on your selection.

VCCT Method

1. Specify the **Failure Criteria Option** property: either **Energy-Release Rate** (default) or **Material Data Table**.

2. Based on the selected **Failure Criteria Option**:

   - If specified as **Energy-Release Rate**: enter a **Critical Rate** value. This value determines the energy release rate in one direction.

   - If specified as **Material Data Table**: specify a **Material**. This property defines the energy release rate in all three fracture modes. This property is defined in Engineering Data. See the Static Structural & Transient Structural section of the Engineering Data Help for additional information about the Cohesive Zone properties used by this feature.

3. Based on the **Generation Method** selected, either **Matched Meshing** (default) or **Node Matching**, perform one of the following:

   Matched Meshing

   If **Matched Meshing**, specify a **Match Control** by selecting a pre-defined **Match Control**. The **Match Control** that is referenced by the property requires that the delamination occurs between two independent parts that have the same element/node pattern.

   Node Matching

   If **Node Matching**, specify:

   a. **Scoping Method**

   b. **Source**

   c. **Target**

   Note:

   This option assumes that the existing mesh is already matched.

4. Define the **Initial Crack** by selecting a user-defined Pre-Meshed Crack.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. Specify the **Auto Time Stepping** property as either **Program Controlled** (default) or **Manual**. The following properties can be modified if **Manual** is selected, otherwise they are read-only.

   a. **Initial Time Step**

   b. **Minimum Time Step**

   c. **Maximum Time Step**

   Note:

   - The **Auto Time Stepping** property *must* be set to **On** in the **Step Controls** (p. 1418) category of the **Analysis Setting** object.
   - Time stepping values take effect when crack growth is detected.

6. If **Node Matching** is selected as the **Generation Type**, the **Node Matching Tolerance** category displays.

   Specify the **Tolerance Type** property as either **Program Controlled** (default) or **Manual**. The **Distance Tolerance** property can be modified if **Manual** is selected, otherwise it is read-only.

CZM Method

1. Specify a **Material**. This property is defined in Engineering Data. See the Static Structural & Transient Structural section of the Engineering Data Help for additional information about the Cohesive Zone properties used by this feature.
2. Define the **Generation Method** property as either **Matched Meshing** (default) or **Node Matching**.
3. Based on the **Generation Method** selected, either **Matched Meshing** or **Node Matching**, perform one of the following:

   Matched Meshing

   For the **Matched MeshingGeneration Method**, select a pre-defined **Match Control**. The **Match Control** that is referenced by the property requires that the delamination occurs between two independent parts that have the same element/node pattern.

   Node Matching

   If **Node Matching** is the **Generation Method**, then specify:

   a. **Scoping Method**

   b. **Source**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

c. **Target**

Note:

This option assumes that the existing mesh is already matched.

4. If **Node Matching** is selected as the **Generation Type**, the **Node Matching Tolerance** category displays.

   Specify the **Tolerance Type** property as either **Program Controlled** (default) or **Manual**. The **Distance Tolerance** property can be modified if **Manual** is selected, otherwise it is read-only.

Contact Debonding Application

Debonding simulations begin by defining contact regions along an interface that will separate. The properties for the contact elements require that the contact **Type** (p. 1234) be **Bonded** or **No Separation** contact and that the **Formulation** (p. 1241) is specified as the **Augmented Lagrange** method or the **Pure Penalty** method.

The **Contact Debonding** object specifies the pre-existing contact region (defined using the Connections (p. 1207) feature) that you intend to separate and it also references the material properties defined in Engineering Data. You must select the material properties from the **Cohesive Zone** category with type **Separation-Distance based Debonding** or **Fracture-Energies based Debonding**. See the Static Structural & Transient Structural section of the Engineering Data Help for additional information about the Cohesive Zone properties used by this feature.

Apply Contact Debonding

To specify **Contact Debonding**:

1. Insert a **Fracture** folder in the **Tree Outline**. The **Fracture** object becomes active by default.
2. On the **Fracture** Context Tab (p. 52): click **Contact Debonding**. Or, right-click:
   - the **Fracture** tree object and select **Insert**>**Contact Debonding**.

   Or...

   - *in* the Geometry window and select **Insert**>**Contact Debonding**.
3. Select a **Material**.
4. Select a **Contact Region**.

Tip:

To automatically generate a **Contact Debonding** object, select a **Contact Region** and drag and drop it onto the Fracture folder.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Also see the Contact Debonding Object Reference Help page for information about the properties of this feature.

Interface Delamination and Ansys Composite PrepPost (ACP)

Mechanical allows you to import interface layer(s) from the Ansys Composite PrepPost (ACP) application. You can define interface layers in ACP, import them into Mechanical, and use them to define **Interface Delamination** objects. You can automatically insert Interface Delamination objects into the **Fracture** folder when importing composite section data into Mechanical by setting the **Create Delamination Objects** property (see Specifying Options (p. 217)) to **Yes**. Alternatively, you can generate Interface Delamination objects automatically after you have imported Composite Section data by selecting the **Generate All Interface Delaminations** option in the context menu of the Fracture object.

Unexpected Penetration during Nonlinear Analysis

If you experience penetration at the interface layers during separation, you may wish to create a Contact condition (p. 1266) for the interface where the penetration is taking place. A Contact Region can be applied (p. 1227) to a **Pre-Generated Interface** provided by ACP. Although all contact Type (p. 1234) settings are supported for **Pre-Generated Interfaces**, the **Frictionless** setting is recommended for this case when specifying the contact condition. Other contact properties can be set to the default, **Program Controlled**, settings.

Apply Interface Delamination via ACP

To specify **Interface Delamination** using the ACP application:

Note:

The following steps assume that you have properly defined your interface layer in the ACP application.

VCCT Method (Default)

1. From the Workbench Project page, link your Static Structural or Transient Structural analysis to the **ACP (Pre)** system and then launch Mechanical.

   A **Fracture** folder is automatically created and includes an **Interface Delamination** object.

2. Select the new **Interface Delamination** object.

3. Specify the **Failure Criteria Option** property: either **Energy-Release Rate** (default) or **Material Data Table**.

4. Based on the selected **Failure Criteria Option**:

   - If specified as **Energy-Release Rate**: enter a **Critical Rate** value. This value determines the energy release rate in one direction.

   - If specified as **Material Data Table**: specify a **Material**. This property defines the energy release rate in all three fracture modes. This property is defined in ACP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

5. The automatic setting for the **Generation Method** property is **Pre-Generated Interface**. Accept this setting.
6. As necessary, select the appropriate Interface Layer from the **Interface** property drop-down menu.
7. Define the **Initial Crack** by selecting the Pre-Meshed Crack created by ACP.
8. Specify the **Auto Time Stepping** property as either **Program Controlled** (default) or **Manual**. The following properties can be modified if **Manual** is selected, otherwise they are read-only.
   a. **Initial Time Step**
   b. **Minimum Time Step**
   c. **Maximum Time Step**

Note:

- The **Auto Time Stepping** property *must* be set to **On** in the **Step Controls** (p. 1418) category of the **Analysis Setting** object.
- Time stepping values take effect when crack growth is detected.

CZM Method

1. From the Workbench Project page, link your Static Structural or Transient Structural analysis to the **ACP (Pre)** system and then launch Mechanical.

   A **Fracture** folder is automatically created and includes an **Interface Delamination** object.
2. Select the new **Interface Delamination** object.
3. Specify the **Material** property. This property provides a fly-out menu to make a material selection that was defined in the **ACP (Pre)** system.
4. The automatic setting for the **Generation Method** property is **Pre-Generated Interface**. Accept this setting.
5. As necessary, select the appropriate Interface Layer from the **Interface** property drop-down menu.

Multi-Point Constraint (MPC) Contact for Fracture

The internally generated crack mesh is created after an initial base mesh is generated. Since the crack mesh is defined based on the crack object, while the base mesh is created based on the geometry and mesh parameters, the two meshes may not perfectly match at the boundaries of the fracture

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

affected zone. For more information on the fracture affected zone, see the Fracture Analysis (p. 406) section in the Meshing User's Guide.

Note:

- The application does not create a contact pair when you set the **Mesh Method** to **Tetrahedrons** when meshing Semi-Elliptical Crack.
- The application does not create a contact pair for Arbitrary crack meshing.

When a solution is performed on an analysis which contains an internally generated crack mesh, a contact region using Multi-Point Constraint (MPC) formulation is automatically created between the crack mesh and the base mesh at the boundaries of the fracture-affected zone. This contact is applicable to static structural analysis, transient structural analysis, modal analysis, steady-state thermal analysis, and transient thermal analysis. For more information about the MPC contact formulation, see Contact Formulation Theory (p. 1222). This contact is only created for a **Semi-Elliptical Crack** object when the **Mesh Method** property is set to **Hex Dominant** and is not applicable to the **Pre-Meshed Crack** object.

The characteristics/settings of the MPC contact are shown below. For more information about the different contact settings, see Advanced Settings (p. 1240).

- Bonded surface-to-surface contact is defined between the crack mesh and the base mesh at the boundary of the fracture-affected zone. The contact element CONTA174 is created on the faces of the base mesh, and the target element TARGE170 is created on the faces of the crack mesh.
- The contact is asymmetric in nature. The contact can be made auto asymmetric by setting the `use auto symmetric` variable to 1 in the Variable Manager (p. 247).
- Nodal contact detection, normal from the contact surface, will be defined.
- The initial gap and penetration are ignored.
- For steady-state thermal and transient thermal analysis, the temperature degree of freedom is selected.

For more information about contact settings, refer to the CONTA174 documentation in the Element Reference. For more information about the MPC constraint, see Multipoint Constraints and Assemblies in the *Contact Technology Guide*.

Solving a Fracture Analysis

Once the crack mesh is generated, you can apply loads and constraints, then solve the analysis. Then, once the solution is done, you can analyze the stress and deformation pattern around the crack. For meshes defined by the **Semi-Elliptical Crack** object or the **Arbitrary Crack** object, you can apply the loads on the crack face top and bottom discontinuity plane using nodal named selections.

For the **Semi-Elliptical Crack** object, the application defines the internally generated crack mesh after the initial base mesh. The base mesh generation is based on a different set of requirements and constraints than the crack mesh. As a result, the crack mesh, generated using the Hex dominant mesh method, may not perfectly match the boundaries of the fracture affected zone. Because they may

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

not match perfectly, kinematic constraints are required to establish a connection between base mesh and crack mesh in the boundaries of the fracture affected zone, which is accomplished using the multi-point constraint (MPC) contact (p. 482). A contact pair is created at the interface of the crack and base meshes, with contact surface created at the interface on the buffer zone side of the base mesh and target surface created at the interface on the fracture affected zone side of the hex dominant mesh. When the solution is performed using internally-generated crack meshes, the MPC contact region is automatically created and sent to the solver.

Important:

For **Arbitrary Cracks** as well as analytical cracks (ring, corner, etc.), the application does not create a contact pair when the **Mesh Method** property is set to **Tetrahedrons**.

Note:

Static Structural and Transient Structural analyses are the only analyses supported for fracture mechanics calculations. However, the mesh with cracks is also supported with a static structural analysis linked to an upstream steady state thermal or transient thermal analysis.

Also, all loads and boundary conditions applicable to the static structural analysis are applicable with the existence of crack in the solution.

Although you can add **Fracture** and crack objects of any definition to a Modal analysis or a Mode Superposition (MSUP) Transient analysis, the application does not compute fracture parameters during the solution.

Computation of Fracture Parameters

The stress and deformation pattern around the crack is not sufficient to evaluate the catastrophic failure of the structure. The computation of fracture parameters and its comparison against fracture toughness is necessary for designing safe structures.

To compute fracture parameters for all cracks defined under the **Fracture** folder, the **Fracture** property in the **Fracture Controls** (p. 1445) of the Analysis Settings must be set to **On**. This entry is visible only if the **Fracture** folder exists in the model. By default, the application does not compute fracture parameters for Material Force and T-Stress. You need to set their properties under the **Fracture Controls** of the Analysis Settings to **Yes**.

The computations used for fracture analysis include Stress Intensity Factors (SIFS), J-Integral (JINT), Energy Release Rates, Material Force, T-Stress and C*-Integral. The Mode 1 Stress Intensity Factor (K1), Mode 2 Stress Intensity Factor (K2), Mode3 Stress Intensity Factor (K3), and T-Stress are computed along the crack front using the interaction integral method. The Mode 1 Energy Release Rate (G1)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

and Mode 2 Energy Release Rate (G2), Mode 3 Energy Release Rate (G3) and Total Energy Release Rate (GT) are computed using the Virtual Crack Closure Technique (VCCT) along the crack front.

Note:

The Energy Release Rate parameters, which are specific to the Pre-Meshed Crack object, are computed using the Virtual Crack Closure Technique (VCCT). When the VCCT technique is used, a specific mesh pattern composed of hexahedral shapes along the crack front is recommended for better accuracy. For more information, see Understanding Fracture Mechanics in the *Fracture Analysis Guide*.

The JINT result is a mixed mode result and is also computed along the crack front using the domain integral method. The fracture parameters, for all cracks defined under the fracture folder, are automatically computed and stored in the results file when the **Fracture** property in the **Fracture Controls** (p. 1445) category of Analysis Settings is set to **On**. The SIFS and JINT results are calculated for all cracks defined under the Fracture folder. The VCCT results are calculated only if the crack mesh generated is of lower order (dropped midside nodes). Material Force and T-Stress results are calculated only when their respective control is set to Yes in the **Fracture Controls** (p. 1445) category of Analysis Settings. You can direct the fracture parameter computation for all cracks to use symmetry by setting the `all cracks symmetric` variable to active with a value of 1 in the Variable Manager. For more information, see Setting Variables (p. 247). Fracture parameter calculation based on SIFS supports linear isotropic elastic material behavior. VCCT based fracture parameter calculation supports linear isotropic elastic, anisotropic elastic and orthotropic elastic material behavior. J-Integral based and T-stress based fracture parameter calculation supports isotropic elastic and isotropic plastic material behaviors. Material force based fracture parameter calculation supports linear isotropic elastic, isotropic hardening plasticity, kinematic hardening plasticity and isotropic hyperelastic material behaviors. C*-Integral based fracture parameter calculation supports secondary (steady-state) creep material behavior and it is computed along the crack front using the domain integral method. You can exclude computation of any fracture parameter (except VCCT) by setting its respective control in the **Fracture Controls** (p. 1445) category of **Analysis Settings** to **No**.

Note:

If you get the following message:

The fracture parameters computed during solution may be incorrect. Check the Solver Output on the Solution Information object (p. 1833) for possible causes.

Check for the following:

- A contact might have been created in the region of the crack contours.
- A load might have been applied in the region of the crack contours that is not supported in the fracture parameter computation. Try replacing it with a Direct FE (p. 1719) load. You can also replace the normal Pressure loads using the **Applied By** property option, **Direct**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Extracting Fracture Results from a Result File

The Fracture Tool (p. 2066) provides the **Scoping Method** option, **Result File Item**, that enables you to read a result file (p. 1861) into the application prior to solution processing. For the associated fracture results and probes, included under the **Fracture Tool**, you specify a desired **Solver ID** from the drop-down menu to evaluate the result from the result file. Furthermore, by reading a results file, you can evaluate the fracture results of a Pre-Meshed Crack without needing to define the crack. For all other crack types, your simulation must include a defined Fracture folder, a matching mesh, as well as defined boundary conditions.

See the Extract Fracture Results from a Result File (p. 2072) topic in the Fracture Results (p. 2066) section for the specific steps to extract fracture results from a result file (`file.rst`).

Requirements

Note the following requirements for this feature:

- You must ensure that the mesh in the result file matches the mesh in Workbench. This includes the Workbench generated mesh from the geometry as well as any nodes or elements defined in the input file (such as for contact or remote boundary conditions). Failure to do so could result in incorrect results and unexpected behavior.
- Fracture probes can be evaluated only if the fracture analysis had cracks associated with a **SMART Crack Growth** object from which the result file is saved and read.
- Once you read a result file, if you do not see an expected **Solver ID** in the property's drop-down menu, this indicates that the result was not available in the results file. You must delete or suppress the result in order to continue with your analysis.

Linear Dynamics Analysis Types

Applying external forces gradually to a structure does not cause it to experience any pulse or motion. You can solve structural responses with a simple static equilibrium analysis. That is, the structural elasticity forces and the external forces equilibrate one another.

In reality, however, structures are subject to rapidly applied forces (or so-called dynamic forces), for example, high-rise buildings, airplane wings, and drilling platforms are subject to wind gusts, turbulences, and ocean waves, respectively. These structures are in a state of motion as a result of the dynamic forces. To simulate and solve for the structural responses in a logical manner, a dynamic equilibrium analysis, or a dynamic analysis, is desirable. In a dynamic analysis, in addition to structural elasticity force, structural inertia and dissipative forces (or damping) are also considered in the equation of motion to equilibrate the dynamic forces. Inertia forces are a product of structural mass and acceleration while dissipative forces are a product of a structural damping coefficient and velocity.

When performing a linear dynamic analysis, the application calculates structural responses based the assumption that a structure is linear.

The following sections discuss the steps and requirements to perform different linear dynamic simulations.

Eigenvalue Buckling Analysis

Harmonic Response Analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Harmonic Response (Full) Analysis Using Pre-Stressed Structural System

Harmonic Response Analysis Using Linked Modal Analysis System

Modal Analysis

Random Vibration Analysis

Response Spectrum Analysis

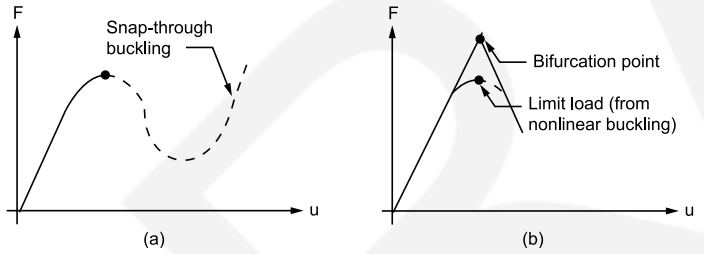
Substructure Generation Analysis

Eigenvalue Buckling Analysis

Background

An Eigenvalue Buckling analysis predicts the theoretical buckling strength of an ideal elastic structure. This method corresponds to the textbook approach to an elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column matches the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Therefore, an Eigenvalue Buckling analysis often yields quick but non-conservative results.

A more accurate approach to predicting instability is to perform a nonlinear buckling analysis. This involves a static structural analysis with large deflection effects turned on. A gradually increasing load is applied in this analysis to seek the load level at which your structure becomes unstable. Using the nonlinear technique, your model can include features such as initial imperfections, plastic behavior, gaps, and large-deflection response. In addition, using deflection-controlled loading, you can even track the post-buckled performance of your structure (which can be useful in cases where the structure buckles into a stable configuration, such as "snap-through" buckling of a shallow dome, as illustrated below).



(a) Nonlinear load-deflection curve, (b) Eigenvalue buckling curve.

Eigenvalue Buckling in Mechanical

In Mechanical, an Eigenvalue Buckling analysis is a linear analysis and therefore cannot account for nonlinearities. It employs the Linear Perturbation Analysis procedure of Mechanical APDL. This procedure requires a pre-loaded environment from which it draws solution data for use in the Eigenvalue Buckling analysis. Based on this requirement, an Eigenvalue Buckling analysis can consider nonlinearities that are present in the pre-stressed environment allowing you to attain a more accurate real-world solution as compared to a traditional linear preloaded state.

Note:

The application supports the use of the Samcef solver for this analysis type. However, the information presented below applies to the use of the Mechanical APDL Solver only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Points to Remember

- An Eigenvalue Buckling analysis must be linked to (proceeded by) a Static Structural Analysis (p. 652). This static analysis can be either linear or nonlinear and the linear perturbation procedure refers to it as the "base analysis" (as either linear or nonlinear).
- The nonlinearities present in the static analysis can be the result of nonlinear:
  - Geometry (the **Large Deformation** property is set to **Yes**)
  - Contact status (A contact condition with the **Type** property set to anything other than **Bonded** or **No Separation** is treated as a non-linearity for contact. In addition, when the **Small Sliding** property set to **Off**, the system is treated as non-linear contact.)
  - Material (such as the definition of nonlinear material properties in Engineering Data, hyperelasticity, plasticity, etc.)
  - Connection (such as nonlinear joints and nonlinear springs)
- A structure can have an infinite number of buckling load factors. Each load factor is associated with a different instability pattern. Typically the lowest load factor is of interest.
- Based upon how you apply loads to a structure, load factors can either be positive or negative. The application sorts load factors from the most negative values to the most positive values. The minimum buckling load factor may correspond to the smallest eigenvalue in absolute value.
- For **Pressure** boundary conditions in the Static Structural analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure to act as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a given pressure value in the upstream static system, the **Normal To** option and the **Component**/**Vector** options can produce significantly different buckling load factors in the follow-on Eigenvalue Buckling analysis.
- Buckling mode shapes do not represent actual displacements but help you to visualize how a part or an assembly deforms when buckling.
- The procedure that the Mechanical APDL solver uses to evaluate buckling load factors is dependent upon whether the pre-stressed Eigenvalue Buckling analysis is linear-based (linear prestress analysis) or nonlinear-based (nonlinear prestress analysis), as described below.

Linear-based Eigenvalue Buckling Analysis

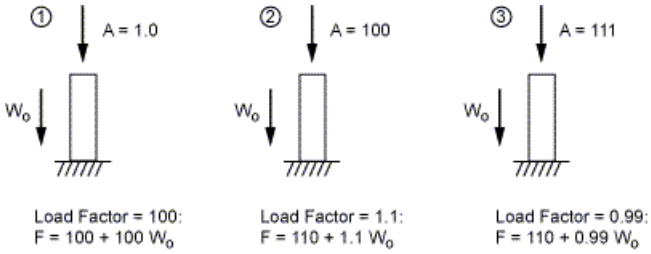
Note the following for an Eigenvalue Buckling analysis when the base analysis is linear:

- You can only define loading conditions in the upstream analysis.
- The results calculated by the Eigenvalue Buckling analysis are buckling load factors that scale **all** of the loads applied in the upstream Static Structural analysis. For example, if you applied a 10 N compressive load on a structure in the static analysis and if the Eigenvalue Buckling analysis calculates a load factor of 1500, then the predicted buckling load is 1500x10 = 15000 N. Because of this, it is typical to apply unit loads in the static analysis that precedes the buckling analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The solver applies the buckling load factor to all the loads specified in the upstream static analysis.
- Note that the load factors represent scaling factors for all loads. If certain loads are constant (self-weight gravity loads) while other loads are variable (externally applied loads), you need to take special steps to ensure accurate results. For example, you can iterate on the Eigenvalue buckling solution, adjusting the variable loads until the load factor becomes 1.0 (or nearly 1.0, within some convergence tolerance). Consider the example below: a pole has a self-weight W_0 that supports an externally-applied load, A. To determine the limiting value of A in an Eigenvalue Buckling analysis, you could solve repetitively, using different values for A, until you find a load factor acceptably close to 1.0.



- If you receive all negative buckling load factor values for your Eigenvalue Buckling analysis and you wish to see them in the positive values, or vice versa, reverse the direction of all of the loads you applied in Static Structural analysis.
- You can apply a nonzero constraint in the Static Structural analysis. The load factors calculated in the buckling analysis should also be applied to these nonzero constraint values. However, the buckling mode shape associated with this load will show the constraint to have zero value.

Nonlinear-based Eigenvalue Buckling Analysis

Note the following for an Eigenvalue Buckling analysis when the base analysis is nonlinear:

- At least one form of nonlinearity must be defined in the upstream static analysis.
- You must define at least one load in the buckling analysis to proceed with the solution. To enable this, set the **Keep Pre-Stress Load-Pattern** property to **Yes** (default). This retains the loading pattern from the Static Structural Analysis in the Eigenvalue Buckling analysis. Setting the property to **No** requires you to define a new loading pattern for the Eigenvalue Buckling analysis. This new loading pattern can be completely different from that of the prestress analysis.
- In a nonlinear-based Eigenvalue Buckling analysis, load multipliers scale the loads applied in buckling analysis *ONLY*. When estimating the ultimate buckling load for the structure, you must account for the loading applied in both analyses. The equation to calculate the ultimate buckling load for the nonlinear-based Eigenvalue Buckling analysis is:

  $F_{BUCKLING} = F_{RESTART} + \lambda_i \cdot F_{PERTRUB}$

  where:

  – $F_{BUCKLING}$ = The ultimate buckling load for the structure.

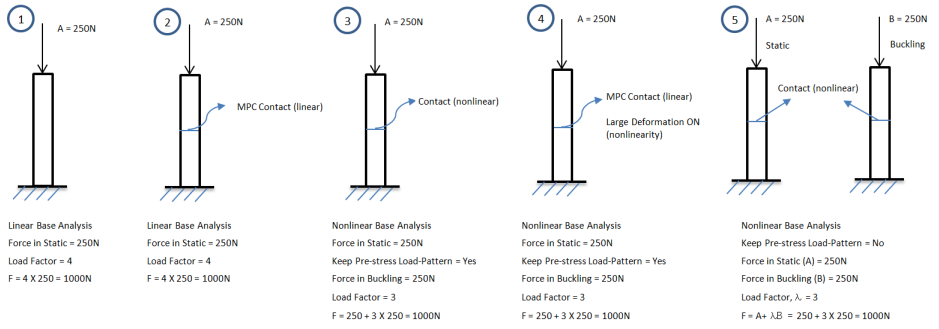
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- $F_{RESTART}$ = Total loads in Static Structural analysis at the specified restart load step.
- λ_i = Buckling load factor for the "i'th" mode.
- $F_{PERTRUB}$ = Perturbation loads applied in buckling analysis.

For example, if you applied a 100 N compressive force on a structure in the static analysis and a compressive force of 10 N in the Eigenvalue Buckling analysis and you get a load factor of 15, then the ultimate buckling load for the structure is 100 + (15 x 10) = 250 N.

You can verify the ultimate buckling load of the above equation using the buckling of a one dimensional column. However, calculating the ultimate buckling load for 2D and 3D problems with different combinations of loads applied in the Static Structural and Eigenvalue Buckling analyses may not be as straightforward as the 1D column example. This is because the $F_{RESTART}$ and $F_{PERTRUB}$ values are essentially the effective loading values in the static and buckling analyses, respectively.

For example, consider a cantilever beam that has a theoretical ultimate buckling strength of 1000N and that is subjected to a compressive force (A) of 250N. The procedure to calculate the ultimate buckling load (F), based on the load factors evaluated by Mechanical for linear-based and nonlinear-based Eigenvalue Buckling analyses is illustrated in the following schematic.



Note:

As illustrated, cases (3) and (5) are identical as the base analysis is nonlinear because of nonlinear contact definition. In Case (3), setting the **Keep Pre-Stress Load-Pattern** property to **Yes** automatically retains the loading from the pre-stress analysis. As a result, there is no need to define new loads for the buckling analysis in Case 3. For Case 5, the **Keep Pre-Stress Load-Pattern** property is set to **No**, enabling you to define a new load pattern in the buckling analysis that can be completely different from that of the Static Structural analysis.

- The buckling load factor evaluated in nonlinear-based Eigenvalue Buckling should be applied to all of the loads used in the buckling analysis.
- If you receive all negative buckling load factor values for your Eigenvalue Buckling analysis and you wish to see them in the positive values, or vice versa, reverse the direction of all of the loads you applied in the Static Structural analysis when the **Keep Pre-Stress Load-Pattern**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

property is set to **Yes**. If this property is set to **No**, reverse the direction of all of the loads that you applied in Eigenvalue Buckling analysis.

Preparing the Analysis

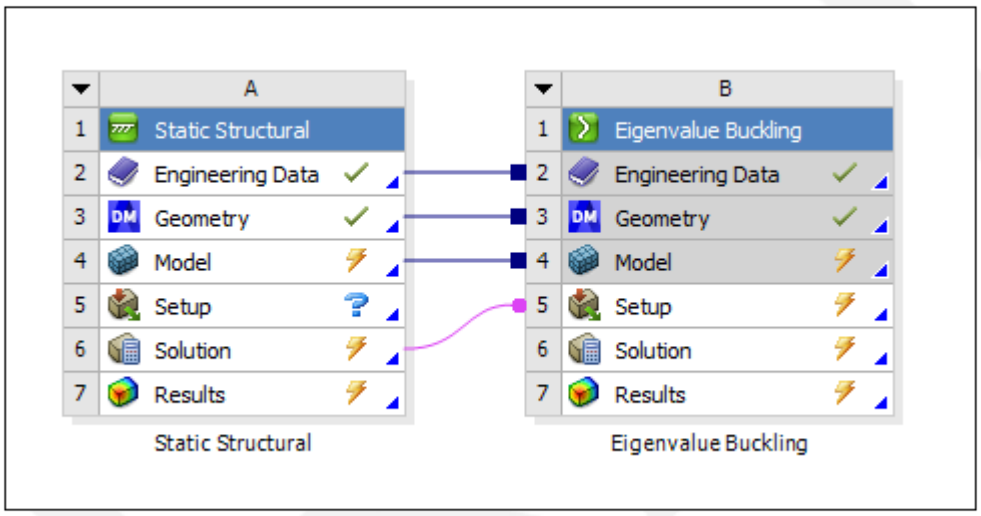
Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

Because this analysis is based on the Static Structural solution, a Static Structural analysis is a prerequisite. This linked setup allows the two analysis systems to share resources such as engineering data, geometry, and boundary condition type definitions.

From the **Toolbox**, drag a **Static Structural** template to the **Project Schematic**. Then, drag an **Eigenvalue Buckling** template directly onto the **Solution** cell of the **Static Structural** template. The proper linking is illustrated below.



Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

- Young's modulus (or stiffness in some form) must be defined.
- Material properties can be linear, nonlinear, isotropic or orthotropic, and constant or temperature-dependent.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

There are no specific considerations for an Eigenvalue Buckling analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

There are no specific considerations for an Eigenvalue Buckling analysis.

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Linear-based Eigenvalue Buckling Analysis

The following contact settings are considered linear contact behaviors for Eigenvalue Buckling analyses. If any other contact settings are used, the analysis will be considered a Nonlinear-based Eigenvalue Buckling analysis.

- The **Formulation** property is set to **MPC** or **Beam**.

  Or...

- The **Type** property is set to **Bonded** or **No Separation** and **Small Sliding** is active.

Springs with linear stiffness definition are taken into account if they are present in the static analysis.

Only **Bushing** and **General** joints enable you to solve an analysis with nonlinear Joint Stiffness (p. 1331). Mechanical considers all other joint types to be linear. The application accounts for linear joints if they are present in the static analysis.

Nonlinear-based Eigenvalue Buckling Analysis

All nonlinear connections (including nonlinear springs and joints) are allowed. Any contact options other than the ones mentioned above would trigger a nonlinear-based Eigenvalue Buckling analysis.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no considerations specifically for an Eigenvalue Buckling analysis.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifics ***... for this analysis type:***

For an Eigenvalue Buckling analysis, the basic Analysis Settings (p. 1417) include:

Options (p. 1459)

- Use the **Max Modes to Find** property to specify the number of buckling load factors and corresponding buckling mode shapes of interest. Typically the first (lowest) buckling load factor is of interest. The default value for this field is **2**. You can change this default setting under the **Buckling** category of the **Frequency** (p. 224) options in the **Options** (p. 217) preference dialog.
- The **Keep Pre-Stress Load-Pattern** property is available for nonlinear-based Eigenvalue Buckling analyses. Use this property to specify whether you want to retain the pre-stress loading pattern to generate the perturbation loads in the Eigenvalue Buckling analysis. The default setting for this property is **Yes**, which automatically retains the structural loading pattern for the buckling analysis (refer to the **ALLKEEP***LoadControl* key setting for **PERTURB** command). Setting the property to **No** requires you to define a new loading pattern for the Eigenvalue Buckling analysis (refer to **PARKEEP***LoadControl* key setting for **PERTURB** command).

Important:

Because the **PARKEEP***LoadControl* key retains all displacements applied in Static Structural analysis for reuse in Eigenvalue Buckling analysis, any non-zero displacements applied in static analysis act as loads in Eigenvalue Buckling analysis. If you specifying different load types in the buckling analysis that are scoped to the same geometric entities and in the same direction, may be ignored. Define your new loading pattern carefully.

Solver Controls (p. 1428)

- **Solver Type**: The default option, **Program Controlled**, enables the application to select the appropriate solver type. Options include **Program Controlled**, **Direct**, and **Subspace**. By default, the **Program Controlled** option uses the **Direct** solver for linear-based Eigenvalue Buckling analyses and **Subspace** solver for nonlinear-based Eigenvalue Buckling analyses.

Note:

Both the **Direct** and **Subspace** solvers evaluate the buckling solutions for most engineering problems. If you experience a solution failure using one of the solvers because it cannot find the requested modes, it may help to switch the solvers. If both of the solvers fail to find the solution, then review your model carefully for possible stringent input specifications or loading conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Include Negative Load Multiplier**: The default option **Program Controlled** and the **Yes** option extract both the negative and positive eigenvalues (load multipliers). The **No** option only extracts positive eigenvalues (load multipliers).

Output Controls (p. 1473)

By default, only buckling load factors and corresponding buckling mode shapes are calculated. You can request **Stress** and **Strain** results to be calculated but note that "stress" results only show the relative distribution of stress in the structure and are not real stress values.

Note:

The **Output Controls** category is only exposed for the Mechanical APDL solver.

Analysis Data Management (p. 1484)

The properties of this category enable you to define whether or not to automatically save the Mechanical APDL database as well as automatically delete unneeded files.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

You must specify a Static Structural analysis that is using the same model in the initial condition environment, and:

- Because an Eigenvalue Buckling analysis must be preceded by a Static Structural analysis, you need to specify the same solver type for each, either Mechanical APDL or Samcef.
- The **Pre-Stress Environment** property in the **Pre-Stress (Static Structural)** initial condition object displays whether the pre-stress environment is considered linear or nonlinear for the Eigenvalue Buckling analysis.
- If the Static Structural analysis has multiple result sets, the value from any restart point available in the Static Structural analysis can be used as the basis for the Eigenvalue Buckling analysis. See the Restarts from Multiple Result Sets (p. 353) topic in the Applying Pre-Stress Effects (p. 352) Help section for more information.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

Loads are supported by Eigenvalue Buckling analysis only when the pre-stressed environment has nonlinearities defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The following loads are supported for a nonlinear-based Eigenvalue Buckling analysis:

- **Loads**: **Thermal Condition** (p. 1594)
- **Direct FE** (p. 1719) (node-based Named Selection scoping and constant loading only):
  - **Nodal Force** (p. 1721)
  - **Nodal Pressure** (p. 1724)
  - **Nodal Displacement** (p. 1728): At least one non-zero **Component** is required for the boundary condition to be fully defined.

Note:

- Choosing to keep the default setting (**Yes**) for the **Keep Pre-Stress Load-Pattern** property retains the pre-stress loading pattern for the buckling analysis and no additional load definition is necessary.
- For **Nodal Pressure**, the only definition option is **Normal To**. This results in the "pressure load stiffness" effect. To avoid the pressure stiffness effect, apply an equivalent **Nodal Force** load to the same surface and set the **Divide Load by Nodes** property to **Yes**. The equivalent force is equal to the value of the pressure multiplied by the area of the scoped surface.
- The node-based Named Selections used with the above **Direct FE Loads** (p. 1719) cannot contain nodes scoped to a rigid body.

No loading conditions can be created in a linear-based Eigenvalue Buckling analysis. The supports as well as the stress state from the linked Static Structural analysis are used in the linear-based Eigenvalue Buckling analysis. See the Apply Pre-Stress Effects for Implicit Analysis (p. 352) section for more information about using a pre-stressed environment.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

Solution Information (p. 1833) continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You can view the buckling mode shape associated with a particular load factor by displaying a contour plot (p. 62) or by animating (p. 2260) the deformed mode shape. The contours represent relative displacement of the part.

Buckling mode shape displays are helpful in understanding how a part or an assembly deforms when buckling, but do not represent actual displacements.

"Stresses" from an Eigenvalue Buckling analysis do not represent actual stresses in the structure, but they give you an idea of the relative stress distributions for each mode. You can make **Stress** and **Strain** results available in the buckling analysis by setting the proper **Output Controls** (p. 1473) *before* the solution is processed.

Harmonic Response Analysis

Harmonic response analyses are used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time, therefore enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Introduction

In a structural system, any sustained cyclic load will produce a sustained cyclic or harmonic response. Harmonic analysis results are used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time, therefore enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

This analysis technique calculates only the steady-state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic analysis.

In this analysis all loads as well as the structure's response vary sinusoidally at the same frequency. A typical harmonic analysis will calculate the response of the structure to cyclic loads over a frequency range (a sine sweep) and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified from graphs of response vs. frequency and stresses are then reviewed at those peak frequencies.

Points to Remember

A harmonic response analysis is a linear analysis. Some nonlinearities, such as plasticity will be ignored, even if they are defined. All loads and displacements vary sinusoidally at the same known frequency (although not necessarily in phase). If the **Reference Temperature** is set as **By Body** and that temperature does not match the environment temperature, a thermally induced harmonic load will result (from the thermal strain assuming a nonzero thermal expansion coefficient). This thermal harmonic loading is ignored for all harmonic analysis.

Mechanical offers the following solution methods for harmonic analyses:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mode Superposition (Default)

For the **Mode Superposition** (also called "MSUP") option, the harmonic response to a given loading condition is obtained by performing the necessary linear combinations of the eigensolutions obtained from a Modal analysis.

For MSUP, it is advantageous for you to select an existing modal analysis directly (although Mechanical can automatically perform a modal analysis behind the scene) since calculating the eigenvectors is usually the most computationally expensive portion of the method. In this way, multiple harmonic analyses with different loading conditions could effectively reuse the eigenvectors. For more details, refer to Harmonic Response Analysis Using Linked Modal Analysis System (p. 509).

Acceleration (p. 1501) and/or Displacement (p. 1666) applied as a base excitation uses the Enforced Motion Method. See the Enforced Motion Method for Mode Superposition Transient and Harmonic Analyses section of the *Mechanical APDL Structural Analysis Guide* for additional information.

Full

Using the **Full** option, you obtain harmonic response through the direct solution of the simultaneous equations of motion. In addition, a Harmonic Response analysis can be linked to, and use the structural responses of, a Static-Structural analysis. See the Harmonic Analysis Using Pre-Stressed Structural System (p. 506) section of the Help for more information.

Include Residual Vector

This property is available when the **Solution Method** is set to **Mode Superposition**. You can turn the **Include Residual Vector** property **On** to execute the **RESVEC** command and calculate residual vectors.

Note:

The following boundary conditions do not support residual vector calculations:

- **Nodal Force**
- **Remote Force** scoped to a **Remote Point** (created via **Model** object)
- **Moment** scoped to a **Remote Point** (created via **Model** object)

Variational Technology

Using the **Variational Technology** option, the application evaluates the harmonic response for each excitation frequency based on one direct solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Program Controlled

Using the **Program Controlled** option, the application selects the best solution method based on the model. Internally, the application chooses either the **Full** or **Variational Technology** solution method.

Note:

- For more technical information about Variational Technology, see the Harmonic Analysis Variational Technology Method section of the *Mechanical APDL Theory Reference*.
- Also see the **HROPT** command in the Mechanical APDL Command Reference for more information about harmonic analysis options.
- If a Command object is used with the MSUP method, object content is sent twice; one for the modal solution and another for the harmonic solution. For that reason, harmonic responses are double if a load command is defined in the object, for example, **F** command.

Preparing the Analysis

As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Define Engineering Data

Both Young's modulus (or stiffness in some form) and density (or mass in some form) must be defined. Material properties must be linear but can be isotropic or orthotropic, and constant or temperature-dependent. Nonlinear properties, if any, are ignored.

Define Connections

Any nonlinear contact such as **Frictional** contact (p. 1234) retains the initial status throughout the harmonic analysis. The stiffness contribution from the contact is based on the initial status and never changes.

The application accounts for spring stiffness as well as damping when using the **Program Controlled**, **Full**, and **Variational Technology** solution methods. For the **Mode Superposition** option, the application ignores damping from springs.

Establish Analysis Settings

For a Harmonic Response analysis, the basic Analysis Settings (p. 1417) include:

Step Controls

The **Step Controls** (p. 1426) category enables you to define step controls for an analysis that includes multiple load steps. You use the properties of this category to define the load steps and their options. When you select the **Analysis Settings** object, the content of the **Step Controls** category automatically displays in the **Worksheet**. You can modify certain properties in either the **Worksheet**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

or in the **Details** pane. For each load step, you can specify solution settings (Frequency Spacing, minimum frequencies, maximum frequencies, etc.). See the Step Controls for Harmonic Analysis Types (p. 1426) section for a description of the properties.

Options

The **Options** category (p. 1447) enables you to specify the frequency range and the number of solution points at which the harmonic analysis will be carried out as well as the solution method to use and the relevant controls.

The **Solution Method** property options for a harmonic response analysis, **Program Controlled**, **Mode Superposition**, Direct Integration (**Full**), and **Variational Technology**, are described below.

- **Program Controlled**: The application selects the best solution method based on the model. Internally, the application chooses either the **Full** or **Variational Technology** solution method.
- **Mode Superposition**: This is the default method and generally provides results faster than the other methods. Using this method, a modal analysis is first performed to compute the natural frequencies and mode shapes. Then the mode superposition solution is carried out where these mode shapes are combined to arrive at a solution. The **Mode Superposition** method cannot be used if you need to apply imposed (nonzero) displacements. When using this method, you can set the **Cluster Results** property to **Yes** to group the structure's natural frequencies. This results in a smoother, more accurate tracing of the response curve. The default method of equally spaced frequency points can result in missing the peak values. See the Options (p. 1447) section for more information about the cluster property.

  The method also includes the following additional properties:

  #### On Demand Expansion Option

  Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**.

  When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. The application evaluates the results using the Modal solution data and calculates any other results "on demand." This improves solution performance and reduces file size. In addition, the application automatically removes mode shape data from the result file as determined by the **On Demand Mode Shape** preference.

  You can change the default setting for the **On Demand Expansion Option** property and the **On Demand Mode Shape** preference using the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) group in the **Options** dialog.

  #### Store Results At All Frequencies

  When you set the **Store Results At All Frequencies** (p. 1453) property to **No**, the application requests that only minimal data be retained. Only the harmonic results requested at the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

time of solution are calculated. The availability of the results is therefore not determined by the settings in the **Output Controls**.

Note:

With this option set to **No**, the addition of new frequency or phase responses to a solved environment requires a new solution. Adding a new contour result of any type (stress or strain) or a new probe result of any type (reaction force or reaction moment) for the first time on a solved environment requires you to solve, but adding additional contour results or probe results of the same type does *not* share this requirement; data from the closest available frequency is displayed (the reported frequency is noted on each result).

New and/or additional displacement contour results as well as bearing probe results do not share this requirement. These results types are basic data and are available by default.

The values of frequency, type of contour results (stress or strain) and type of probe results (reaction force, reaction moment, or bearing) at the moment of the solution determine the contents of the result file and the subsequent availability of data. Planning these choices can significantly reduce the need to re-solve an analysis.

Caution:

Use caution when adding result objects to a solved analysis. Adding a new result invalidates the solution and requires the system to be re-solved, even if you were to add and then delete a result object.

- **Full**: Calculates all displacements and stresses in a single pass. Its main disadvantages are:
  - It is more "expensive" in CPU time than the **Mode Superposition** method.
  - It does not allow clustered results, but rather requires the results to be evenly spaced within the specified frequency range.
- **Variational Technology**: Instead of using the full matrices to compute the results, the application computes the solution at the middle of the requested frequency range and then interpolates the system matrices and loading on the entire frequency range to approximate the results across the range.

Damping Controls (p. 1462)

These properties enable you to specify damping for the structure in the Harmonic Response analysis. Controls include: **Eqv. Damping Ratio From Modal** (MSUP method), **Damping Ratio** (MSUP method), **Constant Structural Damping Coefficient**, **Stiffness Coefficient** (beta damping), and a **Mass Coefficient** (alpha damping). They can also be applied as Material Damping (p. 1467) using the **Engineering Data** tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Damping: You can apply damping through spring-damper elements, however, this is not supported for the **Mode Superposition** solution method.

Note:

If multiple damping specifications are made the effect is cumulative.

Analysis Data Management (p. 1484)

These properties enable you to save solution files from the harmonic analysis. The default behavior is to only keep the files required for postprocessing. You can use these controls to keep all files created during solution or to create and save the Mechanical APDL application database (db file).

Define Initial Conditions

Currently, the initial conditions **Initial Displacement** and **Initial Velocity** are not supported for Harmonic analyses.

For a Pre-Stressed Full Harmonic analysis, the preloaded status of a structure is used as a starting point for the Harmonic analysis. That is, the static structural analysis serves as an Initial Condition for the Full Harmonic analysis. See the Applying Pre-Stress Effects (p. 352) section of the Help for more information.

Note:

- In the Pre-Stressed MSUP Harmonic Analysis, the pre-stress effects are applied using a Modal analysis.
- When you link your Harmonic (Full) analysis to a Structural analysis, all structural loading conditions, including Inertial (p. 1500) loads, such as Acceleration and Rotational Velocity, are deleted from the Full Harmonic Analysis portion of the simulation once the loads are applied as initial conditions (p. 349) (via the **Pre-Stress** object). Refer to the Mechanical APDL command **PERTURB**,**HARM**,,,**DZEROKEEP** for more details.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement**, or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Harmonic solution. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution.

Apply Loads and Supports

The Harmonic Response analysis supports the following boundary conditions:

Inertial

Acceleration (p. 1501) (**Phase Angle** is *not* supported.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Loads

- Pressure (p. 1521)
- Pipe Pressure (p. 1532) (line bodies only) - *Not* supported for MSUP Solution Method.
- Force (p. 1543) (applied to a face, edge, or vertex)
- Moment (p. 1574)
- Remote Force (p. 1552)
- Bearing Load (p. 1561) (**Phase Angle** is *not* supported.)
- Line Pressure (p. 1585)
- Given a specified Displacement (p. 1666)

Supports

Any type of *Support* (p. 1663) can be used in harmonic analyses.

Note:

The **Compression Only** (p. 1685) support is nonlinear but should not be utilized even though it behaves linearly in harmonic analyses.

Conditions

Constraint Equation (p. 1702)

Direct FE (node-based Named Selection scoping and constant loading only)

- Nodal Orientation (p. 1719) (**Phase Angle** is *not* supported.)
- Nodal Force (p. 1721)
- Nodal Displacement (p. 1728)

Base Excitation (MSUP Only)

- Acceleration (p. 1501) as a base excitation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Displacement (p. 1666) as a base excitation.

Important:

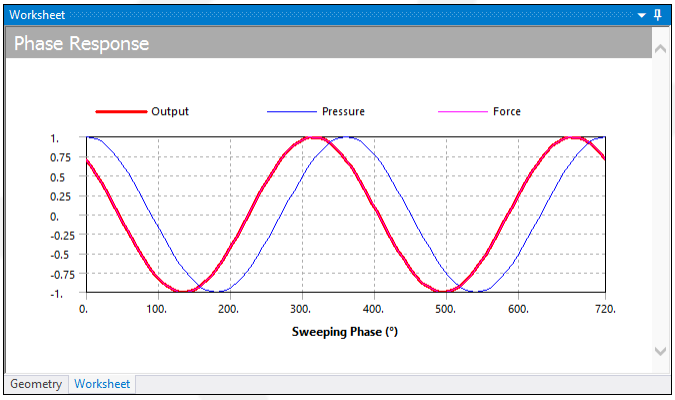
When duplicating an analysis within Mechanical that includes loads with the **Base Excitation** property set to **Yes** (Acceleration and/or Displacement), these loads will lose their scoping during the duplication process.

Note:

Support for boundary conditions varies for a Harmonic Response analysis that is linked to either a Static-Structural or Modal analysis. See the Harmonic Response Analysis Using Linked Modal Analysis System (p. 509) or the Harmonic Analysis Using Pre-Stressed Structural System (p. 506) sections of the Help for specific boundary condition support information.

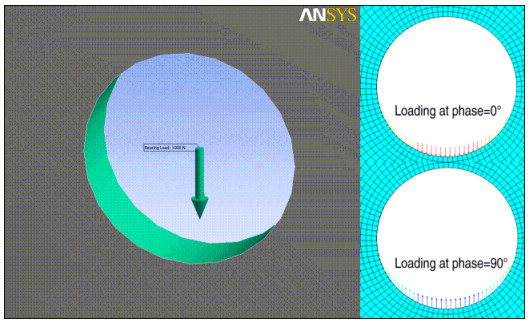
In a Harmonic Response Analysis, boundary condition application has the following requirements:

- You can apply multiple boundary conditions to the same face.
- All boundary conditions must be sinusoidally time-varying.
- Transient effects are not calculated.
- All boundary conditions must have the same frequency.
- Boundary conditions supported with the **Phase Angle** property allow you to specify a phase shift that defines how the loads can be out of phase with one another. As illustrated in the example Phase Response below, the pressure and force are 45^{o} out of phase. You can specify the preferred unit for phase angle (in fact all angular inputs) to be degrees or radians using the **Units** option in the **Tools** (p. 45) group of the **Home** tab.



- An example of a **Bearing Load** (p. 1561) acting on a cylinder is illustrated below. The Bearing Load, acts on one side of the cylinder. In a harmonic analysis, the expected behavior is that the other side of the cylinder is loaded in reverse; however, that is not the case. The applied load simply reverses sign (becomes tension). As a result, Ansys recommends avoid the use of Bearing Loads in this analysis type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Solve

Solution Information (p. 1833) continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis.

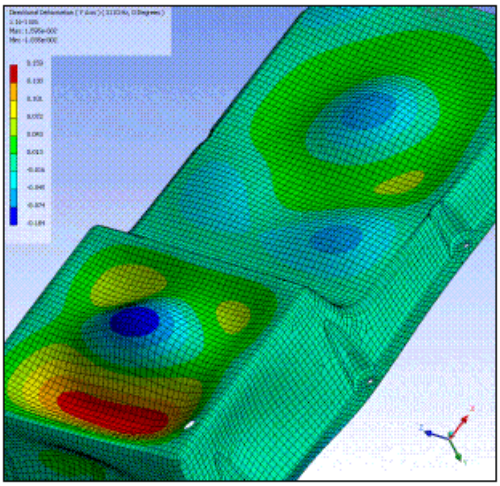
Review Results

Result specification for Harmonic Response analyses includes:

Contour Plots

Contour plots include stress (p. 2026), elastic strain (p. 2026), and deformation (p. 2018), and are basically the same as those for other analyses. If you wish to see the variation of contours over time for these results, you must specify an excitation frequency and a phase. The **Sweeping Phase** property in the details view for the result is the specified phase, in time domain, and it is equivalent to the product of the excitation frequency and time. Because **Frequency** is already specified in the Details view, the **Sweeping Phase** variation produces the contour results variation over time. The **Sweeping Phase** property defines the parameter used for animating the results over time. You can then see the total response of the structure at a given point in time, as shown below.

By setting the **Amplitude** property to **Yes**, you can see the amplitude contour plots at a specified frequency. For additional information about **Amplitude** calculation for derived results, see the Amplitude Calculation in Harmonic Analysis (p. 505) section of the Help.



Since each node may have different phase angles from one another, the complex response can also be animated to see the time-dependent motion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Frequency Response and Phase Response

Frequency Response (p. 2046) and **Phase Response** (p. 2056) charts which give data at a particular location over an excitation frequency range and a phase period (the duration of the Phase Response results, respectively). Graphs can be either **Frequency Response** graphs that display how the response varies with frequency or **Phase Response** plots that show how much a response lags behind the applied loads over a phase period.

Note:

You can create a contour result from a **Frequency Response** result type in a Harmonic Analysis using the Create Contour Result (p. 2053) feature. This feature creates a new result object in the tree with the same **Type**, **Orientation**, and **Frequency** as the **Frequency Response** result type. However, the **Phase Angle** of the contour result has the same magnitude as the frequency result type but an opposite sign (negative or positive). The sign of the phase angle in the contour result is reversed so that the response amplitude of the frequency response plot for that frequency and phase angle matches with the contour results.

Fatigue Tool

You can use the Fatigue Tool to view fatigue results for the repeated loading of a particular Frequency and Phase Angle.

Waterfall Diagrams

If your analysis contains multiple RPM steps, you can use Waterfall Diagram (p. 2135) results. These result types are useful when analyzing the Noise Vibration Harshness (NVH) footprint of a device for the frequencies of all RPMs.

Amplitude Calculation in Harmonic Analysis

A Harmonic analysis result can be expressed using the following complex notation:

$$x = x_{real} + x_{imag} * i \tag{1}$$

The amplitude is calculated as:

$$|x| = \sqrt{(x_{real}2) + (x_{imag}2)} \tag{2}$$

You can verify Equation (2) for component results, such as a **Directional Deformation**, by solving the equation using the real and imaginary components of the given result.

Amplitude of a Derived Result

A derived result is computed from the component results. For example, **Total Deformation**, u_{total}, is a derived result because it is evaluated from the displacement components u_x, u_y, and u_z in X, Y, and Z directions, respectively, as shown in the following equation:

$$u_{total} = \sqrt{(u_x)^2 + (u_y)^2 + (u_z)^2} \tag{3}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For derived results, the following procedure is employed to calculate **Amplitude**. Using the formula for a particular derived result, the real and the imaginary parts of the derived quantity are evaluated from the real and imaginary component results respectively. The **Amplitude** for the derived result is then calculated using Equation (2).

For example, the **Amplitude** of **Total Deformation** is calculated using the formula for **Total Deformation**, shown here:

$$(u_{total})_{real} = \sqrt{(u_x)^2_{real} + (u_y)^2_{real} + (u_z)^2_{real}} \quad (4)$$

$$(u_{total})_{imag} = \sqrt{(u_x)^2_{imag} + (u_y)^2_{imag} + (u_z)^2_{imag}} \quad (5)$$

The **Amplitude** of **Total Deformation**:

$$|u_{total}| = \sqrt{(u_{total})^2_{real} + (u_{total})^2_{imag}} \quad (6)$$

Caution:

Note that for the **Amplitude** results for Minimum, Middle, and Maximum Principal Stresses, the application sorts the three values from highest to lowest before it reports the results. To illustrate this, consider real and imaginary values for Minimum, Middle, and Maximum Principal Stresses, as S1, S2, and S3, at a certain node and frequency. You obtain the result values by setting the Sweeping Phase property to 0 and 90 degrees respectively. The table below shows application generated result values for this example. The amplitude values do not correspond, as applicable to Equation (2), for the real and imaginary components. This is because the application sorts the three amplitude values from highest to lowest, before reporting the result values.

Result	REAL (Phase = 0°)	COMPLEX (Phase = 90°)	AMPLITUDE
S1	3142.8	1.92E-13	3142.8
S2	-124.39	-7.62E-15	145.8
S3	-145.8	-8.93E-15	124.39

Harmonic Response (Full) Analysis Using Pre-Stressed Structural System

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

Because this analysis is linked to (and based on) structural responses, a **Static-Structural** analysis is a prerequisite. This setup allows the two analysis systems to share

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

resources, such as engineering data, geometry, and the boundary condition type definitions that are defined the in the structural analysis.

From the **Toolbox**, drag a **Static-Structural** template to the Project Schematic. Then, drag a **Harmonic Response** template directly onto the **Solution** cell of the Structural template.

Note:

You can create a pre-stress environment in a Harmonic Response system that is already open in Mechanical by:

1. Selecting the **Static Structural** option from the **Analysis** drop-down menu on the **Home** (p. 40) (or displayed) tab.
2. Setting the **Pre-Stress Environment** property (of the **Pre-Stress** object) to the Static Structural system.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For this analysis configuration, the basic Analysis Settings (p. 1417) include:

Step Controls

The **Step Controls** (p. 1426) category enables you to define step controls for an analysis that includes multiple load steps. You use the properties of this category to define the load steps and their options. When you select the **Analysis Settings** object, the content of the **Step Controls** category automatically displays in the **Worksheet**. You can modify certain properties in either the **Worksheet** or in the **Details** pane. For each load step, you can specify solution settings (Frequency Spacing, minimum frequencies, maximum frequencies, etc.). See the Step Controls for Harmonic Analysis Types (p. 1426) section for a description of the properties.

Options

See the Options for harmonic analyses (p. 1447) section for a complete listing of the properties. For a harmonic response analysis linked to an upstream static structural analysis, **Mode Superposition** is **not** an available selection.

Output Controls (p. 1473)

You can specify This category enables you to request **Stress**, **Strain**, **Nodal Force**, and **Reaction** results to be calculated.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifics ***... for this analysis type:***

The **Initial Conditions (Pre-Stress)** object of the Harmonic Response analysis must point to the linked Static Structural analysis.

Note:

- All structural loads, including Inertial (p. 1500) loads, such as Acceleration and Rotational Velocity, are deleted from the Harmonic Analysis portion of the simulation once the loads are applied as initial conditions (p. 349) (via the **Pre-Stress** object). Refer to the Mechanical APDL command **PERTURB**,**HARM**,,,**DZEROKEEP** for more details.
- For **Pressure** boundary conditions in the Static Structural analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different results in the follow-on Full-Harmonic Analysis. See the Pressure Load Stiffness (p. 353) topic in the Applying Pre-Stress Effects for Implicit Analysis (p. 352) Help Section for more information about using a pre-stressed environment.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement**, or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Harmonic solution. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution. If you define a **Nodal Displacement** in the Harmonic analysis at the same location and in the same direction as in the Structural Static analysis, it overwrites the previous loading condition and/or boundary condition in the Harmonic solution.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are allowed for linked Harmonic Response (Full) analysis:

- Inertial: Acceleration (p. 1501) (**Phase Angle** is *not* supported.)
- **Direct FE** (node-based Named Selection scoping and constant loading only):
  - Nodal Force (p. 1721)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– Nodal Pressure (p. 1724) (**Phase Angle** is *not* supported.)

– Nodal Displacement (p. 1728) - At least one non-zero Component is required for the boundary condition to be fully defined.

Note:

Any other boundary conditions must be defined in the prerequisite (parent) Structural Analysis, such as Support Type (p. 1663) boundary conditions.

Harmonic Response Analysis Using Linked Modal Analysis System

This section examines the workflow for a Harmonic Response analysis linked to an upstream Modal analysis. Topics include:

- Create Analysis System (p. 509)
- Establish Analysis Settings (p. 510)
- Define Initial Conditions (p. 512)
- Apply Loads and Supports (p. 512)
- Review Results (p. 514)
- File Management (p. 514)

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

Because this analysis is linked to (or based on) modal responses, a Modal analysis is a prerequisite. This setup allows the two analysis systems to share resources such as engineering data, geometry and boundary condition type definitions made in modal analysis.

Note:

- When you set the **Solution Method** property to **Mode Superposition** (MSUP), the harmonic analysis can be linked to a pre-stressed modal analysis.
- When solving a linked MSUP harmonic system database from a version prior to the most current version of Mechanical, it is possible to encounter incompatibility of the file *file.full* created by the modal system. This incompatibility can cause the harmonic system's solution to fail. In the event you experience this issue, use the **Clear Generated Data** feature and resolve the modal system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Refer to the Obtain the Mode Superposition Harmonic Solution section of the *MAPDL Structural Analysis Guide* for more information.

From the **Toolbox**, drag a **Modal** template to the **Project Schematic**. Then, drag a **Harmonic Response** template directly onto the **Solution** cell of the **Modal** template.

Note:

You can create a modal environment in a Harmonic Response system that is already open in Mechanical by:

1. Selecting the **Modal** option from the **Analysis** drop-down menu on the **Home** (p. 40) (or displayed) tab.
2. Setting the **Modal Environment** property (of the **Pre-Stress/Modal** object) to the Modal system.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For this analysis configuration, the basic Analysis Settings (p. 1417) include:

Step Controls (p. 1426)

This category enables you to define step controls for an analysis that includes rotational velocities in the form of revolutions per minute (RPMs). You use the properties of this category to define RPM steps and their options. Each RPM load is considered as a load step, such as frequency spacing, minimum frequencies, maximum frequencies, etc.

The **Step Controls** category (p. 1426) enables you to define step controls for an analysis that includes multiple load steps. You use the properties of this category to define the load steps and their options. When you select the **Analysis Settings** object, the content of the **Step Controls** category automatically displays in the **Worksheet**. You can modify certain properties in either the **Worksheet** or in the **Details** pane. For each load step, you can specify solution settings (Frequency Spacing, minimum frequencies, maximum frequencies, etc.). See the Step Controls for Harmonic Analysis Types (p. 1426) section for a description of the properties.

Options (p. 1459)

See the Harmonic Analysis Options Group (p. 1447) section for a complete listing of the Details properties for a Harmonic Response analysis. Note that for a Harmonic Analysis Using Linked Modal Analysis System, only the **Mode Superposition** option is applicable for the **Solution Method** property and it is therefore read-only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

In addition, you can turn on the following properties:

- **Include Residual Vectors**. Set this property to **Yes** to execute the **RESVEC** command and calculate residual vectors.

- **Cluster Results**: Set this property to **Yes** to automatically cluster solution points near the structure's natural frequencies ensuring capture of behavior near the peak responses. This results in a smoother, more accurate, response curve.

- **On Demand Expansion Option**: Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**.

  When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. The application evaluates the results using the Modal solution data and calculates any other results "on demand." This improves solution performance and reduces file size. In addition, the application automatically removes mode shape data from the result file as determined by the **On Demand Mode Shape** preference.

  You can change the default setting for the **On Demand Expansion Option** property and the **On Demand Mode Shape** preference using the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) group in the **Options** dialog.

- **Mode Selection Method**: This property enables you to select the modes for the mode expansion. Property options include:

  – **None** (default): This option expands all the extracted modes from the modal analysis. All the modes participate in the Mode Superposition harmonic analysis.

  – **Modal Effective Mass**: When you select this option, the **Significance Threshold** property also displays. For the **Significance Threshold** property value, the application selects the modes that have a ratio for the Modal Effective Mass to the Total Mass that is greater than this value, in all directions, for expansion. That is, only the selected modes participate in the Mode Superposition harmonic response analysis. The application default is **`0.001`**. This feature improves the performance of postprocessing the modal results and reduces the file size.

Also, **Mode Frequency Range** is not applicable because available modes are defined in the linked Modal system.

Note:

The following boundary conditions do not support residual vector calculations:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Nodal Force**
- **Remote Force** scoped to global a **Remote Point** (created via **Model** object)
- **Moment** scoped to global a **Remote Point** (created via **Model** object)

Output Controls (p. 1473)

This category enables you to request **Stress**, **Strain**, **Nodal Force**, and **Reaction** results to be calculated. For better performance, you can also choose to have these results expanded from Harmonic or Modal solutions. To expand reaction forces in the modal solution, set the **Nodal Force** property to **Yes** or **Constrained Nodes**.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

You must specify the desired upstream **Modal** analysis in the **Modal Environment** property of the **Initial Conditions** object for the **Harmonic Response** analysis. This object also indicates whether the upstream Modal analysis is pre-stressed. If it is a pre-stress analysis, the name of the pre-stress analysis system is displayed in the **Pre-Stress Environment** property, otherwise the field indicates **None**. The Modal analysis (p. 515) must extract enough modes to cover the frequency range. A conservative estimate is to extract enough modes to cover 1.5 times the maximum frequency in the excitation.

Note:

- Command objects can be inserted into **Initial Conditions** object to execute a restart of the solution process for the Modal Analysis.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Harmonic solution. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are allowed for the linked analysis:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Inertial

Acceleration (p. 1501) (**Phase Angle** is *not* supported.)

Loads

- Acceleration (p. 1501) as a base excitation
- Bearing Load (p. 1561) (**Phase Angle** is *not* supported.)
- Given a specified Displacement (p. 1666)
- Force (p. 1543) (applied to a face, edge, or vertex)
- Line Pressure (p. 1585)
- Moment (p. 1574)
- Pressure (p. 1521)
- Pipe Pressure (p. 1532) (line bodies only)
- Remote Force (p. 1552)

Direct FE

The **Direct FE** option **Nodal Force** (p. 1721) is supported for node-based Named Selection scoping and constant loading only.

Note:

- For the **Force** and **Pressure** loading conditions, the default setting for the **Applied By** property is **Direct**.
- When you have a supported topology (geometric or mesh entity) selected on your model, and you insert a **Remote Force** or a **Moment**, the application automatically creates (promotes) a corresponding **Remote Point** for the specified load and scopes the load to that **Remote Point**. This promotion feature is only available when none of the analyses in your project have been solved.

Loading and Support Limitations

Note the following load and support requirements and limitations:

- Remote Force is not supported for vertex scoping.
- Moment is not supported for vertex scoping on 3D solid bodies because a beam entity is created for the load application.
- During a linked MSUP Harmonic analysis, if a **Remote Force** or **Moment** scoped to an internal remote point is specified with the **Behavior** property set to **De-**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

formable, the boundary conditions cannot be scoped to the edges of line bodies such that all of their nodes in combination are collinear.

- The **Force Reaction** and **Moment Reaction** probes are not supported for a either a Harmonic Response analysis or a Transient structural analysis using the **Mode Superposition** method Solution Method when 1) the **Damping Ratio** property is defined with a non-zero value and 2) if the Analysis Settings property **Expand Result From** is set to **Harmonic Solution** or **Transient Solution**.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Refer to the **Review Results** (p. 504) topic in the Harmonic Response Analysis (p. 496) section for more information regarding how to set up the harmonic results.

File Management Options

When solving a **Harmonic Response** analysis that includes an upstream linked **Modal** analysis, the application can reference the prerequisite files by specifying the full path of their location (refer to **RESUME** and **MODDIR** commands) instead of making copies. Referencing improves solution time and disk usage. The application will perform referencing once the following conditions and settings are performed:

- You must perform the solution locally (called "In Process (p. 1803)").
- The **On Demand Expansion Option** property of the Options (p. 1447) category of the **Analysis Settings** of the **Modal** analysis must be set to **Yes** manually or through the use of the **Program Controlled** option.

Important:

Make sure that **Modal** and the downstream **Harmonic Response** analysis request the same result data in the Output Controls (p. 1473). Otherwise, you may receive unexpected results.

Note:

If you specify material damping, the **Eqv. Damping Ratio From Modal** property (for the **Harmonic Response** analysis) must also be set to **Yes**.

Limitations

Also note the following limitations for referencing. The **Harmonic Response** analysis cannot include:

- Cyclic Symmetry
- Line Pressure

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Pressure or Force applied using the surface effect elements created on the top of the scoped geometry (**Applied By** property set to **Surface Effect**)
- Remote Force or Moment not scoped to global remote points
- Commands (APDL) object

Modal Analysis

Introduction

A modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. You can also perform a modal analysis on a pre-stressed structure, such as a spinning turbine blade.

If there is damping in the structure or machine component, the system becomes a damped modal analysis. For a damped modal system, the natural frequencies and mode shapes become complex.

For a rotating structure or machine component, the gyroscopic effects resulting from rotational velocities are introduced into the modal system. These effects change the system's damping. The damping can also be changed when a Bearing (p. 1413) is present, which is a common support used for rotating structure or machine component. The evolution of the natural frequencies with the rotational velocity can be studied with the aid of Campbell Diagram Chart Results.

A Modal analysis can be performed using the Ansys, Samcef, or ABAQUS solver. Any differences are noted in the sections below. Rotordynamic analysis is not available with the Samcef or ABAQUS solver.

Points to Remember

- The Rotational Velocity load is not available in Modal analysis when the analysis is linked to a Static Structural analysis.
- Pre-stressed Modal analysis requires performing a Static Structural analysis (p. 652) first. In the modal analysis you can use the **Initial Condition** object (p. 349) to point to the **Static Structural** analysis to include pre-stress effects.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag a **Modal**, **Modal (Samcef)**, or **Modal (ABAQUS)** template to the **Project Schematic**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Due to the nature of modal analyses any nonlinearities in material behavior are ignored. Optionally, orthotropic and temperature-dependent material properties may be used. The critical requirement is to define stiffness as well as mass in some form. Stiffness may be specified using isotropic and orthotropic elastic material models (for example, Young's modulus and Poisson's ratio), using hyperelastic material models (they are linearized to an equivalent combination of initial bulk and shear moduli), or using spring constants, for example. Mass may be derived from material density or from remote masses.

Note:

Hyperelastic materials are supported for pre-stress modal analyses. They are not supported for standalone modal analyses.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

When 2D geometry is used, Generalized Plane Strain is not supported for the Samcef or ABAQUS solver.

When performing a Rotordynamic Analysis, the rotors can be easily generated using the Import Shaft Geometry feature of Ansys DesignModeler. The feature uses a text file to generate a collection of line bodies with circular or circular tube cross sections.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

You can define a **Point Mass** (p. 901) for this analysis type.

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

- Joints are allowed in a modal analysis. They restrain degrees of freedom as defined by the joint definition.
- The stiffness of any spring is taken into account and if specified, damping is also considered.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For the Samcef and ABAQUS solvers, only contacts, springs, and beams are supported. Joints are not supported.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no special considerations for this analysis type.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a Modal analysis, the basic Analysis Settings (p. 1417) include:

Options (p. 1455)

Max Modes to Find

Using the **Max Modes to Find** property, specify the number of frequencies of interest. The default is to extract the first **6** natural frequencies. The number of frequencies can be specified in two ways:

1. The first N frequencies (N > 0), or...
2. The first N frequencies in a selected range of frequencies.

On Demand Expansion Option

You can change the setting of this property as needed to improve performance. Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**.

When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. That is, the application evaluates the results using the Modal solution data and calculates any other results "on demand." This improves solution performance and reduces file size. And as applicable, the application changes parameters of the any linked downstream Harmonic Response and Transient analyses to also improve their performance. In addition, the application automatically removes mode shape data from the result file as determined by the **On Demand Mode Shape** preference (see note below).

If the application detects any of the following conditions, it automatically sets the **On Demand Expansion** property to **No**:

- The system is a standalone modal analysis (no downstream systems).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- You specify any Output Controls (p. 1473).
- A downstream system does not support an **On Demand Expansion Option** property setting of **Yes**.
- A downstream system has the On Demand Expansion Option set to either **No** or **Program Controlled** (**No**).
- Damping is present in the analysis.
- Symmetry present in the analysis.

You can change the default setting for the **On Demand Expansion Option** property and the **On Demand Mode Shape** preference using the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) group in the **Options** dialog.

Solver Controls (p. 1428)

The following two properties are available for this category:

- **Damped**: use this property to specify if the modal system is undamped or damped. Depending upon your selection, different solver options are provided. **Damped** by default, it is set **No** and assumes the modal system is an undamped system.
- **Solver Type** (p. 1429): it is generally recommended that you allow the program to select the type of solver appropriate for your model in both undamped and damped modal systems. When the Solver Type is set to **Reduced Damped**, the following additional properties become available:
  - **Store Complex Solution**: This property is only available when the **Solver Type** property is set to **Reduced Damped**. This property enables you to solve and store a damped modal system as an undamped modal system. By default, it is set to **Yes**.
  - **Mode Reuse**: This property allows the solver to compute complex eigensolutions efficiently during subsequent solve points by reusing the undamped eigensolution that is calculated at the first solve point. The default setting is **Program Controlled**. Set this property to **Yes** to enable it or **No** to disable it.

Note:

If you specify the **Solver Type** property as either **Unsymmetric**, **Full Damped**, or **Reduced Damped** , the modal system does not support a downstream Transient Structural, Harmonic Response, Random Vibration, or Response Spectrum system. However, for a MSUP Harmonic analysis or a MSUP Transient analysis, you can use the **Reduced Damped** setting when the **Store Complex Solution** property is set to **No**. Given these settings, the application also supports the use of the **Include Residual Vectors**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

property (set to **Yes**) as well as the use of **Base Excitation** loading in the downstream systems.

In this case, the mode shapes associated with undamped frequencies are calculated and used for Mode Superposition. However, both damped and undamped frequencies are reported in the **Tabular Data** pane of the Modal analysis. Even if you use the **Reduced Damped** solver with the **Store Complex Solution** property set to **No** in a damped analysis, it is not the equivalent to setting the **Solver Type** property **Undamped**.

If an undamped Modal analysis has a pre-stressed environment from a Static Structural Analysis with the **Newton-Raphson Option** set to **Unsymmetric**, the **Program Controlled** option selects **Unsymmetric** as the **Solver Type** setting (the Mechanical APDL command **MODOPT**,UNSYM is issued).

Cyclic Controls (p. 1446)

When running a cyclic symmetry (p. 1147) analysis, set the **Harmonic Index Range** to **Program Controlled** to solve for all harmonic indices, or to **Manual** to solve for a specific range of harmonic indices.

Output Controls (p. 1473)

By default, only mode shapes are calculated. You can request **Stress** and **Strain** results to be calculated but note that "stress" results only show the relative distribution of stress in the structure and are not real stress values. You can also choose whether or not to have these results stored for faster result calculations in linked systems.

Damping Controls (p. 1462)

The options of the **Stiffness Coefficient Defined By** property, **Direct Input** or **Damping vs. Frequency**, enable you to define the method used to define the **Stiffness Coefficient**. If you select **Damping vs. Frequency**, the **Frequency** and **Damping Ratio** properties appear requiring you to enter values to calculate the **Stiffness Coefficient**. Otherwise, you specify the **Stiffness Coefficient** manually. The **Mass Coefficient** property requires a manual entry.

Rotordynamics Controls (p. 1490)

Specify these properties as needed when setting up a Rotordynamic Analysis.

Analysis Data Management (p. 1484)

These properties enable you to save specific solution files from the Modal analysis for use in other analyses. You can set the **Future Analysis** field to **MSUP Analyses** if you intend to use the modal results in a subsequent Transient Structural, Harmonic Response, Random Vibration (PSD), or Response Spectrum (RS) analysis. If you link a Modal system to another analysis type in advance, the **Future Analysis** property defaults to the setting, **MSUP Analyses**. When a PSD analysis is linked

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

to a modal analysis, additional solver files must be saved to achieve the PSD solution. If the files were not saved, then the modal analysis has to be solved again and the files saved.

Note:

- **Solver Type**, **Damping Controls**, and **Rotordynamic Controls** are not available to the Samcef or ABAQUS solver.
- **Solver Type**, **Scratch Solver Files**, **Save Ansys db**, **Solver Units**, and **Solver Unit System** are only applicable to **Modal** systems.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

You can point to a **Static Structural** analysis in the **Initial Condition** environment field if you want to include pre-stress effects. A typical example is the large tensile stress induced in a turbine blade under centrifugal load that can be captured by a static structural analysis. This causes significant stiffening of the blade. Including this pre-stress effect will result in much higher, realistic natural frequencies in a modal analysis.

If the Modal analysis is linked to a Static Structural analysis for initial conditions and the parent static structural analysis has multiple result sets (multiple restart points at load steps/sub steps), you can start the Modal analysis from any restart point available in the Static Structural analysis. By default, the values from the last solve point are used as the basis for the modal analysis. See Restarts from Multiple Result Sets (p. 353) in the Applying Pre-Stress Effects for Implicit Analysis (p. 352) Help section for more information.

Note:

- When you perform a pre-stressed Modal analysis, the support conditions from the static analysis are used in the Modal analysis. You cannot apply any new supports in the Modal analysis portion of a pre-stressed modal analysis. When you link your Modal analysis to a Structural analysis, all structural loading conditions, including Inertial (p. 1500) loads, such as Acceleration and Rotational Velocity, are deleted from the Modal portion of the simulation once the loads are applied as initial conditions (p. 349) (via the **Pre-Stress** object). Refer to the Mechanical APDL command **PERTURB**,**HARM,,,DZEROKEEP** for more details.
- To account for the Coriolis Effect of rotational velocity applied in the Static analysis, you need to re-apply the rotational velocity in the Modal analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For **Pressure** boundary conditions in the Static Structural analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different modal results in the follow-on Modal Analysis. See the Pressure Load Stiffness (p. 353) topic in the Applying Pre-Stress Effects for Implicit Analysis (p. 352) Help Section for more information about using a pre-stressed environment.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement** or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Modal solution. If the Modal solution is followed by a Harmonic solution, these displacement loads become fixed boundary conditions for the Harmonic solution as well. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

Only the Rotational Velocity (p. 1511) and Thermal Condition (p. 1594) boundary conditions are supported for a stand-alone modal analysis. All structural supports (p. 1663) can be applied except a non-zero **Displacement**, a **Remote Displacement**, and the **Velocity** support. Due to its nonlinear nature, a **Compression Only Support** is not recommended for a modal analysis. Use of compression only supports may result in extraneous or missed natural frequencies.

For the Samcef and ABAQUS solvers, the following supports are not available: Compression Only Support, Elastic Support. When using line bodies, the following Pipe Pressure and Pipe Temperature loads are not available to the Samcef solver. Additionally, the Pipe Idealization object is also unavailable for the Samcef or ABAQUS solver.

Note:

In a pre-stressed modal analysis:

- Any structural supports used in the static analysis persist. Therefore, you are not allowed to add new supports in the pre-stressed modal analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- When creating a Campbell diagram, the Rotational Velocity (p. 1511) in the Static Structural Analysis is used to create normal stress stiffening effects in the Modal Analysis. It is not used to create centrifugal force effects for generating the Campbell diagram.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

Solution Information (p. 1833) continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis.

Important:

If you specify the **Distribute Solution** setting (the default setting on the **Advanced Properties** dialog of the Solve Process Settings (p. 1804)), the files file.full, file.esav and file.emat may not be combined at the end of the Modal analysis solution. As a result, any downstream system, including a Response Spectrum, Mode Superposition Harmonic Respoonse, Mode Superposition Transient, or Random Vibration analysis, or a follow on Mechanical APDL (turn on the **Distributed** property in Project Schematic), must also use a **Distributed Solution** setting as opposed to a shared memory solution, when the setting is turned off.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Highlight the **Solution** object in the tree to view a bar chart of the frequencies obtained in the modal analysis. A tabular data grid is also displayed that shows the list of frequencies, stabilities, modal damping ratios and logarithm decrements of each mode.

Note:

In a Modal Analysis (and other eigenvalue-based analyses such as buckling), the solution consists of a deformed shape scaled by an arbitrary factor. The actual magnitudes of the deformations and any derived quantities, such as strains and stresses, are therefore meaningless. Only the relative values of such quantities throughout the model should be considered meaningful. The arbitrary scaling factor is numerically sensitive to slight perturbations in the analysis; choosing a different unit system, for example, can cause a significantly different scaling factor to be calculated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For an undamped modal analysis, only frequencies are available in the **Tabular Data** window. For a damped modal analysis, real and imaginary parts of the eigenvalues of each mode are listed as **Stability** and **Damped Frequency**, respectively, in the **Tabular Data** window. If the real/stability value is negative, the eigenmode is considered to be stable. For the damped modal analysis, **Modal Damping Ratio** and **Logarithmic Decrement** are also included in the **Tabular Data** window. Like the stability value, these values are an indicator of eigenmode stability commonly used in rotordynamics.

If you select the **Reduced Damped** solver and set the **Store Complex Solution** property to **No**, then the application solves and stores the damped modal system as an undamped modal system. In addition to the undamped **Frequency**, the **Damped Frequency**, **Stability**, **Modal Damping Ratio** and **Logarithmic Decrement** result values are available in the **Tabular Data** window.

Note:

For the **Reduced Damped** solver with the **Store Complex Solution** property set to **No**, the Mechanical APDL Solver only writes undamped frequencies into result file. The solver retrieves the **Damped Frequency**, **Stability**, **Modal Damping Ratio** and **Logarithmic Decrement** from the Ansys database on the fly during the solution process. Use extra caution when using the **/POST1** in a **Command** object and make sure that your command entries and syntax are correct (especially if using the ***GET** command). Incorrect command entries can cause zero values for the **Damped Frequency** and **Stability**. Check the **Solution Information** (p. 1833) and error/warning messages to troubleshooting issues.

If **Campbell Diagram** (p. 1490) is set to On, a Campbell diagram chart result is available for insert under Solution. A Campbell diagram chart result conveys information as to how damped frequencies and stabilities of a rotating structural component evolve/change in response to increased rotational velocities. More detailed information about the result can be found in Campbell Diagram Chart Results (p. 2132). The **Campbell Diagram** function is not available to the Samcef or ABAQUS solver.

Note:

The Campbell diagram result chart is only appropriate for a rotating structural component that is axis-symmetrical. It is supported for all body types: solid, shell, and line bodies, but limited to single spool systems. For a single spool system, all bodies in the modal system are subjected to one and only single rotational velocity.

The contour and probe results are post-processed using set number, instead of mode number. The total set number is equal to number of modes requested multiplied by number of rotational velocity solve points. You can use the **Set**, **Solve Point** and **Mode** columns in the table to navigate between the set number and mode, and rotational velocity solve point and mode.

The ABAQUS solver does not allow modal expansion when post-processing mode shapes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You can choose to review the mode shapes corresponding to any of these natural frequencies by selecting the frequency from the bar chart or tabular data and using the context sensitive menu (right-click) to choose **Create Mode Shape Results**. You can also view a range of mode shapes.

"Stresses" from a Modal analysis do not represent actual stresses in the structure, but they give you an idea of the relative stress distributions for each mode. Stress and Strain results are available only if requested before solution using **Output Controls**.

You can view the mode shape associated with a particular frequency as a contour plot (p. 62). You can also animate (p. 2260) the deformed shape including, for a damped analysis, the option to allow or ignore the time decay animation for complex modes. The contours represent relative displacement of the part as it vibrates.

For complex modes, the Phase Angle associated with a particular frequency represents the specified angle in time domain and is equivalent to the product of frequency and time. Since the frequency is already specified in the results details view for a specific mode, the phase angle variation produces the relative variation of contour results over time.

When running a cyclic symmetry (p. 1147) analysis, additional result object settings in the Details view are available, as well as enhanced animations and graph displays. See Cyclic Symmetry in a Modal Analysis (p. 1159) for more information.

Note:

The use of construction geometry is not supported for the postprocessing of cyclic symmetry results.

Random Vibration Analysis

Introduction

This analysis enables you to determine the response of structures to vibration loads that are random in nature. An example would be the response of a sensitive electronic component mounted in a car subjected to the vibration from the engine, pavement roughness, and acoustic pressure.

Loads such as the acceleration caused by the pavement roughness are not deterministic, that is, the time history of the load is unique every time the car runs over the same stretch of road. Hence it is not possible to predict precisely the value of the load at a point in its time history. Such load histories, however, can be characterized statistically (mean, root mean square, standard deviation). Also random loads are non-periodic and contain a multitude of frequencies. The frequency content of the time history (spectrum) is captured along with the statistics and used as the load in the random vibration analysis. This spectrum, for historical reasons, is called Power Spectral Density or PSD.

In a random vibration analysis since the input excitations are statistical in nature, so are the output responses such as displacements, stresses, and so on.

Typical applications include aerospace and electronic packaging components subject to engine vibration, turbulence and acoustic pressures, tall buildings under wind load, structures subject to earthquakes, and ocean wave loading on offshore structures.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Points to Remember

- The excitation is applied in the form of Power Spectral Density (PSD). The PSD is a table of spectral values vs. frequency that captures the frequency content. The PSD captures the frequency and mean square amplitude content of the load's time history.
- The square root of the area under a PSD curve represents the root mean square (rms) value of the excitation. The application computes this value using the linear trapezoidal rule. The unit of the spectral value of acceleration, for example, is G^2/Hertz.
- The input excitation is expected to be stationary (the average mean square value does not change with time) with a zero mean.
- This analysis is based on the Mode Superposition method. Hence a modal analysis (p. 515) that extracts the natural frequencies and mode shapes is a prerequisite.
- This feature covers one type of PSD excitation only- base excitation.
- The base excitation could be an acceleration PSD (either in acceleration^2 units or in G^2 units), velocity PSD or displacement PSD.
- The base excitation is applied in the specified direction to all entities that have a Fixed Support (p. 1663) boundary condition. Other support points in a structure such as **Frictionless Surface** are not excited by the PSD.
- Multiple uncorrelated PSDs can be applied. This is useful if different, simultaneous excitations occur in different directions.
- If stress/strain results are of interest from the random vibration analysis then you will need to request stress/strain calculations in the modal analysis itself. Only displacement results are available by default.
- Postprocessing:
  - The regular results output by the solver for the Random Vibration analysis, such as Directional Deformation (p. 2018), are by default one sigma values, or one standard deviation values (with zero mean value). These results follow a Gaussian distribution. The interpretation is that 68.3% of the time the response will be less than the standard deviation value. One sigma is indicated by the **Scale Factor** property. All other result are not one sigma values. If you create a User Defined Result (p. 2192) using the Solution Quantities and Result Summary Worksheet (p. 1961) that is not a one sigma value, you will receive informational message indicating the situation.

Note:

→ Currently, one sigma supports the following **Expressions** (p. 2197) for User Defined Results (p. 2192):

UX/UY/UZ

EPELX/EPELY/EPELZ

EPELXY/EPELYZ/EPELXZ

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

SX/SY/SZ

SXY/SYZ/SXZ

SEQV

SPSD

→ The use of the **Scale Factor** property is not supported for force-based results, such as FX/FY/FZ, even though these forces are considered one sigma in the Mechanical APDL application.

– You can scale the result by 2 times to get the 2 sigma values. The response will be less than the 2 sigma values 95.45% of the time and 3 sigma values 99.73% of the time.

– The **Coordinate System** setting for result objects is, by default, set to **Solution Coordinate System** and cannot be changed because the results only have meaning when viewed in the solution coordinate system.

– Since the directional results from the solver are statistical in nature they cannot be combined in the usual way. For example the X, Y, and Z displacements cannot be combined to get the magnitude of the total displacement. The same holds true for other derived quantities such as principal stresses.

– A special algorithm by Segalman-Fulcher is used to compute a meaningful value for equivalent stress.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

Because a random vibration analysis is based on modal responses, a modal analysis is a required prerequisite. The requirement then is for two analysis systems, a modal analysis system and a random vibration analysis system that share resources, geometry, and model data.

From the **Toolbox**, drag a **Modal** template to the **Project Schematic**. Then, drag a **Random Vibration** template directly onto the **Modal** template.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Both Young's modulus (or stiffness in some form) and density (or mass in some form) must be defined in the modal analysis. Material properties must be linear but can be

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

isotropic or orthotropic, and constant or temperature-dependent. Nonlinear properties, if any, are ignored.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

There are no specific considerations for a random vibration analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

You can define rigid bodies for this analysis type.

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Only linear behavior is valid in a random vibration analysis. Nonlinear elements, if any, are treated as linear. If you include contact elements, for example, their stiffnesses are calculated based on their initial status and are never changed.

Only the stiffness of springs is taken into account in a random vibration analysis.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no specific considerations for a random vibration analysis.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

A specific **Modal Environment** must be set as an initial condition/environment for the analysis to be solved.

Note:

The modal analysis (p. 515) must extract enough modes to cover the PSD frequency range. Ansys recommends that you extract enough modes to cover 1.5 times the maximum frequency in the PSD excitation.

In order to process the PSD solution, the application creates additional solver files from the upstream modal analysis. Therefore, to create these required files, you need to solve the upstream Modal analysis before you can solve the PSD analysis. See the Analysis Data Management (p. 1484) section for more information.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a Random Vibration analysis the basic **Analysis Settings** include:

Options for Analyses (p. 1447)

You use this category to specify the number of modes to use from the modal analysis. Ansys recommends that you include modes that cover 1.5 times the maximum frequency in the PSD excitation table. You can also exclude insignificant modes by setting the **Mode Significance Level** property to between 0 (all modes selected) and 1 (no modes selected).

Note:

If you set the **Mode Significance Level** property to **`0.0`**, the application considers all modes in mode superposition of random vibration responses. This can require significant computation time for large systems that use a large number of modes to obtain random vibration displacement responses.

In this case, a **Mode Significance Level** setting that excludes insignificant modes from superimposing random vibration displacement responses is recommended. However, this performance improvement reduces solution accuracy. As a result, you need to use caution and carefully check your solution. Set the **Mode Significance Level** to **`1e-4`** when you are concerned about solution processing time. During Random Vibration analyses, the velocity and acceleration responses are separate calculations, in addition to displacement responses.

To further improve your solution time, do not request velocity and acceleration responses unless needed. The velocity and acceleration responses require approximately the same computation time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Output Controls (p. 1473)

By default, **Displacement** is the only response calculated. To include velocity (**Calculate Velocity** property) and/or acceleration (**Calculate Acceleration** property) responses, set their respective **Output Controls** to **Yes**. By default, modal results are removed from result file to reduce its size.

To keep modal results from an upstream system, set the **Keep Modal Results** property to **Yes**.

Note:

- If your **Random Vibration** system is linked to an upstream **Modal** analysis and you plan to send Modal data to a downstream **Sherlock (Post)** system, you need to set the **Keep Modal Results**, **Calculate Velocity**, **Calculate Acceleration** properties to **Yes**.
- You can change the default setting of these properties using the **Options** dialog. See the Specifying Application Defaults and Preferences (p. 217) section of the Help under the Analysis Settings and Solution (p. 238) category for the specific options.

Damping Controls (p. 1462)

Damping Controls enable you to specify damping for the structure in the Random Vibration analysis. Controls include: **Constant Damping**, **Damping Ratio**, **Stiffness Coefficient** (beta damping), and a **Mass Coefficient** (alpha damping). They can also be applied as Material Damping (p. 1467) using the **Engineering Data** tab. A non-zero damping is required. The **Damping Ratio** has a default setting of 0.01. This value can be modified by setting the **Damping** property to **Manual**.

Note:

For a Random Vibration system, if you choose the **Manual** setting for the **Constant Damping** property and **do not** define damping for one of the above controls, the solver uses a default damping value of **`0.01`**.

Analysis Data Management (p. 1484)

These settings enable you to save solution files from the **Random Vibration** analysis. The default behavior is to only keep the files required for postprocessing. You can use these controls to keep all files created during solution or to create and save a Mechanical APDL application database (db file).

Note:

The **Inertia Relief** option (under **Analysis Settings**) for an upstream Static Structural analysis is not supported in a Random Vibration analysis.

Apply Loads and Supports

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

- Any Support Type (p. 1663) boundary condition must be defined in the prerequisite Modal Analysis.
- The only applicable load is a **PSD Base Excitation** (p. 1589) of spectral value vs. frequency.
- Remote displacement cannot coexist with other boundary condition types (for example, fixed support or displacement) on the same location for excitation. The remote displacement will be ignored due to conflict with other boundary conditions.
- Four types of base excitation are supported: **PSD Acceleration**, **PSD G Acceleration**, **PSD Velocity**, and **PSD Displacement**.
- Each PSD base excitation should be given a direction in the nodal coordinate of the excitation points.
- Multiple PSD excitations (uncorrelated) can be applied. Typical usage is to apply 3 different PSDs in the X, Y, and Z directions. Correlation between PSD excitations is not supported.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

Solution Information (p. 1833) continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. In addition to solution progress you will also find the participation factors for each PSD excitation. The solver output also has a list of the relative importance of each mode in the modal covariance matrix listing.

Note:

When using a random vibration system database from a version prior to the most current version of Mechanical, it is possible to encounter incompatibility of the file(s) *file.mode*, *file.full*, and/or *file.esav*, created by the modal system. This incompatibility can cause the random vibration system's solution to fail. In the event you experience this issue, use the **Clear Generated Data** feature and resolve the modal system.

Refer to the Obtain the PSD Solution section of the *MAPDL Structural Analysis Guide* for more information.

Review Results

General *Basic general information about this topic (p. 356)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifics ***... for this analysis type:***

- If stress/strain results are of interest from the Random Vibration analysis then you will need to request stress/strain calculations in the modal analysis itself. You can use the **Output Controls** under **Analysis Settings** in the modal analysis for this purpose. Only displacement results are available by default.
- Linking a Random Vibration analysis system to a fully solved Modal analysis may result in zero equivalent stress. To evaluate correct equivalent stress in this situation, you need to re-solve the Modal analysis.
- Applicable results are Directional (X/Y/Z) Displacement/Velocity/Acceleration, normal and shear stresses/strains and equivalent stress. These results can be displayed as contour (p. 62) plots.
- For contour results, the **Information** category displays the properties described below. These properties are calculated *after* the associated result is evaluated and they refer to results contained in the results (`file.rst`) file. The following table describes property values and their result.

Example Details Pane Properties

Details of "Directional Acceleration"

Scope	
Definition	
Type	Directional Acceleration
Orientation	Y Axis
Reference	Absolute (including base motion)
Scale Factor	1 Sigma
Probability	68.269 %
Coordinate System	Solution Coordinate System
Identifier	
Suppressed	No
Results	
Options	
Acceleration in G	No
Information	
Load Step	5
Substep	1
Iteration Number	4

Property Output Descriptions

Load Step	Substep	Iteration Number	Result Description
3	1	2	1 Sigma Displacement solution.
4	1	3	1 Sigma Velocity solution (if the **Calculate Velocity** property in Output Controls (p. 1473) is set to **Yes**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Load Step	Substep	Iteration Number	Result Description
5	1	4	1 Sigma Acceleration solution (if the **Calculate Acceleration** property in Output Controls (p. 1473) is set to **Yes**).

See the Review the Results topic in the *Performing a Random Vibration (PSD) Analysis* section in the *Mechanical APDL Structural Analysis Guide* for additional information.

- The displacement results are relative to the base of the structure (the fixed supports).
- The velocity and acceleration results include base motion effects (absolute).
- Since the directional results from the solver are statistical in nature they cannot be combined in the usual way. For example the X, Y, and Z displacements cannot be combined to get the magnitude of the total displacement. The same holds true for other derived quantities such as principal stresses.
- For directional acceleration results, an option is provided to display the Transient Structural Analysis Using Linked in G (gravity) by selecting **Yes** in the **Acceleration in G** field.
- By default the 1 σ results are displayed. You can apply a scale factor to review any multiples of σ such as 2 σ or 3 σ. The Details view as well as the legend for contour results also reflects the percentage (using Gaussian distribution) of time the response is expected to be below the displayed values.
- Meaningful equivalent stress is computed using a special algorithm by Segalman-Fulcher. Note that the probability distribution for this equivalent stress is neither Gaussian nor is the mean value zero. However, the "3 σ" rule (multiplying the RMS value by 3) yields a conservative estimate on the upper bound of the equivalent stress.
- The Fatigue Tool enables you to perform a Spectral Fatigue analysis using the 1, 2, 3 σ stresses.
- For a **User Defined** result, if you want to request equivalent stress, you must specify **SPSD** for the **Expression** property (not SEQV). The SPSD Type uses the Segalman-Fulcher algorithm. SEQV uses a standard method to calculate equivalent stress, and in this instance, is incorrect for the desired 1 Sigma calculation.

  To ensure you properly select the SPSD expression, display results in the Solution Worksheet (p. 2192) and generate your result from the list of solution quantities. See the User Defined Results for the Mechanical APDL Solver (p. 2203) section for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Force Reaction** and **Moment Reaction** probes can be scoped to a **Remote Displacement**, **Fixed Support**, or **Displacement** boundary conditions to view Reactions Results.

  Note:

  – When you scope a **Moment Reaction** probe to a **Fixed Support** or a **Displacement**, the **Summation** property must be set to **Centroid**.

  – The results of **Force Reaction** and **Moment Reaction** probes scoped to a **Fixed Support** or **Displacement** are calculated using the **FSUM** Mechanical APDL solver command. This command reports the vector sum of the elemental nodal forces in the global coordinate system. See the **FSUM** command in the *Mechanical APDL Command Reference* for more information.

- The use of nodal averaging may not be appropriate in a random vibration analysis because the result values are not actual values but standard deviations. Moreover, the element coordinate system for the shell elements in a surface body may not all be aligned consistently when using the Default Coordinate System. Consider using unaveraged results for postprocessing instead.

File Management

When solving a Random Vibration analysis in an "In Process" solve mode, the pre-requisite files from the upstream Modal system are referenced by specifying the full path of their location (refer to **RESUME** and **MODDIR** commands) instead of making copies in order to improve solution time and disk usage. See the Solve Modes and Recommended Usage (p. 1803) section of the Help for more information about the different solve modes.

When you are solving in the "Out of Process" mode or when the **Keep Modal Results** property is set to **Yes**, the application copies the pre-requisite files from the Modal analysis to the Random Vibration Solver Files Directory. This may increase the required solution time for large models.

Using Command Objects within a Random Vibration Analysis

In an effort to minimize disk space usage, only the results from the Random Vibration analysis are kept in the result file. The results from the Modal analysis are removed during the solution.

If your command object contains commands which require this data, set the **Keep Modal Results** property in the **Output Controls** (p. 1473) to **Yes**.

Response Spectrum Analysis

Introduction

This analysis type is ideal for civil structure designs, such as high-rise buildings undergoing loads from wind as well as nuclear power plant designs experiencing seismic loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

A Response Spectrum analysis has similarities to the Random Vibration Analysis (p. 524). However, unlike a random vibration analysis, responses from a response spectrum analysis are deterministic maxima. For a given excitation, the maximum response is calculated based upon the input response spectrum and the method used to combine the modal responses. Combination methods include:

- Square Root of the Sum of the Squares (SRSS)
- Complete Quadratic Combination (CQC)
- Rosenblueth Double Sum Combination (ROSE)

Review the **Response Spectrum Options Category (p. 1458)** topic in the **Analysis Settings** > **Options for Analyses** section for more information about these methods.

Points to Remember

- The response spectrum is calculated based on modal responses. A modal analysis is therefore a prerequisite.
- If response strain/stress is of interest, then the modal strain and the modal stress need to be determined in the modal analysis.
- The excitation is applied in the form of a response spectrum. The response spectrum can have displacement, velocity, or acceleration units. For each spectrum value, there is one corresponding frequency.
- The excitation must be applied at fixed degrees of freedom.
- Because a new solve is required for each requested output (displacement, velocity, and acceleration), the content of a **Commands (APDL)** object is limited to solution (**SOLU**) commands.
- The results from the Mechanical APDL solver are displayed as the model's contour plot. The results are in terms of the maximum response.

Preparing the Analysis

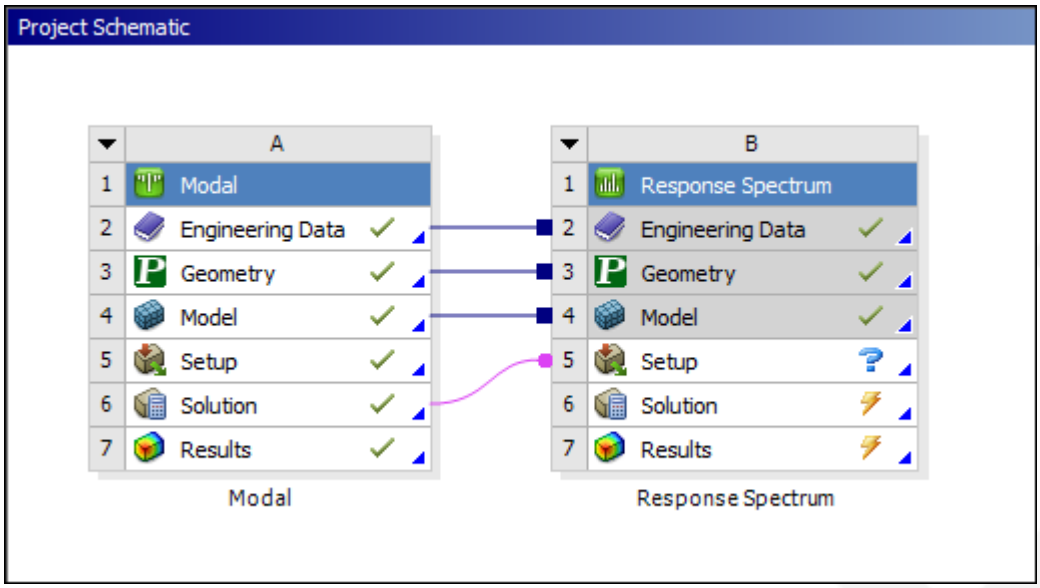
As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Create Analysis System

Because a response spectrum analysis is based on modal responses, a modal analysis is a required prerequisite. The modal analysis system and the response spectrum analysis system must share resources, geometry, and model data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Define Engineering Data

Material properties must be defined in a modal analysis. Nonlinear material properties are not allowed.

Define Part Behavior

You can define rigid bodies for this analysis type.

Define Connections

Nonlinear element types are not supported. They will be treated as linear. For example, the contact stiffness is calculated using the initial status without convergence check.

Define Initial Conditions

A specific **Modal Environment** must be set as an initial condition/environment for response spectrum analysis to be solved.

Establish Analysis Settings

For a Response Spectrum analysis, the basic **Analysis Settings** include:

Options for Analyses - Response Spectrum Analyses (p. 1458)

Perform the following settings for a Response Spectrum analysis:

- Specify the **Number of Modes To Use** property for the response spectrum calculation. Ansys recommends that you include the modes whose frequencies span 1.5 times the maximum frequency defined in the input response spectrum.
- Specify the **Spectrum Type** to be used for response spectrum calculation as either **Single Point** or **Multiple Points**. If the input response spectrum is applied to all fixed degrees of freedom, use **Single Point**, otherwise use **Multiple Points**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Specify the **Modes Combination Type** to be used for response spectrum calculation. In general, the **SRSS** method is more conservative than the **CQC** and the **ROSE** methods.

Note:

The **Inertia Relief** option (under **Analysis Settings**) for an upstream static structural analysis is not supported in a response spectrum analysis.

Output Controls (p. 1473)

By default, only displacement responses are calculated. To include velocity and/or acceleration responses, set their respective **Output Controls** to **Yes**.

Damping Controls (p. 1462)

allow you to specify damping for the structure in the response spectrum analysis. Controls include: **Damping Ratio**, **Stiffness Coefficient** (beta damping), and a **Mass Coefficient** (alpha damping). They can also be applied as Material Damping (p. 1467) using the **Engineering Data** tab. For the CQC mode combination type, non-zero damping is required.

Note:

Damping is not applicable to the SRSS combination method. Damping Controls are not available when the **Modes Combination Type** property is set to **SRSS**.

Analysis Data Management (p. 1484)

These settings enable you to save solution files from the response spectrum analysis. An option to save the Mechanical APDL application database (db file) from the analysis is provided.

Apply Loads and Supports

Note the following for boundary condition use and processing:

- Supported boundary condition types include Fixed Support, Displacement, Remote Displacement and body-to-ground Spring. If one or more fixed supports are defined in the model, the input excitation response can be applied to all fixed supports.
- A Remote Displacement cannot coexist with other boundary condition types, such as a Fixed Support or a Displacement, at the same location (scoping) as the excitation. The Remote Displacement will be ignored due to conflict with other boundary conditions.
- Note that the All boundary condition types for **Single Point** Response Spectrum only includes those fixed degree of freedoms defined using Fixed Support, Displacement, Remote Displacement and Body-to-Ground Spring. To apply an RS load to All boundary condition types for **Single Point** Response Spectrum, at least one allowed boundary condition must be defined.
- For a **Single Point** spectrum type, input excitation spectrums are applied to all boundary condition types defined in the model. For **Multiple Points** however, each input excitation spectrum is associated to only one boundary condition type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Three types of input excitation spectrum are supported: displacement input excitation (**RS Displacement**), velocity input excitation (**RS Velocity**) and acceleration input excitation (**RS Acceleration**). See RS Base Excitation (p. 1590) for further details.
- The input excitation spectrum direction is defined in the global coordinate system for Single Point spectrum analysis. For Multiple Points spectrum analysis, however, the input excitation is defined in the nodal coordinate systems (if any) attached to the constrained nodes.
- More than one input excitation, with any different combination of spectrum types, is allowed for the response spectrum analysis.
- Specify option to include or not include contribution of high frequency modes in the total response calculation by setting **Missing Mass Effect** (p. 1590) to **Yes** or **No**. The option for including the modes is normally required for nuclear power plant design.
- Specify option to include or not include rigid responses to the total response calculation by setting **Rigid Response Effect** (p. 1590) to **Yes** or **No**. The rigid responses normally occur in the frequency range that is lower than that of missing mass responses, but is higher than that of periodic responses.
- **Missing Mass Effect** is only applicable to **RS Acceleration** excitation. See the RS Base Excitation (p. 1590) section of the Help for more information.
- For a **Single Point** spectrum type, the entire table of input excitation spectrum can be scaled using the **Scale Factor** setting. The factor must be greater than 0.0. The default is 1.0.

Perform Solution

Ansys recommends that you review the **Solution Information** (p. 1833) page for any warnings or errors that might occur during the solution. You may receive some warning messages and still be able to solve the analysis.

Note:

When using a Response Spectrum system from a previous version of the application, it is possible you could encounter incompatibility of for the modal system file(s) *file.mode*, *file.full*, and/or *file.esav*. This incompatibility can cause the Response Spectrum system's solution to fail. In the event you experience this issue, use the **Clear Generated Data** feature and resolve the modal system.

Refer to the Obtain the Spectrum Solution section of the *MAPDL Structural Analysis Guide* for more information.

Review Results

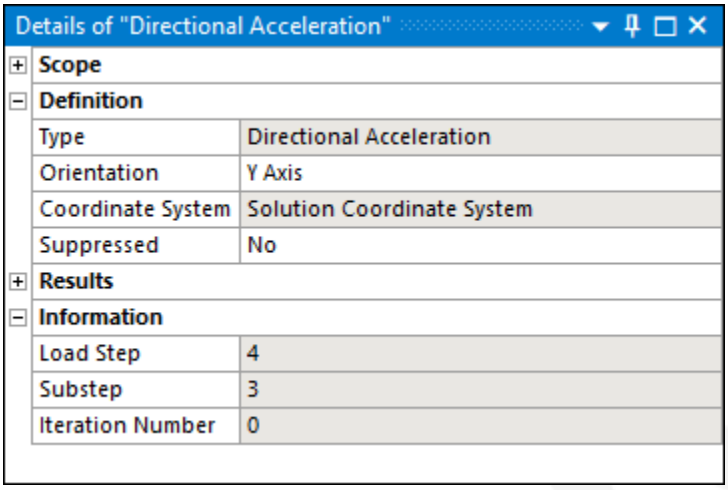
Note the following for result use and processing:

- To view strain/stress results, a selection must be made in **Output Controls** of the Modal analysis. By default, only **Deformation** drop-down menu results are available.
- Applicable **Deformation** results are **Total**, **Directional** (X/Y/Z), **Directional Velocity** and **Directional Acceleration**. If strain/stress are requested, applicable results are normal strain and stress, shear strain and stress, and equivalent stress.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Equivalent stress is a derived stress calculated using component stresses.
- Results are displayed as a contour plot on the model.
- For contour results, the **Information** category includes the following properties. These properties are calculated *after* the associated result is evaluated and they refer to results contained in the results (`file.rst`) file.

Example Details Properties



Property Output Descriptions

Load Step	Substep	Result Description
4	1	Displacement solution.
4	2	Velocity solution (if the **Calculate Velocity** property in Output Controls (p. 1473) is set to **Yes**).
4	3	Acceleration solution (if the **Calculate Acceleration** property in Output Controls (p. 1473) is set to **Yes**).

Note:

The value of the **Iteration Number** property varies based on analysis specifications. It is application defined, is information-based only, and has no impact on results.

You can also see the Review the Results topic in the *Performing a Random Vibration (PSD) Analysis* section in the *Mechanical APDL Structural Analysis Guide* for additional information.

- If you set the **Store Modal Result** property of the **Output Controls** to **No** in the upstream Modal analysis, the application creates additional auxiliary files in addition to standard files. The application automatically creates the `Displacement.mcom` file. And, if you set the **Calculate Velocity** and **Calculate Acceleration** properties (also **Output Controls**) of the downstream spectrum analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to **Yes**, then the application also creates the files `Velocity.mcom` and/or `Acceleration.mcom`. These files contain combination instructions as well as mode coefficients.

Note:

This configuration increases the computation time of spectrum analysis.

- **Force Reaction** and **Moment Reaction** probes can be scoped to a **Remote Displacement**, **Fixed Support**, or **Displacement** boundary conditions to view Reactions Results.

  Note:

  – When you scope a **Moment Reaction** probe to a **Fixed Support** or a **Displacement**, the **Summation** property must be set to **Centroid**.

  – The results of **Force Reaction** and **Moment Reaction** probes scoped to a **Fixed Support** or **Displacement** are calculated using the **FSUM** Mechanical APDL solver command. This command reports the vector sum of the elemental nodal forces in the global coordinate system. See the **FSUM** command in the *Mechanical APDL Command Reference* for more information.

- When the **Missing Mass Effect** property is set to **Yes**, the **Deformation** results that include the data from property in their result calculation are the **Directional** (Deformation/Displacement) and **Directional Acceleration** results. Note that the application supports the **Directional Velocity** result; however, it does not incorporate **Missing Mass Effect** conditions for its calculation.
- The **Directional** (Deformation/Displacement) result is a relative result whereas **Directional Velocity** and **Directional Acceleration** are absolute results.
- A **Directional Acceleration** result at or near fixed supports scoped to an **RS Acceleration** are correctly calculated by setting the **Missing Mass Effect** property and the **Rigid Response Effect** property to **Yes**. Corresponding properties display and require definition when these two properties are included.

File Management

When solving a Response Spectrum analysis with the "In Process" solve mode, the pre-requisite files from the upstream Modal system are referenced by specifying the full path of their location (refer to **RESUME** and **MODDIR** commands) instead of making copies in order to improve solution time and disk usage. See the Solve Modes and Recommended Usage (p. 1803) section of the Help for more information about the different solve modes.

When you are solving in the "Out of Process" mode, the application copies the pre-requisite files from the Modal analysis to the Response Spectrum **Solver Files Directory**. This may increase the required solution time for large models.

For additional technical information, refer to the Spectrum Analysis section of the *Mechanical APDL Structural Analysis Guide* as well as the **MMASS** command and the **RIGRESP** command in the *Mechanical APDL Command Reference*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Substructure Generation Analysis

Introduction

The **Substructure Generation** analysis uses the component mode synthesis (CMS) to:

1. Condense all the finite elements of your model into one "superelement" that is represented as a matrix.
2. Create load vectors from the loading conditions and constraints that you apply. In addition, you can use an upstream Static Structural analysis to create pre-stress effects in the Substructure Generation analysis solution.

Once you have solved the analysis, you export this generated data as an Exported Condensed Part (`.cpa`) file for use in another appropriate simulation .

Note:

The load vectors extracted from the imported condensed part (`.cpa`) file are applied as loads in the use pass of the MSUP Harmonic Response analysis. No other analysis consumes these load vectors

---

Benefits

Substructuring reduces computer time and enables the solution of very large problems with limited computer resources. In a nonlinear analysis, you can substructure the linear portion of the model so that the element matrices for that portion do not need to be recalculated at every equilibrium iteration. In a structure with repeated patterns (such as the four legs of a table), you can generate one substructure to represent the pattern and simply make copies of it at different locations, thereby saving a significant amount of computer time.

Limitations

Note the following limitations for this analysis:

- This analysis type is supported for the Mechanical APDL solver only.
- This analysis supports External Model when Beta Options are active.
- The Master Degrees of Freedom Nodes (MDOFs) specified in the Mechanical application may be reduced by the solver during the solution of the analysis. This could be the result of a constraint defined on the MDOF nodes. The application reports this with a warning. For this case, there could be a discrepancy between Mechanical selected MDOFs and solver computed MDOFs as it might have been reduced due to over constraint. This is not recommended, and you should review the model before proceeding with use pass. A warning message will also display while eliminating the redundant degrees of freedom.
- The contact setting **Normal Lagrange** is not supported for the **Formulation** property.
- The only supported Joint types are **Fixed** and **Bushing** (with the **Formulation** property set to **Bushing**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Most element types can be used to generate a substructure. In general, the only restriction is that elements within the superelement are assumed to be linear and cannot use Lagrange multipliers. See the Generation Pass section of the *Mechanical APDL Substructuring Analysis Guide* for additional information.

Points to Remember

- All active bodies in the model will be condensed into one substructure (superelement).
- The substructure analysis can be linked to upstream Static Structure (p. 652) analysis in order to consider the prestressed effects of the structure. However, when you specify a pre-stress analysis, the **Substructure Generation** analysis only supports the following loading conditions:
  - Acceleration (p. 1501)
  - Standard Earth Gravity (p. 1509)
  - Nodal Force (p. 1721)
  - Nodal Pressure (p. 1724)
  - Nodal Orientation (p. 1719)
- This analysis supports Imported Pressure (p. 1752), including both real and imaginary values.

> **Note:**
>
> Imported Body Temperature (p. 1740) is also supported when Beta Options are active.

Preparing the Analysis

As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Establish Analysis Settings

The analysis includes the following Analysis Settings (p. 1417) categories:

- Options (p. 1458)

> **Note:**
>
> The maximum value for the **Number of Load Vectors** property is limited by the memory available on your computer.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Solver Controls (p. 1428): **Direct** is the only supported **Solver Type** property option.

  Note:

  The options **Supernode** and **Subspace** are available when Beta Options are active.

- Analysis Data Management (p. 1484)

Define Pre-Stress

If your analysis includes an upstream Static Structural (p. 652) system, specify the properties of the **Pre-Stress (Static Structural)** object. The analysis considers the pre-stressed inertial loads and boundary conditions during the generation pass. This is a result of the **Load Control** property of the **Pre-Stress** object being set (by default) to the **Keep Inertia and Displacement Constraints**. See the description of the **Load Control** property in the Pre-Stress object reference for more information.

Specify Interfaces

You specify interfaces on the **Substructure Definition** object using Named Selections (p. 1057) and Remote Points (p. 1089).

Tip:

To automatically create a **Named Selection** and a corresponding interface (in the **Worksheet**):

1. Select the **Substructure Definition** object.
2. From the **Geometry** window, select a supported geometric or mesh entity on the model.
3. Right-click and select **Create Selection and Add Interface**.

Define Substructure Definition Object

Specify the properties for the Substructure Definition object, including:

Category	Properties/Options/Descriptions
Geometry	**Geometry**: Read-only property that displays the number of bodies of the model.
Definition	**Physics Type**: Read-only property indicating the physics type (Structural). **Matrix Reduction Method**: Indicates the currently supported matrix reduction method - Component Mode Synthesis. **Interface Method**: Options include **Fixed** (default) or **Free**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Descriptions
	Lumped Mass Formulation: Specify the desired lumped mass matrix formulation. Options include **Program Controlled** (default), **Off**, and **On**. The **Off** setting uses the element-dependent default mass matrix formulation and the **On** setting uses a lumped mass approximation. **Generate Damping Matrix**: Specify whether to generate a damping matrix. Options include **Program Controlled** (default), **Yes**, and **No**.
Interfaces	**Number of Interfaces**: Read-only display of the number of interface entries detected by the application. **Number of Master Nodes**: Read-only display of the number of superelement master nodes resulting from interfaces.

Note:

The application treats added masses (from Point Masses (p. 901) and Distributed Masses (p. 902)) as internal to the superelement.

Apply Loads and Supports

Without considering the Limitations (p. 540), the analysis supports the following boundary conditions:

Inertial

- Acceleration (p. 1501)
- Standard Earth Gravity (p. 1509)

Loads

- Pressure (p. 1521)
- Force (p. 1543) (applied to a face, edge, or vertex)

Supports

The following *Supports* (p. 1663) can be used in this analysis:

- Fixed Support
- Displacement
- Remote Displacement
- Frictionless Support
- Compression Only Support
- Cylindrical Support
- Elastics Support

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Conditions

Constraint Equation (p. 1702)

Direct FE (node-based Named Selection scoping and constant loading only)

- Nodal Force (p. 1721)
- Nodal Pressure (p. 1724)
- Nodal Displacement (p. 1728)
- Nodal Orientation (p. 1719)

Imported Loads

Imported Pressure (p. 1752)

Note:

Imported Body Temperature (p. 1740) is also supported when Beta Options are active.

Note:

The application computes load vectors for each defined load. Once you export the solution (see below), you use these load vectors in another analysis through the **Imported Condensed Parts** (p. 790) object. Load vector information displays in the Worksheet when you select the **Imported Condensed Parts** object.

Specify Results

The only result supported by this analysis is Total Deformation (p. 2018).

Perform Solution

The **Solution Information** object presents continuous updates of any output from the solver and provides valuable information on the behavior of the structure during the analysis.

Select the **Solution** object to view the frequencies obtained from the solution process in the **Graph** and **Tabular Data** windows.

Exporting Substructure Solution

Once you have solved your analysis, you can export the generated superelement for use in a different session of the application. Right-click the **Solution** object, select **Export Substructure (.cpa)**, and then select one of the following options:

- **Use Pass Only**: Using this option, the application generates solution files required to perform a Use Pass analysis. No Expansion Pass can be performed when this option is selected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

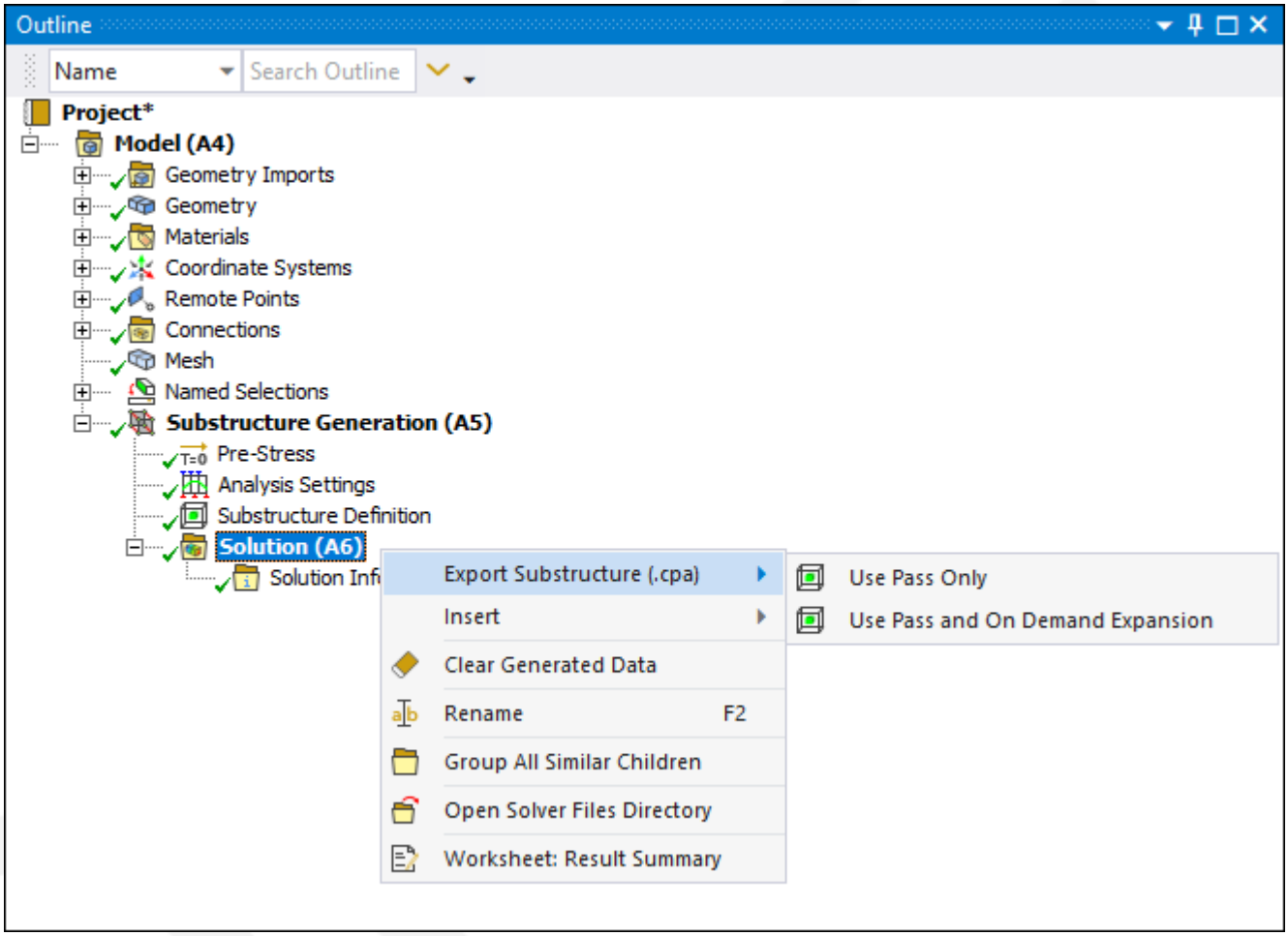
Published: 2023-10-23

- **Use Pass and On Demand Expansion**: Using this option, the application generates solution files required to perform a Use Pass analysis as well as **On Demand Expansion**.

A progress bar displays during the export operation and offers an option to stop (red square icon) the process.

Note:

If you set the **Future Analysis** property (**Analysis Settings**>Analysis Data Management (p. 1484)) to **Ansys Rigid Dynamics (Beta)**, the application automatically includes the additional files necessary to perform a downstream Rigid Dynamics (p. 550) analysis in the exported file.



References

For a more detailed look at this analysis type, see the:

- Substructure Analysis (p. 781) section of the *Mechanical User's Guide*.
- Mechanical APDL Substructuring Analysis Guide

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Magnetostatic Analysis

Introduction

Magnetic fields may exist as a result of a current or a permanent magnet. In the Mechanical application you can perform 3D static magnetic field analysis. You can model various physical regions including iron, air, permanent magnets, and conductors.

Typical uses for a magnetostatic analysis are as follows:

- Electric machines
- Transformers
- Induction heating
- Solenoid actuators
- High-field magnets
- Nondestructive testing
- Magnetic stirring
- Electrolyzing cells
- Particle accelerators
- Medical and geophysical instruments.

Points to Remember

- This analysis is applicable only to 3D geometry.
- The geometry must consist of a single solid multibody part (p. 847).
- A magnetic field simulation requires that air surrounding the physical geometry be modeled as part of the overall geometry. The air domain can be easily modeled in DesignModeler using the Enclosure feature. Ensure that the resulting model is a single multibody part which includes the physical geometry and the air.
- In many cases, only a symmetric portion of a magnetic device is required for simulation. The geometry can either be modeled in full symmetry in the CAD system, or in partial symmetry. DesignModeler has a Symmetry feature that can slice a full symmetry model, or identify planes of symmetry for a partial symmetry model. This information is passed to the Mechanical application for convenient application of symmetry plane boundary conditions.
- A Magnetostatic analysis supports a multi-step solution.

Preparing the Analysis

Create Analysis System

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag the **Magnetostatic** template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

- Magnetic field simulation supports 4 categories of material properties:

  1. Linear "soft" magnetic materials - typically used in low saturation cases. A **Relative Permeability** is required. This may be constant, or orthotropic with respect to the coordinate system of the body (See Details view). Orthotropic properties are often used to simulate laminate materials.

  2. Linear "hard" magnetic materials - used to model permanent magnets. The demagnetization curve of the magnet is assumed to be linear. **Residual Induction** and **Coercive Force** are required.

  3. Nonlinear "soft" magnetic material - used to model devices which undergo magnetic saturation. A B-H curve is required. For orthotropic materials, you can assign the B-H curve in any of the orthotropic directions, while specifying a constant **Relative Permeability** in the other directions. (Specifying a value of "0" for **Relative Permeability** will make use of the B-H curve in that direction.)

  4. Nonlinear "hard" magnetic material - used to model nonlinear permanent magnets. A B-H curve modeling the material demagnetization curve is required.

- When using an Ansys license that includes the Emag license feature, only the following material properties are allowed: Isotropic Resistivity, Orthotropic Resistivity, Relative Permeability, Relative Permeability (Orthotropic), Coercive Force & Residual Induction, B-H Curve, B-H Curve (Orthotropic), Demagnetization B-H Curve. You may have to turn the filter off in the **Engineering Data** tab to suppress or delete those material properties/models that are not supported for the license.

- Conductor bodies require a **Resistivity** material property. Solid source conductor bodies can be constant or orthotropic with respect to the coordinate system of the body. Stranded source conductor bodies can only be modeled as isotropic materials.

- For convenience, a library of common B-H curves for soft magnetic material is supplied with the product. Use the **Import** tool in Engineering Data to review and retrieve curves for use.

Note:

In a magnetostatic analysis, you can orient a polarization axis for a Linear or Nonlinear Hard material in either the positive or negative x direction with respect to a local or global coordinate system (p. 1191). Use the **Material Po-**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

larization setting in the Details view for each body to establish this direction. The **Material Polarization** setting appears only if a hard material property is defined for the body. For a cylindrical coordinate system, a positive x polarization is in the positive radial direction, and a negative x polarization is in the negative radial direction.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

There are no specific considerations for a magnetostatic analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

Mechanical does not support Rigid Bodies in Magnetostatic analyses. For more information, see the *Stiffness Behavior* documentation for Rigid Bodies (p. 852).

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Connections are not supported in a magnetostatic analysis.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

- Although your body is automatically meshed at solve time, it is recommended that you select the **Electromagnetic Physics Preference** in the Details view of the Mesh object folder.
- Solution accuracy is dependent on mesh density. Accurate force or torque calculations require a fine mesh in the air regions surrounding the bodies of interest.
- The use of pyramid elements in critical regions should be minimized. Pyramid elements are used to transition from hexagonal to tetrahedral elements. You can eliminate pyramid elements from the model by specifying **Tetrahedrons** using a **Method** mesh control tool.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifics ***... for this analysis type:***

For a Magnetostatic Analysis, the basic **Analysis Settings** include:

Step Controls for Static and Transient Analyses (p. 1418)

Step Controls are used to specify the end time of a step in a single or multiple step analysis.

Multiple steps are needed if you want to change load values, the solution settings, or the solution output frequency over specific steps. Typically you do not need to change the default values.

Solver Controls (p. 1428)

Solver Controls enable you to select either a direct or iterative solver. By default the program will use the direct solver. Convergence is guaranteed with the direct solver. Use the Iterative solver only in cases where machine memory is an issue. The solution is not guaranteed to converge for the iterative solver.

Nonlinear Controls (p. 1468)

Nonlinear Controls enable you to modify convergence criteria and other specialized solution controls. These controls are used when your solution is nonlinear such as with the use of nonlinear material properties (B-H curve). Typically you will not need to change the default values for this control. CSG convergence is the criteria used to converge the magnetic field. CSG represents magnetic flux. AMPS convergence is only used for temperature-dependent electric current conduction for solid conductor bodies. AMPS represents current.

Output Controls (p. 1473)

Output Controls enable you to specify the time points at which results should be available for postprocessing. A multi-step analysis involves calculating solutions at several time points in the load history. However you may not be interested in all of the possible results items and writing all the results can make the result file size unwieldy. You can restrict the amount of output by requesting results only at certain time points or limit the results that go onto the results file at each time point.

Analysis Data Management (p. 1484)

The **Analysis Data Management** settings enable you to save solution files from the magnetostatic analysis. The default behavior is to only keep the files required for postprocessing. You can use these controls to keep all files created during solution or to create and save the Mechanical APDL application database (db file).

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

There is no initial condition specification for a magnetostatic analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

- You can apply electromagnetic boundary conditions and excitations in the Mechanical application. See Electromagnetic Boundary Conditions and Excitations (p. 1635) for details.
- Boundary conditions may also be applied on symmetry planes via a **Symmetry** (p. 1127). A **Symmetry** folder allows support for Electromagnetic Symmetry (p. 1130), Electromagnetic Anti-Symmetry (p. 1131), and Electromagnetic Periodicity (p. 1131) conditions.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

The **Solution Information** (p. 1833) object provides some tools to monitor solution progress in the case of a nonlinear magnetostatic analysis.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

Adaptive mesh refinement (p. 1853) is available for magnetostatic analyses.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

A magnetostatic analysis offers several results (p. 2157) for viewing. Results may be scoped to bodies and, by default, all bodies will compute results for display. For **Inductance** or **Flux Linkage**, define these objects prior to solution. If you define these after a solution, you will need to re-solve.

Rigid Dynamics Analysis

Introduction

You can perform a Rigid Dynamics analysis in the Mechanical application using the Ansys Rigid Dynamics solver. This type of analysis is used to determine the dynamic response of an assembly of rigid bodies linked by joints and springs. You can use this type of analysis to study the kinematics of a robot arm or a crankshaft system for example.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Points to Remember

- Inputs and outputs are joint forces, moments, displacements, velocities and accelerations.
- On rigid parts, there are no stresses and strain results produced, only forces, moments, displacements, velocities and accelerations.
- The solver is tuned to automatically adjust the time step. Doing it manually is often inefficient and results in longer run times.

This section contains the following topics:

Preparing a Rigid Dynamics Analysis
Command Reference for Rigid Dynamics Systems
Using the Variable Load Add-on
Using the Motion Load Transfer Add-on
Multibody Dynamics Theory Guide

Preparing a Rigid Dynamics Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag a **Rigid Dynamics** template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Density is the only material property utilized by rigid bodies. Models that use zero or nearly zero density fail to solve with the Ansys Rigid Dynamics solver.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

Sheet, solid, and line bodies are supported by the Ansys Rigid Dynamics solver, but line bodies can only be flexible and included in a condensed part (p. 781). Plane bodies cannot be used.

Rigid line bodies are not supported in RBD because the mass moment of inertia is not available.

Define Part Behavior

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

You can define a **Point Mass** (p. 901) for this analysis type.

Stiffness behavior can be rigid or flexible.

Note:

If the part behavior is flexible, it must be included in a condensed part.

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Applicable connections are joints (p. 1292), springs (p. 1395), and contacts (p. 1235).

When an assembly is imported from a CAD system, joints or constraints are not imported, but joints may be created automatically after the model is imported. You can also choose to create the joints manually.

Each joint is defined by its coordinate system of reference. The orientation of this coordinate system is essential as the free and fixed degrees of freedom are defined in this coordinate system.

Automatic contact generation is also available after the model is imported.

For information on contact specifically oriented for Rigid Dynamics, see Contact in Rigid Dynamics (p. 1275) and Best Practices for Contact in Rigid Body Analyses (p. 1278).

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Mesh controls apply to surfaces where contact is defined as well as to deformable parts.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For Rigid Dynamics analyses the basic controls are:

Step Controls (p. 1418) enable you to create multiple steps. Multiple steps (p. 1418) are useful if new loads are introduced or removed at different times in the load history.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

A Rigid Dynamics analysis can use an explicit time integration scheme, especially if the model is made only of rigid parts. Unlike the implicit time integration, there are no iterations to converge in an explicit time integration scheme. The solution at the end of the time step is a function of the derivatives during the time step. As a consequence, the time step required to get accurate results is usually smaller than is necessary for an implicit time integration scheme. Another consequence is that the time step is governed by the highest frequency of the system. A very smooth and slow model that has a very stiff spring will require the time step needed for the stiff spring itself, which generates the high frequencies that will govern the required time step. Stiff models can be more efficiently solved using the Implicit Generalized-α, Implicit Stabilized Generalized-α, or MJ Time Stepping time integration schemes.

Because it is not easy to determine the frequency content of the system, an automatic time stepping algorithm is available, and should be used for the vast majority of models. This automatic time stepping algorithm is governed by **Initial Time Step**, **Minimum Time Step**, and **Maximum Time Step** under **Step Controls**; and **Energy Accuracy Tolerance** under **Nonlinear Controls**.

- **Initial Time Step**: If the initial time step chosen is vastly too large, the solution will typically fail, and produce an error message that the accelerations are too high. If the initial time step is only slightly too large, the solver will realize that the first time steps are inaccurate, automatically decrement the time step and start the transient solution over. Conversely, if the chosen initial time step is excessively small, and the simulation can be accurately performed with higher time steps, the automatic time stepping algorithm will, after a few gradual increases, find the appropriate time step value. Choosing a good initial time step is a way to reduce the cost of having the solver figure out what time step size is optimal to minimize run time. While important, choosing the correct initial time step typically does not have a large influence on the total solution time due to the efficiency of the automatic time stepping algorithm.

- **Minimum Time Step**: During the automatic adjustment of the time step, if the time step that is required for stability and accuracy is smaller than the specified minimum time step, the solution will not proceed. This value does not influence solution time or its accuracy, but it is there to prevent the solver from running forever with an extremely small time step. When the solution is aborting due to hitting this lower time step threshold, that usually means that the system is over constrained, or in a lock position. Check your model, and if you believe that the model and the loads are valid, you can decrease this value by one or two orders of magnitude and run again. That can, however generate a very large number of total time steps, and it is recommended that you use the **Output Controls** settings to store only some of the generated results.

- **Maximum Time Step**: Sometimes the time step that the automatic time stepping settles on produces too few results outputs for precise postprocessing needs. To avoid these postprocessing resolution issues, you can force the solution to use time steps that are no bigger than this parameter value.

Solver Controls (p. 1428): For this analysis type, enables you to select a time integration algorithm (Program Controlled, Runge-Kutta order 4, Implicit Generalized-α, Stabilized Generalized-α, MJ Time Stepping) and select whether to use constraint stabilization. The default time integration option, **Program Controlled**, provides the appropriate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

accuracy for most applications. The default, **Program Controlled** is valid for most applications, however; you may wish to set this option to **User Defined** and manually enter customized settings for weak spring and damping effects. The default is **Off**.

Nonlinear Controls (p. 1468) enable you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control.

- **Energy Accuracy Tolerance**: This is the main driver to the automatic time stepping. The automatic time stepping algorithm measures the portion of potential and kinetic energy that is contained in the highest order terms of the time integration scheme, and computes the ratio of the energy to the energy variations over the previous time steps. Comparing the ratio to the **Energy Accuracy Tolerance**, Workbench will decide to increase or decrease the time step. Energy accuracy tolerance is program controlled by default. It is enabled with the Explicit Runge-Kutta method and disabled by default with Implicit Generalized-α, Implicit Stabilized Generalized-α, and MJ Time Stepping.

  Note:

  For systems that have very heavy slow moving parts, and also have small fast moving parts, the portion of the energy contained in the small parts is not dominant and therefore will not control the time step. It is recommended that you use a smaller value of integration accuracy for the motion of the small parts.

  Spherical (p. 1302), slot (p. 1301) and general (p. 1310) joints with three rotation degrees of freedom usually require a small time step, as the energy is varying in a very nonlinear manner with the rotation degrees of freedom.

- **Force Residual Relative Tolerance:** (Only available with Implicit Generalized-α or Stabilized Generalized-α time integration or MJ Time Stepping integration) This option controls the threshold used in Newton-Raphson for force residual convergence. The default value is 1.e-7. A smaller value will lead to a smaller residual, but it will require more iterations. The convergence of force residual can be monitored in **Solution Information** using Force Convergence.

- **Constraint Equation Residual Relative Tolerance:** (Only available with Implicit Generalized-α or Stabilized Generalized-α time integration or MJ Time-Stepping integration) This option controls the threshold used in Newtom-Raphson to check convergence of constraint equations violations. The default value is 1.e-8. The convergence of this criterion can be checked in **Solution Information** using Displacement Convergence

Output Controls (p. 1473) enable you to specify the time points at which results should be available for postprocessing. In a transient nonlinear analysis it may be necessary to perform many solutions at intermediate time values. However i) you may not be interested in reviewing all of the intermediate results and ii) writing all the results can make the results file size unwieldy. This group can be modified on a per step basis.

Define Initial Conditions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

Before solving, you can configure the joints and/or set a joint load to define initial conditions.

1. Define a **Joint Load** (p. 1592) during a short initial load step to set initial conditions on the free degrees of freedom of a joint.

   For the Ansys Mechanical APDL solver to converge, it is recommended that you ramp the angles and positions from zero to the real initial condition over one step. The Ansys Rigid Dynamics solver does not need these to be ramped. For example, you can directly create a joint load for a revolute (p. 1299) joint of 30 degrees, over a short step to define the initial conditions of the simulation. If you decide to ramp it, you have to keep in mind that ramping the angle over 1 second, for example, means that you will have a non-zero angular velocity at the end of this step. If you want to ramp the angle and start at rest, use an extra step maintaining this angle constant for a reasonable period of time or, preferably, having the angular velocity set to zero. Another way to specify the initial conditions in terms of positions and angles is to use the Configure tool (p. 1380), which eliminates the time steps needed to apply the initial conditions.

   To fully define the initial conditions, you must define position and velocities. Unless specified by joint loads, if your system is initially assembled, the initial configuration will be unchanged. If the system is not initially assembled, the initial configuration will be the "closest" configuration to the unassembled configuration that satisfies the assembly tolerance (p. 219) and the joint loads.

   Unless specified otherwise, *relative* joint velocity is, if possible, set to zero. For example, if you define a double pendulum and specify the angular velocity of the grounded revolute joint, by default the second pendulum will not be at rest, but will move rigidly with the first one.

2. Configure a joint (p. 1380) to graphically put the joint in its initial position.

See Joint Initial Conditions (p. 1296) for further details.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads and supports can be used in a Rigid Dynamics analysis:

- **Acceleration** (p. 1501)
- **Standard Earth Gravity** (p. 1509)
- **Joint Load** (p. 1592)
- **Remote Displacement** (p. 1674)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Remote Force** (p. 1552)
- **Constraint Equation** (p. 1702)

Joint Load

Magnitude

For a **Joint Load**, the joint condition's magnitude could be a constant value, could vary with time as defined in a table or via a function, or could depend on other values measured on the model during the solution. See Using the Variable Load Add-on (p. 611) to define such a load variation. Details of how to apply a tabular or function load are described in Defining Boundary Condition Magnitude (p. 1771). Details on the **Joint Load** are included below.

In addition, see the Apply Loads and Supports (p. 354) section for more information about time stepping and ramped loads.

Interpolation/Derivation

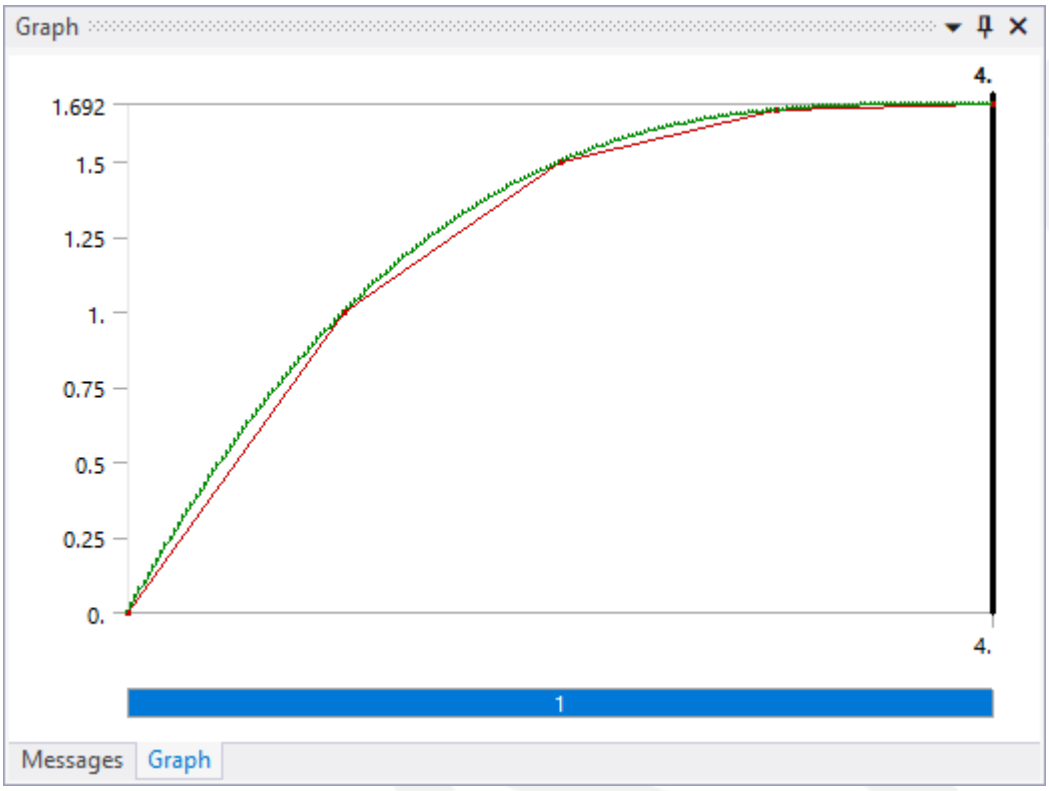
For joint loads applied through tabular data values, the number of points input will most likely be less than the number of time steps required to solve the system. As such, an interpolation is performed. When a joint load requires fitting, the result of the fitting is displayed in the joint load chart along with the user-specified values. The number of segments used to display the curve is driven by the **Number Of Segments** property.

The underlying fitting method used for interpolation can be configured using the **Fitting Method** field (specific to Rigid Dynamics analysis). Options include:

- **Program Controlled (default):** Depending on the **Joint Load** type, the solver chooses the appropriate interpolation method. Accelerations and Force joint loads use a piecewise linear. Displacement/Rotation/Velocity/Rotational Velocity joint loads use either a linear fitting when only two points are given per load step, or a cubic spline fitting when there are more than two points as shown on the following graph:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



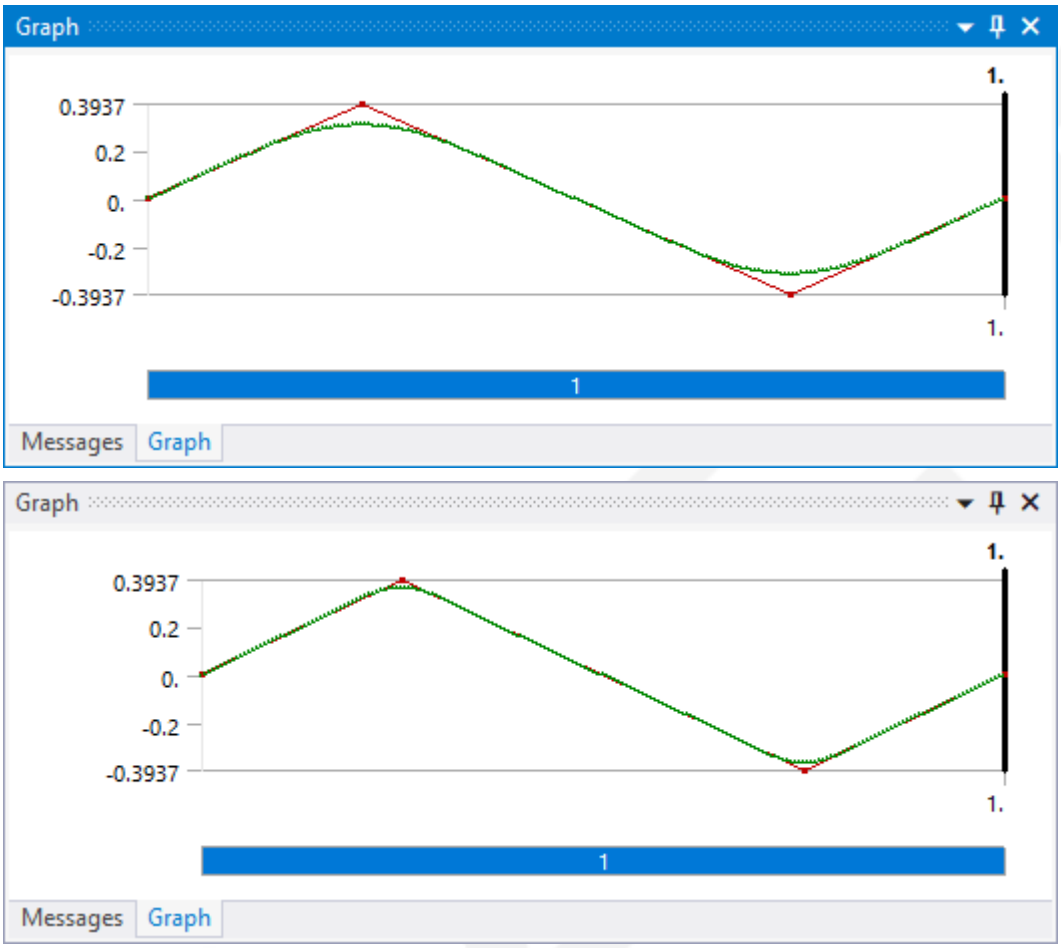
A large difference between the interpolated curve and the linear interpolation may prevent the solution from completing. If this is the case and you intend to use the linear interpolation, you can simply use multiple time steps, as the interpolation is done in one time step.

- **Fast Fourier Transform:** Fast Fourier Transform is performed to fit tabular data. Unlike cubic spline fitting, no verification on the fitting quality is performed. The additional cutoff frequency parameter specifies the threshold (expressed in Hz) used to filter high frequencies. Higher cutoff frequency results in a better fitting, but leads to smaller time steps. The following graphs show the effect of cutoff frequency on FFT fitting on a triangular signal using 5 Hz and 10 Hz, respectively.

Note:

The Fourier Fitting implicitly expects cyclic data. Lack of continuity between end time and start time values may lead to oscillations known as Gibbs phenomenon. Similarly, the lack of continuity for the derivatives may also lead to unwanted oscillations around jumps. For instance, an imposed displacement requires the continuity of the first and second derivatives. The Rigid Dynamics solver implements several dedicated treatments to prevent Gibbs phenomenon. However, results of Fourier Fitting must be carefully reviewed to check there are no artificial oscillations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

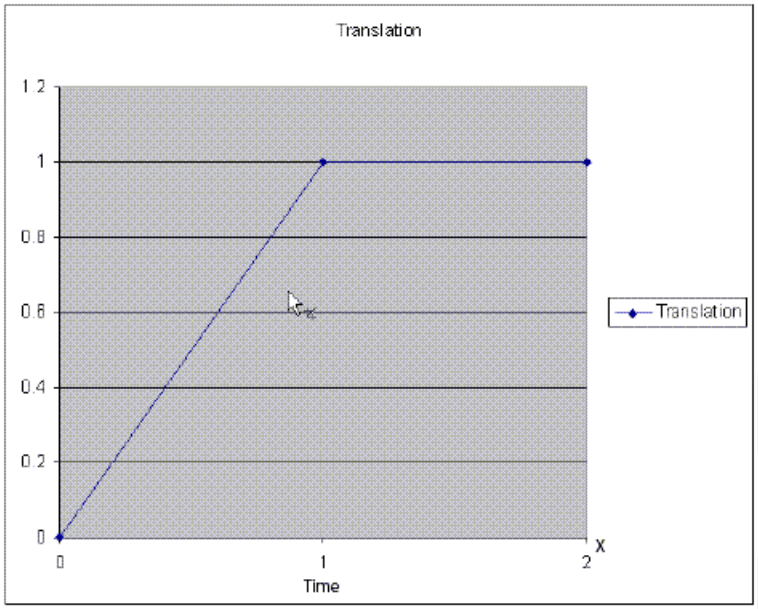


Discontinuities

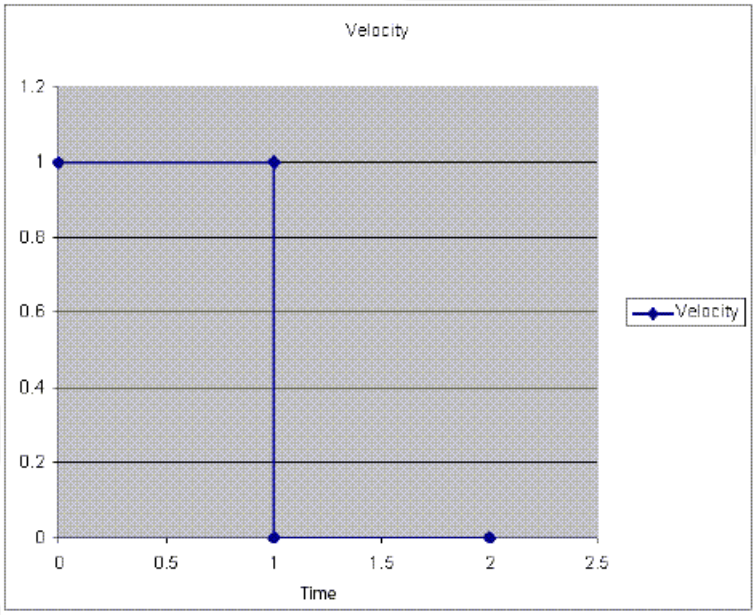
When defining a joint load for a position and an angle, the corresponding velocities and accelerations are computed internally. When defining a joint load for a translational and angular velocity, corresponding accelerations are also computed internally. By activating and deactivating joint loads, you can generate some forces/accelerations/velocities, as well as position discontinuities. Always consider what the implications of these discontinuities are for velocities and accelerations. Force and acceleration discontinuities are perfectly valid physical situations. No special attention is required to define these velocity discontinuities. Discontinuities can be obtained by changing the slope of a relative displacement joint load on a translational joint, as shown on the following graphs using two time steps:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The corresponding velocity profile is shown here.



This discontinuity of velocity is physically equivalent to a shock, and implies infinite acceleration if the change of slope is over a zero time duration. The Ansys Rigid Dynamics solver will very often handle these discontinuities, and redistribute velocities after the discontinuity according to all active joint loads. This process of redistribution of velocities usually provides accurate results; however, no shock solution is performed, and this process is not guaranteed to produce a physically acceptable energy redistribution. A closer look at the total energy probe will tell you if the solution is valid. In case the redistribution is not done properly, use one step instead of two to use an interpolated, smooth position variation with respect to time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Discontinuities of positions and angles are not a physically acceptable situation. Results obtained in this case may not be physically sensible. Workbench cannot detect this situation up front. If you proceed with position discontinuities, the solution may abort or produce false results.

Rotations

For fixed axis rotations, it is possible to count a number of turns. For 3D general rotations, it is not possible to count turns. In a single axis case, although it is possible to prescribe angles higher than 2π, it is not recommended because Workbench can lose count of the number of turns based on the way you ramp the angle. You should avoid prescribing angular displacements with angles greater than Pi when loading bushing joints, because the angle-moment relationship could differ from the stiffness definition if the number of turns is inaccurate, or in case of Euler angles singularity. It is highly recommended that you use an angular velocity joint load instead of an angle value to ramp a rotation, whenever possible.

For example, replace a rotation joint load designed to create a joint rotation from an angle from 0 to 720 degrees over 2 seconds by an angular velocity of 360 degrees/second. The second solution will always provide the right result, while the behavior of the first case can sometimes lead to the problems mentioned above.

For 3D rotations on a general joint for example, no angle over 2π can be handled. Use an angular velocity joint load instead.

Multiple Joint Loads On The Same Joint

When prescribing a position or an angle on a joint, velocities and acceleration are also prescribed. The use of multiple joint loads on the same joint motion can cause for joint loads to be determined inaccurately.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

Only synchronous (p. 1803) solves are supported for Rigid Dynamics analyses.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Use a **Solution Information** object to track, monitor, or diagnose problems that arise during solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Applicable results are Deformation (p. 2018) and Probe (p. 1946) results.

Note:

If you highlight **Deformation** results in the tree that are scoped to rigid bodies, the corresponding rigid bodies in the **Geometry** window are not highlighted.

To plot different results against time on the same graph or plot one result quantity against a load or another results item, use the Chart and Table (p. 1929) feature.

If you duplicate (p. 40) a Rigid Dynamics analysis, the results of the duplicated branch are also cleared (p. 1981).

Joint Conditions and Expressions

When a rotation, position, velocity or angular velocity uses an expression that user the power (^) operator, such as (x)^(y), the table will not be calculated properly if the value x is equal to zero. This is because its time derivative uses log(x), which is not defined for x = 0. An easy workaround is to use x*x*x... (y times), which assumes that y is an integer number and thus can be derived w.r.t time without using the log operator.

Remote Force

Remote Force (p. 1552) direction can be configured in Rigid Dynamics analyses using the **Follower Load** option. Remote direction can be either constant (**Follower Load**=**No**, Default), or it can follow the underlying body/part (**Follower Load**=**Yes**).

Command Reference for Rigid Dynamics Systems

The Rigid Dynamics solver uses an object-based approach that uses Python-based commands that follow Python syntax. This section explains this approach and the role of Python in rigid body commands. It also provides a library of commands for Rigid Dynamics analyses (arranged by parent object) and examples of command usage. Topics available in this section include:

An ACT extension is provided to facilitate the creation of complex joint and body loads that would otherwise require using Python command snippets. You can find information about how to load and use the extension in Using the Variable Load Add-on (p. 611).

IronPython References

The Rigid Dynamics Object Model

Rigid Dynamics Command Objects Library

Command Use Examples

Debugging RBD Commands with Visual Studio

Using RBD Commands with Excel

Using RBD Commands from the IronPython Console

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

IronPython References

Rigid Dynamics uses an object-based approach, so it is useful to have experience with object oriented programming and the Python language when writing commands for the solver. Ansys Workbench scripting is based on IronPython, which is well integrated with the rest of the .NET Framework (on Windows) and Mono CLR (on Linux). This makes all related libraries easily available to Python programmers while maintaining compatibility with the Python language. For more information on IronPython, see http://ironpython.net.

IronPython is compatible with existing Python scripts, but not all C-based Python library modules are available under IronPython. Refer to the IronPython website for more information. For more information on Python, including a standard language reference, see http://www.python.org/.

The Rigid Dynamics Object Model

In the Rigid Dynamics object-based approach, the **Environment** is the top level object that allows access to all other underlying objects. The environment is associated with an environment object in the **Mechanical** tree. Many environments can exist on the same model. The model is called the **System** in the **Rigid Dynamics Object** model. The system contains the physical representation of the model, and the environment contains the representation of a given simulation done on the model. This means that **Bodies** and **Joints** belong to the systems, and **Joint Conditions** or **Loads** are available on the environment.

You can access an object using its unique ID, which is the same ID used by Mechanical. Global object tables help you to access an object for which you have an ID.

For example, a **Joint** with the ID **`_jid`** can be accessed using the following call:

```
Joint= CS_Joint.Find(_jid)
```

`CS_xxx` is the table of all **`xxx`** type objects.

If the ID of an object is not known, or if only one occurrence of the object exists in the object model, query the object table to find the first occurrence of a given object type. This is explained in the following example:

```
Environment = CS_Environment.FindFirstNonNull()
```

`GetId()`

This call returns the object ID.

`GetName()`

This call returns the object name.

`SetName(name)`

This call sets or changes the object name.

Some objects have to be created by calling the object constructor. For example, to create a constant variable:

```
Var = CS_ConstantVariable()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Rigid Dynamics Command Objects Library

The following Rigid Dynamics command objects are available:

Actuator (p. 563)
Basis (p. 564)
Body (p. 565)
Body Coordinate System (p. 566)
Body Load (p. 567)
CMSBody (p. 568)
Condition (p. 569)
Contact (p. 570)
ContactDebugMask (p. 570)
ContactOptions (p. 571)
Driver (p. 572)
Environment (p. 572)
Flexible Body (p. 574)
GILTable (p. 576)
Joint (p. 578)
JointDOFLoad (p. 582)
Load (p. 583)
Measure (p. 583)
MSolverDB (p. 587)
PointsTable (p. 588)
PolynomialTable (p. 588)
Relation (p. 589)
Spring (p. 592)
SolverOptions (p. 590)
System (p. 592)
Table (p. 593)
UserTable (p. 594)
Variable (p. 594)

Actuator

The actuator is the base class for all Loads (p. 583), Body Loads (p. 567), and Drivers (p. 572).

ID table: **`CS_Actuator`**

DRAFT

Members:

`Condition`

All actuators can be conditional. See Condition (p. 569) to create this condition.

`AppliedValue`

Measure that stores the evaluation of the actuator variable. Can be useful when the applied value depends on a measure other than time.

`EnergyMeasure`

Measure that stores the energy generated by the actuator.

Member Functions:

There are two ways to define the value of the load: using a variable, or by defining a table of input measures (in which case a variable is defined automatically).

`SetVariable(variable)`

`variable` is a list of input measures in table form.

`SetInputMeasure(measure)`

`measure` is typically the time measure object, but other measures can be used as well. When using an expression to define a load variation, the measure must have only one component (it cannot be a vector measure). The variation can be defined by a constant, an expression, or a table.

`SetConstantValues(value)`

`value` is a Python float constant. See Relation (p. 589) object for defining a constant.

`SetTable(table)`

`table` is a **`CS_Table`** .

`SetFunc(string, is_degree)`

`string` is similar to the expression used in the user interface to define a joint condition by a function. Note that the literal variable is always called **`time`**, even if you are using another measure as input.

`is_degree` is a boolean argument. If the expression uses trigonometric function, it specifies that the input variable should be expressed in degrees.

Basis

A basis is a material frame moving with a body. Each coordinate system has a basis, but multiple coordinate systems can share the same basis.

ID table: **`CS_Basis`**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Constructors:

CS_Basis()

CS_Basis(Angle1, Angle2, Angle3)

Members:

double [,]Matrix

Sets or gets function of the transformation matrix

Body

A body corresponds to a **Part** in the geometry node of the Mechanical tree, or can be created by a command snippet. The preset **_bid** variable can be used to find a corresponding body.

ID table: **CS_Body**

Example:

```
MyBody = CS_Body.Find(_bid)
print MyBody.Name
```

Constructors:

CS_Body()

CS_Body(Id)

Members:

Name

Name of the body.

Origin

Origin Coordinate System of the body. This Coordinate System is the moving coordinate system of one of the joints connected to the body. The choice of this joint, called parent joint, is the result of an optimization that minimizes the number of degrees of freedom of the system.

InertiaBodyCoordinateSystem

Inertia body coordinate system of the body.

BodyType

Type of body, values in **E_UnknownType**, **E_Ground**, **E_Rigid**, **E_CMS**, **E_General**, **E_Fictitious**, **E_RigidLeaf**, **E_RigidSubModel**, **E_PointMass**, **E_Beam**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Member Functions:

`SetMassAndInertia(double mass, double Ixx, double Iyy, double Izz, double Ixy, double Iyz, double Ixz)`

Overwrites the mass and inertia values of a body.

`SetCenterOfMassAndOrientationAngles(double Xg, double Yg, double Zg, double XYAngle, double YZAngle, double XZAngle)` and `SetCenterOfMassAndOrientationMatrix(double Xg, double Yg, double Zg, double mxx, double mxy, double mxz, double myx, double myy, double myz, double mzx, double mzy, double mzz)`

Overwrites the position of the center of mass and the orientation of the inertia coordinate system.

`SetVariableMassAndPrincipalInertia(CS_Variable mass, CS_Variable Ixx, CS_Variable Iyy, CS_Variable Izz)`

Overwrites the constant mass and principal inertia properties by variable properties. During the solution process, the mass and inertia variation rate needs to be evaluated. Therefore, only Point Table, Polynomial and Function can be used to define the variation. Python user tables cannot be used to define kinetic properties variations. You can make some of the properties (mass, Ixx, Iyy and Izz) constants by using constant variables.

Note:

The principal axis needs to be defined when the principal inertia is being assigned. If the body is created by a command, **`SetCenterOfMassAndOrientationAngles`** or **`SetCenterOfMassAndOrientationMatrix`** must be called before calling **`SetVariableMassAndPrincipalInertia`**.

This function only applies to rigid bodies.

Note:

All quantities used in the solver must use a consistent unit system, which sometimes differs from the user interface unit system. For example if the user interface unit system is "mm,kg,N,s", the solver unit system will be “mm,t,N,s". When using `SetMassAndInertia` or `SetVariableMassAndPrincipalInertia`, the values of mass and inertia have to be entered using the solver unit system.

Derived Classes:

`CS_FlexibleBody`

Body Coordinate System

The body coordinate system is used to connect a body to joints, hold the center of mass, or define a load. See Joint (p. 578) or Body (p. 565) to access existing coordinate systems. Coordinate systems can also be created.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

ID table: **CS_BodyCoordinateSystem**

Constructors:

CS_BodyCoordinateSystem(body, type, xyz, basis)

Members:

Basis (p. 564)

Member Functions:

RotateArrayThroughTimeToLocal(MeasureValues)

Rotates the transient values of a measure to a coordinate system. **MeasureValues** is a python two-dimensional array, such as that coming out of **FillValuesThroughTime** or **FillDerivativesThroughTime**. This function works for 3D vectors such as relative translation between two coordinate systems or 6-D vectors such as forces/moments.

RotateArrayThroughTimeToGlobal(MeasureValues)

Rotates the transient values of a measure from a coordinate system to the global coordinate system.

Type

Type of coordinate system, values in **E_Unknown**, **E_Ground**, **E_Part**, **E_Joint**, **E_Inertia**, **E_BodyTransform**, **E_Contact**, **E_SplitJoint**.

Derived Classes:

None

Example:

```
forceInGlobal=joint.GetForce()

valuesInGlobal=forceInGlobal.FillValuesThroughTime()

for i in range(0,valuesInGlobal.GetLength(0)):
    print '{0:e} {1:e} {2:e} {3:e}'.format(valuesInGlobal[i,0],
 valuesInGlobal[i,1],valuesInGlobal[i,2],valuesInGlobal[i,3])

mobileCS=joint.MobileCoordinateSystem

valuesInLocal=valuesInGlobal.Clone()

mobileCS.RotateArrayThroughTimeToLocal(valuesInLocal)

for i in range(0,valuesInGlobal.GetLength(0)):
    print '{0:e} {1:e} {2:e} {3:e}'.format(valuesInLocal[i,0],
    valuesInLocal[i,1],valuesInLocal[i,2],valuesInLocal[i,3])
```

Body Load

A body load is a load that is applied to all bodies in the system. Gravity or global acceleration are body loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The body load must implement a **`GetAccelerationVector`** method. This vector is applied to the center of mass of each body. In order to maintain the energy balance of the system, the body load must also implement a **`ComputeEnergy`** method.

Example: Acceleration varying with time

```
HalfTime = 1.0
HalfAmplitude = 10.0

Env=CS_Environment.GetDefault()
Sys=Env.System
(ret,found,time) = Sys.FindOrCreateInternalMeasure(CS_Measure.E_MeasureType.E_Time)

class MyBodyLoad(CS_UserBodyLoad):
    def __init__(self):
        CS_UserBodyLoad.__init__(self)
        self.value = 0.0

    def GetAccelerationVector(self,Mass,xyz,vel,bodyLoadForce):
        values = time.Values
        print 'MyBodyLoad::GetAccelerationVector'
        bodyLoadForce[0] = 0.0
        bodyLoadForce[1] = 0.0
        bodyLoadForce[2] = Mass*HalfAmplitude*math.sin(values[0]*3.14/(2.*HalfTime))

    def ComputeEnergy(self,Mass,xyz,vel):
        print 'MBodyLoad::ComputeEnergy'
        return 0.0

load=MyBodyLoad()
load.value = 10.0

Env=CS_Environment.GetDefault()

Env.BodyLoads.Add(load)
```

CMSBody

A CMSBody represents a condensed part in the Mechanical tree.

Constructors:

None.

Members:

`CondensedPartId (read only)`

The ID of the condensed part in the Mechanical tree.

`PartIds (read only)`

The vector of the IDs of the Mechanical parts that are used in the condensed part.

Member Functions:

None.

Derived Classes:

None.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Condition

Condition causes a load or a joint condition to be active only under defined circumstances. A condition is expressed in one of the following forms:

1. **`MeasureComponent`** operator **`threshold`**
2. **`LeftThreshold`** < **`MeasureComponent`** < **`RightThreshold`**
3. **`LeftCondition`** operator **`RightCondition`**

For case 1:

- **`MeasureComponent`** is a scalar Measure (p. 583).
- **`Operator`** is one of the following math operators:

  `E_GreaterThan`
  `E_LessThan`
  `E_DoubleEqual`
  `E_ExactlyEqual`

- **`Threshold`** is the threshold value.

Note:

A condition cannot be shared between various actuators. For example, if two joint conditions must be deactivated at the same time, two conditions must be created.

Example:

```
DispCond = CS_Condition(CS_Condition.E_ConditionType.E_GreaterThan,DispX,0.1)
```

For case 2:

- **`MeasureComponent`** is a scalar Measure (p. 583).
- **`LeftThreshold`** and **`RightThreshold`** are the bounds within which the condition will be true.

Example:

```
RangeCond = CS_Condition(DispX,0.0,0.1)
```

For case 3:

- **`LeftThreshold`** and **`RightThreshold`** are two conditions (case 1, 2 or 3).
- **`Operator`** is one of the following boolean operators:

  `E_Or`
  `E_And`

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example:

```
BoolCond = CS_Condition(CS_Condition.E_ConditionType.E_Or, RangeCond, DispCond)
```

Contact

A Contact corresponds to a contact pair between two bodies.

Corresponding ID table: **`CS_Contact`**

Note:

If multiple contact objects have been defined between the same two bodies (with different surfaces), the solver merges them into one single pair. In that case, only one of the contact pairs exists and the call to **`CS_Contact.Find(_cid)`** will fail for all contact objects other than the one that was used to handle the pair of bodies.

Constants:

None

Members:

None

Member Functions:

`GetOutputContactForce()`

Retrieves a measure that contains the total contact force between the two linked bodies.

ContactDebugMask

The ContactDebugMask object allows you to activate and customize the output of contact points. It can also be used to modify the default behaviour of contact. ContactDebugMask uses a set of switches that can be toggled on or off.

ID table: **`CS_ContactDebugMask`**

Constants:

```
      E_DEBUG_Flag.E_None,
 (*)E_DEBUG_Flag.E_Point1: point on the side 1 (contact)
   E_DEBUG_Flag.E_Point2: point on the side 2 (target)
   E_DEBUG_Flag.E_Normal: contact normal
   E_DEBUG_Flag.E_Normal1: normal on side 1 (Reference)
   E_DEBUG_Flag.E_Normal2: normal on side 2 (Target)
   E_DEBUG_Flag.E_Violation: contact violation (rd.n = P1P2.n)
   E_DEBUG_Flag.E_MaterialVelocity: material normal velocity (V2-V1).n
(*)E_DEBUG_Flag.E_TotalVelocity: total normal velocity (material velocity + sliding velocity)
   E_DEBUG_Flag.E_EntityId1: geometric entity Id on side 1 (contact)
   E_DEBUG_Flag.E_EntityId2: geometric entity Id on side 2 (target)
   E_DEBUG_Flag.E_SurfaceId1: surface Id on side 1 (contact)
   E_DEBUG_Flag.E_SurfaceId2: surface Id on side 2 (target)
(*)E_DEBUG_Flag.E_EntityType: type of geometric entities (vertex/edge/surface)
(*)E_DEBUG_Flag.E_GeometricStatus: status of the contact position and velocity (touching,separated,.
   E_DEBUG_Flag.E_Accepted: points that are finally kept
   E_DEBUG_Flag.E_InconsistentPoint: points not consistent with rank analysis
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    E_DEBUG_Flag.E_ReceivedPoint: all points send by the contact
    E_DEBUG_Flag.E_DeletedPoint: points deleted during Geometric Filtering
    E_DEBUG_Flag.E_TrackedPoint: points successfully tracked
    E_DEBUG_Flag.E_TrackedPointFailure: points that failed for tracking
    E_DEBUG_Flag.E_NormalAngle: angle between normal (in degrees)
    E_DEBUG_Flag.E_SlidingVelocity1: sliding velocity on side 1 (contact) in global coordinates
    E_DEBUG_Flag.E_SlidingVelocity2: sliding velocity on side 2 (target) in global coordinates
    E_DEBUG_Flag.E_FailSafeFilteringMode: adjust contact radius to accept at least one point
    E_DEBUG_Flag.E_CheckIntegration: check consistency of integration between solver and contact
    E_DEBUG_Flag.E_RankAnalysis: result from rank analysis
    E_DEBUG_Flag.E_Transition: result from edge transitions analysis
  (*)E_DEBUG_Flag.E_NewTimeStep: at beginning of time step
    E_DEBUG_Flag.E_BeforeCorrection: before external loop of correction
    E_DEBUG_Flag.E_BeforeCorrectionPlus: before geometric correction
    E_DEBUG_Flag.E_All
```

Members:

None

Member Functions:

`SetOn(E_DEBUG_Flag flag)`

Enable output of contact points information specified by flag.

`SetOff(E_DEBUG_Flag flag)`

Disable output of contact points information specified by flag.

Example:

```
CS_ContactDebugMask.SetOn(E_DEBUG_Flag.E_Accepted)
```

ContactOptions

The ContactOptions object allows you to customize the behaviour of a contact server. ContactOptions uses a set of numerical values (real or integer) that can be get or set. When used as a switch, 0 means off and 1 is on.

Constants:

None

Members:

TimeOut

Time in second (=30.0 by default)

Verbose

Enable verbose mode in contact.out file (=0, disabled by default)

NumberOfThreads

Number of parallel threads used for contact detection (=2 by default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Member Functions:

None

Example:

```
cOpts=CS_ContactOptions()
cOpts.Verbose=1
```

Driver

A driver is a position, velocity or acceleration, or translational or rotational joint condition. Drivers derive from the Actuator class.

Corresponding ID table: **`CS_Actuator`**

Constants:

`E_Acceleration`, **`E_Position`**, **`E_Velocity`**

Members:

None

Member Functions:

`CS_Driver(CS_Joint joint, int[] components, E_MotionType driverMotionType)`

Creation of a joint driver, on joint **`joint`**, degree of freedom **`components`**, and with motion type **`driverMotionType`**. Note that the same driver can prescribe more than one joint motion at the same time. This is useful if you want to add the same condition to all components of a prescribed motion for example. Components must be ordered, are zero based, and refer to the actual free degrees of freedom of the joint.

Environment

This is the top level of the Rigid Dynamics model.

ID table: **`CS_Environment`**

Members:

System:

Corresponding system.

Example:

```
Env=CS_Environment.FindFirstNonNull()
Sys = Env.System
```

Ground:

Ground body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Example:

```
Env = CS_Environment.FindFirstNonNull()
Ground = Env.Ground
```

Loads:

The vector of existing loads. This includes **Springs** that are considered by the solver as loads, as well as force and torque joint conditions.

Example:

```
Xdof = 0
Friction=CS_JointDOFLoad(PlanarJoint,Xdof)
Env.Loads.Add(Friction)
```

BodyLoads:

The vector of **Body Loads**.

Example:

```
MyBodyLoad = CS_BodyLoad()
…
Env.BodyLoads.Add(MyBodyLoad)
```

Relations:

The vector of external constraint equations.

Example:

```
rel3=CS_Relation()
rel3.MotionType=CS_Relation.E_MotionType.E_Velocity

var30=CS_ConstantVariable()
var30.SetConstantValues(System.Array[float]([0.]))

var31=CS_ConstantVariable()
var31.SetConstantValues(System.Array[float]([23.]))

var32=CS_ConstantVariable()
var32.SetConstantValues(System.Array[float]([37.]))

var33=CS_ConstantVariable()
var33.SetConstantValues(System.Array[float]([-60.+37.]))

rel3.SetVariable(var30)
rel3.AddTerm(jp,0,var31)
rel3.AddTerm(js3,0,var32)
rel3.AddTerm(jps,0,var33)

Env.Relations.Add(rel3)
```

Drivers:

The vector of **Displacements**, **Velocity** and **Acceleration** joint conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

InitialConditions:

The vector of **Displacements**, **Velocity**, and **Acceleration** joint conditions to be used only at **time**=**0**.

PotentialEnergy:

Gets the Potential Energy Measure.

KineticEnergy:

Gets the Kinetic Energy Measure.

TotalEnergy:

Gets the Total Energy Measure.

ActuatorEnergy:

Gets the Actuator Energy Measure.

RestartTime

Specifies the starting time in a restart analysis

Member Functions:

`FindFirstNonNull()`:

Returns the first environment in the global list. The table usually contains only one environment, thus it is a common way to access the current environment.

Example:

```
Env=CS_Environment.FindFirstNonNull()
```

`AlterSimulationEndTime(endTime)`

Overwrites the end time of the simulation.

`Solve()`

Solves the current analysis.

Derived Classes:

None

FlexibleBody

A Flexible Body is used by RBD for bodies that have flexible behavior, for instance a CMS-Body (p. 568).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Constructors:

None.

Members:

AlphaDamping

Uses a variable to define the amount of alpha Rayleigh damping (proportional to the mass matrix) to be considered for the flexible body. The variable can be either dependent or constant.

Example:

```
aFlexibleBody.AlphaDamping=100
```

Or equivalently:

```
var=CS_Variable()
var.SetConstantValues(System.Array[float]([100.]))
aFlexibleBody.AlphaDamping=var
```

BetaDamping

Uses a variable to define the amount of beta Rayleigh damping (proportional to the mass matrix) to be considered for the flexible body. The variable can be either dependent or constant.

Example:

```
Env=CS_Environment.GetDefault()
Sys=Env.System

array=System.Array.CreateInstance(float,4,2)
array[0,0]=0.0
array[0,1]=5.e-6
array[1,0]=0.05
array[1,1]=5.e-6
array[2,0]=0.051
array[2,1]=1.e-4
array[3,0]=0.1
array[3,1]=1.e-4

table=CS_PointsTable(array)

(err,found,time)=Sys.FindOrCreateInternalMeasure(CS_Measure.E_MeasureType.E_Time)

var=CS_Variable()
var.AddInputMeasure(time)
var.SetTable(table)

aFlexibleBody.BetaDamping = var
```

CMatrixScaleFactor

Define a factor to be used to multiply the default damping matrix. For instance, with a CMSBody (p. 568), this matrix can be created during the generation pass. When the damping matrix is generated for a Condensed Part (CMSBody (p. 568)), it will be automatically taken into account in the RBD use pass with a factor equal to 1.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Member Functions:

`SetModalDamping(iDof, variable)`

Define the amount of damping used for the degree of freedom specified by **`iDof`** (index starts at 0). The variable can be either dependent or constant.

`GetModalDamping(iDof)`

Retrieve the damping variable defined for the degree of freedom **`iDof`** (index starts at 0).

`SetLoadVectorScaleFactor(iLV, variable)`

Define a scale factor applied to the flexible body internal load specified by **`iLV`** (index starts at 0). By default, the first load vector uses a constant scale factor equal to 1.0.

`GetLoadVectorScaleFactor(iLV, variable)`

Retrieve the variable associated to the factor specified by **`iLV`** (index starts at 0).

Derived Classes:

`CS_CMSBody`

GILTable

A general multi-input interpolated table based on an unstructured cloud of points.

Corresponding ID table: **`CS_GILTable`**

Member Functions:

`CS_GILTable(sizeIn,sizeOut)`

Creates a GIL table with **`sizeIn`** inputs and **`sizeOut`** outputs

`CS_GILTable(sizeIn, sizeOut, filename, scale, separator, noHeader)`

Creates a GIL table from a text file; **`filename`** is the name of the file containing the points (typically a `.CSV` file). This file must be in ASCII format, with one data point per row. Each row must contain **`sizeIn`** + **`sizeOut`** columns. The columns must be separated by a character specified by the argument **`separator`**. The default value of **`separator`** is **`,`**. **`scale`** is an optional argument that scales all the output values. The default value, used if the optional argument is not specified, is 1.0. **`noHeader`** is a boolean, optional argument that should be **`true`** if there is no first row with labels.

Example file:

```
Velocity, Deflection, Force
0.,0.,10.0
100.,0.,200.0
...
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

AddInterpolationPoint(values)

Adds an interpolation point to the General Interpolation Table. **values** is a one dimensional array of size **sizeIn**+**sizeOut**. The first **sizeIn** values in array **values** corresponds to the values of the input variables. The following **sizeOut** values in array **values** correspond to the output values.

Example 1: Creation of a Nonlinear Stiffness Value That Depends on Spin Velocity (Omega) and on Deflection (dY)

```
VarForceY = CS_Variable();
#
# Variable 0: spin
VarForceY.AddInputMeasure(SpinMeasure )
#
# Variable 1: Y displacement
VarForceY.AddInputMeasure( TransY )
#
# Create table with 2 input and 1 output
EvalY = CS_GILTable(2,1)

Omega = -1.0
dY = -1e-4
stiff = -9.0

values=System.Array.CreateInstance(float,3)

values[0] = Omega
values[1] = dY
values[2] = stiff
EvalY.AddInterpolationPoint( values )

Omega = 11.0
dY = -1e-4
stiff = -21.0
values[0] = Omega
values[1] = dY
values[2] = stiff
EvalY.AddInterpolationPoint( values )
…
```

AddInterpolationPointArray(values)

Adds a set of points to the General Interpolation Table. **values** is a two dimensional array of size (**numberOfPoints**, **sizeIn**+**sizeOut**). On each row of the array, first **sizeIn** values in array **values** corresponds to the values of the input variables. The following **sizeOut** values in array **values** correspond to the output values. Each row contains a single interpolation point in the cloud of points.

Example 2: Creation of a Nonlinear Force Value (F) That Depends on Deflection (dX)

```
ForceVariable = CS_Variable()
ForceVariable.AddInputMeasure( TransX )
Evaluator = CS_GILTable( 1,1 )
values = System.Array.CreateInstance( float, 6, 2 )
dX = 0.0
F = 0.0
values[0,0] = dX
values[0,1] = F

dX = 10.0
F = 1.0
values[1,0] = dX
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
values[1,1] = F

dX = 30.0
F = 2.0
values[2,0] = dX
values[2,1] = F

dX = 60.0
F = 3.0
values[3,0] = dX
values[3,1] = F

dX = 90.0
F = 4.0
values[4,0] = dX
values[4,1] = F

dX = 130.0
F = 5.0
values[5,0] = dX
values[5,1] = F
Evaluator.AddInterpolationPointArray( values )
```

`SetVerbosity(bVerbose)`

If **`bVerbose`** is set to true, the GILTable will print the output value every time it is evaluated. This can be used for debugging purposes, but it will affect the performance if used on a table in a long simulation.

Limitations:

These tables can only be used to apply forces and moments, not for other joint conditions or remote displacements.

Joint

ID table: **`CS_Joint`**

Constants: For the joint type (**`E_JointType`**):

`E_2DSlotJoint`, **`E_BushingJoint`**, **`E_CylindricalJoint`**, **`E_GeneralJoint`**, **`E_FixedJoint`**, **`E_FreeJoint`**, **`E_PlanarJoint`**, **`E_PointOnCurveJoint`**, **`E_RevoluteJoint`**, **`E_ScrewJoint`**, **`E_SingleRotationGeneralJoint`**, **`E_SlotJoint`**, **`E_SphericalJoint`**, **`E_TranslationalJoint`**, **`E_TwoRotationGeneralJoint`**, **`E_UniversalJoint`**,

Members:

`Name`

Name of the joint

`ReferenceCoordinateSystem`

Joint reference coordinate system

Example:

```
J1 = CS_Joint.Find(_jid)
CSR = J1.ReferenceCoordinateSystem
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

MovingCoordinateSystem

Joint moving coordinate system

Example:

```
J1 = CS_Joint.Find(_jid)
CSM = J1. MovingCoordinateSystem
```

Type

Joint type

IsRevert

The internal representation of the joint can use flipped reference and mobile coordinate systems. In that case, all the joint results (for example, forces, moments, rotation, velocities and acceleration) must be multiplied by -1 to go from their internal representation to the user representation. As transient values of joint measures are giving the internal representation, use this **IsRevert** information to know if results should be negated.

AccelerationFromVelocitiesDerivatives

When extracting joint degrees of freedom on joints that return true, accelerations should be done using the time derivatives of the joint velocity measure. On joints that return false, joint DOF derivatives should be extracted using the joint acceleration measure. It is important to check this flag first. Use of the wrong method to query joint acceleration can result in failure or incorrect results.

Example:

```
if Universal.AccelerationFromVelocitiesDerivatives:
 UniversalAccelerationValues=UniversalVelocityM.FillDerivativesThroughTime()
else:
 UniversalAcceleration = Universal.GetAcceleration()
 UniversalAccelerationValues=UniversalAcceleration.FillValuesThroughTime()
```

Stops

Returns the list of the stops defined on the joint.

Member Functions:

GetVelocity()

Returns the joint velocity measure. The size of this measure is the number of degrees of freedom of the joint. The derivatives of this measure give access to the joint accelerations.

GetRotation()

Returns the joint rotation measure. The type of measure depends on the joint number of rotational degrees of freedom (**E_1DRotationMeasure**, **E_3DRotationMeasure**, **E_UniversalAngles**). These rotations components are relative to the reference coordinate system of the joint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

GetTranslation()

Returns the joint translation measure. The length of this measure is the number of translational degrees of freedom of the joint. The translation components are expressed in the reference coordinate system of the joint.

GetForce()

Returns the joint force measure. The length of this measure is always 6 (3 forces components, 3 torque component). This force measure is the total force/moment, including constraint forces/moment, external forces/moment applied to the joint, and joint internal forces/moment, such as elastic moment in a revolute joint that has a stiffness on the Z rotation axis. The force measure components are expressed in the global coordinate system. Note that the sign convention is different from the sign convention used in the Joint Probes in Mechanical.

GetAcceleration()

Returns the joint acceleration measures on the joints that are constraint equations based. See the **AccelerationFromVelocitiesDerivatives** member to see when this function should be used.

Example:

```
J1 = CS_Joint.Find(_jid)
jointRotation = J1.GetRotation()
jointVelocity = J1.GetVelocity()
jointAcceleration = J1.GetAcceleration()
jointForce = J1.GetForce()
```

SetFrictionVariable(var)

Replaces the constant value already given to the friction coefficient with the expression given by **var**.

Example:

```
Joint = CS_Joint.Find(_jid)
Var = CS_Variable()
u0 = 0.1
u1 = 0.2
alpha = 0.5
Var.SetFunc('u0+u1exp(-alpha*time)',0)
Var.AddInputMeasure(Joint.GetVelocity())
Joint.SetFrictionVariable(Var)
```

The command has no effect if no value for the friction coefficient has been provided in the UI. For more information, see Joint Friction (p. 1335)

SetFrictionTolerance(tol)

Sets the friction tolerance.

Example:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
Joint = CS_Joint.Find(_jid)
Joint = Joint.SetFrictionTolerance(1e-4)
```

Derived Classes:

On **SphericalJoint, SlotJoint, BushingJoint, FreeJoint, General-Joint**.

Member Function

AddStop(angle_max, restitution_factor)

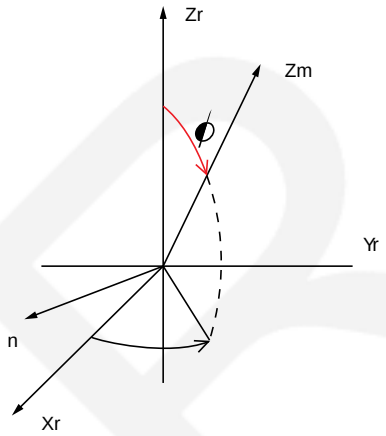
Adds a spherical stop to a joint that has three rotations. A spherical stop constrains the motion of the X and Y rotational degrees of freedom, to give to the joint the behavior of a loose revolute joint, with a rotational gap. This will allow easier handling of over-constrained systems and building higher fidelity models without having to use contact.

angle_max

The angle between the reference coordinate system **Zr** axis and the moving coordinate system **Zm**. **Zr** is the natural **revolute** axis.

restitution_factor

The restitution factor, similar to other joint stops (p. 1360).



On CylindricalJoint:

ReplaceByScrew(pitch)

Creates a relation between the translational and the rotational degrees of freedom of a cylindrical joint.

Note:

- The pitch is in the current length unit. Any stop and/or lock defined on the original cylindrical joint is not transferred to the screw joint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Similarly, any constraint equation defined on the original cylindrical joint is not converted and so will prevent a proper solution.

- The ReplaceByScrew command is deprecated. It is replaced by the Screw Joint (p. 1307) provided in the Mechanical UI.

On Bushing Joint:

`GetBushingAngles()`

Returns the measure of the joint angles. This measure is used to compute the forces and torques developed in the joint. Note that this is only available for post-processing operations, as the measure does not exist before the solve has been performed.

Creating New Joints:

The following joint can be created by commands:

```
CS_GeneralJoint(from, to, FreeX, FreeY, FreeZ, FreeRX,
FreeRY, FreeRZ)
```

Where **`from`** and **`to`** are of type **`CS_BodyCoordinateSystem`** and **`Free*`** are integers where 0 is no available motion and nonzero is available motion. Selecting two free rotations is not allowed.

JointDOFLoad

JointDOFLoads are loads applied on a given degree of freedom of a joint. The load is applied in the joint reference coordinate system.

JointDOFLoad derives from Load (p. 583).

Constructor:

`CS_JointDOFLoad(joint,dof)`

`joint`

A joint object

`dof`

An integer that defines the joint degree of freedom to be included in the term. The ordering of the degrees of freedom sets the translation degrees of freedom first. The degrees of freedom numbering is zero based. For example, in a slot joint, the translational degree of freedom is 0, while the third rotational degree of freedom is 3.

Members:

None

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Member functions:

None

Load

Loads derive from the Actuator (p. 563) class. They are derived from various types of loads, such as the **CS_JointDOFLoad**.

Corresponding ID table: **CS_Actuator**

Members:

None

Members Functions:

None

Measure:

Most useful measures are pre-existing in the Rigid Dynamics model, and can be accessed using other object "get" functions. Additional measures can be created before solving for use in custom post-processing or as input values for joint conditions. For example, measures can be created to express conditions. In this case, the measure must be added to the system to be computed at each time step (see component measure example below).

ID table: **CS_Measure**

Constants: For the measure type (**E_MeasureType**):

E_1DRotationJoint, **E_3DRotationBody**, **E_3DRotationJoint**, **E_Acceleration**, **E_ActuatorStatus**, **E_ActuatorEnergy**, **E_AnsysJointForceAndTorque**, **E_AXPY**, **E_BodyAcceleration**, **E_BodyIntertialBCSQuaternion**, **E_BodyRotation**, **E_BodyTranslation**, **E_CenterOfGravity**, **E_Component**, **E_Constant**, **E_Contact**, **E_ContactForce**, **E_ContactVelocity**, **E_Counter**, **E_Displacement**, **E_Distance**, **E_DistanceDot**, **E_Divides**, **E_EigenValue**, **E_DOFSensitivity**, **E_Dot**, **E_ElasticEnergy**, **E_Energy**, **E_EulerAngles**, **E_ForceMagnitude**, **E_Forces**, **E_IntegratedOmega**, **E_JointAcceleration**, **E_JointDOFFrictionCone**, **E_JointDriverForce**, **E_JointForce**, **E_JointMBDVelocity**, **E_JointNormalForce**, **E_JointTranslation**, **E_JointRotation**, **E_JointVelocity**, **E_KineticEnergy**, **E_MassMomentsOfInertia**, **E_MeasureDotInDirectionOfLoad**, **E_Minus**, **E_Modulus**, **E_Multiplies**, **E_Norm**, **E_Omega**, **E_OmegaDot**, **E_OutputContactForce**, **E_Plus**, **E_PointOnCurveGeometryMeasure**, **E_PointOnCurveJointSigmaMeasure**, **E_PointToPointRotation**, **E_PointToPointRotationDot**, **E_Position**, **E_PotentialEnergy**, **E_RadialGap**, **E_ReferenceEnergy**, **E_RelativeAcceleration**, **E_RelativePosition**, **E_RelativeVelocity**, **E_RotationalRelativeDOF**, **E_RotationMatrix**, **E_SphericalStop**, **E_StopVelocity**, **E_StopStatus**, **E_Time**, **E_TimeStep**, **E_TranslationalJoint**, **E_UniversalAngles**, **E_UnknownType**, **E_User**, **E_Velocity**, **E_Violation**, **E_XYZAnsysRotationAngles**, **E_ZYXRotationAngles**,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Members:

Length:

Number of components of the measure

Example:

```
nbValues = Measure.Length
```

Type:

Measure type

Calculation Method:

A measure can use direct calculation or be time integrated. On a measure that uses direct calculation, it is possible to retrieve the measure value through time. On a measure that is time-integrated, both values and time derivatives can be retrieved.

Name:

Measure Name

Member Functions:

`FillValuesThroughTime()`

Returns a two dimensional array. This function is to be called after the solution has been performed. The first dimension of the returned array is the number of time values in the transient. The second dimension is the size of the measure plus one. The first column contains the time values, while the subsequent columns contain the corresponding measure values.

Example:

```
jointRotation = J1.GetRotation()
jointVelocity = J1.GetVelocity()
jointAcceleration = J1.GetAcceleration()
jointForce = J1.GetForce()

jointRotationValues =jointRotation.FillValuesThroughTime()
jointVelocityValues =jointVelocity.FillValuesThroughTime()
jointAccelerationValues =jointAcceleration.FillValuesThroughTime()
jointForceValues =jointForce.FillValuesThroughTime()

nbValues = jointRotationValues.GetLength(0)

print jointRotation.Id

print '  Time     Rotation Velocity Acceleration'
for i in range(0,nbValues):
    print jointRotationValues[i,0],jointRotationValues[i,1],jointVelocityValues[i,1],jointAccelerat

fich.close()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

`FillDerivativesThroughTime()`

Returns a two dimensional array. This function is to be called after the solution has been performed. The first dimension of the returned array is the number of time values in the transient. The second dimension is the size of the measure plus one: the first column contains the time values, while the subsequent columns contain the corresponding measure derivatives. These derivatives are available on measures that are time integrated. To know if a measure is time integrated, use the **`CalculationMethod`** member.

Derived Classes:

`CS_JointVelocityMeasure`

Both translational and rotational joint velocities are expressed in the joint reference coordinate system. The number of components is the number of translational degrees of freedom plus the number of rotational degrees of freedom. For example, the size of the joint velocity measure for a revolute joint is 1. It contains the relative joint rotation velocity along the z axis of the joint reference coordinate system. The size of the measure for a slot joint is 4: one component for the relative translational velocity, and the 3 components of the relative rotational velocity. The joint velocity measure can be obtained from the joint using the **`GetVelocity`** function. Rotational velocities are expressed in radians/second.

`CS_JointAccelerationMeasure`

Both translational and rotational joint accelerations are expressed in the joint reference coordinate system. The number of components is the number of translational degrees of freedom plus the number of rotational degrees of freedom. The joint acceleration measure can be obtained from the joint using the **`GetAcceleration`** function.

`CS_JointRotationMeasure`

- For revolute joints, cylindrical joints, or single rotation general joints, this measure has only one component: the relative angle between the reference and the moving coordinate system of the joint. Rotations are expressed in radians.
- For slots, spherical joints, bushing joints, and 3 rotation vectors, this measure contains values that are not directly usable.
- For universal joints, this measure contains the two joint axis rotational velocities. (The first one along the X axis of the reference coordinate system and the second along the Z axis of the moving coordinate system.) These angles are expressed in radians.

`CS_JointTranslationMeasure`

This measure contains only the joint relative translations, expressed in the joint reference coordinate system. The joint translation measure can be obtained from the joint using the **`GetTranslation`**function.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

CS_JointForceMeasure

This measure contains the total forces and moment that develop in the joint. This includes constraint forces, elastic forces, and external forces. The joint velocity measure can be obtained from the joint using the **GetForce**function.

CS_PositionMeasure

This measure allows for tracking of the position of a Body Coordinate System over time.

Example:

```
CoMBCS = OneBody.InertiaBodyCoordinateSystem
Pos = CS_PositionMeasure(CoMBCS)
Env=CS_Environment.FindFirstNonNull()
Sys = Env.System
Sys.AddMeasure(Pos)
```

CS_ComponentMeasure

This measure allows the extraction of one component of an existing measure. This component can be expressed in a non default coordinate system. A component of -2 will compute the norm 2 of the vector of values of the measure.

Example:

```
Planar = CS_Joint.Find(_jid)
Vel = Planar.GetVelocity()
Xglobaldirection = 0
VelX = CS_ComponentMeasure(Vel,Xglobaldirection)
Sys.AddMeasure(VelX)
```

CS_AXPYMeasure

This measure allows a linear transformation from another measure with a scaling factor and an offset. This can be useful to transform an internal rotation measure that is expressed in radians to a measure in degrees used as an input to a load calculation, for example.

Example:

```
Revolute = CS_Joint.Find(_jid)
Rot = Revolute.GetRotation()
RotInDegrees = CS_AXPYMeasure( Rot, 180.0/math.pi, 0. )
Sys.AddMeasure(RotInDegrees)
```

CS_ModulusMeasure

This measure allows you to compute the floating point remainder of value/modulus.

Example:

```
Revolute = CS_Joint.Find(_jid)
Rot = Revolute.GetRotation()
Rot02pi = CS_ModulusMeasure( Rot, 2.0*math.pi )
Sys.AddMeasure(Rot02pi)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

`CS_OutputContactForceMeasure`

This measure contains four 3D vectors:

- Values 0 to 2 are the total contact force components between the two bodies, including the normal and tangential contributions.
- Values 3 to 5 are the coordinates of the point where the interaction between the two bodies is reduced to a force; in other words, the total torque is zero.
- Values 6 to 8 are the frictional force between the two bodies.
- Values 9 to 11 are frictional moment components at the reduction point.

MSolverDB

Solver database. The database is both the input and the results file to the solver. It can be used to solve outside the Mechanical session (for example, for co-simulation purposes) or to restart from a previous run.

Members:

`SetFileName(FileName)`

Set the database file name.

`SetDirectoryName(DirectoryName)`

Set the database directory.

`ReadDatabase()`

Read the content of the database.

`WriteDatabase()`

Write the current database to a file.

`DeleteDatabase()`

Delete the database.

`CloseDatabase()`

Close the database file.

`OpenDatabase()`

Open the database file and reads the database content table.

`Dispose()`

Clear the content of the database and free memory used by the database.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

PointsTable

Corresponding ID table: **`CS_PointsTable`**

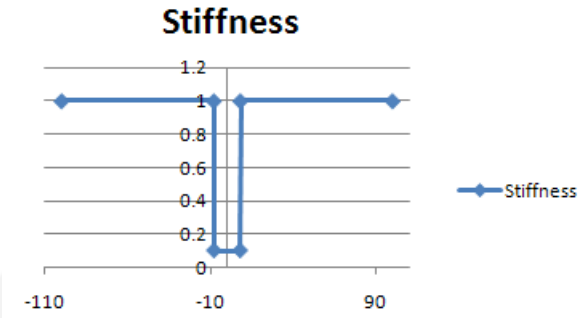
Members Functions:

`CS_PointsTable( tab )`

`tab` is a two dimensional array where the first column contains the input values and the second column contains the corresponding output values.

Example:

```
tab = System.Array.CreateInstance(float,6,2)
tab[0,0]=-100.
tab[1,0]=-8.
tab[2,0]=-7.9
tab[3,0]= 7.9
tab[4,0]= 8.
tab[5,0]= 100.

tab[0,1]=1.0
tab[1,1]=1.0
tab[2,1]=0.1
tab[3,1]=0.1
tab[4,1]=1.0
tab[5,1]=1.0
Table = CS_PointsTable(tab);
```



Here, the output (shown as Stiffness in the chart above) varies in a linear, piece-wise manner. For values of input less than -8.0 or greater than 8.0, the output is equal to 1.0. For values between -7.9 and +7.9, the output is 0.1. The transition is linear between -8.0 and -7.9, and as well between +7.9 and +8.0.

PolynomialTable

Corresponding ID: **CS_PolynomialTable**

Create a polynomial relation between **`sizeIn`** inputs and **`sizeOut`** outputs using the following function:

$$y_j = c_j^0 + \sum_k c_j^k x_1^{\alpha_1^k} \cdots x_i^{\alpha_i^k} \cdots x_{sizeIn}^{\alpha_{sizeIn}^k}$$

Where i denotes the index of input and goes from 1 to n (**`sizeIn`**), j denotes the index of output (from 1 to **`sizeOut`**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Member Functions:

`CS_PolynomialTable()`

Creates an empty polynomial table.

`Initialize(constant)`

Specialized for 1x1 table. Initializes the table to be a 1 input, 1 output table, and sets the constant term (constant is a float value).

`Initialize(sizeIn,sizeOut,constantValues)`

(generic version) Initializes the table with **`sizeIn`** inputs and **`sizeOut`** outputs and sets the constant terms. **`sizeIn`** and **`sizeOut`** are two integer values, and **`constantValues`** is an array of **`sizeOut`** float values.

`AddTerm(coefficient,order)`

Specialized for 1x1 table. Adds one monomial term to the table. The coefficient is a float value and order is an integer value giving the power of the input.

`AddTerm(coefficients,orders)`

(generic version) Adds one monomial term to the table. The coefficients are given by a **`sizeOut`** float array and the power for each input by an array of **`sizeIn`** integers.

Relation

The relation object enables you to write constraint equations between degrees of freedom of the model. For example, two independent lines of shaft can be coupled using a relation between their rotational velocities.

If you have a gear coupling between two shafts where the second shaft rotates twice as fast as the first one, you can write the following equation:

$2.0 \times \Omega 1 + \Omega 2 = 0$

where $\Omega 1$ and $\Omega 2$ are joint rotational velocities.

This relation contains two terms and a constant right hand side equal to zero.

The first term ($2 \times \Omega 1$) can be described using the following information:

- A joint selection
- A joint degree of freedom selection
- The nature of motion that is used in the equation (joint velocities, which is the most common case). For convenience, the nature of motion upon which the constraint equation is formulated is considered as being shared by all the terms in the relation.

This information defines $\Omega 1$

- The factor 2.0 in the equation can be described by a constant variable, whose value is 2.0

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

ID table: **CS_Actuator**

The coefficients of the relation can be constant or variable; however, the use of non-constant coefficients is limited to relations between velocities and relations between accelerations. If non-constant coefficients are used for relations between positions, the solution will not proceed.

Constants:

E_Acceleration, **E_Position**, **E_Velocity**

Members:

None

Member Functions:

SetRelationType(type)

Type of relation, with type selected in the previous enumeration.

AddTerm(joint, dof, variable)

Adds a term to the equation.

joint

A joint object

dof

An integer that defines the joint degree of freedom to be included in the term. The ordering of the degrees of freedom sets the translation degrees of freedom first, and that the degrees of freedom numbering is zero based. For example the translational degrees of freedom in a slot joint is **0**, while the third rotational degree of freedom is **3**.

variable

A variable object

SetVariable(variable)

Sets the right hand side of the relation. "variable" is a variable object.

SolverOptions

The SolverOptions object allows you to customize the behaviour of the RBD solver. The option uses a group of numerical values (real or integer) that can be get or set. When used as a switch, 0 means off and 1 is on.

Corresponding ID table: **CS_SolverOptions**

Constants:

None

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Member Functions:

`VelocityToleranceFactor`

Multiplicative factor used to determine zero velocity tolerance (=100.0 by default);

`ContactRadiusFactor`

Contact radius factor used in contact failsafe mode (=2.0 by default);

`MaximumNumberOfCorrectionAttempts`

Number of external loops for geometric correction (=2 by default));

`FrictionForShock`

Enable friction for shock solve (=0, disabled by default);

`MaximumNumberOfDiagnostics`

Number of diagnostics messages given in Mechanical UI (=10 by default);

`InactiveTouchingInDynamics`

Prevent inactive contact pair from being violated (=1, enabled by default);

`DisablePolygonEvent`

Disable polygon event for contact (=0, active by default);

`PrintDynamicSystem`

Print the dynamics system (=0 by default);

`PurgeGST`

Purge GST file every n steps (=0, never by default);

`PrintErrorEstimation`

Force output of error estimation (=0, disabled by default);

`ExportXLSFileForCMS`

Export generalized coordinates for CMS bodies in a CSV file (=0, disabled by default)

`HandlePOCTransitionsWithEnergyMinimization`

When point on curve joints are used, different solutions (depending on the topology) may be found when crossing curve connections. Furthermore, these solutions do not guarantee the conservation of the kinetic energy at the transition. To remedy this issue, this option makes the transitions using a method that minimizes the kinetic energy in a way similar to the assembly process using the inertia matrix (p. 644). This solution works well for explicit time integration schemes, but it is not guaranteed for implicit ones. (=0, disabled by default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example:

```
sOpts=CS_SolverOptions()
sOpts.ExportXLSFileForCMS=1
```

Spring

Corresponding ID table: **`CS_Actuator`**

Members:

None

Member Functions:

`ToggleCompressionOnly()`

Calling this function on a translational spring will make the spring develop elastic forces only if its length is less than the spring free length. The free length has to be defined in the regular spring properties.

`ToggleTensionOnly()`

Calling this function on a translational spring will make the spring develop elastic forces only if its length is greater than the free length of spring. The free length has to be defined in the regular spring properties.

`SetLinearSpringProperties(system, stiffness, freeLength)`

Enables you to overwrite the stiffness and free length of a translational spring. This can be useful to parameterize these properties. For example, system is the system object, stiffness and free length are the double precision values of stiffness and free length.

`SetNonLinearSpringProperties(table_id)`

Enables you to replace the constant stiffness of a spring with a table of ID **`table_id`** that gives the force as a function of the elongation of the spring. The table gives the relation between the force and the relative position of the two ends.

`GetDamper()`

The user interface has stiffness and damping properties of the spring. Internally, the **Spring** is made of two objects; a spring and a damper. This function enables you to access the internal damper using the **Spring** object in the GUI.

Derived Classes:

None

System

Corresponding ID table: **`CS_System`**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Members:

`Bodies`

Gets the list of bodies.

`Joints`

Gets the list of joints.

Member Functions:

`AddBody(body)`

Adds a body to the system.

`AddJoint(joint)`

Adds a joint to the system.

`PrintTopology()`

Prints the topology of the systems (parent/child relation).

`AddMeasure(measure)`

Adds a measure to the system, to be calculated during the simulation. This function must be called prior to solving so that the measure values through time can be retrieved.

`(istat,found,measure)=FindOrCreateInternalMeasure( MeasureType)`

Extracts an existing global measure on the system. Supported measure types are: E_Energy, E_PotentialEnergy, E_ElasticEnergy, E_KineticEnergy, and E_Time.

Derived Classes:

None

Table

A table is the base class for Points Tables, Polynomial Tables, User Tables, and GIL Tables.

ID table: **`CS_Table`**

Members:

None

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Member Functions:

`Evaluate(In, Out)`

Allows evaluating a table in Python. In and Out are arrays of float, with sizes corresponding to the table input and output sizes. This function can be called from a user table for example.

`Dispose()`

Explicit destruction of the table. This explicit destructor should be used only when the table hasn't been assigned to an actuator. When the table is assigned to an actuator, the actuator is calling this destructor. Omitting to call this destructor can cause the evaluation of the results to fail.

UserTable

A user table is a function with *i* input values and *o* output values, with an evaluator that is defined in IronPython, allowing complex variation, or even evaluation performed outside the solver.

Example:

```
LeftVarCoefX = CS_Variable();

# input 0,1,2 of the variable
LeftVarCoefX.AddInputMeasure( LeftRelTrans )

# input 3 to 8 of the variable
LeftVarCoefX.AddInputMeasure( LeftRelVelo )

class XForceTable(CS_UserTable):
    def __init__(self,sizeIn,sizeOut):
        CS_UserTable.__init__(self,sizeIn,sizeOut)

    def Evaluate(self,In,Out):
        TX = In[0]
        VX = In[3]

        Force = 1000.0*TX
        Out[0] = Force
        print 'ForceX = {0:e}'.format(Out[0])
        return 0

LeftForceTableX = XForceTable( 9, 1 )
LeftVarCoefX.SetTable( LeftForceTableX )
```

Variable

A variable is an n-dimensional vector quantity that varies over time. It is used to define the variation of a load or a joint condition, or to express the coefficients in a relation between degrees of freedom. For convenience, the solver allows the creation of constant variables, where only the value of the constant has to be provided. More complex variables can be built using a function variable. A function variable is a function of input, where input is given by a measure (p. 583) and function is described by a table. In some cases, you are able to replace the table or the measure of an internal variable as used in a joint condition.

ID table: **`CS_Variable`**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Members:

None

Member Functions:

`SetConstantValues(value)`

`value` is an array, whose size is equal to the size of the table. To create a constant scalar variable, the value can be defined as shown in the following example:

```
value = System.Array[float]([1.0])
```

`System`, **`Array`**, and **`float`** are part of the **Python** language. The result of this is an array of size one, containing the value 1.0.

`AddInputMeasure(measure)`

`measure` is a measure object. The same variable can have more than one measure. The input variable of the variable is formed by the values of the input measure in the order that they have been added to the list of input measures.

`SetTable(table)`

`table` is a CS_PointsTable.

`SetFunc(string, is_degree)`

`string` is similar to the expression used in the user interface to define a joint condition by a function. Note that the literal variable is always called "time", even if you are using another measure as input. "is_degree" is a boolean argument. If the expression uses a trigonometric function, it specifies that the input variable should be expressed in degrees.

Note:

Variables cannot be shared by different actuators.

Derived Classes:

ConstantVariable

Command Use Examples

The following command use examples are included in this section:

Constraint Equation

Joint Condition: Initial Velocity

Joint Condition: Control Using Linear Feedback

Non-Linear Spring Damper

Spherical Stop

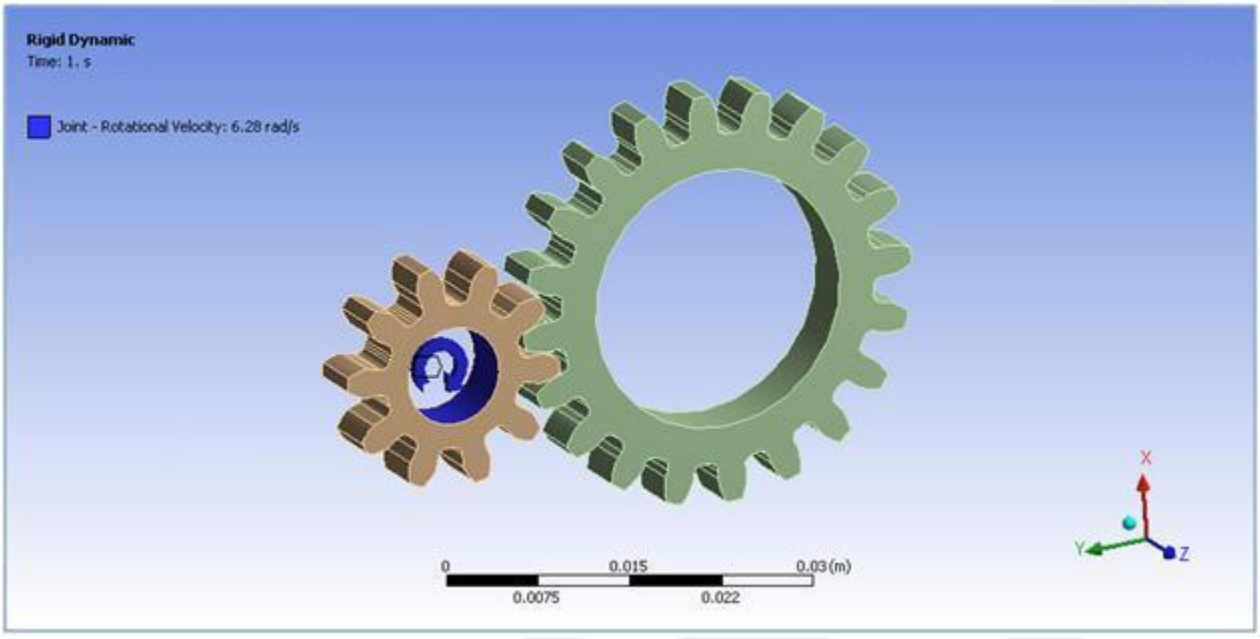
Export of Joint Forces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Breakable Joint

Constraint Equation

This example considers the gear mechanism shown below.



A relation is created between two revolute joints to simulate a gear with a ratio 2 M. Commands are used to enforce the ratio of velocities between the two wheels, and create a linear relation between rotational velocities, defined by:

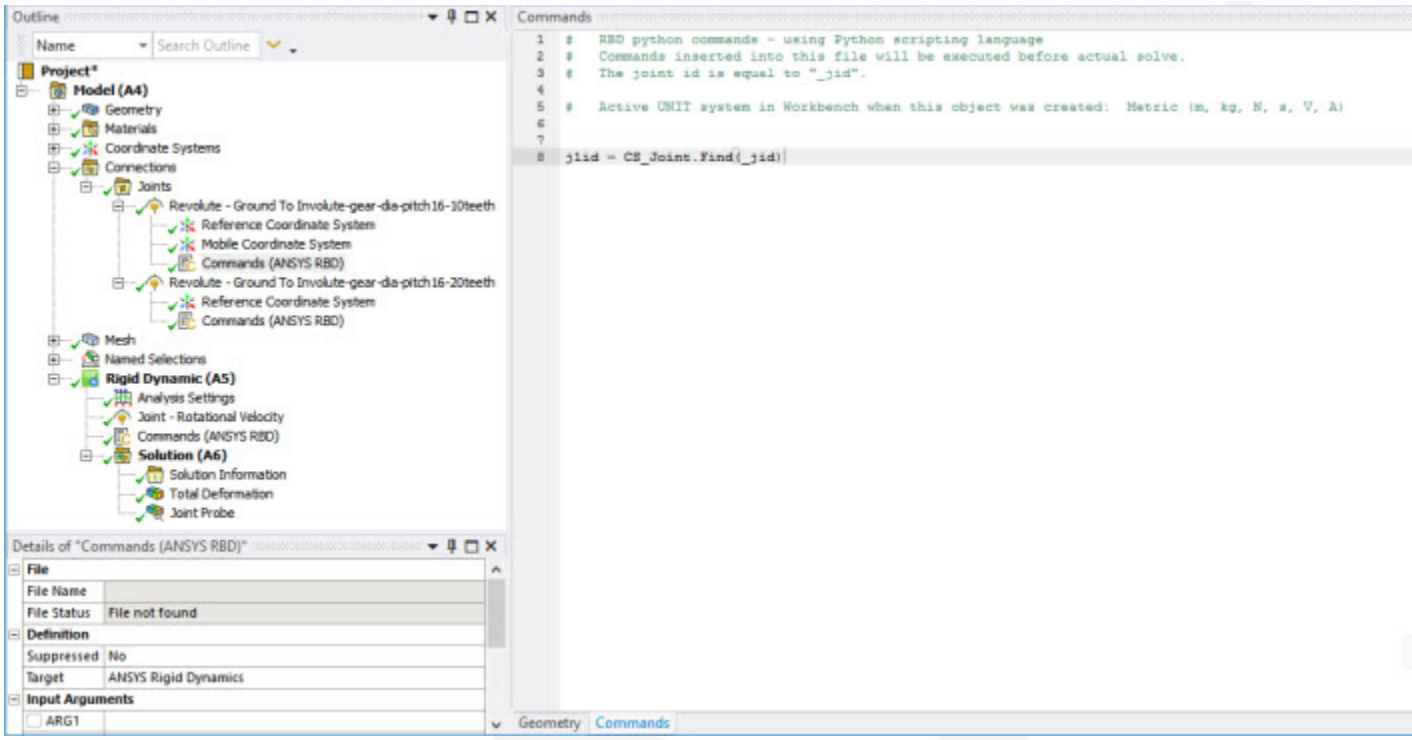
$(1)*\omega_1 + (-2)*\omega_2 = 0$

First, the joint objects are retrieved using their IDs:

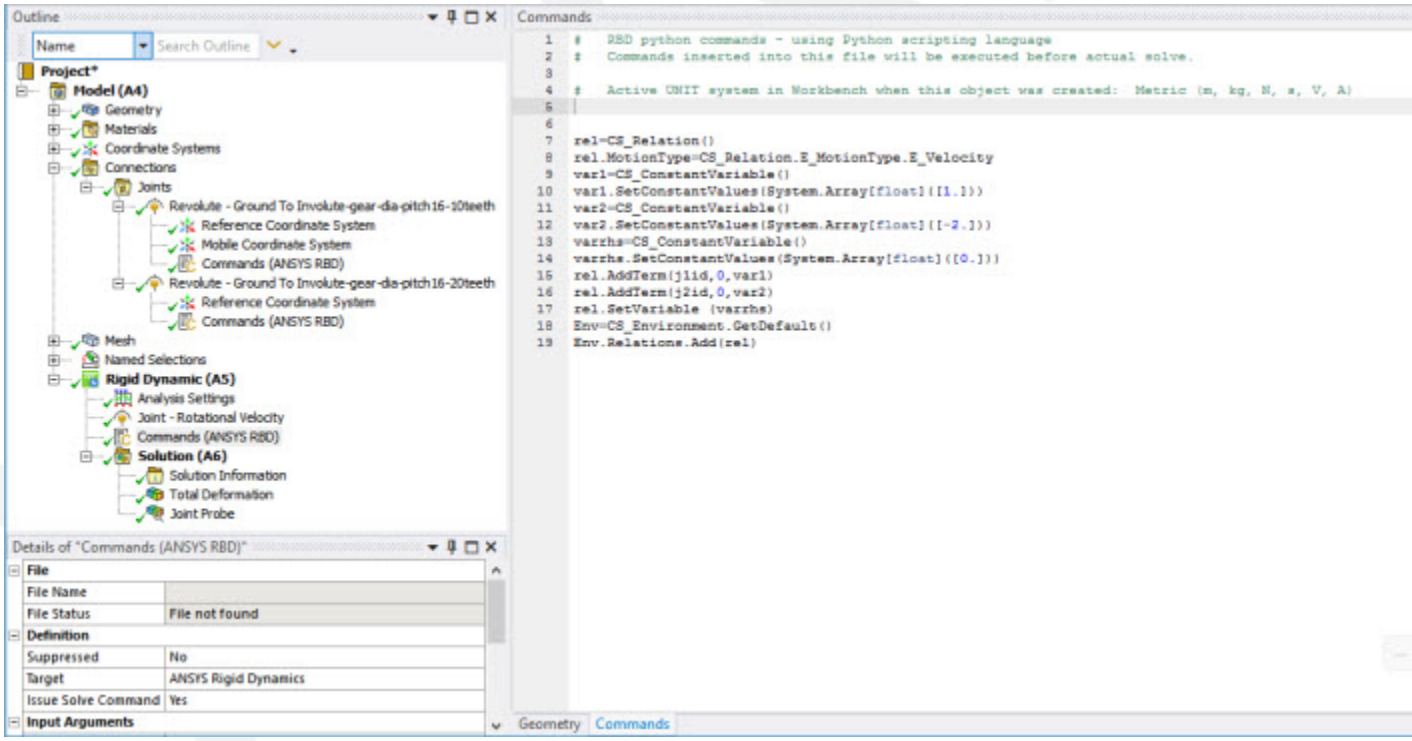
```
j1id = CS_Joint.Find(_jid)
j2id = CS_Joint.Find(_jid)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Next, the relationship between the two wheels is defined. The complete list of commands is shown below. A description of these commands follows.



1. A relation object is created and specified as a relation between velocities:

```
rel=CS_Relation()
rel.MotionType=CS_Relation.E_MotionType.E_Velocity
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. The constant coefficients that appear in the relation are created. The first constant term is created by:

```
var1=CS_ConstantVariable()
var1.SetConstantValues(System.Array[float]([1.]))
```

3. The second coefficient and constant right hand side are created by:

```
var2=CS_ConstantVariable()
var2.SetConstantValues(System.Array[float]([-2.]))
varrhs=CS_ConstantVariable()
varrhs.SetConstantValues(System.Array[float]([0.]))
```

4. The first term of relation (1) X ω_1 is added to the relation object:

```
rel.AddTerm(j1id,0,var1)
```

   The first argument is the joint object. The second argument defines the DOF (degrees of freedom) of the joint that are involved in the relation. Here, **0** represents the rotation, which is the joint's first and only DOF is the rotation.

5. The second term and right hand side are introduced in the same manner:

```
rel.AddTerm(j2id,0,var2)
rel.SetVariable (varrhs)
```

6. The relation is added to the list of relations:

```
Env=CS_Environment.GetDefault()
Env.Relations.Add(rel)
```

Joint Condition: Initial Velocity

This example shows how to impose an initial velocity to a joint. A velocity driver (joint condition) is created using commands and added to the list of initial conditions. During the transient solve, initial conditions are applied only at t=0.

The complete list of commands and their explanation follows.

```
Joint=CS_Joint.Find(_jid)
driver=CS_Driver(Joint,System.Array[int]([0]),CS_Driver.E_MotionType.E_Velocity)
Env=CS_Environment.GetDefault()
Sys=Env.System
(ret,found,time) = Sys.FindOrCreateInternalMeasure(CS_Measure.E_MeasureType.E_Time)
driver.SetInputMeasure(time)
driver.SetConstantValues(System.Array[float]([-4.9033]))
Env.InitialConditions.Add(driver)
```

1. The joint is retrieved using its ID(_jid):

```
Joint=CS_Joint.Find(_jid)
```

2. A velocity driver (imposed velocity) is created on this joint:

```
driver=CS_Driver(Joint,System.Array[int]([0]),CS_Driver.E_MotionType.E_Velocity)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The driver constructor takes the joint instance as the first argument. The second argument is an array of integer that defines which DOFs are active. The physical meaning of these integers is dependent of the joint. For instance, if the underlying joint is a translation joint, **0** is the translation along x. But if the joint is revolute, **0** now is the rotation along z axis. Similarly, for a cylindrical joint, **0** is is the translation along z, and **1** is the rotation. The last argument gives the type of driver here velocity. Drivers can be one of three types: position, velocity, or acceleration:

3. The default environment and corresponding system are retrieved

```
Env=CS_Environment.GetDefault()
Sys=Env.System
```

4. This command returns an instance on an internal measure. It is often used to obtain the instance of the time measure:

```
(ret,found,time) = Sys.FindOrCreateInternalMeasure(CS_Measure.E_MeasureType.E_Time)
```

5. The time measure is specified as the input measure for the driver and a constant value is given to the driver. As the driver may be applied to several components of the joint, the values are given as an array of float:

```
driver.SetInputMeasure(time)
driver.SetConstantValues(System.Array[float]([-4.9033]))
```

6. The driver is added to the list of initial conditions. Consequently, it will be active only at t=0 and will give an initial velocity to the joint:

```
Env.InitialConditions.Add(driver)
```

Joint Condition: Control Using Linear Feedback

In this example, an existing load is modified to apply a torque proportional to the joint velocity. Two Methods are discussed:

Method 1

Obtain the velocity measure from the joint:

```
joint = CS_Joint.Find(_jid)
vel=joint.GetVelocity()
```

Next, modify an existing moment in order to use the velocity measure as its input measure:

```
Env=CS_Environment.FindFirstNonNull()

ids=Env.DSToInternalIds[_jcid]

load=CS_Actuator.Find(ids[0])
load.SetInputMeasure(vel)
```

Method 2

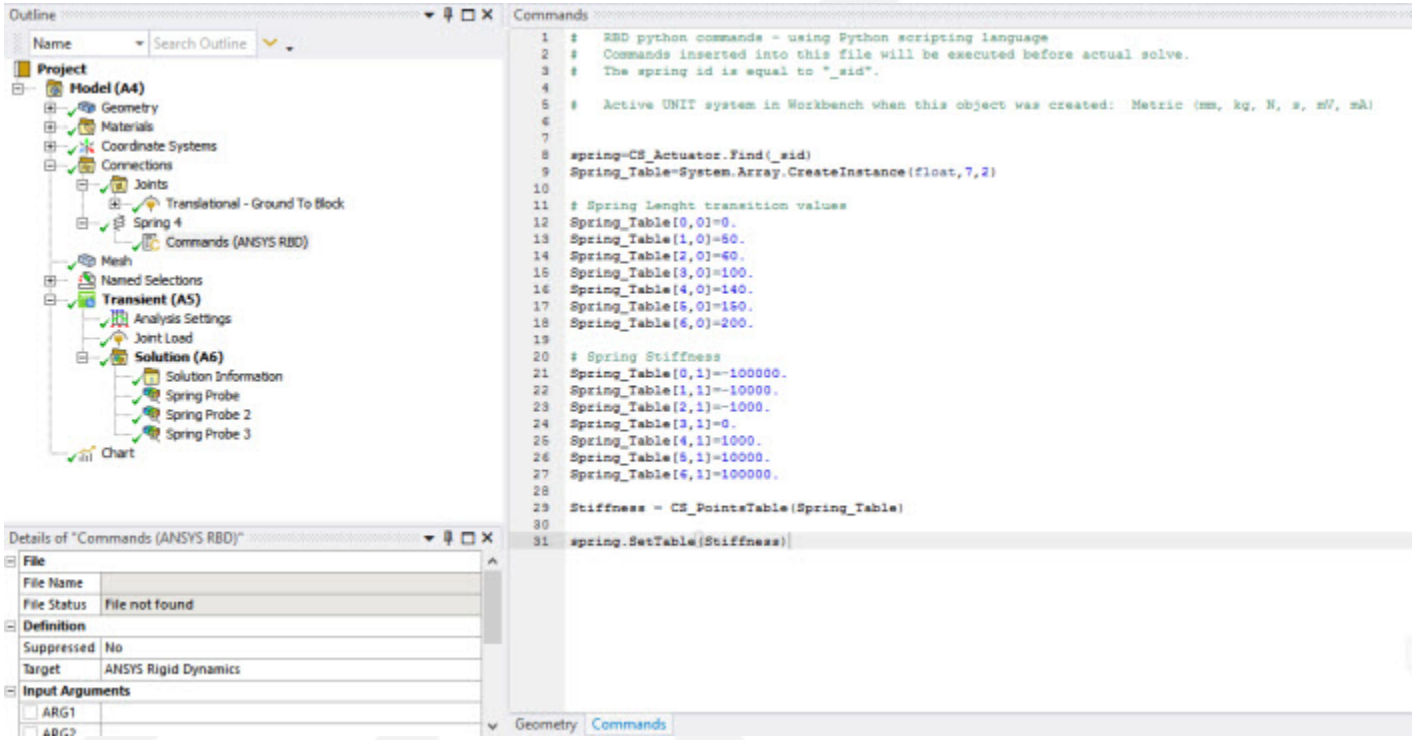
Using this method, the load is created entirely using commands. These commands are shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
Env=CS_Environment.FindFirstNonNull()

load=CS_JointDOFLoad(joint,0)
load.SetInputMeasure(vel)
load.SetFunc('0.1*(-2*acos(-1)-time)',0)

Env.Loads.Add(load)
```

Non-Linear Spring Damper

This example shows how the behavior of a spring can be altered to introduce a non-linear force-displacement relationship. The complete list of commands is shown below. A description of these commands follows.



1. Retrieve the spring object using its ID:

   ```
   Spring=CS_Actuator.Find(_sid)
   ```

2. Create an array of real values and fill it with the pairs of values (elongation, force):

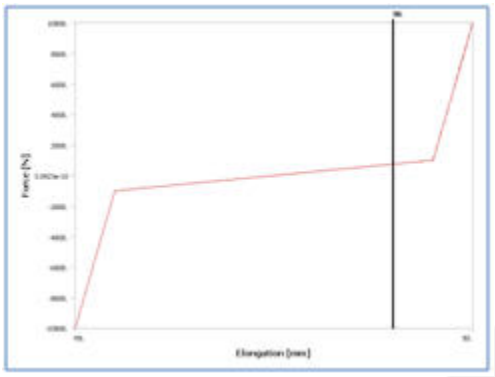
   ```
   Spring_table=System.Array.CreateInstance(float,7,2)

   #
   # LENGTH transition values
   spring_table[0,0]=0.
   spring_table[1,0]=50.
   spring_table[2,0]=60.
   spring_table[3,0]=100.
   spring_table[4,0]=140.
   spring_table[5,0]=150.
   spring_table[6,0]=200.

   #
   # Spring Stiffness
   spring_table[0,1]=-100000.
   spring_table[1,1]=-10000.
   spring_table[2,1]=-1000.
   ```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

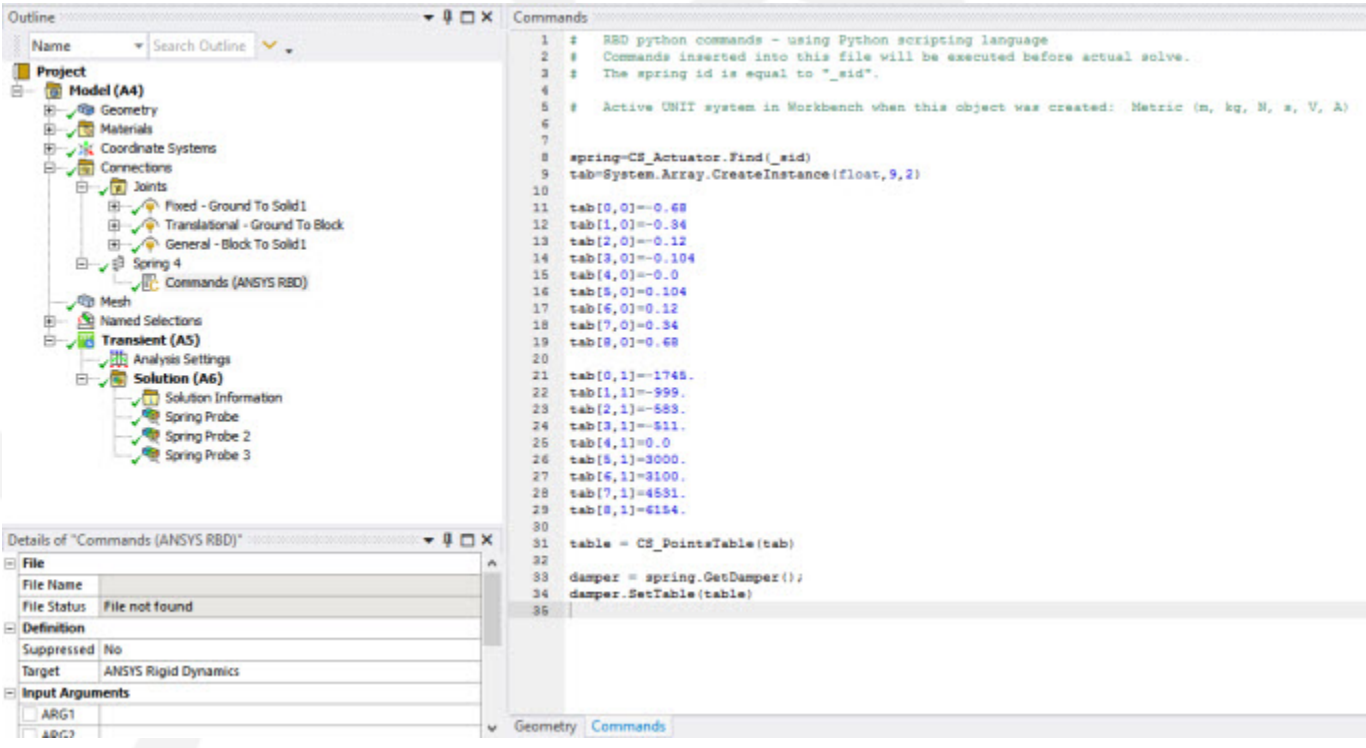
Published: 2023-10-23

```
spring_table[3,1]=0.
spring_table[4,1]=1000.
spring_table[5,1]=10000.
spring_table[6,1]=100000.

stiffness = CS_PointsTable(spring_table)

spring.SetTable(stiffness)
```

In this command, **7** represents the number of rows and **2** for the number of columns. The first column gives elongation and the second, the corresponding force value. This command generates a PointsTable assigned to the spring, as shown below.



Each spring object in the Mechanical GUI is actually a combination of a spring and a damper. The **`GetDamper`** method enables you to retrieve the damper object on a given spring, as shown below.



3. Introduce a table is to define a non-linear force velocity relation:

```
Damper=spring.GetDamper()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Spherical Stop

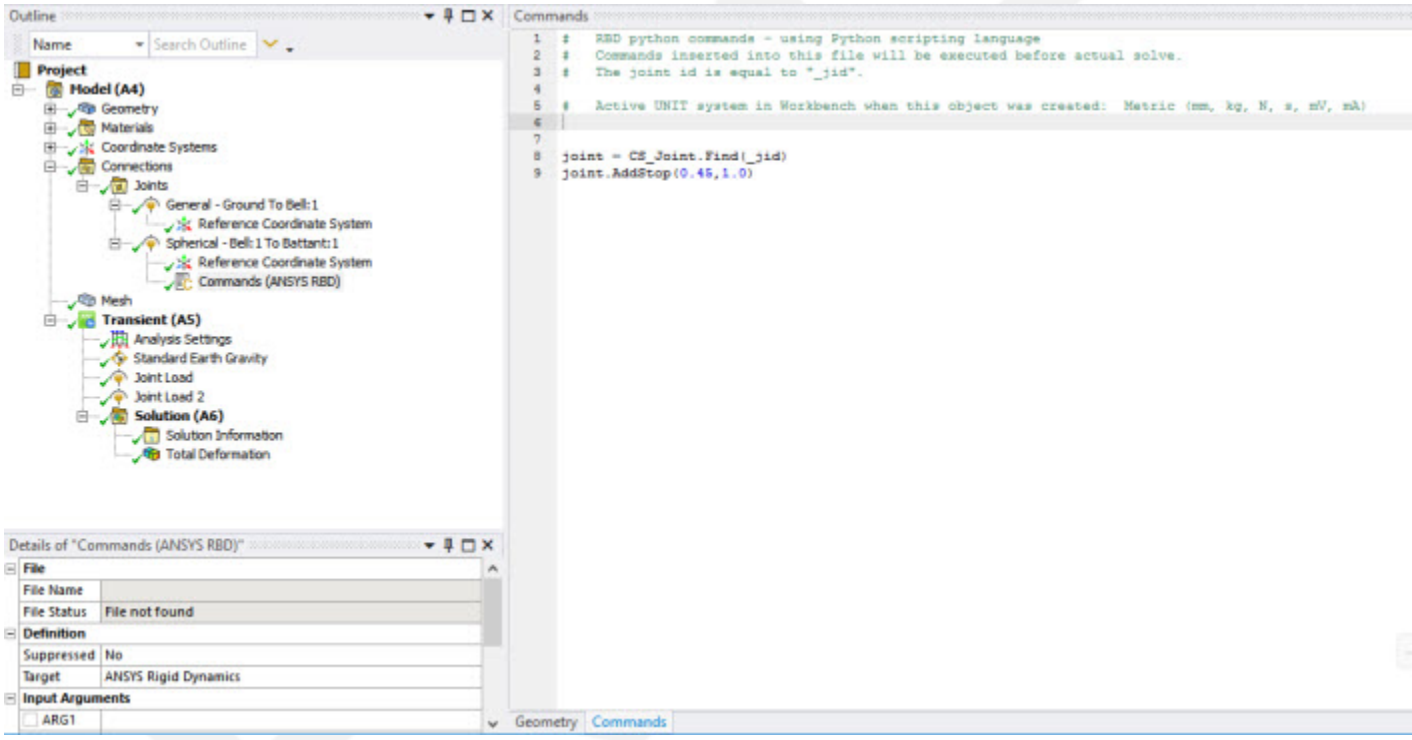
This example describes the implementation of a spherical stop. A spherical stop is a joint that has 3 rotations (joints include spherical, slot, bushing, free and general joints). This specific type of stop creates a limit to the angle between the z-axis of the reference frame and the z-axis of the moving frame. This functionality is available using the following command:

```
AddStop(angle_max, restitution_factor)
```

For example, to add a spherical stop for an angle value equal to 0.45 radians and a restitution factor equal to 1.0, the following command would be issued:

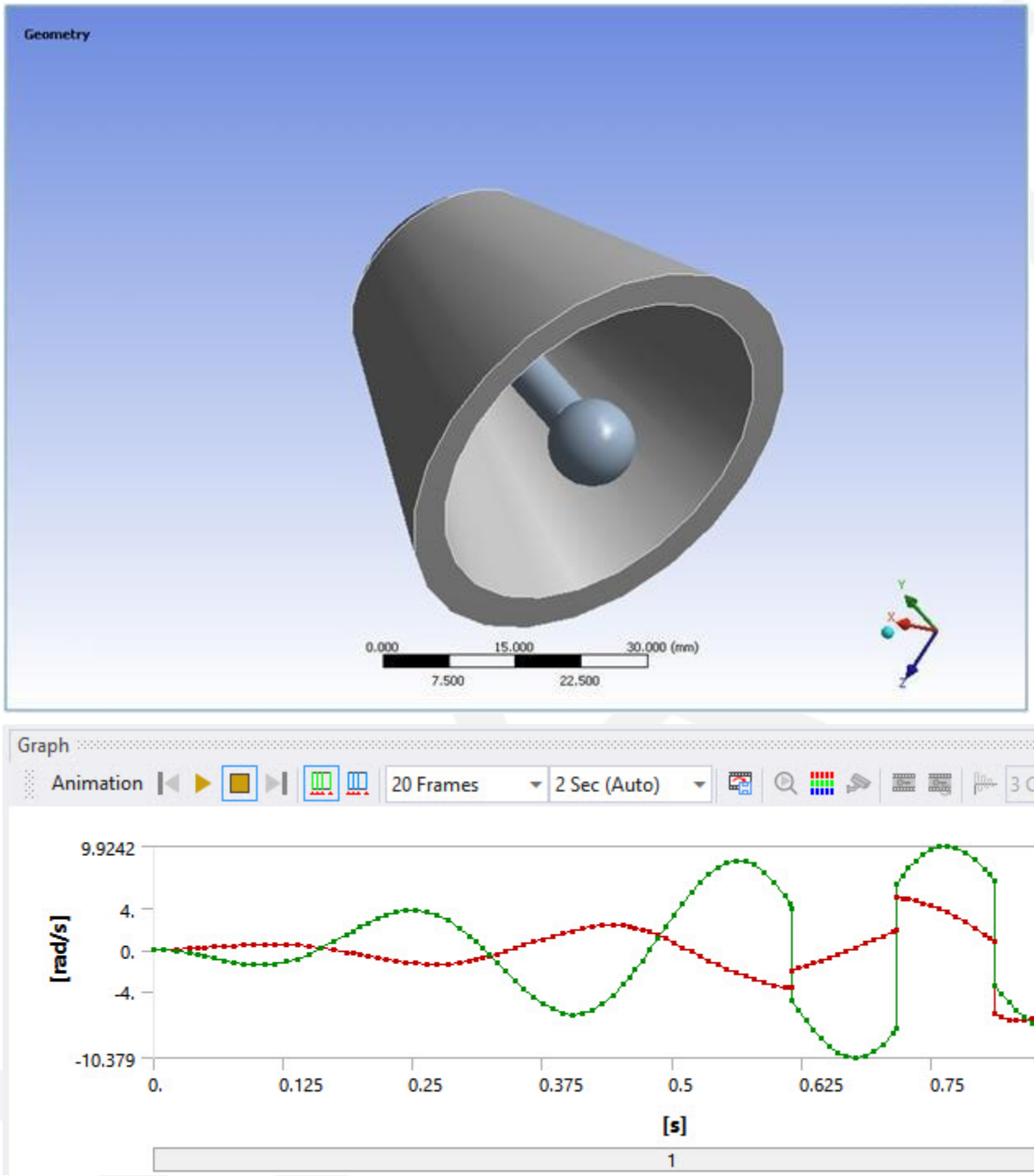
```
Joint.AddStop(0.45,1.0)
```

An example of the model and the results of this command are shown below.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Export of Joint Forces

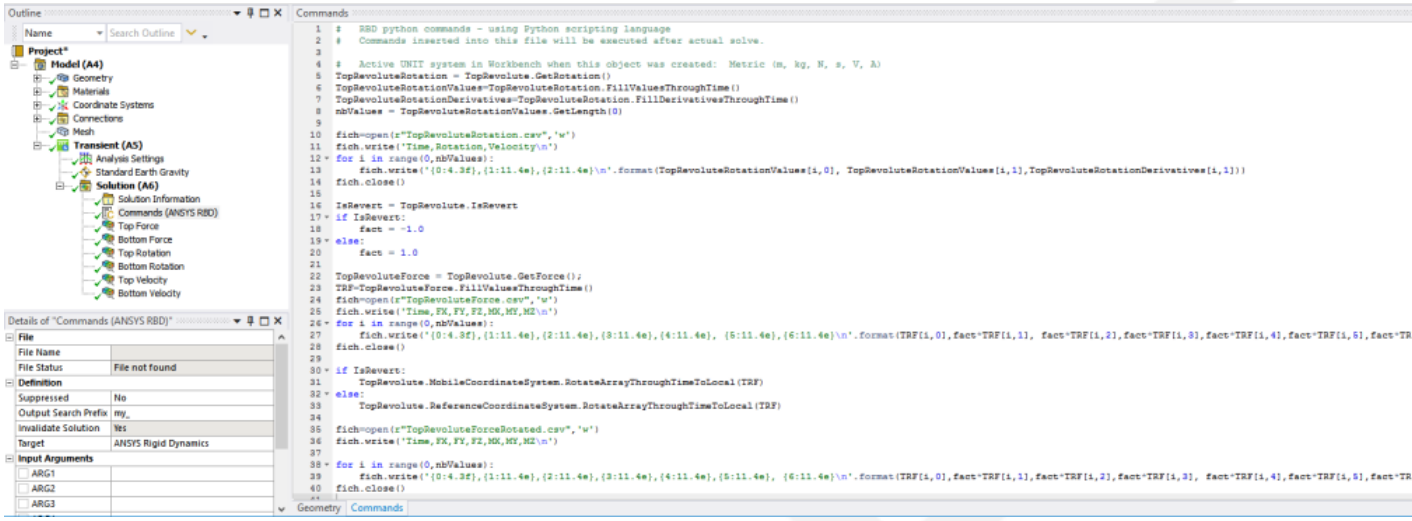
In this example joint forces are extracted in the local coordinate system, rotated into the global coordinate system, and written into an ASCII File

First, the joint is retrieved by inserting the following command on the corresponding joint in the tree:

```
TopRevolute = CS_Joint.Find(_jid)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Next, the commands object shown below is inserted in the result node. An explanation of these commands follows.



1. Get measures from the joint:

```
TopRevoluteRotation = TopRevolute.GetRotation()
```

2. Extract transient values for this measure:

```
TopRevoluteRotationValues=TopRevoluteRotation.FillValuesThroughTime()
```

3. Get angle derivatives by extracting the time derivatives of the measure:

```
TopRevoluteRotationDerivatives=TopRevoluteRotation.FillDerivativesThroughTime()
```

4. Count the number of components of this array:

```
nbValues = TopRevoluteRotationValues.GetLength(0)
```

5. Open the ASCII output file:

```
fich=open(r"TopRevoluteRotation.csv",'w')
fich.write('Time,Rotation,Velocity\n')
```

6. Loop over all time values, and write values:

```
for i in range(0,nbValues):
 fich.write('{0:4.3f},{1:11.4e},{2:11.4e}\n'.format(TopRevoluteRotationValues[i,0],
 TopRevoluteRotationValues[i,1],TopRevoluteRotationDerivatives[i,1]))
fich.close()
```

7. Check if joint is « revert » or not:

```
IsRevert = TopRevolute.IsRevert
if IsRevert:
 fact = -1.0
else:
 fact = 1.0
```

8. Extract Force Measure and write them into the file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
TopRevoluteForce = TopRevolute.GetForce();
TRF=TopRevoluteForce.FillValuesThroughTime()
fich=open(r"TopRevoluteForce.csv",'w')
fich.write('Time,FX,FY,FZ,MX,MY,MZ\n')
for i in range(0,nbValues):
 fich.write('{0:4.3f},{1:11.4e},{2:11.4e},{3:11.4e},{4:11.4e},
{5:11.4e},{6:11.4e}\n'.format(TRF[i,0],fact*TRF[i,1],
fact*TRF[i,2],fact*TRF[i,3],fact*TRF[i,4],fact*TRF[i,5],fact*TRF[i,6]))




fich.close()
```

9. Get the joint reference coordinate system, and rotate the forces from the global coordinate system to the joint coordinate system:

```
if IsRevert:
    TopRevolute.MobileCoordinateSystem.RotateArrayThroughTimeToLocal(TRF)
else:
    TopRevolute.ReferenceCoordinateSystem.RotateArrayThroughTimeToLocal(TRF)

fich=open(r"TopRevoluteForceRotated.csv",'w')
fich.write('Time,FX,FY,FZ,MX,MY,MZ\n')

for i in range(0,nbValues):
 fich.write('{0:4.3f},{1:11.4e},{2:11.4e},{3:11.4e},{4:11.4e},{5:11.4e},
{6:11.4e}\n'.format(TRF[i,0],fact*TRF[i,1],fact*TRF[i,2],fact*TRF[i,3],
fact*TRF[i,4],fact*TRF[i,5],fact*TRF[i,6]))
fich.close()
```

Breakable Joint

This example considers a breakable joint. A breakable joint is a joint that cannot withstand an internal force higher than a given value. To create a breakable joint:

1. Get the joint by inserting a command on a planar joint:

```
joint=CS_Joint.Find(_jid)
```

2. Create a joint condition to prescribe zero velocity on the two translational degrees of freedom:

```
driver=CS_Driver(Joint,System.Array[int]([0,1]),CS_Driver.E_MotionType.E_Velocity)
```

3. Define the value of the velocity, then retrieve the time measure:

```
Env=CS_Environment.GetDefault()
Sys=Env.System
(ret,found,time)=Sys.FindOrCreateInternalMeasure(CS_Measure.E_MeasureType.E_Time)
```

4. Define the time as variable, and use constant values for the two components:

```
driver.SetInputMeasure(time)
driver.SetConstantValues(System.Array[float]([0.,0.]))
```

   Next, make the driver only active if the force in the joint is less than a maximum threshold of 3N. To do that, create a Condition based on the joint force measure norm.

5. Retrieve the force on the joint:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
force=joint.GetForce()
```

6. Create a component measure, that is the norm 2 of the force. To be computed at each time step, this measure has to be added to the system.

```
norm=CS_ComponentMeasure(force,-2)
Sys.AddMeasure(norm)
```

7. Now, create the condition and assign it to the driver:

```
cond=CS_Condition(CS_Condition.E_ConditionType.E_LessThan,norm,3.0)
driver.Condition=cond
```

8. Finally, add the driver to the environment:

```
Env.Drivers.Add(driver)
```

Debugging RBD Commands with Visual Studio

You can debug RBD command snippets Using Microsoft Visual Studio. This allows you to execute commands line by line and review variable values.

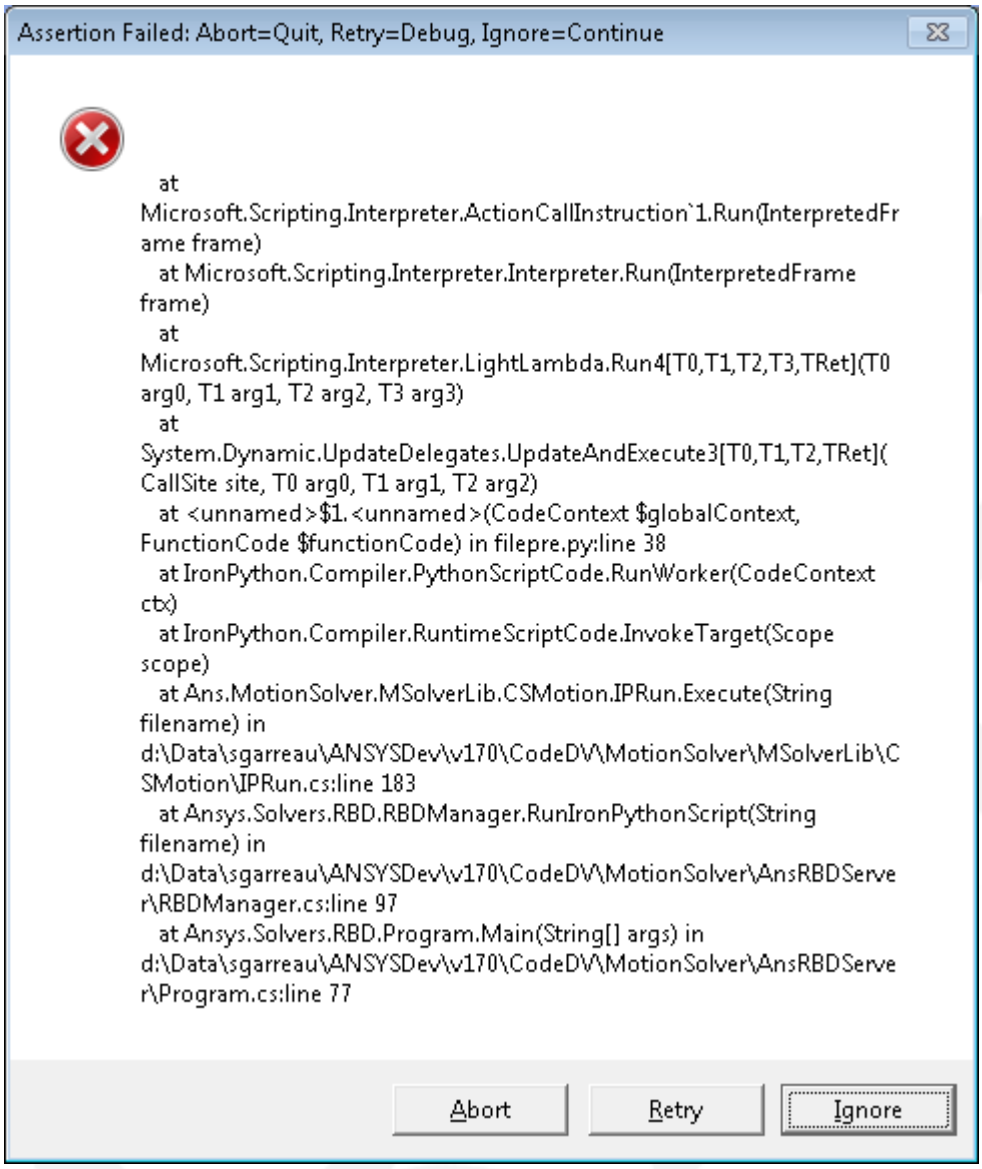
1. To begin, insert the following lines before the commands snippet you want to debug. (Note that if there are several commands snippet, they are executed in the order they appear in the Mechanical tree.)

```
from System import Diagnostics
Diagnostics.Debug.Assert(0)
```

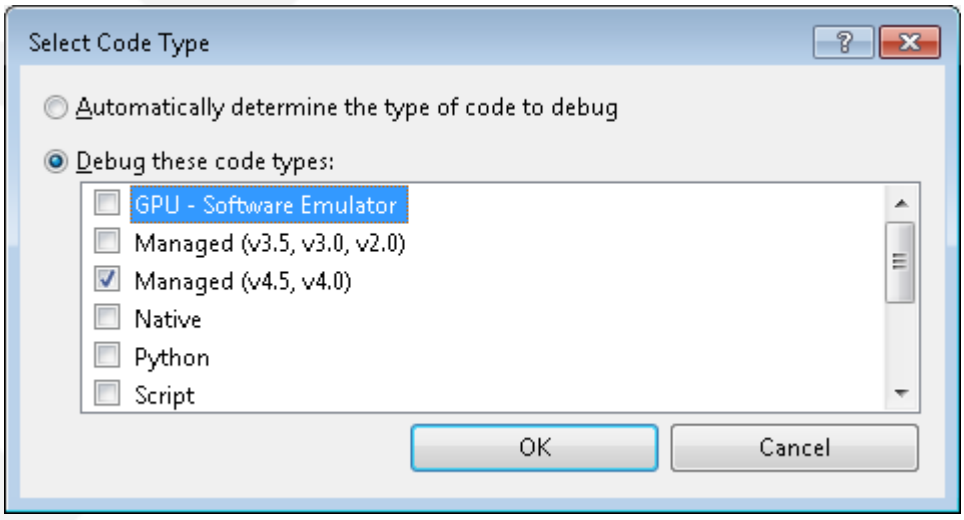
2. Begin the solve. When the solution begins, the following warning dialog appears. Do not close this dialog. The dialog will pause the solver and allow you to attach the Visual Studio debugger and set breakpoints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

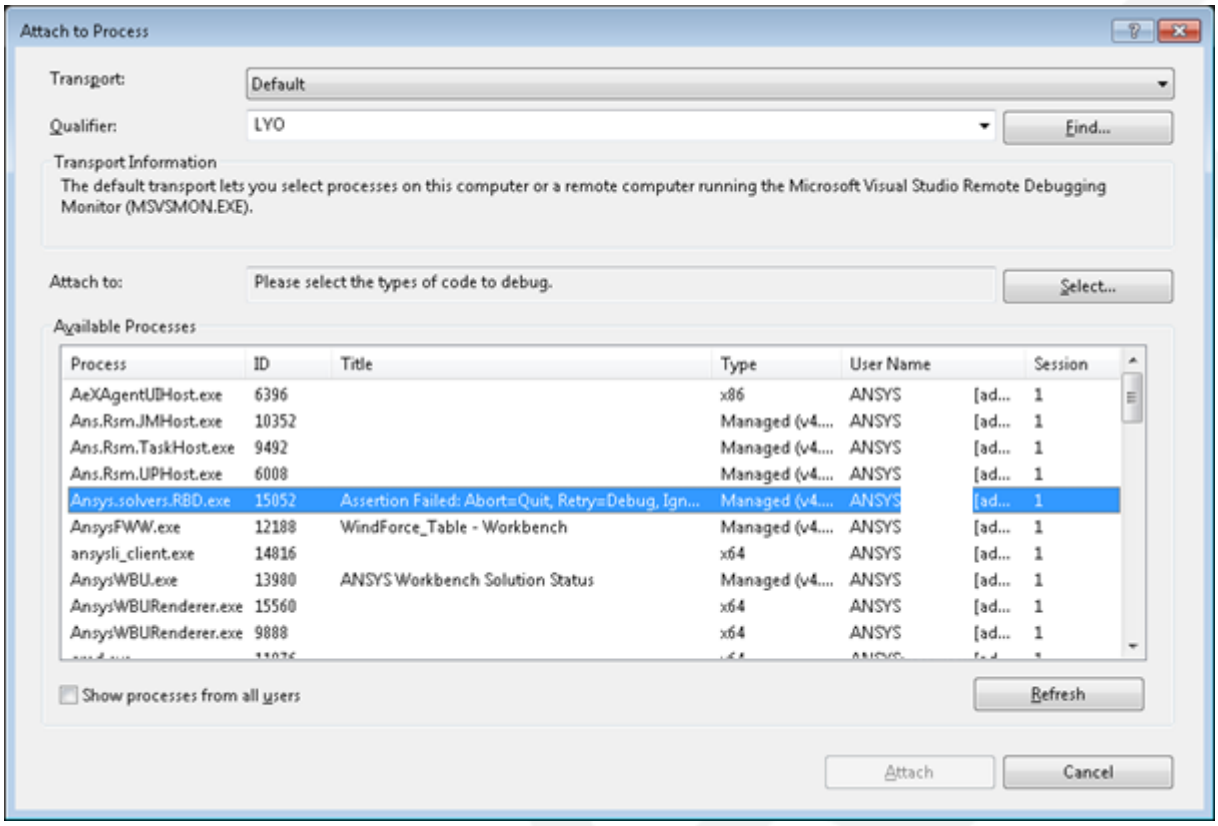
Published: 2023-10-23



3. In Visual Studio, select **Attach To Process...** from the **DEBUG** menu. In the **Select Code Type** dialog, select **Managed**, then click **OK**.



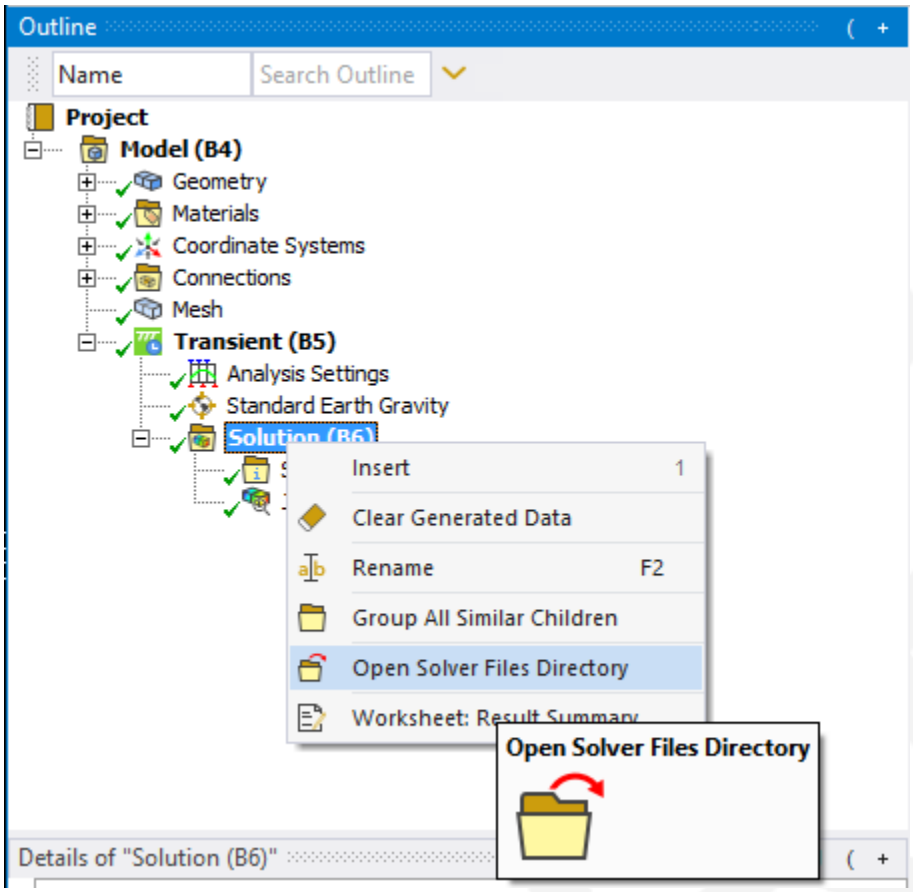
4. In the **Attach to Process** dialog, select the RBD solver process (**Ansys.solvers.RBD.exe**), then click **Attach**.



5. Once Visual Studio is attached to the RBD solver, open the script file in Visual Studio. To locate script files, in Mechanical, right-click the **Solution** object and select **Open Solver Files Directory**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



6. In the solver files directory, commands are written to two python files: `filepre.py` and `filepost.py`. `filepre.py` contains the commands that are executed before solve (all command snippets except those at solution level). `filepost.py` is executed after the solve (only command snippets at solution level).

   Open the desired file in Visual Studio. You can insert breakpoints as desired and click **Ignore** on the warning dialog to resume the solve.

Using RBD Commands with Excel

It is possible to call Microsoft Office Excel from an RBD command to read and write data to and from Excel. If Excel is available, RBD will automatically detect and load the Excel interopt. If Excel is detected, the solver output will contain the following lines at the beginning.

```
Processing Python commands
import base modules and macros
Microsoft.Office.Interop.Excel is loaded
Ans.Customize.Misc interop is not available
ready to process commands
```

The Excel application is initiated using:

```
excel=Microsoft.Office.Interop.Excel.ApplicationClass()
excel.Visible=True
excel.DisplayAlerts=False
```

Use the following line to open an Excel file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
workbook=excel.Workbooks.Open(r"E:\RBD\MODELS\COMMANDES\EXCEL\Excel_v160_files\Velocity.xlsx")
ws=workbook.Worksheets[1]
```

To read data from the current worksheet, use the following:

```
xlrange = ws.Range["A2", "B66"]
values=xlrange.Value2
```

Before using the values in an RBD script, it may be necessary to convert them to real values:

```
realValues=System.Array.CreateInstance(float,2,values.GetLength(0))
for i in range(0,values.GetLength(0)):
    print '{0:e} {1:e}'.format(values[i,0],values[i,1])
    realValues[0,i]=values[i,0].real
    realValues[1,i]=values[i,1].real
```

Similarly, it is possible to write values to the current worksheet. The following sequence of commands shows how to create a new worksheet and write the joint force in the new worksheet:

```
# retrieve joint force measure
force=joint.GetForce()

# obtain time values for this measure
values=force.FillValuesThroughTime()

# create a new worksheet
ws2=workbook.Worksheets.Add()
ws2.Name='Reaction forces'

len=values.GetLength(0)

cell=ws2.Range["A1"]
cell.Value2='Time'
cell=ws2.Range["B1"]
cell.Value2='Fx'
cell=ws2.Range["C1"]
cell.Value2='Fy'
cell=ws2.Range["D1"]
cell.Value2='Fz'
cell=ws2.Range["E1"]
cell.Value2='Mx'
cell=ws2.Range["F1"]
cell.Value2='My'
cell=ws2.Range["G1"]
cell.Value2='Mz'

# put values into the new worksheet
end="G"+str(1+len)
cells=ws2.Range("A2",end)
cells.Value2=values
```

Using RBD Commands from the IronPython Console

It is possible to use any rigid body dynamics commands you would use during a simulation from the IronPython console, outside of the simulation environment. The following command snippets and instructions provide a demonstration of this capability:

You can access the IronPython console by clicking the **File** > **Scripting** > **Open Command Window** menu item.

The following code snippets load the RBD Command module into IronPython:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
import clr
import os
import Ansys
import sys

clr.AddReference('Ans.Utilities')
ver=Ansys.Utilities.ApplicationConfiguration.DefaultConfiguration.VersionInfo.VersionString

awp_root=os.getenv('AWP_ROOT'+ver)
sys.path.Add(awp_root+r'\aisol\bin\winx64')
clr.AddReference('Ans.MotionSolver.MSolverLib.CSMotion')

from Ans.MotionSolver.MSolverLib.CSMotion import *
```

You can read an already-solved rigid body dynamics model using the following code:

```
dbIn=CS_MSolverDB()
dbIn.SetFileName(GetProjectDirectory()+'/TestRestart_files/dp0/SYS/MECH/file.mbd')
dbIn.OpenDataBase(0)
dbIn.ReadDB()
dbIn.Dispose()
```

The environment and system objects are accessed in the following way:

```
environment=CS_Environment.GetDefault()
system=environment.System
```

It is now possible to alter properties of the simulation. For example, you could modify the end time and restart from 0.5 s:

```
environment.AlterSimulationEndTime(2.0)
environment.RestartTime=0.5
environment.Solve()
```

Once you have made your changes, make use the following code snippet to save the modified database:

```
dbOut=CS_MSolverDB()
dbOut.SetFileName(GetProjectDirectory()+'/TestRestart_files/dp0/SYS/MECH/file.mbd')
dbOut.OpenDataBase(1)
dbOut.WriteDB()
dbOut.Dispose()
```

Note:

You cannot restart a Rigid Dynamics analysis using this procedure if the model has contact or a Point On Curve joint, or if there are multiple load steps.

Using the Variable Load Add-on

The Variable Load Add-on greatly simplifies the definition of complex loading for Rigid Dynamics systems. It allows you to create loads that depend on the state of the model, and to make these loads conditionally applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Without the add-on, such complex loads can only be defined using command objects. The Variable Load Add-on will similarly write commands to define these loads.

Note:

The add-on is only available on Windows platforms.

The following topics explain the use of the Variable Load Add-on:

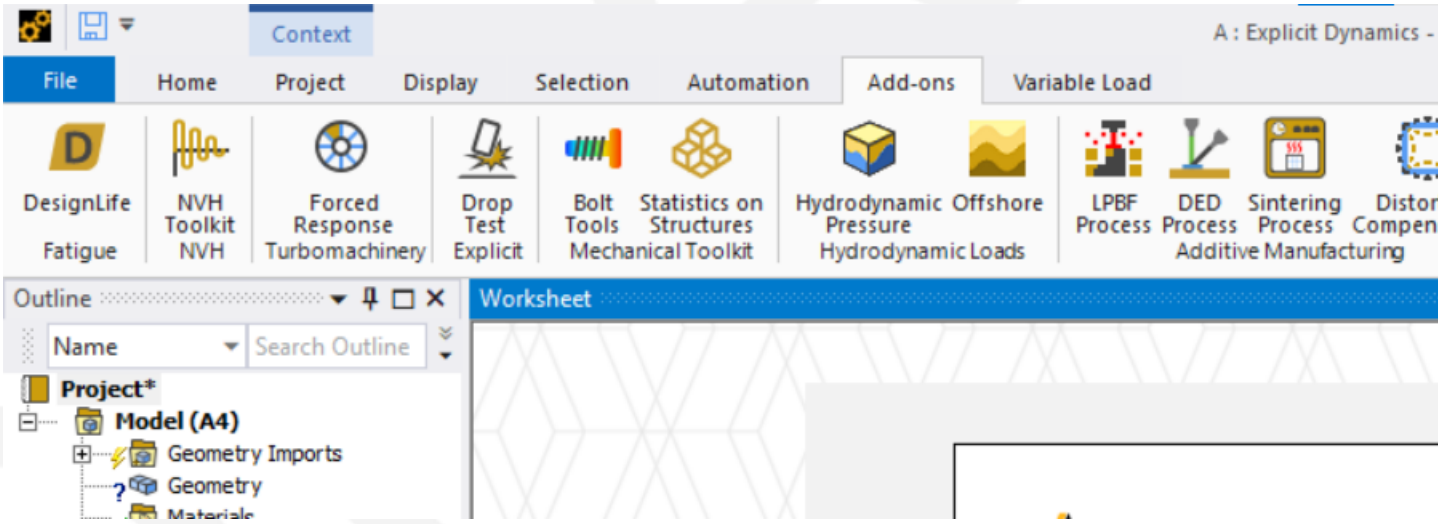
How to Load the Add-on
Creating Measures
Defining Joint Loads Dependent on one or more Measures
Defining Force Loads Dependent on one or more Measures
Known Issues and Limitations

How to Load the Add-on

The Variable Load Add-on is included in the Ansys product installation but must be loaded into Workbench.

To make the capabilities of the Variable Load Add-on available, click the **Variable Load** Icon in the **Add-ons** Ribbon. The icon will be highlighted in blue, indicating that the add-on is loaded.

Figure 1: Variable Load Add-on Showing Loaded Status



Once the add-on is loaded, the **Variable Load** Toolbar will appear in the Mechanical Ribbon.

Creating Measures

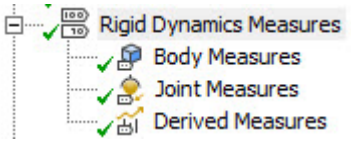
Measures can be thought of as sensors used to instrument the models. The workflow consists of instrumenting the model with measures that capture the state of the model, and then using these measure values to compute the value of the applied loads, or to activate and deactivate them.

Joints and Bodies can be instrumented with Measures. For these base measures, derived measures that transform the base measures can be introduced.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To create the Rigid Dynamics Measures folder in the Project tree, click the Insert Measures icon .

The corresponding folder appears in the tree.

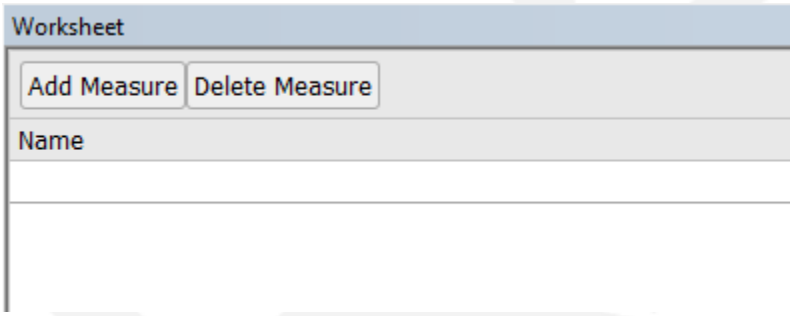


Note:

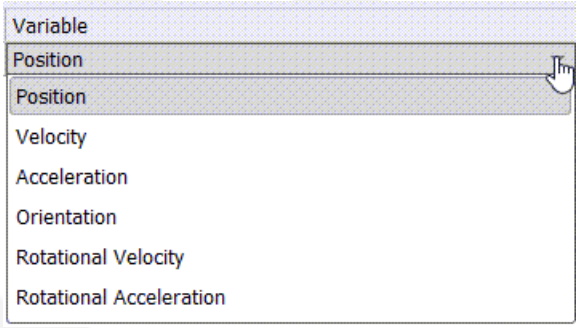
- All measures are computed and consumed during the solution. The solver uses the consistent unit system that is associated with the user unit system. Therefore, you should define operations done on derived measures, or the tables that consume the measure values, with the consistent unit system in mind.
- All rotations are in radians and all rotations velocities are in rad/s.

Body Measures

1. Select **Body Measure** in the tree. The Body Measure worksheet appears.
2. To add a new measure, click **Add Measure**.



3. Give a **Name** to this measure. You should use unique names, as the name of the measure will be used in selections later on.
4. Select the body that you want to instrument in the **Selection** column.
5. Select the quantity that you want to measure from the **Variable** dropdown list.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

6. Select the coordinate system that defines the position of the point where position, velocity and acceleration are reported.

7. If you need to delete a measure that you previously created, select its row in the worksheet and click **Delete Measure**.

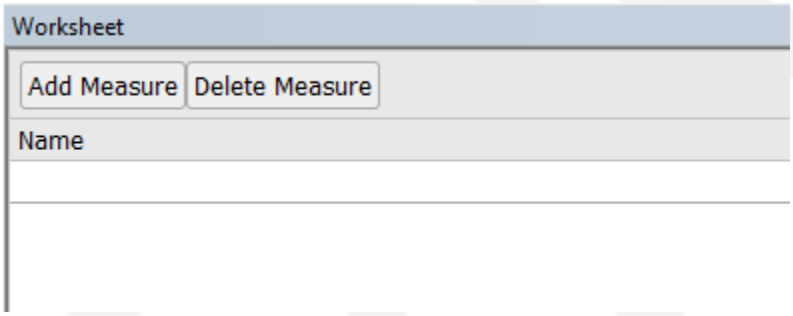
Note:

- Position, Velocity, Acceleration, Rotational Velocity and Rotational Accelerations are 3D vectors, with X, Y and Z components.
- For Orientation, Rotational Velocity and Rotational Acceleration, the coordinate system is not used.
- The coordinate system just gives the position. X, Y and Z components of Position, Velocity, and Acceleration are global coordinates.

Joint Measures

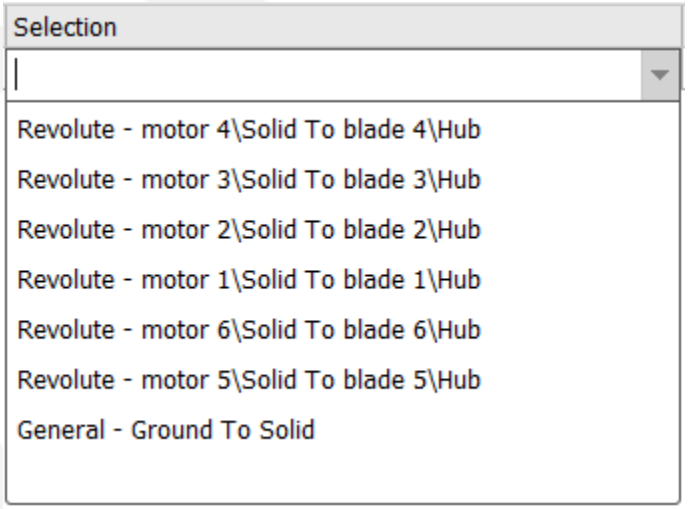
1. Select **Joint Measure** in the tree. The Joint Measure worksheet appears.

2. To add a new measure, click **Add Measure**.



3. Give a **Name** to this measure. You should use unique names, as the name of the measure will be used in selections later on.

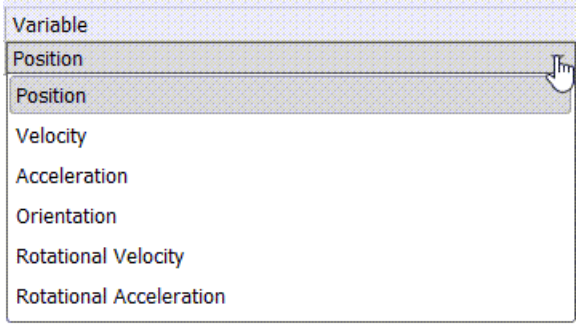
4. Select the joint that you want to instrument from the dropdown in the **Selection** column.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. Select the quantity that you want to measure from the **Variable** dropdown list.



6. If you need to delete a measure that you previously created, select its row in the worksheet and click **Delete Measure**.

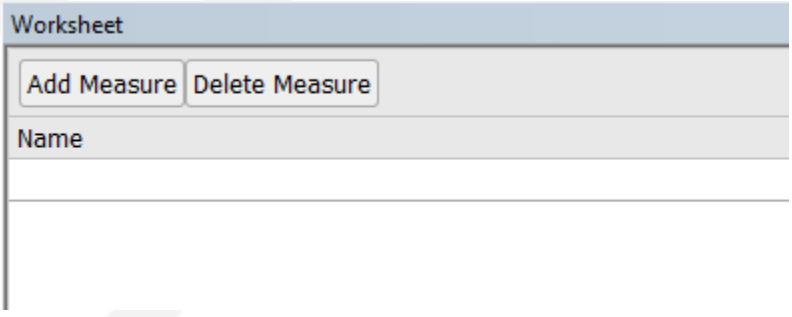
Note:

- The quantities available will depend on the type of joint that is selected. Select the joint first so that the Variable drop-down menu updates the list with relevant quantities.
- The number of components of the position, rotation, velocity and acceleration measure depends on the joint type.
- During the solution, the solver sometimes flips the reference and mobile coordinate system of the joint. If the joint is reverted, force load must be negated. See the model topology tool from the connection menu to see which joints are reverted. See the IsRevert property in the CS_joint class of the scripting manual.

Derived Measures

Based on how the loads consuming the measures are defined, one-dimensional measures might be needed. Measures also sometimes need to be transformed by some math operators. Derived measures are made for these operations.

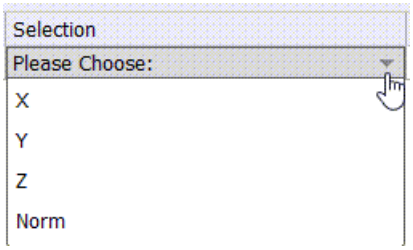
1. Select **Derived Measure** in the tree. The Derived Measure worksheet appears.
2. To add a new measure, click **Add Measure**.



3. Give a **Name** to this measure. You should use unique names, as the name of the measure will be used in selections later on.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Select the **Base Measure** the operator applies to.
5. Select select a transformation **Type**, Component or Operator.
   - When using Component, select the component from among the components available. The list of component depends on the base measure type.



   - When using Operator, choose from Modulo or Scale Factor.
     – Modulo corresponds to a modulus operation. It can for example be used to remove the number of turns from the rotation angle of a revolute joint. The modulus is defined by the **Operator Value** property.
     – Scale Factor scales the value of the input measure by a constant defined by the **Operator Value** property.
6. If you need to delete a measure that you previously created, select its row in the worksheet and click **Delete Measure**.

Note:

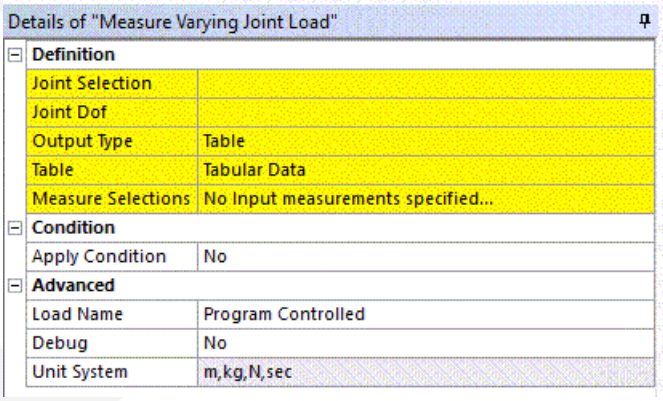
The base measure of a derived measure can be a derived measure.

Defining Joint Loads Dependent on one or more Measures

To define a joint load that depends on one or more measures:

1. Click the Insert Measure Varying Joint Load icon .

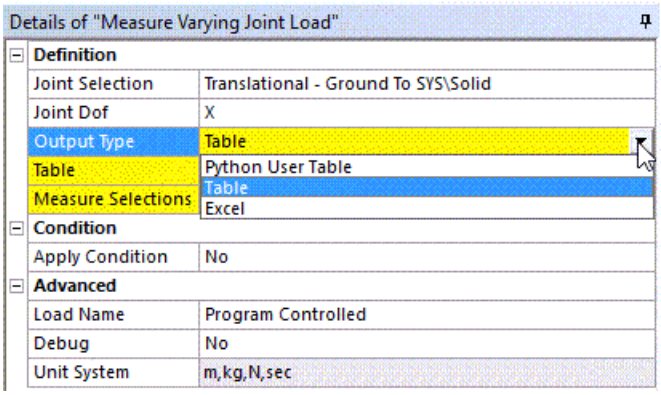
   The properties of this load can then be edited in the **Details** panel.



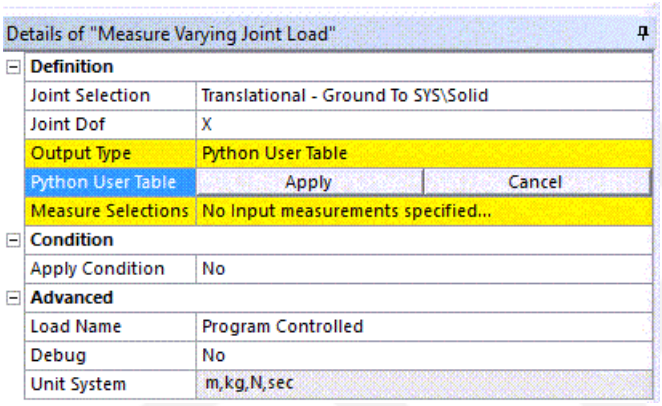
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. Select the joint on which to apply the load in the **Joint Selection** field.
3. Select the joint degree of freedom on which the load is applied in the **Joint Dof** field.
4. Define the **Output Type** from the dropdown list.



- *Python User Table*



When you select Python User Table, a text editor will open, allowing you to define the evolution of the joint load with respect to the input variable defined in the **Measure Selections** field.

The text editor will contain a pre-defined IronPython function.

```
class UserTable_33(CS_UserTable):
   def __init__(self, sizeIn, sizeOut):
       CS_UserTable.__init__(self, sizeIn, sizeOut)

   def Evaluate(self, In, Out):
       ## define Out[] as a function of In[] here
       Stiffness = 43.0
       Damping = 0.1
       Preload = 100.0
       Displacement = In[0]
       Velocity = In[1]
       Out[0] = -Stiffness*Displacement -Damping*Velocity + Preload

       return 0

#define input size and output size here
SizeIn=1
SizeOut=1
table_33 = UserTable_33(SizeIn,SizeOut)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
print "    *** CS_UserTable UserTable_33 created ***"
```

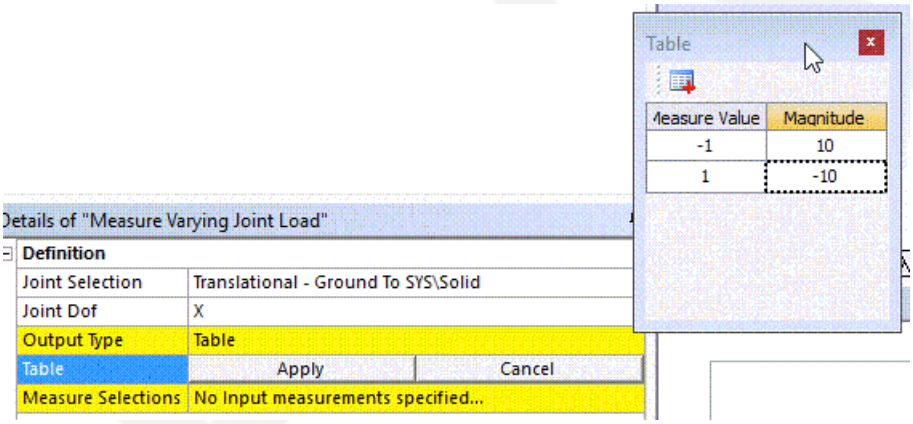
Note:

- The string _33 is given here as example. Actual value will vary.
- The text editor that will be used is that associated with the `.txt` add-on in your user preferences.
- This feature relies on the CS_UserTable class of the scripting manual.

In general, you would only modify the Evaluate function. Its role is to compute *Out[0]*. The input measures come into this function in the *In[]* array. In the example above, the load has two input measures: one is the joint displacement and the other is the joint velocity, for a joint that has only one degree of freedom. The force is computed as if a preloaded spring and damper were acting together.

- *Table*

  This option will allow you to define tabular data defining output as a function of input. This option requires that you have only one input variable. The table will have two columns, the first one corresponding to the input values and the second one corresponding to the output values.



- *Excel*

  You can also use Excel to define tabular output, and the data will be read from an Excel comma separated value file (`.csv`). Click the field next to **Excel** to browse to your Excel file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details of "Measure Varying Joint Load"	
Definition	
Joint Selection	Translational - Ground To SYS\Solid
Joint Dof	X
Output Type	Excel
Excel	
Skip Header	Yes
Output Scale Factor	1
Column Separator	,
Measure Selections	No Input measurements specified...
Condition	
Apply Condition	No
Advanced	
Load Name	Program Controlled
Debug	No
Unit System	m,kg,N,sec

The Excel table must have two columns, the first one corresponding to the input values and the second one corresponding to the output values.

– By default, the first row of the table is assumed to contain labels that describe the columns.

	A	B
1	Displacement	Force
2	0	-3.55E-15
3	0.01	1.00E+00
4	0.03	3.00E+00
5	0.07	7.00E+00
6	0.12	2.00E+00
7	0.17	7.00E+00
8	0.22	2.00E+00
9	0.27	7.00E+00
10	0.32	2.00E+00
11	0.37	7.00E+00
12	0.42	2.00E+00
13	0.47	7.00E+00
14	0.52	2.00E+00
15	0.57	7.00E+00
16	0.6	1.00E+01
17		

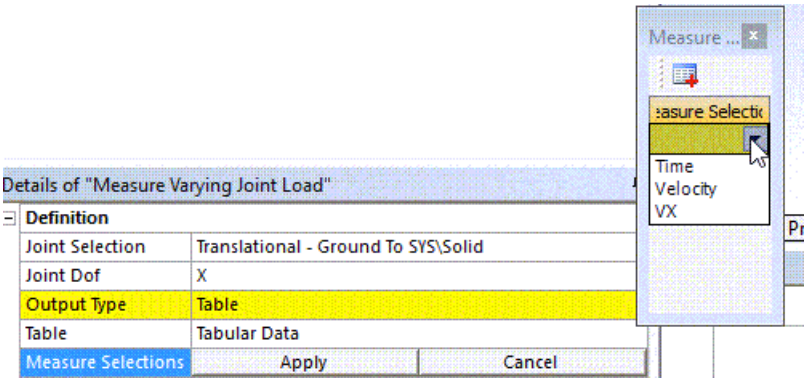
If this first row contains data (it is not a header), you should set the **Skip Header** option to No.

– Values from the spreadsheet can be scaled by the **Output Scale Factor**. This can be useful, for example, if the `.csv` file hasn't been created in the same unit system as the solution.

– By default, the Comma Separated Values file format uses "," as a separator. However, depending on the language of your operating system, or of the machine that has generated the `.csv` file, Excel can use a different character to delimit the fields. You can specify this character using the **Column Separator** field.

5. Define one or more Input Measures using the **Measure Selection** field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



If **Output Type** is set to Table only one measure can be defined. If **Output Type** is Excel or Table, the Input Measure(s) must be scalar. Use a derived measure to extract one particular component of a multi-dimensional base measure.

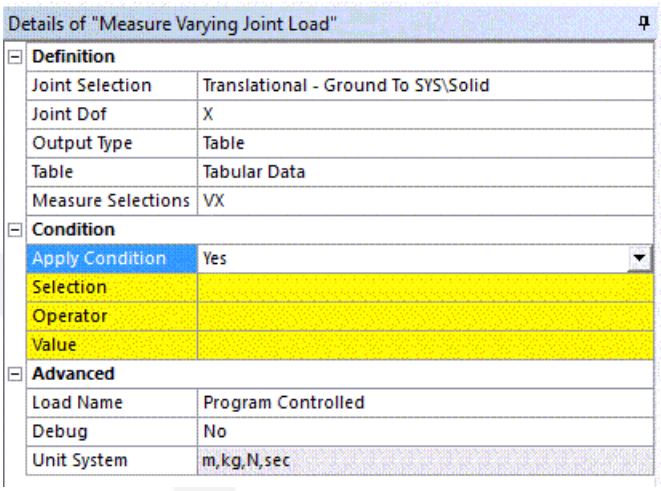
The measure Time is always available. While time-varying loads can be defined on all transient Mechanical loads, the use of IronPython can be very powerful for complex loads.

Note:

- You cannot specify a Force measure or a measure derived from a Force measure as an input used to compute loads.
- You should not use accelerations as inputs because it could lead to very small time steps or lack of convergence. If the intent is to link the accelerations of two joints, it is more efficient to use a constraint equation that links the two accelerations.

---

6. Joint loads can be applied conditionally, based on the value of any derived measure.



- Use **Selection** to define the scalar measure that the condition is based on.
- Select the **Operator** from the list.
- Select the right-hand side of the condition in the **Value** field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

You can, for example, define a "breakable" spring that will act only if the joint displacement is less than a maximum displacement.

7. The load has a default name. This name can be overwritten using the **Load Name** field in order to increase the readability of the created script.

8. Use the **Debug** option to print the value of the input values and calculated output values of the table. This option should be used only for short simulations as it will slow down the evaluation of the load.

Important:

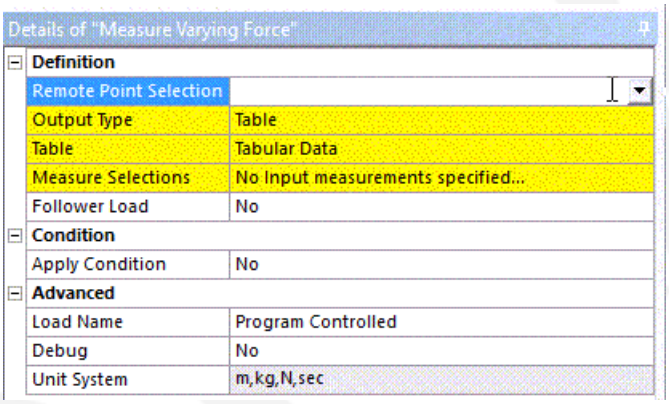
The read-only property **Unit System** is shown for information, as the **Measure Varying Joint Load** object does not change if you change the unit system in the Mechanical Application. It is strongly recommended that you use one single unit system to define all the measure varying loads and to solve using this same unit system.

Defining Force Loads Dependent on one or more Measures

To define a force load that depends on one or more measures:

1. Click the Insert Measure Varying Force icon .

2. The properties of this load can then be edited in the **Details** panel.



- Select the **Remote Point** on which the load will be applied.

Note:

The Remote Point must be connected to a geometric entity or a named selection. It cannot be a freestanding remote point. The corresponding body must be a rigid body.

- Refer to Defining Joint Loads Dependent on one or more Measures (p. 616) step 2 to 8 to define the load. The only difference is that the resulting forces has 3 components (3-dimensional vector) rather than a single value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Use the **Follower Load** property if the load's application and orientation varies with the rigid body it is applied to.

Known Issues and Limitations

Using **Clear Generated Data** at the project level after a first solve may prevent further solutions. There is no workaround to this problem, and it may require that all of the variable loads be recreated.

Using the Motion Load Transfer Add-on

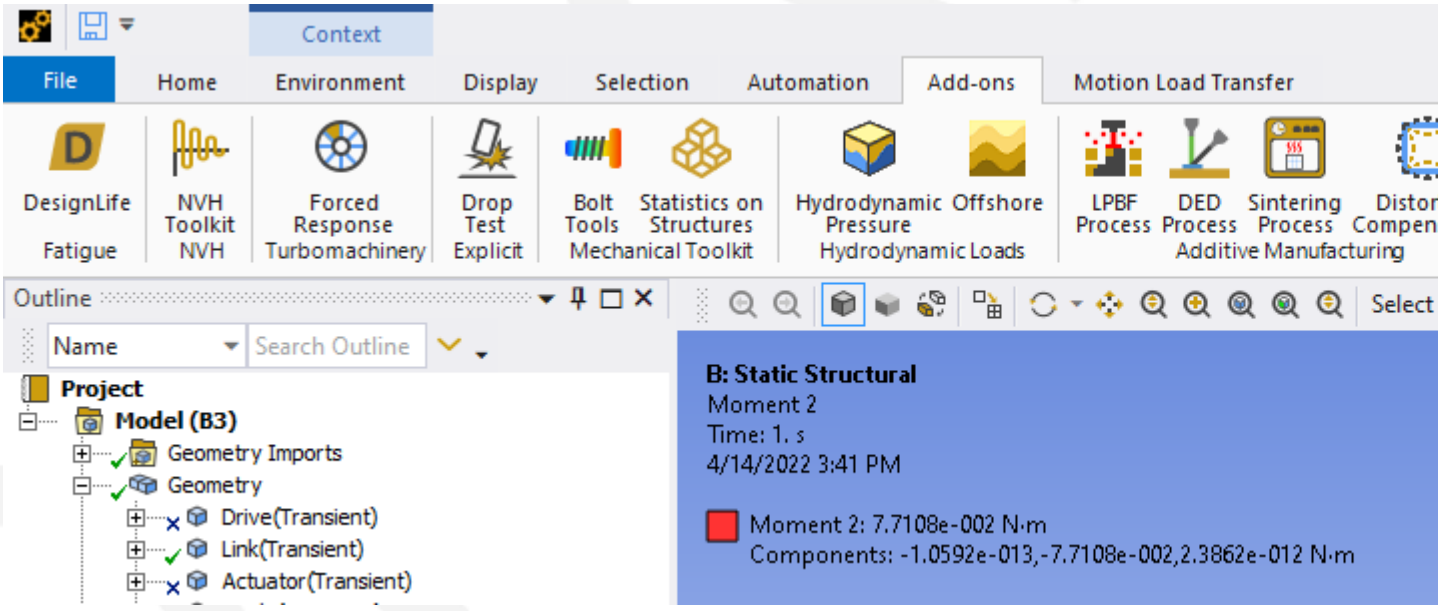
The Motion Load Transfer Add-on allows you to apply the loads created in a Rigid Dynamics analysis on the flexible bodies in a Static Structural analysis. For more information about exporting geometry from deformation results, see Geometry From Rigid Dynamics Results (p. 1027).

How to Load the Add-on

The Motion Load Transfer Add-on is included in the Ansys product installation but must be loaded into Workbench.

To make its capabilities available, click the **Motion Load Transfer** icon in the **Add-ons** Ribbon. The icon will be highlighted in blue, indicating that the add-on is loaded.

Figure 2: Motion Load Transfer Add-on Showing Loaded Status



Once the add-on is loaded, the **Motion Load Transfer** toolbar appears in the Mechanical Ribbon. Next, navigate to the Project window and add a Rigid Dynamics analysis system to the project. The system will have the Motion Loads features available.

Setting up the Motion Loads Transfer

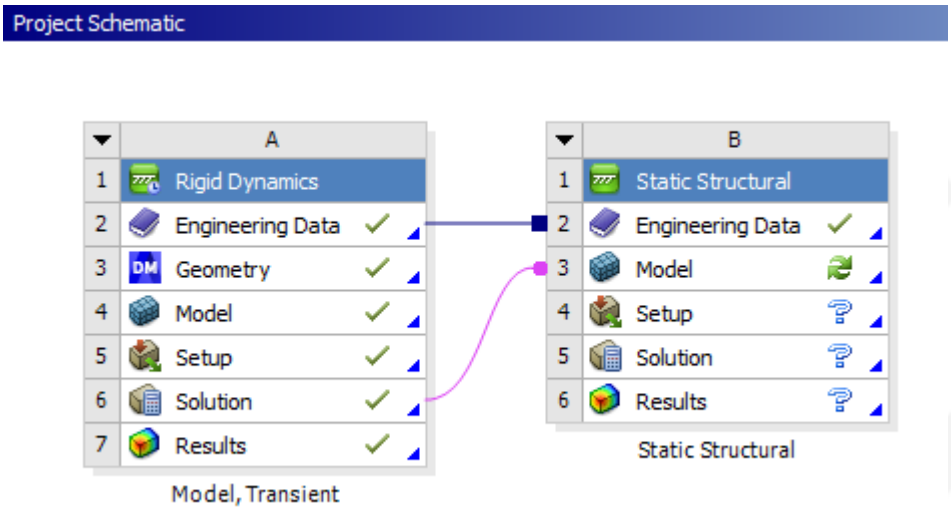
Follow these steps to set up the Motion Loads transfer:

1. Perform a Rigid Dynamics simulation.

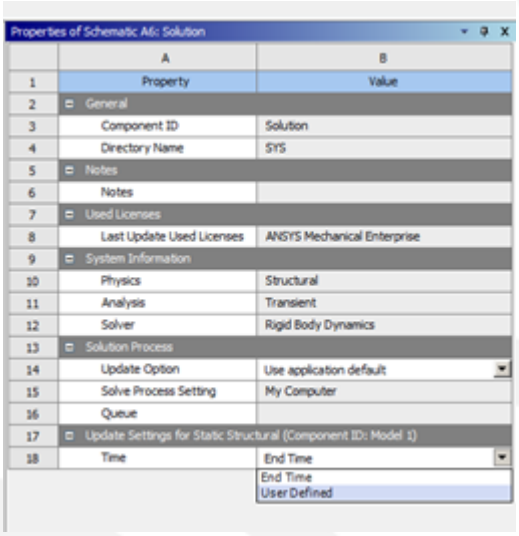
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. On the Project Schematic, add a Static Structural system and link the Rigid Dynamics system Engineering Data cell to the Static Structural Engineering Data cell, and the Solution cell to the Model cell.



3. Right-click the **Solution** cell of the Rigid Dynamics system and choose Properties. Under **Update Settings for Static Structural (Component)** set **Time** to User Defined, then enter the time at which you want to transfer the loads to the system.



4. Update the Rigid Dynamics system, then update the Model cell of the Static Structural system.

Note:

After a Save and Resume of a Rigid Dynamics analysis containing a motion load transfer, the motion load transfer won't work unless you delete the connection between the Rigid Dynamics Solution cell and the downstream system and recreate it using the steps in this section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Transferring the Motion Loads

Open the Static Structural system and follow these steps:

1. Select a body in the model by picking it in the Graphics view.

2. Click the Motion Loads icon in the Motion Loads tab.

   This automatically:

   - Creates Remote Points where Forces and Moments issued from the Rigid Dynamic Solution are applied.
   - Creates Rotational Velocity and Rotational Acceleration corresponding to the Angular Velocity and Angular Acceleration issued from the Rigid Dynamic Solution.
   - Changes the behavior of the selected body to flexible.
   - Suppresses all the other bodies.

3. While the loads applied by remote forces and moments are balanced by the inertia forces, the part is nevertheless not "supported" and the analysis needs to be adapted to be stable. This can be achieved, in general, by adding weak springs (p. 1433) and/or with inertia relief (p. 1435). In some instances, for example when the part has a fixed joint to ground, it can be preferable to replace the force and torque corresponding to this joint to ground by a fixed support. When accurate contact representation between bodies is important to properly capture stresses, you could choose to keep more than one body in the analysis. In that case, use separate static systems for each part and assemble them together.

4. Now solve the Static Structural system.

To Perform the same operation on another body, unsuppress all bodies and repeat steps 1 through 4.

Note:

The modification of the Rigid Dynamics system or the export time requires the Motion Loads to be re-imported.

Multibody Dynamics Theory Guide

Multibody dynamics is the study of the motion of assemblies of bodies, rigid or flexible, that undergo large motion in the 3D space. The free motion of bodies is restrained by joints. Every joint links two bodies in two points. These joints are idealizations of the contact between the two bodies. Joints are characterized by the motion that they allow between the two bodies that they connect. For example, a revolute joint allows one relative rotation between two bodies, constrains all three relative translations, and blocks the two other relative rotations.

The primary unknowns of a multibody dynamics solution are the translation and rotation of each body and the motion in the joints themselves. The output quantities on rigid bodies are the forces that develop in the joints and flow through the rigid bodies, as opposed to a structural analysis where

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the output quantities are strains or stresses. Flexible bodies can be included in a multibody analysis. These flexible bodies will have both joint forces and stress and strain results.

The following topics are discussed in this section:

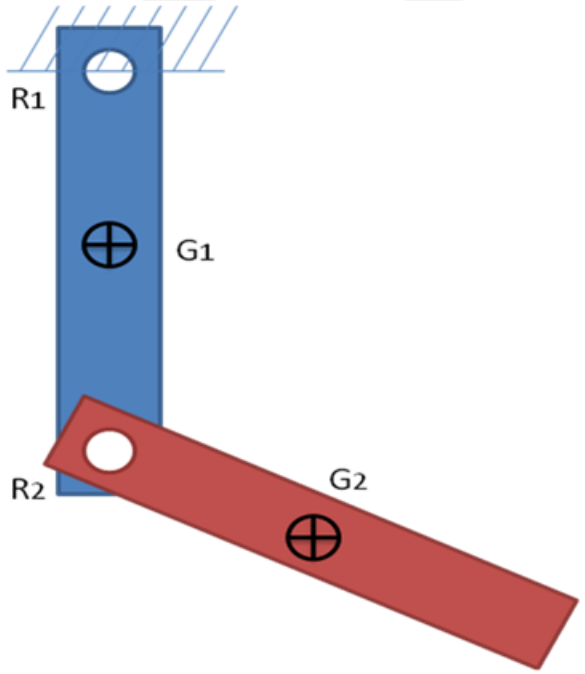
Rigid Degrees of Freedom
Rigid Shape Functions
Flexible Shape Functions
Equations of Motion
Time Integration with Explicit Runge-Kutta
Implicit Generalized-α Method
Stabilized Implicit Generalized-α Method
Moreau-Jean Method
Geometric Correction
Contact and Stops
References

Rigid Degrees of Freedom

This section discusses the options available when selecting degrees of freedom (DOFs) in a rigid body assembly and their effect on simulation time.

The double pendulum model shown below is considered in this section. The first body in this model (in blue) has center of gravity G1. This body is linked to the ground through revolute joint R1, and linked to a second body through revolute joint R2. The second body (in red) has center of gravity G2, and is linked to the first body through revolute joint R2.

Figure 3: Double Pendulum Model



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The two bodies in this model are rigid, meaning that the deformations of these bodies are neglected. The distance between any two points on a single rigid body is constant regardless of the forces applied to it. All the points on the body can move together, and the body can translate and rotate in every direction.

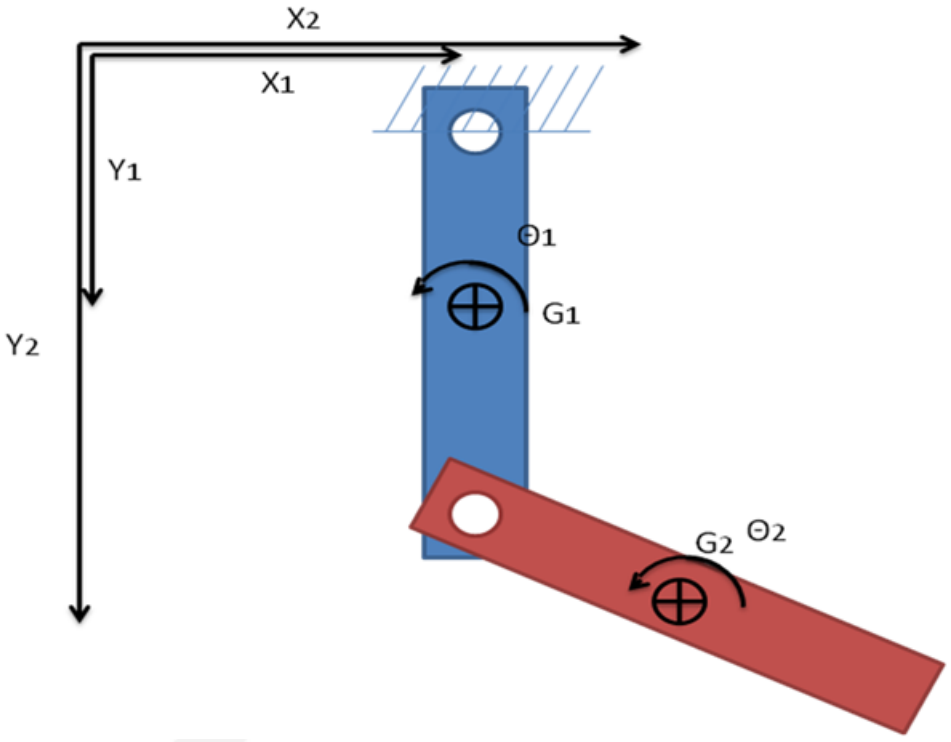
Many parameters are available to describe the body position and orientation, but the parameter usually chosen for the translation is the position of the center of mass with respect to a ground coordinate system. It is extremely difficult to represent 3D rotations for the orientation in a universal way. A sequence of angles is often used to describe the orientation, but some configurations are singular. An option frequently used to describe the orientation in computer graphics is the use of quaternion (also known as Euler-Rodrigues parameters); however, this option uses four parameters instead of three, and does not have a simple interpretation.

A natural choice of parameters to describe the position and orientation of the double pendulum model, is to use the position and orientation of the two individual bodies. In other words, use three translational and rotational degrees of freedom for each body, and introduce the joints using constraint equations.

The constraint equations used state that the two points belonging to the two bodies linked by the revolute joint are always coincident, and that the rotation axis of the joint remains perpendicular to the other body. This requires five constraint equations for each revolute joint.

The selected degrees of freedom (six DOFs per body and certain joints based on constraint equations) are considered "absolute" parameters.

Figure 4: Absolute Degrees of Freedom



The model shown in Figure 4: Absolute Degrees of Freedom (p. 626) depicts global parameters in 2-D for the double pendulum. Body 1 and 2 are respectively parameterized by X and Y translation

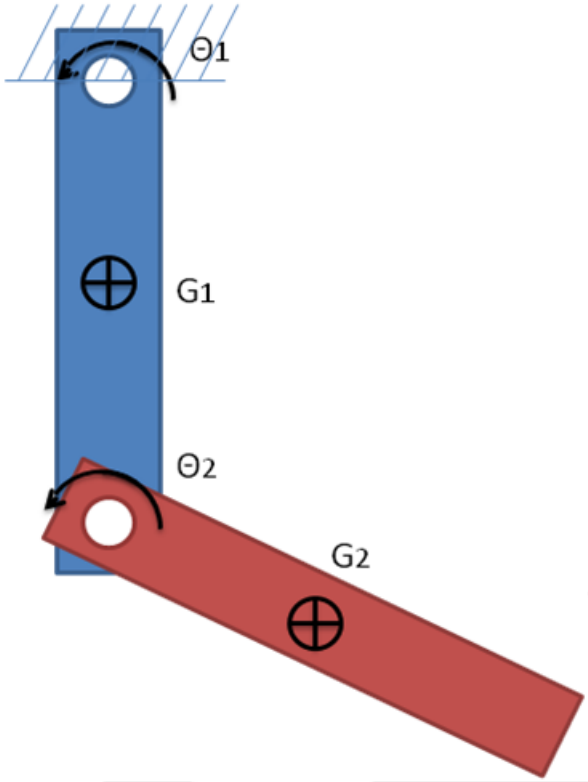
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

and theta rotation. Because the model has only two degrees of freedom, it does not require any additional constraint equations.

Global parameters for the body are chosen independently of the joints that exist between those bodies. When these joints are known, parameters for the joints can be chosen that reduce the number of parameters and constraint equations needed. For this example, the first degree of freedom is defined as the relative orientation of the first body with respect to the ground. The second degree of freedom is defined as the relative orientation of the second body with respect to the first body. Relative degrees of freedom are shown in the figure below:

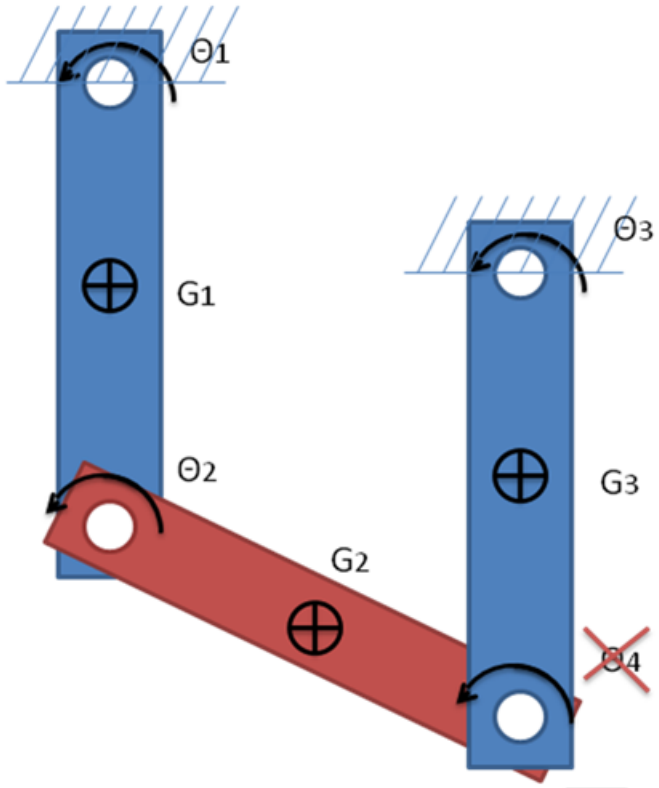
Figure 5: Relative Degrees of Freedom



Next, a third body is added to the model that is grounded on one side and linked to the second body with another revolute joint, as shown below:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 6: Closed Loop Model



The closed loop model shown above has three bodies (plus the ground) and four revolute joints. The degrees of freedom can be chosen for the example as follows:

- Θ1 - The relative rotation of body 1 with respect to ground
- Θ2 - The relative rotation of body 2 with respect to body 1
- Θ3 - The relative rotation of body 3 with respect to ground

The fourth revolute joint cannot be based on degrees of freedom because both the motions of Body 2 and Body 3 are already defined by existing degrees of freedom. For this joint, constraint equations are added to the relative degree of freedom parameters.

Θ1, Θ2, and Θ3 will be the degrees of freedom, and the corresponding joints will be topological joints. The fourth joint will be based on a constraint equation. Constraint equation-based joints are also known as kinematic joints. Kinematic joints are needed when the model has closed loops, that is, when there is more than one way to reach the ground from a given body in the system.

To determine which joints will be topological joints and which will be kinematic joints, a graph is constructed to show connections where the bodies are vertices and the joints are arcs. This graph is decomposed into a tree, and the joints corresponding to arcs that are not used in the tree are transformed into kinematic joints.

The Model Topology (p. 1369) view displays whether joints are based on degrees of freedom or constraint equations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Kinematic Variables vs. Geometry Variables

Euler's theorem on rotations states that an arbitrary rotation can be parameterized using three independent parameters. The choice of these three parameters is not unique, and many choices are possible. For example:

- A sequence of three rotations, as introduced by Euler (the first rotation around X, the second rotation around the rotated Y' axis, and the third rotation around the updated Z'' axis). Many other sequences of rotations exist, among them the Bryant angles.
- The 3 components of the rotation vector
- Etc…

Unfortunately, these minimal sets of parameters are not perfect. Sequences of angles usually have some singular configurations, and the composition of rotations using these angles is simple. This composition of rotation is intensively used in transient simulation. For example, it can be used to prevent the use of the rotation vector.

Another option is to use the 3x3 rotation matrix. Composition of rotations is easy with this option, as it corresponds to matrix multiplication; however, this matrix is an orthogonal matrix, and time integration must be done carefully to maintain the matrix properties.

A good compromise is to use quaternion, which have 4 parameters and a normalization equation.

Once rotation parameters have been selected, the time derivatives of these parameters have to be established:

$$\{\dot{g}\} = f(g, \vec{\Omega}) \tag{7}$$

where $\vec{\Omega}$ is the angular velocity vector.

Two sets of variables exist:

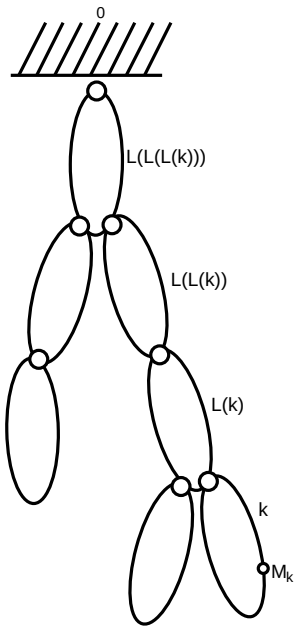
- Kinematic variables, expressed as $\{q\}$: $\vec{\Omega}$ as long as the translational velocities.
- Geometric variables, expressed as $\{g\}$, as well as the position variables for the translations. The geometric variables are obtained by time-integration of the kinematic variables.

Rigid Shape Functions

Shape functions, also called generalized velocities, are the projections of the velocity of material point M_k attached to body k on the kinematic variables of the model. Generalized velocities of a material point are depicted in the figure below:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

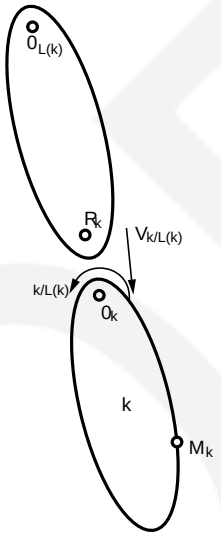
Figure 7: Generalized Velocities of a Material Point



Because of the choice of relative degrees of freedom, the velocity of M_k is a function of kinematic variables of the joint located between body *k* and its parent body *L(k)*, as well as those of the joint between *L(k)* and *L(L(k))*, continuing until the ground is reached.

To understand how these generalized velocities are formed, it helps to first focus on the contribution of the first joint of the chain (pictured below). This joint is located between body *k* and its parent, *L(k)*.

Figure 8: Contribution of the Parent Joint to the Generalized Velocities



Because body *k* is rigid, the velocity of point M_k with respect to the ground 0 can be expressed from the velocity of point O_k. Point O_k is the material point on the mobile coordinate system of the joint between body *k* and its parent, *L(k)*. This is expressed as follows:

$$\vec{V}_{Mk/0} = \vec{V}_{Ok/0} + \vec{\Omega}_{k/0} \wedge \overrightarrow{O_k M_k} \tag{8}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The angular velocity of body *k* with respect to the ground $\vec{\Omega}_{k/0}$ can be expressed as the angular velocity of its parent, plus the contribution of the joints linking body *k* and its parent, *L(k)*. This is expressed as follows:

$$\vec{\Omega}_{k/0}=\vec{\Omega}_{L(k)/0}+\vec{\Omega}_{k/L(k)} \tag{9}$$

Similarly, $\vec{V}_{Ok/0}$ can be expressed using point R_k , which is the reference coordinate system of the joint between body *k* and its parent, *L(k)*. Note that R_k is a material point on body *L(k)*. This is expressed as follows:

$$\vec{V}_{Ok/0}=\vec{V}_{Rk/0}+\vec{\Omega}_{L(k)/0}\wedge\overrightarrow{R_kO_k}+\vec{V}_{k/L(k)} \tag{10}$$

where $\vec{V}_{k/L(k)}$ is the joint relative velocity, i.e. the translational velocity between body *k* and its parent, *L(k)*.

It is important to realize that the vector $\overrightarrow{R_kO_k}$ has an angular velocity of $\vec{\Omega}_{L(k)/0}$. Joints can have translational degrees of freedom, and rotational degrees of freedom. The translation is expressed in the reference coordinate system, while the rotation center is the moving coordinate system. In other words, the joint translation is applied first, and the rotation is applied after the coordinate system is updated with the results of the joint translation. The decomposition of the Model Topology graph into a tree results in an oriented parent-child relationship. When the joint has both translational and rotational degrees of freedom and its reference coordinate system is on the child side, the joint must be split into a rotational joint linked to the parent side, and a translational joint linked to the child side, with a fictitious mass-less body between these two joints. While this is an internal representation of that "reverted" joint (that is, a joint that has both translational and rotational degrees of freedom and a link to the ground on the mobile coordinate system side), results are reported on the original user-defined joint.

Because R_k is a material point of body *L(k)*, the same methodology can be used to decompose the velocity into the contribution of the parent joint located between *L(k)* and *L(L(k))* and the contribution of the parent.

Two important quantities have been introduced in this process:

- $\vec{\Omega}_{k/L(k)}$ is the joint contribution to the angular velocity of body *k*.
- $\vec{V}_{k/L(k)}$ is the joint contribution to the translational velocity of point M_k

The concept of recursive calculation of the generalized velocities has also been introduced. The generalized velocities on body *k* can be computed by adding the contribution of the parent joint to the generalized velocities of body *L(k)*.

The contribution of each joint in the chain between body *k* and the ground can be found and expressed as:

$$\vec{V}_{Mk/0}=\sum_i \vec{V}_i q_i \tag{11}$$

$$\vec{\Omega}_{k/0}=\sum_i \vec{\Omega}_i q_i \tag{12}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Vector $\vec{V}_i$, which is associated with the kinematic variable q_i, is the "partial velocity" of the variable expressed at point M_k. It is configuration dependent, that is, it varies with the geometric variables of the joints located between body *k* and the ground.

The translational and accelerations can similarly be derived to obtain:

$$\vec{\Gamma}_{Mk/0}=\sum_i\left(\vec{V}_{Ok\,i}\dot{q}_i+\dot{\vec{V}}_{Ok\,i}q_i\right) \tag{13}$$

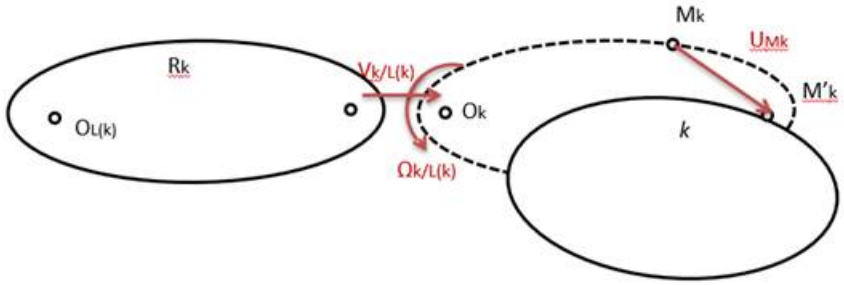
$$\dot{\vec{\Omega}}_{k/0}=\sum_i\left(\vec{\Omega}_i\dot{q}_i+\dot{\vec{\Omega}}_i q_i\right) \tag{14}$$

Flexible Shape Functions

We assume here that the body is not rigid anymore, but undergoes small elastic deflections.

To define the position of a point of body k, we use a floating reference rigid body and define a small displacement vector between the point and its reference position on the floating reference body.

Figure 9: Flexible Bodies Kinematics



With the assumption of small deflections and elastic behavior, sub-structuring can be used to reduce the flexible body to a small set of DOFs. We will define a set of generalized coordinates q_i such that:

$$\{u_M\}=\sum_1^n\{N_{i\,M}\}y_i=[N]u$$

The global position of M' becomes:

$$\overrightarrow{OM'}=\overrightarrow{OO_k}+\overrightarrow{O_kM}+\overrightarrow{MM'}=\overrightarrow{OO_k}+\overrightarrow{O_kM}+[N]u \tag{15}$$

The global velocity of M' is expressed by:

$$\vec{V}_{M_{k/0}}=\vec{V}_{O_{k/0}}+\vec{\Omega}_{k/0}\wedge\overrightarrow{O_kM}+\vec{\Omega}_{k/0}\wedge[N]u+[N]\dot{u} \tag{16}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Compared to the rigid case, the translational shape functions are modified to be expressed at the flexible point location, and the flexible generalized coordinates now contribute to the shape functions.

The basis of vectors [*N*] is obtained using a Component Mode Synthesis analysis with Fixed Interfaces (see Component Mode Synthesis (CMS) in the *Theory Reference* for more details). Master nodes are created for each joint connected to the condensed part. The internal modes and attachment modes Φ are orthogonalized to form the N basis.

The point O_k can be any point in the condensed part. However, in practice, it can be either on a joint or on the center of gravity of the condensed part.

Note:

Before simulating a model that contains a condensed part, the Rigid Dynamics solver first runs a sanity check. An eigenvalue decomposition called free-free modal analysis in the log is calculated with free master nodes based on the reduced mass and stiffness matrices provided by the MAPDL solver.

The purpose of this decomposition is to check that the condensed part contains exactly six rigid modes, which are near zero frequency. The first flexible frequency is used to estimate the time step required by the Rigid Dynamics analysis, which must be smaller than the inverse of that frequency.

Equations of Motion

Equations of Motion for Rigid Bodies

Many methods are available to derive the equations of motion, such as Newton Euler equations, Gibbs-Appell equations, and Lagrange equations.

The combination of Gibbs-Appell equations with generalized velocities is often referred to as Kane's equations [KAN61 (p. 652)]. Kane's equations are used for this example.

Open Loop Equations of Motion

The positional variation of a point M_k on body *k* is written as a reduction point using the origin of the body O_k :

$$\delta\vec{M}_k=\delta\vec{O}_k+\delta\vec{\Omega}_{k/0}\wedge\overrightarrow{O_kM_k} \tag{17}$$

Similarly, the translational acceleration of point M_k can be expressed using reduction point O_k :

$$\vec{\Gamma}_{Mk/0}=\sum_i\left(\vec{V}_{Oki}\dot{q}_i+\dot{\vec{V}}_{Oki}q_i\right)+\sum_i\left(\vec{\Omega}_i\dot{q}_i+\vec{\Omega}_iq_i\right)\wedge\overrightarrow{O_kM_k}+\vec{\Omega}_iq_i\wedge(\vec{\Omega}_iq_i\wedge\overrightarrow{O_kM_k}) \tag{18}$$

The virtual work of the acceleration can be formed and integrated over body *k*, and summed over the bodies as follows:

$$\sum_k\int_k\vec{\Gamma}_{Mk/0}.\delta\vec{M}_kdm \tag{19}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The integration over the body leads to integrating quantities as follows:

$$\int_k \overrightarrow{O_k M_k}\ dm \tag{20}$$

These terms can be easily pre-calculated as follows:

$$\int_k \overrightarrow{O_k M_k}\ dm = m_k \overrightarrow{O_k G_k} \tag{21}$$

In this equation, M_k stands for the mass of body *k*, and G_k stands for the center of gravity of that body. Other terms lead to:

$$\int_k \overrightarrow{O_k M_k}\ \hat{}\ v\ \hat{}\ \overrightarrow{O_k M_k}\ dm \tag{22}$$

where v is a constant vector. Those terms can be expressed as a function of the inertia tensor of body *k*.

Similarly, the virtual work of external distributed forces is computed as follows:

$$\int \vec{F}_{ext} . \delta \vec{M}_k \tag{23}$$

Finally, the open loop equations of motion lead to the following algebraic system:

$$M\{\dot{q}\} = F \tag{24}$$

Both the mass matrix *M* and the force vector *F* are dependent on the geometric variables and time *t*. The force vector is also a function of the generalized velocities.

$$M(g,t)\{\dot{q}\} = F(g,q,t) \tag{25}$$

When the mass and inertia properties of a rigid body are not constant, the force vector includes some additional terms dependent on the mass matrix time derivatives $\dot{M}$.

Equations of Motion with Flexible Bodies

Assuming that body k is flexible, the variation of the position of a point M'_k on body k is written, using the origin of the body O_k as a reduction point:

$$\vec{\delta}_{M'} = \vec{V}_m \delta q_m + \vec{\Omega}_m \delta q_m \wedge \left(\overrightarrow{O_k M} + [N]u \right) + \vec{N}_j \delta u_j \tag{26}$$

Similarly, the translational acceleration of point M'_k can be expressed using a reduction point O_k

$$\overrightarrow{\Gamma_{M'/O}} = \vec{V}_l \ddot{q}_l + \vec{V}_l \dot{q}_l + \left(\vec{\Omega}_l \ddot{q}_l + \vec{\Omega}_l \dot{q}_l \right) \wedge \left(\overrightarrow{O_k M} + [N]u \right) + \vec{\Omega}_l \dot{q}_l \wedge \left(\vec{\Omega}_l \dot{q}_l \wedge \overrightarrow{O_k M'} \right) + 2.* \vec{\Omega}_l \dot{q}_l \wedge \vec{N}_i \dot{u}_i + \vec{N}_i \ddot{u}_i \tag{27}$$

As in the case of rigid bodies, the virtual work of the acceleration can be formed and integrated over body k, and summed over the bodies:

$$\sum_k \int_k \vec{\Gamma}_{M'_{k/0}} . \overrightarrow{\delta M'}_k dm \tag{28}$$

In presence of flexible bodies, the equations of motion are modified by 2 sets of terms:

- Terms that involve only the set of flexible degrees of freedom only,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Coupling terms, involving flexible degrees of freedom and rigid degrees of freedom.

Please refer to [SHA13 (p. 652)] for more detailed information about the equations of motion.

Because the equilibrium is written on the current (deformed) configuration, the mass matrix and right hand side depend on the flexible degrees of freedom. To avoid having to go back to the finite element model to compute these integrals, these terms are decomposed over a basis of invariant terms, which are computed only once in the generation pass.

These invariants are expressed below. These terms are approximated using a lumped mass approach.

$$I_1 = \int \vec{N}_i dm \approx \sum_{i=0}^{n} M_{i,i} \cdot \Phi(i) \tag{29}$$

$$I_2 = \int \overrightarrow{OM} \wedge \vec{N}_i dm \approx \sum_{i=0}^{n} M_{i,i} \cdot \left(\vec{P}_i \wedge \overrightarrow{\Phi(i)} \right) \tag{30}$$

$$I_3 = \int \overrightarrow{OM} \cdot \vec{N}_i dm \approx \sum_{i=0}^{n} M_{i,i} \cdot \left(\vec{P}_i \cdot \overrightarrow{\Phi(i)} \right) \tag{31}$$

$$I_4 = \int \left(\vec{N}_i \right)_x \cdot \overrightarrow{OM} dm \approx \sum_{i=0}^{n} M_{i,i} \cdot \left(\Phi_i x \vec{P} \right) \qquad resp.\ y,z \tag{32}$$

$$I_5 = \int \overrightarrow{N_i N_j} dm \approx \sum_{i=0}^{n} \sum_{j=0}^{m} M_{i,i} \cdot \Phi_i \cdot \Phi_j \tag{33}$$

$$I_6 = \int \vec{N}_i \wedge \vec{N}_j dm \approx \sum_{i=0}^{n} \sum_{j=0}^{m} M_{i,i} \cdot \Phi_i \wedge \Phi_j \tag{34}$$

Where the *Φ(i)* are the Component Mode Synthesis base vectors.

Closed Loop Equations of Motion

When the model has some closed loops, not all joints can be treated as topological joints, thus requiring constraint equations to be added to the system. These constraint equations are usually written in terms of velocities as follows:

$$B(g)\{q\} = H(g,t) \tag{35}$$

Each kinematic joint generates up to six of these equations, depending on the motion direction that the joint fixes.

To be introduced in the equations of motion, a time derivative of these equations must be written as follows:

$$B(g)\{\dot{q}\} = \dot{H}(g,t) = G(g,t) \tag{36}$$

The equations of motion for the closed loop system become:

$$M(g,t)\{\dot{q}\} = F(g,q,t) \tag{37}$$

Subject to:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$B(g)\{\dot{q}\}=\dot{H}(g,t)=G(g,t) \tag{38}$$

An additional scalar variable λ (called a Lagrange Multiplier) is introduced for each constraint equation. These constraint equations are introduced in the algebraic system, which then becomes:

$$\begin{pmatrix} M & B^T \\ B & 0 \end{pmatrix}\begin{Bmatrix} \dot{q} \\ \lambda \end{Bmatrix}=\begin{Bmatrix} F \\ G \end{Bmatrix} \tag{39}$$

M, B, F, and G can be formed from a set of known geometric variables and kinematic variable values. The above system can be resolved, providing both accelerations $\dot{q}$ and Lagrange multipliers λ.

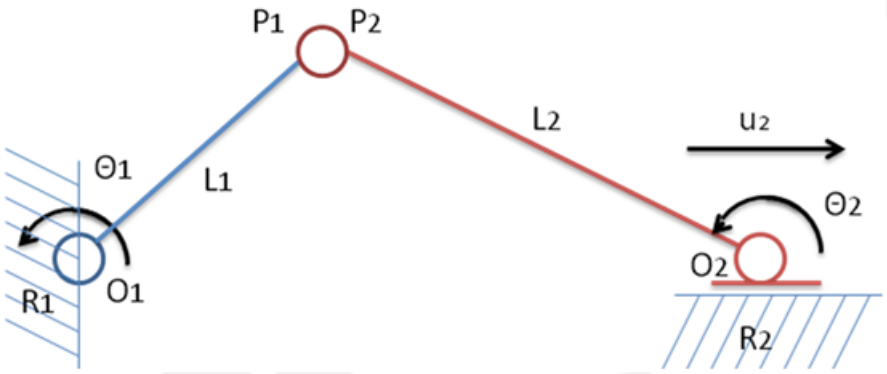
These Lagrange multipliers can be interpreted as constraint forces, equivalent to the amount of force needed to prevent motion in the direction of the constraint equations.

Redundant Constraint Equations

The system matrix shown in Equation 39 (p. 636) has size n+m where n is the number of degrees of freedom, and m is the number of constraint equations in *B*. The mass matrix *M* is usually positive-definite, but the full matrix including the constraint equation will retain that property only if there are no redundant constraint equations in *B*.

The constraint equations are applied to the piston/crankshaft system shown below to demonstrate how the *B* matrix can contain redundant constraint equations.

Figure 10: Crankshaft Mechanism



The revolute joint between point P1 on body 1 and point P2 on body 2 generates five constraint equations. For the sake of simplicity, these equations are written below in the global coordinate system, even if it is not always possible in general cases. The equations are:

1. $\left(\vec{v}_{P_2}-\vec{v}_{P_1}\right).\vec{x}=0$
2. $\left(\vec{v}_{P_2}-\vec{v}_{P_1}\right).\vec{y}=0$
3. $\left(\vec{v}_{P_2}-\vec{v}_{P_1}\right).\vec{z}=0$
4. $\left(\vec{\Omega}_2-\vec{\Omega}_1\right).\vec{x}=0$
5. $\left(\vec{\Omega}_2-\vec{\Omega}_1\right).\vec{y}=0$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

These equations must be projected on the degrees of freedom. This is achieved in the code by writing the shape functions on each body on points P1 and P2:

$$\vec{v}_{P_2} = \dot{u}_2\vec{x} + \dot{\Theta}_2\vec{z} \wedge \overrightarrow{O_2P_2} \tag{40}$$

$$\vec{\Omega}_2 = \dot{\Theta}_2\vec{z} \tag{41}$$

and:

$$\vec{v}_{P_1} = \dot{\Theta}_1\vec{z} \wedge \overrightarrow{O_1P_1} \tag{42}$$

$$\vec{\Omega}_1 = \dot{\Theta}_1\vec{z} \tag{43}$$

Replacing the velocities in the five constraint equations leads to:

1. $\dot{u}_2 + \dot{\Theta}_2 L_2 sin\Theta_2 = \dot{\Theta}_1 L_1 sin\Theta_1$
2. $\dot{\Theta}_2 L_2 cos\Theta_2 = \dot{\Theta}_1 L_1 cos\Theta_1$
3. $0 = 0$
4. $0 = 0$
5. $0 = 0$

The five equations above only generate two nontrivial constraints. The third equation indicates that the mechanism cannot shift along the z axis. It also indicates that the mechanism cannot be assembled if the z-coordinate of O_2 and O_2 are not the same. Similarly, the fourth and fifth equations indicate that the orientation of the axis of the revolute joint in P1/P2 is already entirely dependent on the axis of the two other revolute joints. A manufacturing error in the parallelism of the axis would result in a model that cannot be assembled. As such, this system is redundant.

Because introducing the five equations into Equation 39 (p. 636) would make the system matrix singular, some processing must be done on the full set of equations to find a consistent set of equations. Equations that are trivial need to be removed, as well as equations that are colinear. An orthogonalization technique is used to form a new set of equations that keep the matrix invertible. The matrix is decomposed into two orthogonal matrices, B_f and R:

$$[B] = [B_f][R] \tag{44}$$

where the $[B_f]$ matrix has a full rank and $[R]$ is a projection matrix. This matrix can then used in Equation 39 (p. 636):

$$\begin{pmatrix} M & B_f{}^T \\ B_f & 0 \end{pmatrix} \begin{Bmatrix} \dot{q} \\ \lambda_f \end{Bmatrix} = \begin{Bmatrix} F \\ G \end{Bmatrix} \tag{45}$$

Joint Forces Calculation

A benefit of using Kane's equations and relative parameters is that joint forces in topological joints are eliminated from the algebraic system. Joint forces can be calculated explicitly by writing the dynamic equilibrium of each body recursively, starting from the leaves of the tree associated with the connection graph, with the unknown being the body parent joint's forces and torque.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When the system has redundancies, that is, the [*B*] matrix does not have a full rank, some forces cannot be calculated. In the crankshaft example, no information is available in the forces developing in the revolute joint in P1/P2 in the z direction, and the moments cannot be calculated in this joint. These values will be reported as zero, but it is recommended that you avoid such situations by releasing some of the degrees of freedom in the system.

Time Integration with Explicit Runge-Kutta

Equation 25 (p. 634) (open loop) and Equation 39 (p. 636) (closed loop) provide a relation between generalized accelerations $\{\dot{q}\}$ and generalized velocities {q}.

Equation 7 (p. 629) provides a relation between generalized velocities $\{q\}$ and the time derivatives of the geometry variables $\{\dot{g}\}$

These two sets of equations form a system of first order explicit ordinary differential equations (ODE).

$$\dot{y}=f\left(y,t\right) \tag{46}$$

This system is integrated using the explicit Runge-Kutta method RK4.

RK4 Method

The fourth order method is based on four estimations. Given an initial value *y* at time value *t*, and a time step value *dt*, the following four estimations are formed:

$$\dot{y}_1=f\left(y,t\right) \tag{47}$$

$$\dot{y}_2=f\left(y+\frac{dt}{2}\dot{y}_1,t+\frac{dt}{2}\right) \tag{48}$$

$$\dot{y}_3=f\left(y+\frac{dt}{2}\dot{y}_2,t+\frac{dt}{2}\right) \tag{49}$$

$$\dot{y}_4=f\left(y+dt\dot{y}_3,t+dt\right) \tag{50}$$

A fourth order approximation of *y(t+dt)* is given by:

$$y\left(t+dt\right)\simeq y+\frac{dt}{6}\left(\dot{y}_1+2\dot{y}_2+2\dot{y}_3+\dot{y}_4\right) \tag{51}$$

Adaptive Time Stepping (for explicit time integration)

Time step *dt* must be chosen carefully for the integration of the ODEs to ensure that it is stable (that is, not becoming exponentially large), and accurate (that is, the difference between the approximation of the solution and the exact solution is controlled).

RK4 is conditionally stable, meaning that stability can be guaranteed if the time step is small enough. While the algorithm is accurate when it is stable, the time step chosen must be large enough to maintain computational efficiency.

For both integration schemes, quantifying the amount of kinetic energy contained in the highest order term of the polynomial approximation can give a good indication of whether the time step should be reduced or increased.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If the energy in the high order term is too large, it is likely that the approximation is inaccurate, and the time step should smaller.

If this energy is significant and controlled, the time step can be accepted, but the time step used will be smaller.

If the energy is low, then the next time step can be increased.

Rigid body systems usually have relatively slow motion, but the following factors can lead to smaller time steps:

- Existence of stiff springs and bushing in the model
- Three-dimensional rotations
- Proximity to geometrically singular configurations, such as the top, dead center position of a piston/crankshaft mechanism

These factors imply that the optimal time step varies with the system velocities and configuration, and thus cannot be determined before running the solution. As a consequence, automatic time stepping generally should not be turned off.

When automatic time stepping is used, the energy balance of the system is maintained within the tolerance that is requested. Note that impacts and shock can be non-conservative, and will affect the energy balance. This loss during impact is detailed in Contact and Stops (p. 645).

Implicit Generalized-α Method

Implicit Generalized-α Method

This family of methods was initially developed by Chung and Hilbert for the resolution of dynamics in the context of computational mechanics of solids. Cardona and Géradin adapted the method to compute the dynamics of multibody systems. Many extensions have been developed in the past, such as the extension developed by O. Brüls and M. Arnold for dynamics equations formulated as an index-3 DAE. The dynamics is written at time t_{n+1} as:

$$\begin{cases} M\ddot{q}_{n+1} = F(q_{n+1}, \dot{q}_{n+1}, t_{n+1}) + G^T(q_{n+1})\lambda_{n+1} \\ g(q_{n+1}) = 0 \end{cases} \tag{52}$$

The acceleration-like variable a_n is defined by the recurrence relation as:

$$(1-\alpha_m)a_{n+1} + \alpha_m a_n = (1-\alpha_f)\ddot{q}_{n+1} + \alpha_f \ddot{q}_n \tag{53}$$

At the beginning of the simulation, this variable is initialized as $a_0 = \ddot{q}_0$. The following difference equations relate q_{n+1}, $\dot{q}_{n+1}$, and a_n, a_{n+1}:

$$\begin{cases} q_{n+1} = q_n + h\dot{q}_n + h^2\left(\frac{1}{2} - \beta\right)a_n + h2\beta a_{n+1} \\ \dot{q}_{n+1} = \dot{q}_n + h(1-\gamma)a_n + h\gamma a_{n+1} \\ g(q_{n+1}) = 0 \end{cases} \tag{54}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

where the constants of α_f, α_m, β, and γ are suitably chosen so that the scheme is stable. The algorithm is unconditionally stable if the coefficients are chosen such that for $\rho_\infty<1$,

$$\begin{cases} \alpha_m=\frac{2\rho_\infty-1}{\rho_\infty+1}<\frac{1}{2} \\ \alpha_f=\frac{\rho_\infty}{\rho_\infty+1}<\frac{1}{2} \\ \gamma=\frac{1}{2}+\alpha_f-\alpha_m>\frac{1}{2} \\ \beta=\frac{1}{4}\left(\gamma+\frac{1}{2}\right)^2 \end{cases} \tag{55}$$

The scheme is based on a prediction step and a correction step where some Newton iterations are performed in order to solve the dynamical and the constraint residuals R_q and R_λ, defined by:

$$\begin{cases} R_q(q_{n+1}^k,\dot{q}_{n+1}^k,t_{n+1})=M(q_{n+1},t_{n+1})\ddot{q}_{n+1}-F(q_{n+1},\dot{q}_{n+1},t_{n+1})-G^T(q_{n+1})\lambda_{n+1}=0 \\ R_\lambda(q_{n+1})=g(q_{n+1})=0 \end{cases} \tag{56}$$

The Newton iterations solve the following linear system:

$$\begin{pmatrix} \beta' M(q_{n+1}^k,t_{n+1})+\gamma' C_t\left(q_{n+1},\dot{q}_{n+1}^k,t_{n+1}\right)+K_t\left(q_{n+1}^k,\dot{q}_{n+1}^k,t_{n+1}\right) & -G^T\left(q_{n+1}^k\right) \\ G\left(q_{n+1}^k\right) & 0 \end{pmatrix}\begin{pmatrix}\Delta q \\ \Delta\lambda\end{pmatrix}=-\begin{pmatrix} R_q\left(q_{n+1}^k,\dot{q}_{n+1}^k,t_{n+1}\right) \\ R_\lambda\left(q_{n+1}^k\right)\end{pmatrix} \tag{57}$$

where $\beta'=\frac{1-\alpha_m}{h^2\beta(1-\alpha_f)}$, $\gamma'=\frac{\gamma}{h\beta}$, M is the inertia matrix, $K_t=\frac{\partial\left(M\ddot{q}-F+G^T\lambda\right)}{\partial q}$ is the stiffness matrix, and $C_t=-\frac{\partial F}{\partial\dot{q}}$ is the damping matrix, and *k* denotes the number of the Newton iterations. Note that this algorithm maintains the constraints at the position level, but it can also be reformulated to write the constraints at the velocity level or the acceleration level.

When considering the constraints at the velocity level, the problem is formulated as an index-2 DAE, whose discretization is given by the first two equations of Equation 54 (p. 639), as well as:

$$G\left(q_{n+1}\right)v_{n+1}=0 \tag{58}$$

In this case, the correction step is performed using Newton iterations to solve the following linear system:

$$\begin{pmatrix} \beta' M(q_{n+1},t_{n+1})+\gamma' C_t\left(q_{n+1},\dot{q}_{n+1},t_{n+1}\right)+K_t\left(q_{n+1},\dot{q}_{n+1},t_{n+1}\right) & -G^T\left(q_{n+1}\right) \\ G\left(q_{n+1}\right) & 0 \end{pmatrix}\begin{pmatrix}\Delta\dot{q} \\ \Delta\lambda\end{pmatrix}=-\begin{pmatrix} R_q\left(q_{n+1}^k,\dot{q}_{n+1}^k,t_{n+1}\right) \\ G\left(q_{n+1}\right)v_{n+1}\end{pmatrix} \tag{59}$$

where:

$$\beta'=\frac{1-\alpha_m}{h\beta(1-\alpha_f)}$$

$$\gamma'=\frac{h\beta}{\gamma}$$

The integration error is estimated using the methodology proposed by Géradin and Cardona for the **HHT** schemes family. The exact value of the positions vector can be approximated using a truncated Taylor series development around t_n as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

$$q_e(t_n+h)=q_n+h\dot{q}_n+\frac{h^2}{2}\ddot{q}_n+\frac{h^3}{6}\dddot{q}_n+O(h^4) \tag{60}$$

The integration error is computed as:

$$err=q_{n+1}-q_e(t_n+h) \tag{61}$$

By substituting the expression q_{n+1} from Equation 54 (p. 639) into Equation 61 (p. 641), we obtain:

$$err=h^2\left(\frac{1}{2}-\beta\right)a_n+h^2\beta a_{n+1}-\frac{h^2}{2}\ddot{q}_n-\frac{h^3}{6}\dddot{q}_n+O(h^4) \tag{62}$$

The third derivative $\dddot{q}_n$ of the position can be approximated by:

$$\dddot{q}_n=\frac{\ddot{q}_{n+1}-\ddot{q}_n}{h}+O(h) \tag{63}$$

By substituting a_{n+1} and $\dddot{q}_n$ into Equation 62 (p. 641), and by using the relations in Equation 54 (p. 639), we obtain:

$$err=q_{n+1}-q_n-\frac{1}{h}\dot{q}_n-\frac{h^2}{3}\ddot{q}_n-\frac{h^2}{6}\ddot{q}_{n+1}+O(h^4) \tag{64}$$

Adaptive Time Stepping (for implicit time integration)

Much like RK4, the Implicit Generalized-α method also implements an adaptive time stepping strategy based on the energy conservation. However, the order of the Implicit Generalized-α method is lower than RK4. Consequently, the Implicit Generalized-α method may require smaller time steps than explicit method to achieve the same accuracy.

Unlike RK4, the Implicit Generalized-α method is unconditionally stable. This method remains stable even for large time steps, regardless the accuracy. As such, it is particularly suitable to address situations where the explicit RK4 method requires small time steps. The energy tolerance control can be relaxed in these situations, and the time step is driven only by the Newton-Raphson convergence. If a large number of iterations was required for the last time step, the time step decreases. Conversely, if fewer iterations were required for convergence of the previous time step, the time step increases.

Stabilized Implicit Generalized-α Method

At the end of each time step, violated constraints equations are corrected as explained in Geometric Correction (p. 643). However, this correction perturbs the dynamics of the system and may introduce artificial energy to the system, which leads to vibrations when flexible bodies are involved. These artificial vibrations cause the time integration method to use tight time steps and more iterations which leads to long simulation times. The Stabilized Generalized-α method remedies this issue.

The equations of the dynamics are discretized as in Equation 59 (p. 640). A new Lagrange multiplier is introduced which satisfies the position-level constraints equations:

$$M(q_n)U_n-G^T(q_n)\mu_n=\vartheta(h^2) \tag{65}$$

Where U_n represents the position gap needed to vanish the violation $g(q_n)$. The correction of this position drift is achieved using:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\begin{pmatrix} M(q_n) & -G^T(q_n) \\ G(q_n) & 0 \end{pmatrix} \begin{pmatrix} U_n \\ \mu_n \end{pmatrix} = - \begin{pmatrix} 0 \\ g(q_n) \end{pmatrix} \tag{66}$$

This correction is followed by another dynamics solve using Equation 59 (p. 640). In this way, both the dynamics and the applied constraints are satisfied, which leads to more stable numerical systems.

Compared to the Generalized-α method, the Stabilized Generalized-α method uses larger average time steps and needs less iterations to converge.

Moreau-Jean Method

In time-stepping schemes, the formulation of the dynamics equations enables the scheme to simultaneously handle the smooth dynamics and non-smooth dynamics. The idea behind these schemes is to consider the dynamics equations as a measure differential inclusion. On the velocity level, they are expressed as:

$$\begin{cases} \dot{q} = v \\ M(q)\frac{dv}{dt} = F(q,v,t) + di \\ -di \in N_{T_C(q)}(v^+) \end{cases} \tag{67}$$

where di is the reaction measure.

In the case $C = \mathbb{R}^+$:

$$0 \le \lambda \perp g \ge 0 \Leftrightarrow -\lambda \in N_{\mathbb{R}^+}(g) \supseteq N_{T_{\mathbb{R}^+}(g)}(\dot{g}^+) \tag{68}$$

therefore:

$$\begin{cases} g > 0 \Rightarrow \dot{g}^+ \in T_{\mathbb{R}^+}(g) = \mathbb{R} \text{ and } N_{\mathbb{R}^+}(g) = \{0\} \\ g = 0 \Rightarrow \dot{g}^+ \in T_{\mathbb{R}^+}(g) = \mathbb{R}^+ \text{ and } N_{\mathbb{R}^+}(g) = \mathbb{R}^- \end{cases} \tag{69}$$

In the Moreau-Jean time stepping method (MJ Time Stepping (p. 1436)), unilateral contact is modeled with a Signorini condition at the velocity level, that is a complementarity between the contact impulse and the relative velocity, which ensures impenetrability. In order to handle multiple contacts with potentially changing status, an implicit algorithm is used to integrate the dynamics. As a simplified example, consider $F_{\text{int}}(q,v) = -C\dot{q} - Kq$ and $M(q) = M$. The discrete form of the equations of motion is:

$$\begin{cases} q(t_{i+1}) = q(t_i) + \int_{]t_i,t_{i+1}]} v^+ \, dt \\ M(v^+(t_{i+1}) - v^+(t_i)) = \int_{t_i}^{t_{i+1}} ((-Cv(t) - Kq(t)) + F_{ext}) \, dt + \int_{]t_i,t_{i+1}} di \end{cases} \tag{70}$$

The smooth terms may be discretized using a θ-method as:

$$\begin{cases} q_{i+1} = q_i + h[\theta v_{i+1} + (1-\theta) v_i] \\ \int_{t_i}^{t_{i+1}} (Cv^+ + Kq) \, dt = h(\theta(Cv_{i+1} + Kq_{i+1}) + (1-\theta)(Cv_i + Kq_i)) \\ \int_{t_i}^{t_{i+1}} F_{ext} \, dt = h(\theta(F_{ext})_{i+1} + (1-\theta)(F_{ext})_i \end{cases} \tag{71}$$

then:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

$$\begin{cases} q_{i+1}=q_i+h(\theta v_{i+1}+(1-\theta)v_i) \\ v_{i+1}=v_{\text{free}}+\hat{M}^{-1}p_{i+1} \\ \widehat{M}=M+h\theta C+h^2\theta^2 K \\ v_{\text{free}}=v_i+\hat{M}^{-1}(-hCv_i-hKq_i-h^2\theta K v_i+h[\theta(F_{ext})_{i+1}+(1-\theta)(F_{ext})_i]) \end{cases} \tag{72}$$

where $p_{i+1}\approx\int_{]t_i,t_{i+1}]}di$ is the approximation of the impulse over the time step. Note that the acceleration of the system is never explicitly computed because it becomes infinite for impulsive forces. The kinematic equations are discretized as follows for contact α:

$$\begin{cases} \dot{g}^{\alpha}_{n+1}=G^{\alpha}(q_{n+1})v_{n+1} \\ P^{\alpha}_{n+1}\approx\int_{]t_n,t_{n+1}}d\lambda^{\alpha} \\ p^{\alpha}_{n+1}=G^{\alpha T}P^{\alpha}_{n+1} \\ g^{\alpha}_{n+1}=g^{\alpha}_{n+1}+h(\theta\dot{g}^{\alpha}_{n+1}+(1-\theta)\dot{g}^{\alpha}_{n}) \\ \tilde{q}_{n+1}=q_n+\frac{h}{2}v_n \\ \text{if } g^{\alpha}(\tilde{q}_{n+1})\le 0 \text{ then } 0\le g^{\alpha}_{n+1}\perp P^{\alpha}_{n+1}\ge 0 \end{cases} \tag{73}$$

Finally, Newton's law is implicitly formulated as:

$$\begin{cases} \text{if } g^{\alpha}(\tilde{q}_{ni+1})\le 0 \text{ then } 0\le U^{\alpha}_{n+1}+eU^{\alpha}_{n}\perp P^{\alpha}_{n+1}\ge 0 \\ \text{if } g^{\alpha}(\tilde{q}_{ni+1})>0 \text{ then } P^{\alpha}_{n+1}=0 \\ U^{\alpha}_{n}=\dot{g}^{\alpha}_{n} \\ U^{\alpha}_{n+1}=\dot{g}^{\alpha}_{n+1} \end{cases} \tag{74}$$

Because the Moreau-Jean time stepping method is formulated in terms of non-smooth velocities, it better handles the acceleration discontinuities that can happen when the geometry is non-smooth. Consequentially, it is well-suited to work with mesh-based contact (p. 648).

Geometric Correction

Using relative parameters, the majority of joints are introduced in the system using their natural degrees of freedom. As a consequence, no matter how big the variation of the degrees of freedom is over the time step, these joints only allow motions that are consistent with the joint kinematics. For example, a revolute joint that has one single rotational degree of freedom can have an increment with a rotation of 3600 degrees in one time step, and it will still not generate out of plane motion, nor will the two points linked by the joint separate. Conversely, some joints are constraint equation based (all the graph closed-loop joints) and need special attention to satisfying proper joint kinematics.

The time integration schemes that are used provide a piecewise polynomial approximation of the solution. These schemes realize a polynomial approximation of the solution. The constraint equations such as those developed in the crankshaft example shown in Figure 10: Crankshaft Mechanism (p. 636) are not polynomial expressions of the geometric variables. Similarly the relation between kinematic variables and geometric variables, expressed as $\{\dot{g}\}=f(g,\vec{\Omega})$, is usually not polynomial. As a con-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

sequence, using for example the RK4 time integration, the constraint equations that are exactly satisfied in terms of accelerations at each of the Runge-Kutta estimations might not be satisfied in terms of velocities and positions at the end of the time step. After a number of time steps, closed loops will not be closed anymore, and points P1 and P2 in the crankshaft example will slowly drift away from each other.

To avoid these violations of constraint equations, various strategies can be used.

The drift of the violated constraints equations is corrected by a projection of the numerical solution. Two methods are available to perform it:

Pure Kinematic Method

As proposed by Dehombreux [DEH95 (p. 652)], this method is performed by projecting back the solution of the constraint equation.

Both positions and velocities can be projected back using this correction.

By projecting the solution, an increment of the geometry variables can be found iteratively:

$$B(g)\{\delta q\} = h_p \tag{75}$$

$$\delta g = f(g, \delta q) \tag{76}$$

$$g := g + \delta g \tag{77}$$

Note that the relation between the kinematic variables and the geometric variables is reused in an incremental form.

Because of the dependency of the constraint equations on the geometric variables, this solution is nonlinear, and must be performed until the increment δg is small enough.

Once the position has been corrected, another step can be done to correct velocities:

$$B(g)\{\Delta q\} = h_v \tag{78}$$

$$q := q + \Delta q \tag{79}$$

As these equations are not velocity dependent, there is no need to iterate on this system.

Inertia Matrix Method

In this method, geometry variables are calculated iteratively by solving:

$$\begin{pmatrix} M & B^T \\ B & 0 \end{pmatrix} \begin{pmatrix} \Delta q \\ \lambda \end{pmatrix} = - \begin{pmatrix} 0 \\ g \end{pmatrix} \tag{80}$$

Once convergence is reached, the velocity increments are obtained using

$$\begin{pmatrix} M & B^T \\ B & 0 \end{pmatrix} \begin{pmatrix} \Delta v \\ \lambda \end{pmatrix} = - \begin{pmatrix} 0 \\ \frac{dg}{dt} \end{pmatrix} \tag{81}$$

This method results from the minimization of kinetic energy. Using the inertia matrix for correction of violated constraints leads to more consistent results than when using the pure kinematic method. Furthermore, scaling with the inertia matrix M enables you to perform a resolution which is consistent with the metrics of the problem.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

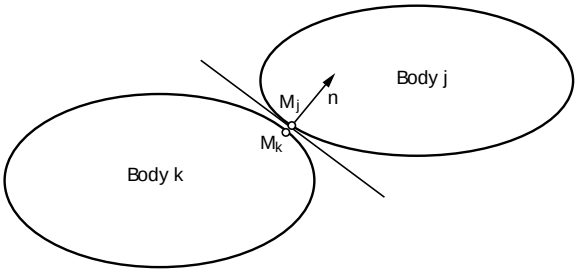
Contact and Stops

So far, the only interaction between bodies that has been considered was permanent joints; however, impacts and contacts will also play a significant role.

Contact Kinematics

The figure below depicts the contact between convex bodies *j* and *k*.

Figure 11: Contact Between Two Convex Bodies



The non-penetration equation below describes the contact between these bodies, and is written along the shared normal at the contact point:

$$\left(\frac{d\overrightarrow{OM_j}}{dt}-\frac{d\overrightarrow{OM_k}}{dt}\right).\vec{n}=0 \tag{82}$$

In this equation, the two points M_j and M_k are the points that minimize the distance between the two bodies, and thus are not material points, that is, their location varies over the bodies with time.

For more information on the definition of the contact point, refer to Pfeiffer [PFE96 (p. 652)] in References (p. 652).

Special Cases

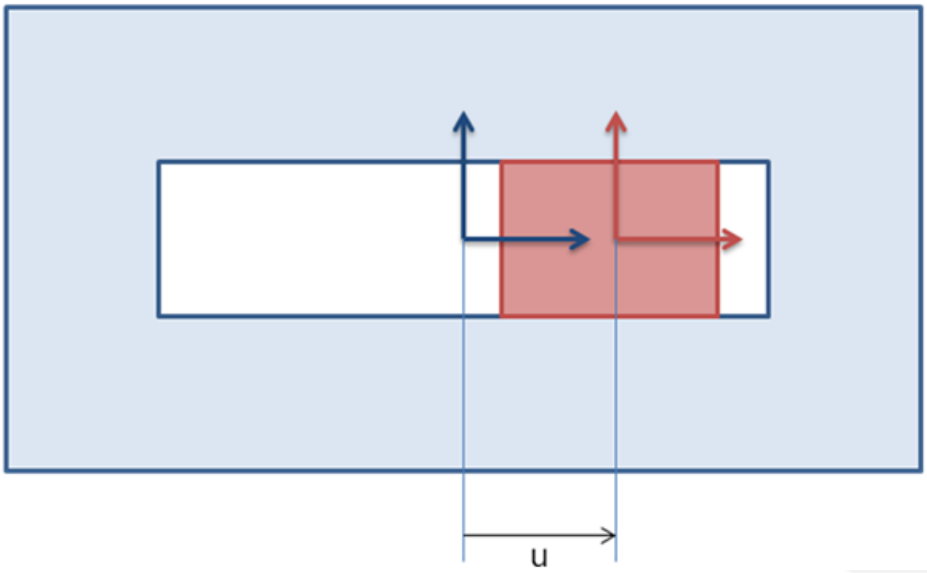
Some special cases are worth mentioning. For instance, when contact occurs in a joint between two bodies linked by that joint, the contact points become material points, and Equation 82 (p. 645) can become dependent on one single degree of freedom. Figure 12: Stops on a Translational Joint (p. 646) shows an example of stop on a translational joint. Both left and right vertical surfaces can impact the red body, but this translates very easily into a simple double inequality:

$$u_m \leq u \leq u_M \tag{83}$$

where subscript $_m$ stands for the minimum bound, and M stands for the maximum bound. The normal here is replaced by the projection on the joint degree of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 12: Stops on a Translational Joint

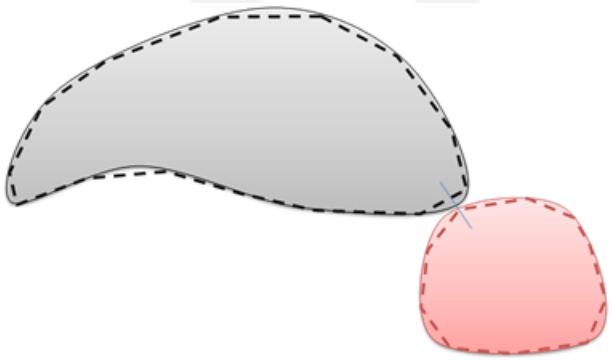


Another case of specialized contact geometry is the radial gap where contact points can be computed explicitly. In the general case of complex geometries, the strategy for computing the contact points and the impact times is more complex.

General Cases

In general cases, geometries that are potentially in contact are neither simple nor convex. It is however required to find the accurate position of the contact points between two bodies. Sometimes the contact point is unique, as shown in the figure below.

Figure 13: One Contact Point

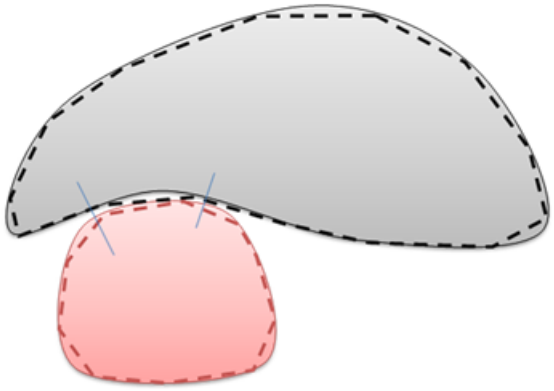


But for the same pair, the contact can occur in more than one point, as shown in the figure below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

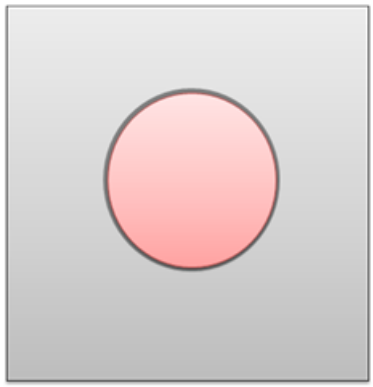
Published: 2023-10-23

Figure 14: Two Contact Points



Finally, the contact can exist along a full line for some geometries, or even on an entire surface, as shown in the figure below. In this case, there is an infinite number of contact points.

Figure 15: Cylinder/Cylinder Contact



Contact Formulation

Two bodies will impact when their distance is equal to zero. Once the distance is equal to zero and the bodies are touching, forces can develop in the contact. When the contact distance is greater than zero, there is no interaction between the bodies. Introducing interaction in the equations of motion results in the addition of inequalities to the system:

$$B_u(g)\{q\} \geq G_u(g,t) \tag{84}$$

where the subscript $_u$ stands for unilateral.

In the Rigid Dynamics solver, contact and stops always use a pure Lagrange formulation. Unlike the penalty based approaches, pure Lagrange prevents any penetration.

Note:

Although a pure Lagrange formulation prevents penetration, a small penetration (linear with the time step) may be visible when using the MJ Time Stepping time integration due to the lack of exact collision detection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Rigid Contact Detection

To control the density of contact points that will need to be adjusted, a surface mesh is used on the bodies that has contact defined. Mesh based contact points are first computed.

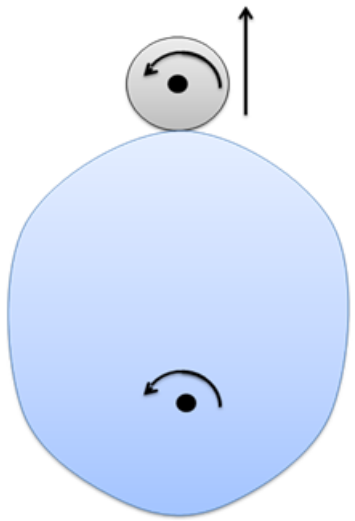
When the contact is geometry-based, these discrete points are then adjusted on the actual geometrical surfaces.

Based on whether the contact is geometry-based or mesh-based, the requirements in terms of mesh are different:

Geometry-Based Contact

It is important to understand that contact will create constraints between the two bodies. The relative motion between these two bodies varies in a 6-dimensional space, so 6 contact points at most will be used to constrain the relative motion of two bodies. These constraints will be added to already existing constraint, so contact can create additional redundancies. For example, two cams with parallel axis will contact along a line (as shown in the figure below). However, if the two axes are maintained parallel by existing joints in the model, one single point through the thickness of the cam is necessary to properly represent the kinematics of the assembly. To avoid useless calculation, the mesh through the thickness can be coarse.

Figure 16: Contact Requiring One Single Point



If the mesh is very refined, many points through thickness can satisfy the contact equations. An automatic filtering of the contact points will also be performed, but the position of the points through thickness might vary from one step to the next. This can cause some unexpected changes in the moment developed in the contact. To avoid this situation, it can be useful to modify the joints or the geometry itself, and include a draft angle in the cam profile extrusion for force the contact along a line.

Mesh-Based Contact

Similar to geometry-based contact, the mesh defines the density of contact points defined between the bodies. Because the points are on the mesh and not on the geometry, the contact

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

happens between faceted geometries. To avoid spikes in the forces, it is recommended that you refine the mesh further when contact is mesh-based.

Caution:

When the mesh-based contact detection method is used, the behavior of the contact is not symmetric. Results may change when the contact is flipped.

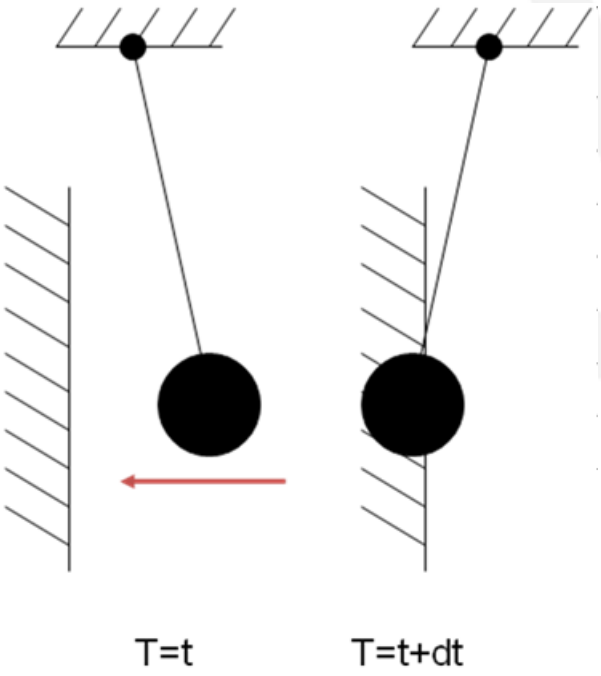
Using mesh-based contact with Runge-Kutta (p. 638) may lead to computationally expensive simulations. Mesh-based contact is recommended for use with the Moreau-Jean method (p. 642).

Time Integration and Contact

In the Rigid Dynamics solver, two alternate methods are available. The first method is suited for situations where contact is mostly intermittent, the second method is suited where contact is mostly established, for example the contact involved in a pair of gears.

Method 1: Event Capturing Contact Formulation

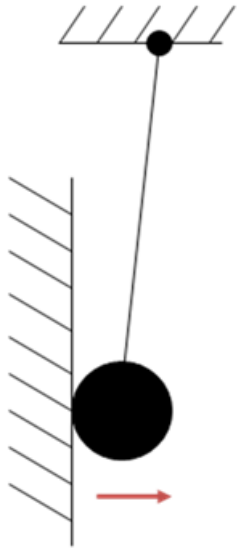
A second method of contact formulation is to detect the transition between the separated space of a given pair of bodies and the configuration where they are overlapping. The image below depicts a point mass approaching a separate wall, and the overlapping configuration following impact.



Determining the time of the transition using this point mass model involves advancing in time without introducing non-penetration constraint equations, and realizing at the end of the time step that the penetration is not acceptable. By using the polynomial interpolation that the time integration scheme provides over the time step, the moment where the penetration reaches zero can be found fairly accurately. This time value can be expressed as a fraction of the time

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

step. To determine this time value, find α such that $p(t+\alpha\Delta t)=0$ where p is the penetration distance.



$T=t0+\alpha dt$

Advancing in time up to $\alpha\Delta t$ will position the system exactly at the impact time and position, where an impact occurs between the bodies. This impact is assumed to have a very short duration, orders of magnitude smaller than the simulation time. During the impact, the interaction forces between the bodies are first increasing in a compression phase, and then decreasing in the expansion phase until they vanish entirely. This impact will lead to a certain amount of energy loss determined by the material of the bodies interacting.

Newton's impact laws are idealized in this impact process. They relate the relative velocity before the impact to the "bouncing" velocity after the impact using a restitution factor. This restitution factor varies from zero to one. A restitution factor of one indicates that the normal velocity after the impact is equal to the velocity before the impact.

$$v^{+}=-v^{-} \tag{85}$$

Where the superscript $^{+}$ represents quantities after the impact, and the superscript $^{-}$ represents quantities before the impact.

A restitution of zero leads to:

$$v^{+}=0 \tag{86}$$

And the general formula will be:

$$v^{+}-v^{-}=-(1+r)v^{-} \tag{87}$$

where r is the restitution factor.

This equation is written as a scalar equation at the impact point. Combined with the conservation of momentum it leads to the following system:

$M(g,t)\{\Delta q\}=\{0\}$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

$B(q)\{\Delta q\}=0$ for all permanent equations and active contacts, and $B(q)\{\Delta q\}=-(1+r)v^{-}$ for the impacting contact.

Each impact with a restitution factor less than one will introduce an energy loss in the system. In a model with multiple imperfect impacts over time, the total energy will be constant piecewise with a drop at each impact.

Runge-Kutta 4, Generalized-α, and Stabilized Generalized-α integration methods are event capturing schemes (see the table below).

Method 2: Event Absorbing Contact Formulation

When the density of events increases (small bounces, with phases where the contact is sliding, such as the contact in a pair of gears) the event detection method starts to lose efficiency and robustness, for two reasons:

- It requires accurate detection of the transition time, thus forcing the reduction of the time step to avoid missing changes. If events are changed, inconsistencies between the state of the contact (whether the contact is touching, separated, or in-between) and the actual relative position of the bodies.
- Each event processing phase has a computational cost.

To work around these difficulties, it is possible to reformulate what happens during the time step in terms of variation of velocities. These variations come from both smooth dynamics (the variation due to finite accelerations) and from non-smooth dynamics (the variation due to infinite accelerations over a zero duration, which corresponds to a shock). For specific theory information, see Moreau-Jean Method (p. 642).

MJ Time Stepping is an event absorbing scheme.

Summary of Time Integration Methods

The following table summarizes the characteristics of the Time Integration Types available for the Rigid Dynamics solver and the corresponding recommended setting for the Contact Detection.

Time Integration type		RK4, Generalized-α, Stabilized Generalized-α	MJ Time Stepping
Event treatment		Event capturing	Event absorbing
RBD Contact Detection	Geometry based	Best choice	Not recommended
	Mesh based	Not recommended	Best choice

Note:

When the Time Integration Type is set to Program Controlled, the Rigid Dynamics solver chooses the time integration type based only on the use of flexible parts, regardless of the contact definition. If the contact uses only rigid bodies, the solver will use RK4. If there are any Condensed Parts, it will use Stabilized Generalized-α. The contact definition has no influence on this logic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The use of mesh-based contact detection with explicit time integration (RK4) is likely to lead to solver problems and is not recommended.

References

1. [CAS90] J. R. Cash, A. H. Karp, "A variable order Runge-Kutta method for initial value problems with rapidly varying right-hand sides", ACM Transactions on Mathematical Software, 1990, Vol 16, p.201-222
2. [DEH95] P. Dehombreux, "Simulation Dynamique des systemes multicorps constraints", These de Doctorat, Faculte Polytechnique de Mons
3. [PFE96] F. Pfeiffer, C. Glocker, "Multibody Dynamics with Unilateral Contacts", Wiley, New. York, 1996.
4. [KAN61] Kane, T.R., Dynamics of nonholonomic systems, Transactions of the ASME, J. App. Mech., 1961, Vol. 28, December, p.574-578
5. [WIT77] Wittenburg, J., Dynamics of Systems of Rigid Bodies. Stuttgart. B. G. Teubner. 1977.
6. [SHA13] Shabana, A., Dynamics of Multibody Systems, 4th edition, Cambridge University Press, 2013

Static Structural Analysis

Introduction

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. A static structural load can be performed using the Ansys, Samcef, or ABAQUS solver. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)

Point to Remember

Note the following for this analysis type:

- A static structural analysis can be either linear or nonlinear. All types of nonlinearities are allowed - large deformations, plasticity, stress stiffening, contact (gap) elements, hyperelasticity and so on. This chapter focuses on linear static analyses, with brief references to nonlinearities. Details of how to handle nonlinearities are described in Nonlinear Controls (p. 1468).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- When 2D geometry is used, Generalized Plane Strain is not supported for the Samcef or ABAQUS solver.
- Note that available nonlinearities can differ from one solver to another.
- As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Define Engineering Data

Material properties can be linear or nonlinear, isotropic or orthotropic, and constant or temperature-dependent. You must define stiffness in some form (for example, Young's modulus, hyperelastic coefficients, and so on). For inertial loads (such as **Standard Earth Gravity** (p. 1509)), you must define the data required for mass calculations, such as density.

Define Part Behavior

You can define a **Point Mass** (p. 901) for this analysis type.

A "rigid" part is essentially a point mass connected to the rest of the structure via joints. Hence in a static structural analysis the only applicable loads on a rigid part are acceleration and rotational velocity loads. You can also apply loads to a rigid part via joint loads. The output from a rigid part is the overall motion of the part plus any force transferred via that part to the rest of the structure. Rigid behavior cannot be used with the Samcef or ABAQUS solver.

If your model includes nonlinearities such as large deflection or hyperelasticity, the solution time can be significant due to the iterative solution procedure. Hence you may want to simplify your model if possible. For example you may be able to represent your 3D structure as a 2-D plane stress, plane strain, or axisymmetric model or you may be able to reduce your model size through the use of symmetry or antisymmetry surfaces. Similarly if you can omit nonlinear behavior in one or more parts of your assembly without affecting results in critical regions it will be advantageous to do so.

Define Connections

This analysis supports:

- Contact, joints, springs, beams, mesh connections, and end releases.
- (Samcef and ABAQUS solvers only), contacts, springs, and beams are supported. Joints are **not** supported.

Apply Mesh Controls/Preview Mesh

Provide an adequate mesh density on contact surfaces to allow contact stresses to be distributed in a smooth fashion. Likewise, provide a mesh density adequate for resolving stresses; areas where stresses or strains are of interest require a relatively fine mesh compared to that needed for displacement or nonlinearity resolution. If you want to include nonlinearities, the mesh should be able to capture the effects of the nonlinearities. For example, plasticity requires a reasonable integration point density (and therefore a fine element mesh) in areas with high plastic deformation gradients.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Establish Analysis Settings

For simple linear static analyses you typically do not need to change these settings. For more complex analyses the basic **Analysis Settings** include:

Large Deflection (p. 1435)

Large Deflection is typically needed for slender structures. Use large deflection if the transverse displacements in a slender structure are more than 10% of the thickness.

Small deflection and small strain analyses assume that displacements are small enough that the resulting stiffness changes are insignificant. Setting **Large Deflection** to **On** will take into account stiffness changes resulting from changes in element shape and orientation due to large deflection, large rotation, and large strain. Therefore the results will be more accurate. However this effect requires an iterative solution. In addition it may also need the load to be applied in small increments. Therefore, the solution may take longer to solve.

You also need to turn on large deflection if you suspect instability (buckling) in the system. Use of hyperelastic materials also requires large deflection to be turned on.

Step Controls for Static and Transient Analyses (p. 1418)

Step Controls are used to i) control the time step size and other solution controls and ii) create multiple steps when needed. Typically analyses that include nonlinearities such as large deflection or plasticity require control over time step sizes as outlined in the Automatic Time Stepping (p. 1493) section. Multiple steps are required for activation/deactivation of displacement loads or pretension bolt loads. This group can be modified on a per step basis.

Note:

Time Stepping is available for any solver.

Rotordynamics Controls (p. 1490)

Specify these properties as needed when setting up a Rotordynamic analysis.

Output Controls (p. 1473)

Output Controls allow you to specify the time points at which results should be available for postprocessing. In a nonlinear analysis it may be necessary to perform many solutions at intermediate load values. However i) you may not be interested in all the intermediate results and ii) writing all the results can make the results file size unwieldy. This group can be modified on a per step basis except for **Stress** and **Strain**.

Nonlinear Controls (p. 1468)

Nonlinear Controls allow you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control. This group can be modified on a per step basis. If you are performing a *nonlinear* Static Structural analysis, the **Newton-Raphson Type** property becomes available. This property only affects nonlinear analyses. Your selections execute the Mechanical APDL **NROPT** command. The default option, **Program Controlled**, allows

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the application to select the appropriate **NROPT** option or you can make a manual selection and choose **Full**, **Modified**, or **Unsymmetric**.

See the Help section for the **NROPT** command in the *Mechanical APDL Command Reference* for additional information about the operation of the **Newton-Raphson Type** property.

Analysis Data Management (p. 1484)

Settings enable you to save specific solution files from the **Static Structural** analysis for use in other analyses. You can set the **Future Analysis** field to **Pre-Stressed Analysis** if you intend to use the static structural results in a subsequent **Harmonic Response**, **Modal**, or **Eigenvalue Buckling** (**Eigenvalue Buckling** is applicable to **Static Structural** systems only) analysis. If you link a structural system to another analysis type in advance, the **Future Analysis** field defaults to **Pre-Stressed Analysis**. A typical example is the large tensile stress induced in a turbine blade under centrifugal load. This causes significant stiffening of the blade resulting in much higher, realistic natural frequencies in a modal analysis. More details are available in the section Define Initial Conditions (p. 349).

Note:

Scratch Solver Files, **Save Ansys db**, **Solver Units**, and **Solver Unit System** are applicable to **Static Structural** systems only.

Define Initial Conditions

Initial condition is not applicable for **Static Structural** analyses.

Apply Loads and Supports

For a Static Structural analysis applicable loads include:

- Inertial (p. 1500)
- Structural (p. 1520)
- Interaction (p. 1521)
- Imported (p. 1736)

Applicable structural Supports (p. 1663), certain Conditions Type Boundary Conditions (p. 1697), as well as supported Direct FE Type Boundary Conditions (p. 1719) are also available.

For the Samcef and ABAQUS solvers, the following loads and supports are not available: Hydrostatic Pressure, Bearing Load, Bolt Pretension, Joint Load, Interface, Motion Loads, Compression Only Support, Elastic Support.

Loads and supports vary as a function of time even in a static analysis as explained in the Role of Time in Tracking (p. 1491). In a static analysis, the load's magnitude could be a constant value or could vary with time as defined in a table or via a function. Details of how to apply a tabular or function load are

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

described in Defining Boundary Condition Magnitude (p. 1771). In addition, see the Apply Loads and Supports (p. 354) section for more information about time stepping and ramped loads.

Note:

A static analysis can be followed by a "pre-stressed" analysis such as modal or linear (eigenvalue) buckling analysis. In this subsequent analysis the effect of stress on stiffness of the structure (stress-stiffness effect) is taken into account. If the static analysis has a pressure or force load applied on faces (3D) or edges (2D) this could result in an additional stiffness contribution called "pressure load stiffness" effect. This effect plays a significant role in linear (eigenvalue) buckling analyses. This additional effect is computed during the Eigenvalue Buckling analysis using the pressure or force value calculated at the time in the static analysis from which the perturbation occurs. See the Applying Pre-Stress Effects (p. 352) section for more information on this topic.

When using the Samcef or ABAQUS solver, **Direct FE** boundary conditions are not available.

Solve

When performing a nonlinear analysis you may encounter convergence difficulties due to a number of reasons. Some examples may be initially open contact surfaces causing rigid body motion, large load increments causing non-convergence, material instabilities, or large deformations causing mesh distortion that result in element shape errors. To identify possible problem areas some tools are available under **Solution Information** (p. 1833) object Details view.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

You can display contour plots of **Newton-Raphson Residuals** in a nonlinear static analysis. Such a capability can be useful when you experience convergence difficulties in the middle of a step, where the model has a large number of contact surfaces and other nonlinearities. When the solution diverges identifying regions of high Newton-Raphson residual forces can provide insight into possible problems.

Result Tracker (p. 1844) (applicable to **Static Structural** systems only) is another useful tool that enables you to monitor displacement and energy results as the solution progresses. This is especially useful in case of structures that possibly go through convergence difficulties due to buckling instability. **Result Tracker** is not available to the Samcef or ABAQUS solver.

Review Results

All structural result types (p. 2017) except frequencies are available as a result of a static structural analysis. You can use a **Solution Information** (p. 1833) object to track, monitor, or diagnose problems that arise during a solution.

Once a solution is available you can contour the results (p. 62) or animate the results (p. 2260) to review the response of the structure.

As a result of a nonlinear static analysis you may have a solution at several time points. You can use probes (p. 1946) to display the variation of a result item as the load increases. An example might be large deformation analyses that result in buckling of the structure. In these cases it is also of interest to plot

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

one result quantity (for example, displacement at a vertex) against another results item (for example, applied load). You can use the Charts (p. 1929) feature to develop such charts.

Steady-State Thermal Analysis

Introduction

You can use a steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineers often perform a steady-state analysis before performing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished. A steady-state thermal analysis can be performed using the Mechanical APDL, Samcef, or ABAQUS solver.

Important:

By default, the application does not write thermal gradient results to the result file. To have these results written to the results file, use a Command object (p. 2303) and insert the command **OUTRES**,**ERASE**.

Point to Remember

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects or temperature dependent convection coefficient also makes the analysis nonlinear.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag a **Steady-State Thermal**, **Steady-State Thermal (Samcef)**, or **Steady-State Thermal (ABAQUS)** template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Thermal Conductivity must be defined for a steady-state thermal analysis. Thermal Conductivity can be isotropic or orthotropic, and constant or temperature-dependent.

Attach Geometry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

There are no specific considerations for a steady-state thermal analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

Mechanical does not support Rigid Bodies in thermal analyses. For more information, see the *Stiffness Behavior* documentation for Rigid Bodies (p. 852).

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

In a thermal analysis only contact is valid. Any joints or springs are ignored.

With contact the initial status is maintained throughout the thermal analysis, that is, any closed contact faces will remain closed and any open contact faces will remain open for the duration of the thermal analysis. Heat conduction across a closed contact face is set to a sufficiently high enough value (based on the thermal conductivities and the model size) to model perfect contact with minimal thermal resistance. If needed, you can model imperfect contact by manually inputting a **Thermal Conductance** (p. 1253) value.

By default, Contact Results (p. 2043) (accessible through User Defined Results (p. 2192) via CONTSTAT or CONTFLUX – see the User Defined Results for the Mechanical APDL Solver (p. 2203) section.) are not written to the result file in a thermal analysis.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no specific considerations for steady-state thermal analysis itself. However if the temperatures from this analysis are to be used in a subsequent structural analysis the mesh must be identical. Therefore in this case you may want to make sure the mesh is fine enough for structural analysis.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For a steady-state thermal analyses you typically do not need to change these settings. The basic Analysis Settings include:

Step Controls for Static and Transient Analyses (p. 1418)

Step Controls enable you to control the rate of loading which could be important in a steady-state thermal analysis if the material properties vary rapidly with temperature. When such nonlinearities are present it may be necessary to apply the loads in small increments and perform solutions at these intermediate loads to achieve convergence. You may wish to use multiple steps if you a) want to analyze several different loading scenarios within the same analysis or b) if you want to change the analysis settings such as the time step size or the solution output frequency over specific time ranges.

Output Controls (p. 1473)

Output Controls enable you to specify the time points at which results should be available for postprocessing. In a nonlinear analysis it may be necessary to perform many solutions at intermediate load values. However i) you may not be interested in all the intermediate results and ii) writing all the results can make the results file size unwieldy. In this case you can restrict the amount of output by requesting results only at certain time points.

Nonlinear Controls (p. 1468)

Nonlinear Controls enable you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control.

Nonlinear Controls are exposed for the Mechanical APDL solver only.

Analysis Data Management (p. 1484)

Analysis Data Management settings enable you to save specific solution files from the steady-state thermal analysis for use in other analyses.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a steady-state thermal analysis you can specify an initial temperature value. This uniform temperature is used during the first iteration of a solution as follows:

- To evaluate temperature-dependent material properties.
- As the starting temperature value for constant temperature loads.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifics ***... for this analysis type:***

The following loads are supported in a steady-state thermal analysis:

- Temperature (p. 1598)
- Convection (p. 1602)
- Radiation (p. 1609)
- Heat Flow (p. 1614)
- Perfectly Insulated (p. 1614)
- Heat Flux (p. 1617)
- Internal Heat Generation (p. 1620)
- Imported Temperature (p. 1756)
- Imported Convection Coefficient (p. 1744)
- Fluid Solid Interface (p. 1648)
- System Coupling Region (p. 1651)

Loads and supports vary as a function of time even in a static analysis as explained in the Role of Time in Role of Time in Tracking (p. 1491). In a static analysis, the load's magnitude could be a constant value or could vary with time as defined in a table or via a function. Details of how to apply a tabular or function load are described in Defining Boundary Condition Magnitude (p. 1771). In addition, see the Apply Loads and Supports (p. 354) section for more information about time stepping and ramped loads.

Fluid Solid Interface (p. 1648) is not available for the Samcef or ABAQUS solver.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

The **Solution Information** (p. 1833) object provides some tools to monitor solution progress.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

You can also insert a **Result Tracker** (p. 1844) object under **Solution Information**. This tool enables you to monitor temperature at a vertex as the solution progresses. **Result Tracker** is not available to the Samcef or ABAQUS solver.

Review Results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Applicable results are all thermal result types (p. 2153).

Once a solution is available you can contour the results (p. 62) or animate the results (p. 2260) to review the response of the structure.

As a result of a nonlinear analysis you may have a solution at several time points. You can use probes (p. 1946) to display the variation of a result item over the load history. Also of interest is the ability to plot one result quantity (for example, maximum temperature on a face) against another results item (for example, applied heat generation rate). You can use the Charts (p. 1929) feature to develop such charts.

Note that Charts (p. 1929) are also useful to compare results between two analyses of the same model.

Thermal-Electric Analysis

Introduction

A Steady-State Thermal-Electric Conduction analysis allows for a simultaneous solution of thermal and electric fields. This coupled-field capability models joule heating for resistive materials and contact electric conductance as well as Seebeck, Peltier, and Thomson effects for thermoelectricity, as described below.

- **Joule heating** - Heating occurs in a resistive conductor carrying an electric current. Joule heating is proportional to the square of the current, and is independent of the current direction. Joule heating is also present and accounted for at the contact interface between bodies in inverse proportion to the contact electric conductance properties. (Note however that the **Joule Heat** results object will not display contact joule heating values. Only solid body joule heating is represented).
- **Seebeck effect** - A voltage (Seebeck EMF) is produced in a thermoelectric material by a temperature difference. The induced voltage is proportional to the temperature difference. The proportionality coefficient is known as the **Seebeck Coefficient** (α).
- **Peltier effect** - Cooling or heating occurs at a junction of two dissimilar thermoelectric materials when an electric current flows through that junction. Peltier heat is proportional to the current, and changes sign if the current direction is reversed.
- **Thomson effect** - Heat is absorbed or released in a non-uniformly heated thermoelectric material when electric current flows through it. Thomson heat is proportional to the current, and changes sign if the current direction is reversed.

Points to Remember

Electric loads may be applied to parts with electric properties and thermal loads may be applied to bodies with thermal properties. Parts with both physics properties can support both thermal and electric loads. See the Steady-State Thermal Analysis (p. 657) section and the Electric Analysis (p. 403) section of the help for more information about applicable loads, boundary conditions, and results types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In addition to calculating the effects of steady thermal and electric loads on a system or component, a Steady-State Thermal-Electric analysis supports a multi-step solution.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag the **Thermal-Electric** template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

To have Thermal and/or Electrical effects properly applied to the parts of your model, you need to define the appropriate material properties. For a steady-state analysis, the electrical property **Resistivity** is required for Joule Heating effects and **Thermal Conductivity** for thermal conduction effects. Seebeck/Peltier/Thomson effects require you to define the **Seebeck Coefficient** material property.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

Note that 3D shell bodies and line bodies are not supported in a thermal-electric analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

Mechanical does not support Rigid Bodies in thermal-electric analyses. For more information, see the *Stiffness Behavior* documentation for Rigid Bodies (p. 852).

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Contact across parts during a thermal-electric analysis consider thermal and/or electric effects based on the material properties of adjacent parts. That is, if both parts have thermal properties, thermal contact is applied and if both parts have electric properties, electric contact is applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no specific considerations regarding meshing for a thermal-electric analysis.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a Thermal-Electric analysis, the basic **Analysis Settings** include:

Step Controls for Static and Transient Analyses (p. 1418)

: used to specify the end time of a step in a single or multiple step analysis. Multiple steps are needed if you want to change load values, the solution settings, or the solution output frequency over specific steps. Typically you do not need to change the default values.

Nonlinear Controls (p. 1468)

Typical thermal-electric problems contain temperature dependent material properties and are therefore nonlinear. Nonlinear Controls (p. 1468) for both thermal and electrical effects are available and include Heat and Temperature convergence for thermal effects and Voltage and Current convergence for electric effects.

Output Controls (p. 1473)

Output Controls enable you to specify the time points at which results should be available for postprocessing. A multi-step analysis involves calculating solutions at several time points in the load history. However you may not be interested in all of the possible results items and writing all the results can make the result file size unwieldy. You can restrict the amount of output by requesting results only at certain time points or limit the results that go onto the results file at each time point.

Analysis Data Management (p. 1484)

Review these settings as needed.

Solver Controls (p. 1428)

The default Solver Controls (p. 1428) setting for thermal-electric analysis is the **Direct** (Sparse) solver. The Iterative (PCG) solver may be selected as an alternative solver. If Seebeck effects are included, the solver is automatically set to Direct.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifics ***... for this analysis type:***

There is no initial condition specification for a thermal-electric analysis.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are supported in a **Thermal-Electric** analysis:

- Voltage (p. 1627)
- Current (p. 1630)
- Coupling Condition (p. 1698)
- Temperature (p. 1598)
- Convection (p. 1602)
- Radiation (p. 1609)
- Heat Flow (p. 1614)
- Perfectly Insulated (p. 1614)
- Heat Flux (p. 1617)
- Internal Heat Generation (p. 1620)
- System Coupling Region (p. 1651)

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

The **Solution Information** (p. 1833) object provides some tools to monitor solution progress.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the model during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Applicable results include all thermal and electric results (p. 1927).

Once a solution is available, you can contour the results (p. 62) or animate the results (p. 2260) to review the responses of the model.

For the results of a multi-step analysis that has a solution at several time points, you can use probes (p. 1946) to display variations of a result item over the steps.

You may also wish to use the Charts (p. 1929) feature to plot multiple result quantities against time (steps). For example, you could compare current and joule heating. Charts can also be useful when comparing the results between two analysis branches of the same model.

Transient Structural Analysis

A transient analysis, by definition, involves loads that are a function of time. In the Mechanical application, you can perform a transient analysis on either a flexible structure or a rigid assembly. For a flexible structure, the Mechanical application can use the Ansys Mechanical APDL, Samcef, or ABAQUS solver to solve a Transient Structural analysis.

You can perform a transient structural analysis (also called time-history analysis) in the Mechanical application using the transient structural analysis that specifically uses the Ansys Mechanical APDL solver. This type of analysis is used to determine the dynamic response of a structure under the action of any general time-dependent loads. You can use it to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any transient loads. The time scale of the loading is such that the inertia or damping effects are considered to be important. If the inertia and damping effects are not important, you might be able to use a static analysis instead.

Go to a section topic:

- Points to Remember (p. 666)
- Preparing the Analysis (p. 666)
- Define Engineering Data (p. 667)
- Define Part Behavior (p. 667)
- Define Connections (p. 667)
- Apply Mesh Controls/Preview Mesh (p. 667)
- Establish Analysis Settings (p. 668)
- Define Initial Conditions (p. 669)
- Apply Loads and Supports (p. 672)
- Solve (p. 673)
- Review Results (p. 673)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Points to Remember

A transient structural analysis can be either linear or nonlinear. All types of nonlinearities are allowed - large deformations, plasticity, contact, hyperelasticity, and so on. Ansys Workbench offers an additional solution method of Mode-Superposition to perform linear transient structural analysis. In the Mode-Superposition method, the transient response to a given loading condition is obtained by calculating the necessary linear combinations of the eigenvectors obtained in a modal analysis. For more details, refer to Transient Structural Analysis Using Linked Modal Analysis System (p. 674) section. The Mode Superposition method is not available to the Samcef or ABAQUS solver.

A transient dynamic analysis is more involved than a static analysis because it generally requires more computer resources and more of your resources, in terms of the "engineering" time involved. You can save a significant amount of these resources by doing some preliminary work to understand the physics of the problem. For example, you can:

1. Try to understand how nonlinearities (if you are including them) affect the structure's response by doing a static analysis first. In some cases, nonlinearities need not be included in the dynamic analysis. Including nonlinear effects can be expensive in terms of solution time.

2. Understand the dynamics of the problem. By doing a modal analysis, which calculates the natural frequencies and mode shapes, you can learn how the structure responds when those modes are excited. The natural frequencies are also useful for calculating the correct integration time step.

3. Analyze a simpler model first. A model of beams, masses, springs, and dampers can provide good insight into the problem at minimal cost. This simpler model may be all you need to determine the dynamic response of the structure.

Note:

Refer to the following sections of the Mechanical APDL application documentation for a more thorough treatment of dynamic analysis capabilities:

- The *Transient Dynamic Analysis* chapter of the *Structural Analysis Guide* - for a technical overview of nonlinear transient dynamics.

- The *Multibody Analysis Guide* - for a reference that is particular to multibody motion problems. In this context, "multibody" refers to multiple rigid or flexible parts interacting in a dynamic fashion.

Although not all dynamic analysis features discussed in these manuals are directly applicable to Mechanical features, the manuals provide an excellent background on general theoretical topics.

Preparing the Analysis

As needed throughout the analysis, refer to the Steps for Using the Application (p. 323) section for an overview the general analysis workflow.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Define Engineering Data

Material properties can be linear or nonlinear, isotropic or orthotropic, and constant or temperature-dependent. Both Young's modulus (and stiffness in some form) and density (or mass in some form) must be defined.

Define Part Behavior

You can define a **Point Mass** (p. 901) for this analysis type.

In a transient structural analysis, rigid parts are often used to model mechanisms that have gross motion and transfer loads between parts, but detailed stress distribution is not of interest. The output from a rigid part is the overall motion of the part plus any force transferred via that part to the rest of the structure. A "rigid" part is essentially a point mass connected to the rest of the structure via joints. Hence in a transient structural analysis the only applicable loads on a rigid part are acceleration and rotational velocity loads. You can also apply loads to a rigid part via joint loads. Rigid behavior cannot be used with the Samcef or ABAQUS solver.

If your model includes nonlinearities such as large deflection or hyperelasticity, the solution time can be significant due to the iterative solution procedure. Hence, you may want to simplify your model if possible. For example, you may be able to represent your 3D structure as a 2-D plane stress, plane strain, or axisymmetric model, or you may be able to reduce your model size through the use of symmetry or antisymmetry surfaces. Similarly, if you can omit nonlinear behavior in one or more parts of your assembly without affecting results in critical regions, it will be advantageous to do so.

Define Connections

This analysis supports Contact (p. 1207), Joints (p. 1292), and Springs (p. 1395).

Note:

You can specify a damping coefficient property in longitudinal springs that will generate a damping force proportional to velocity.

Samcef and ABAQUS Solvers

Only contacts, springs, and beams are supported. Joints are not supported.

Apply Mesh Controls/Preview Mesh

Provide an adequate mesh density on contact surfaces to allow contact stresses to be distributed in a smooth fashion. Likewise, provide a mesh density adequate for resolving stresses; areas where stresses or strains are of interest require a relatively fine mesh compared to that needed for displacement or nonlinearity resolution. If you want to include nonlinearities, the mesh should be able to capture the effects of the nonlinearities. For example, plasticity requires a reasonable integration point density (and therefore a fine element mesh) in areas with high plastic deformation gradients.

In a dynamic analysis, the mesh should be fine enough to be able to represent the highest mode shape of interest.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Establish Analysis Settings

For a Transient Structural analysis, the basic **Analysis Settings** include:

Large Deflection (p. 1435)

Large Deflection is typically needed for slender structures. It is typical to use large deflection if the transverse displacements in a slender structure are more than 10% of the thickness.

Small deflection and small strain analyses assume that displacements are small enough that the resulting stiffness changes are insignificant. Setting **Large Deflection** to **On** will take into account stiffness changes resulting from change in element shape and orientation due to large deflection, large rotation, and large strain. Therefore the results will be more accurate. However this effect requires an iterative solution. In addition it may also need the load to be applied in small increments. Therefore the solution may take longer to solve.

You also need to turn on large deflection if you suspect instability (buckling) in the system. Use of hyperelastic materials also requires large deflection to be turned on.

Step Controls for Static and Transient Analyses (p. 1418)

Step Controls enable you to control the time step size in a transient analysis. Refer to the Guidelines for Integration Step Size (p. 1493) section for further information. In addition this control also enables you to create multiple steps. Multiple steps (p. 1418) are useful if new loads are introduced or removed at different times in the load history, or if you want to change the analysis settings such as the time step size at some points in the time history. When the applied load has high frequency content or if nonlinearities are present, it may be necessary to use a small time step size (that is, small load increments) and perform solutions at these intermediate time points to arrive at good quality results. This group can be modified on a per step basis.

Output Controls (p. 1473)

Output Controls enable you to specify the time points at which results should be available for postprocessing. In a transient nonlinear analysis it may be necessary to perform many solutions at intermediate time values. However, i) you may not be interested in all the intermediate results, and ii) writing all the results can make the results file size unwieldy. This group can be modified on a per step basis except for **Stress** and **Strain**.

Nonlinear Controls (p. 1468)

Nonlinear Controls enable you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control. This group can be modified on a per step basis. If you are performing a *nonlinear* Full Transient Structural analysis, the **Newton-Raphson Type** property becomes available. This property only affects nonlinear analyses. Your selections execute the Mechanical APDL **NROPT** command. The default option, **Program Controlled**, allows the application to select the appropriate **NROPT** option or you can make a manual selection and choose **Full**, **Modified**, or **Unsymmetric**.

See the Help section for the **NROPT** command in the *Mechanical APDL Command Reference* for additional information about the operation of the **Newton-Raphson Type** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Damping Controls (p. 1462)

Damping Controls enable you to specify damping for the structure in the Transient analysis. Controls include: **Stiffness Coefficient** (Beta Damping) and **Mass Coefficient** (Alpha Damping). They can also be applied as Material Damping (p. 1467) using the **Engineering Data** tab. In addition, **Numerical Damping** is also available for handling result accuracy. Damping controls are not available to the Samcef or ABAQUS solver.

Analysis Data Management (p. 1484)

Analysis Data Management settings enable you to save specific solution files from the transient structural analysis for use in other analyses. The default behavior is to only keep the files required for postprocessing. You can use these controls to keep all files created during solution or to create and save the Mechanical APDL application database (db file).

Define Initial Conditions

The structure of a transient analysis is, by default, "at rest." That is, both initial displacement and initial velocity are zero. Using the Initial Conditions object, you can specify a **Velocity** at time = zero, as a **Vector** or axially, using **Components**.

In many analyses one or more parts will have an initial known velocity such as in a drop test, metal forming analysis, or kinematic analysis. In these analyses, you can specify a constant Velocity initial condition if needed. The constant velocity could be scoped to one or more parts of the structure. The remaining parts of the structure which are not part of the scoping will retain the "at rest" initial condition.

Note:

When you are using the Mechanical APDL solver, you can also specify initial conditions using step controls, that is, by specifying multiple steps in a transient analysis and controlling the time integration effects along with activation/deactivation (p. 1423) of loads. This is useful when different parts of your model have different initial velocities or more complex initial conditions.

The following are common initial condition scenarios:

1. **Initial Displacement = 0, Initial Velocity ≠ 0 for some parts:** The nonzero velocity is established by applying small displacements over a small time interval on the part of the structure where velocity is to be specified.

   a. Specify two steps in your analysis. The first step will be used to establish initial velocity on one or more parts.

   b. Choose a small end time (compared to the total span of the transient analysis) for the first step. The second step will cover the total time span.

   c. Specify displacement(s) on one or more faces of the part(s) that will give you the required initial velocity. This requires that you do not have any other boundary condition on the part that will interfere with rigid body motion of that part. Make sure that these displacements are ramped from a value of 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

d. Deactivate or release the specified displacement load in the second step so that the part is free to move with the specified initial velocity.

For example, if you want to specify an initial Y velocity of 5 inch/second on a part, and your first step end time is 0.001 second, then specify the following loads. Make sure that the load is ramped from a value of 0 at time = 0 so that you will get the required velocity.

Tabular Data

	Steps	Time [s]	Y [in]
1	1	0.	0.
2	1	1.e-003	5.e-003
3	2	30.	= 5.e-003
*			

In this case the end time of the actual transient analysis is 30 seconds. Note that the Y displacement in the second step is deactivated.

e. In the **Analysis Settings** Details view, set the following for first step:

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1.e-003 s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	1.
Time Integration	Off

f. You can choose appropriate time step sizes for the second step (the actual transient). Make sure that time integration effects are turned on for the second step.

In the first step, inertia effects will not be included but velocity will be computed based on the displacement applied. In the second step, this displacement is released by deactivation and the time integration effects are turned on.

2. **Initial Displacement ≠ 0, Initial Velocity ≠ 0:** This is similar to case a. above except that the imposed displacements are the actual values instead of "small" values. For example if the initial displacement is 1 inch and the initial velocity is 2.5 inch/sec then you would apply a displacement of 1 inch over 0.4 seconds.

a. Specify two steps in your analysis. The first step will be used to establish initial displacement and velocity on one or more parts.

b. Choose a small end time (compared to the total span of the transient analysis) for the first step. The second step will cover the total time span.

c. Specify the initial displacement(s) on one or more faces of the part(s) as needed. This requires that you do not have any other boundary condition on the part that will interfere with rigid body motion of that part. Make sure that these displacements are ramped from a value of 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

d. Deactivate or release the specified displacement load in the second step so that the part is free to move with the specified initial velocity.

For example if you want to specify an initial Z velocity on a part of 0.5 inch/sec and have an initial displacement of 0.1 inch, then your first step end time = (0.1/0.5) = 0.2 second. Make sure that the displacement is ramped from a value of 0 at time = 0 so that you will get the required velocity.

Tabular Data

	Steps	Time [s]	Z [in]
1	1	0.	0.
2	1	0.2	0.1
3	2	2.	= 0.1
*			

In this case the end time of the actual transient analysis is 5 seconds. Note that the Z displacement in the second step is deactivated.

e. In the **Analysis Settings** Details view, set the following for first step:

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	0.2 s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	4.
Time Integration	Off

f. You can choose appropriate time step sizes for the second step (the actual transient). Make sure that time integration effects are turned on for the second step.

In the first step, inertia effects will not be included but velocity will be computed based on the displacement applied. In the second step, this displacement is released by deactivation and the time integration effects are turned on.

3. **Initial Displacement ≠ 0, Initial Velocity = 0:** This requires the use of two steps also. The main difference between b. above and this scenario is that the displacement load in the first step is not ramped from zero. Instead it is step applied as shown below with 2 or more substeps to ensure that the velocity is zero at the end of step 1.

a. Specify two steps in your analysis. The first step will be used to establish initial displacement on one or more parts.

b. Choose an end time for the first step that together with the initial displacement values will create the necessary initial velocity.

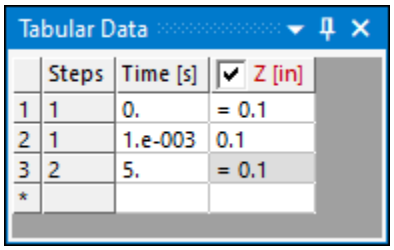
c. Specify the initial displacement(s) on one or more faces of the part(s) as needed. This requires that you do not have any other boundary condition on the part that will interfere with rigid

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

body motion of that part. Make sure that this load is step applied, that is, apply the full value of displacements at time = 0 itself and maintain it throughout the first step.

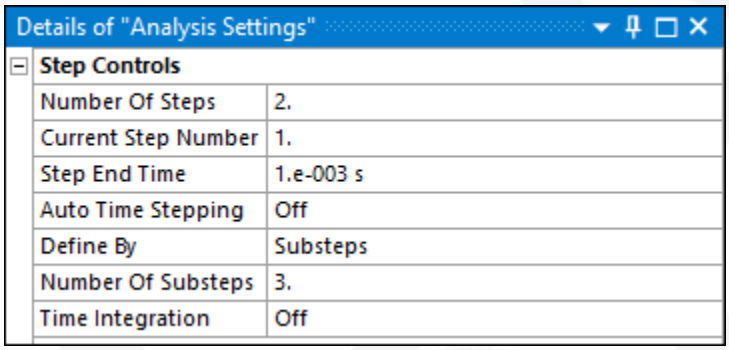
d. Deactivate or release the specified displacement load in the second step so that the part is free to move with the initial displacement values.

   For example if you want to specify an initial Z displacement of 0.1 inch and the end time for the first step is 0.001 seconds, then the load history displays as shown below. Note the step application of the displacement.



   In this case the end time of the actual transient analysis is 5 seconds. Note that the Z displacement in the second step is deactivated.

e. In the **Analysis Settings** Details view, set the following for first step. Note that the number of substeps must be at least 2 to set the initial velocity to zero.



f. You can choose appropriate time step sizes for the second step (the actual transient). Make sure that time integration effects are turned on for the second step.

In the first step, inertia effects will not be included but velocity will be computed based on the displacement applied. But since the displacement value is held constant, the velocity will evaluate to zero after the first substep. In the second step, this displacement is released by deactivation and the time integration effects are turned on.

Apply Loads and Supports

For this analysis type, loading is always step-applied (regardless of the setting of the **Time Integration** property). The Environment (p. 61) Context tab provides the various groups of loads, supports, and conditions (p. 1500).

In this analysis, the load's magnitude can be constant or vary with time as defined in **Tabular Data** or as a **Function**. Details of how to apply a tabular or function load are described in Defining Boundary

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Condition Magnitude (p. 1771). In addition, see the Apply Loads and Supports (p. 354) section for more information about time stepping and ramped loads.

Note:

- You can use a Joint Load (p. 1592) to kinematically drive a joint. See For the solver to converge, it is recommended that you ramp joint load angles and positions from zero to the real initial condition over one step.
- Joint loads are not supported for the Samcef or ABAQUS solver.
- Acceleration (p. 1501) and/or Displacement (p. 1666) can be defined as a base excitation only in a Transient Structural Analysis Using Linked Modal Analysis System.

Solve

When performing a nonlinear analysis, you may encounter convergence difficulties due to a number of reasons. Some examples may be initially open contact surfaces causing rigid body motion, large load increments causing non-convergence, material instabilities, or large deformations causing mesh distortion that result in element shape errors. To identify possible problem areas some tools are available under **Solution Information** (p. 1833) object Details view.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

You can display contour plots of **Newton-Raphson Residuals** in a nonlinear static analysis. Such a capability can be useful when you experience convergence difficulties in the middle of a step, where the model has a large number of contact surfaces and other nonlinearities. When the solution diverges, identifying regions of high Newton-Raphson residual forces can provide insight into possible problems.

Result Tracker (p. 1844) is another useful tool that enables you to monitor displacement and energy results as the solution progresses. This is especially useful in the case of structures that may go through convergence difficulties due to buckling instability. **Result Tracker** is not available to the Samcef or ABAQUS solver.

Review Results

All structural result types (p. 2017) except frequencies are available as a result of a transient structural analysis. You can use a **Solution Information** (p. 1833) object to track, monitor, or diagnose problems that arise during a solution.

Once a solution is available you can contour the results (p. 62) or animate the results (p. 2260) to review the response of the structure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

As a result of a nonlinear static analysis, you may have a solution at several time points. You can use probes (p. 1946) to display the variation of a result item as the load increases.

Note:

Fixed body-to-body joints between two rigid bodies will not produce a joint force or moment in a transient structural analysis.

Also of interest is the ability to plot one result quantity (for example, displacement at a vertex) against another result item (for example, applied load). You can use the Charts (p. 1929) feature to develop such charts. Charts (p. 1929) are also useful to compare results between two analyses of the same model. For example, you can compare the displacement response at a vertex from two transient structural analyses with different damping characteristics.

Transient Structural Analysis Using Linked Modal Analysis System

This section examines the workflow for a Transient Structural analysis linked to an upstream Modal analysis. Topics include:

- Create Analysis System (p. 674)
- Establish Analysis Settings (p. 675)
- Define Initial Conditions (p. 677)
- Apply Loads and Supports (p. 677)
- Support Limitations (p. 678)
- Analysis Notes (p. 679)
- File Management (p. 679)

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

Because this analysis is linked to (or based on) modal responses, a Modal analysis is a prerequisite. This linked setup allows the transient system to share resources such as engineering data, geometry, and boundary condition type definitions made in the Modal analysis.

Linking the Modal Solution cell to the Transient Setup cell configures the Transient analysis so that the application selects, by default, the **Mode-Superposition** (MSUP) Solution Method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The Samcef and ABAQUS solvers do not support a linked Modal/Transient Structural analysis. For additional support requirements and notes regarding this analysis configuration, see the references at the bottom of this section.

Note:

- For a Mode Superposition Transient Structural analysis, the Modal analysis can be pre-stressed (p. 352) by a Static Structural analysis.
- When solving a linked MSUP transient system database from a version prior to the most current version of Mechanical, it is possible to encounter incompatibility of the file *file.full* created by the modal system. This incompatibility can cause the transient system's solution to fail. In the event you experience this issue, use the **Clear Generated Data** feature and resolve the modal system.

  Refer to the Obtain the Mode-Superposition Transient Solution section of the *MAPDL Structural Analysis Guide* for more information.

From the **Toolbox**, drag a **Modal** template to the **Project Schematic**. Then, drag a **Transient Structural** template directly onto the **Solution** cell of **Modal** template.

Note:

You can create a modal environment in **Transient Structural** system that is already open in Mechanical by:

1. Selecting the **Modal** option from the **Analysis** drop-down menu on the Home (p. 40) tab (or displayed tab).
2. Setting the **Modal Environment** property of the **Modal (None)** object to the Modal system. The **Modal** object is included under the **Initial Conditions** object.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a Transient Structural system linked to a Modal system, the basic Analysis Settings (p. 1417) include:

Step Controls (p. 1418)

This analysis is only compatible with constant time stepping. So, auto time stepping is turned off and will always be in read only mode. The user specified substep or time step value is applicable to all the load steps. All of the **Step Controls** settings applied to this analysis are not step aware. The time integration is turned on by

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

default and will always be in read-only mode. A Time Step value that results in an integral number of sub steps over the load step must be selected.

Options (p. 1459)

You can turn on the following properties:

- **Include Residual Vectors**. Set this property to **Yes** to execute the **RESVEC** command and calculate residual vectors.
- **On Demand Expansion Option**: If you set this property to **Yes** manually or through the **Program Controlled** setting, the application does not create a result file. Your results are evaluated using the Modal solution data and otherwise calculated "on demand." This property supports specific result types. See the property's description in the **Options** (p. 1459) section for a listing.
- **Mode Selection Method**: This property enables you to select the modes for the mode expansion. Property options include:
  - **None** (default): This option expands all the extracted modes from the modal analysis. All the modes participate in the mode-superposition transient dynamic analysis.
  - **Modal Effective Mass**: When you select this option, the **Significance Threshold** property also displays. For the **Significance Threshold** property value, the application selects the modes that have a ratio for the Modal Effective Mass to the Total Mass that is greater than this value, in all directions, for expansion. That is, only the selected modes participate in the mode-superposition transient dynamic analysis. The application default is **`0.001`**. This feature improves the performance of postprocessing the modal results and reduces the file size.

Note:

The following loads do not support residual vector calculations:

- **Nodal Force**
- **Remote Force** scoped to a **Remote Point** (created via **Model** object)
- **Moment** scoped to a **Remote Point** (created via **Model** object)

Output Controls (p. 1473)

These controls enable you to request **Stress**, **Strain**, **Nodal Force**, **Reaction**, **Velocity and Acceleration** results to be calculated. To expand reaction forces in the modal solution, set the **Nodal Force** property to **Yes** or **Constrained Nodes**. The **General Miscellaneous** property needs to be set to **Yes** in order to apply a Beam Tool (p. 2087) and/or to calculate Beam Results (p. 2089). In addition, this setting is required to correctly produce twisted beam shapes. For better performance, you can also choose

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to have these results expanded from transient or modal solutions. The **Contact Miscellaneous** option is not available.

Note:

Solver Controls (p. 1428), Restart Controls (p. 1438), Nonlinear Controls (p. 1468) and Creep Controls (p. 1445) are not applicable to the current analysis.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

The Transient Structural analysis must point to a Modal analysis in the **Modal** (Initial Conditions) object. This object also indicates whether the upstream Modal analysis is pre-stressed. If it is a pre-stress analysis, the name of the pre-stress analysis system is displayed in the **Pre-Stress Environment** field, otherwise the field indicates **None**. The Modal Analysis (p. 515) must extract all modes that may contribute to the dynamic response.

Note:

Command (ADPL) objects can be inserted into **Initial Conditions** object to execute a restart of the solution process for the Modal Analysis.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are allowed for the linked analysis:

- Acceleration (p. 1501)
- Acceleration (p. 1501) as a base excitation
- Pressure (p. 1521)
- Pipe Pressure (p. 1532)
- Force (p. 1543) (applied to a face, edge, or vertex)
- Line Pressure (p. 1585)
- Moment (p. 1574)
- Remote Force (p. 1552)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Standard Earth Gravity (p. 1509)

Direct FE

The **Direct FE** option **Nodal Force** (p. 1721) is supported for node-based Named Selection scoping.

Note:

- For the **Force** and **Pressure** loading conditions, the default setting for the **Applied By** property is **Direct**.
- When you have a supported topology (geometric or mesh entity) selected on your model, and you insert a **Remote Force** or a **Moment**, the application automatically creates (promotes) a corresponding **Remote Point** for the specified load and scopes the load to that **Remote Point**. This promotion feature is only available when none of the analyses in your project have been solved.

Support Limitations

Note the following limitations:

- If the **Reference Temperature** is set as **By Body** and that temperature does not match the environment temperature, a thermally induced transient load will result (from the thermal strain assuming a nonzero thermal expansion coefficient). This thermal transient loading is ignored for Transient Structural Analysis using Linked Modal Analysis System.
- **Remote Force** is not supported for vertex scoping.
- During a linked MSUP Transient analysis, if a **Remote Force** or **Moment** scoped to an internal remote point is specified with the **Behavior** property set to **Deformable**, the boundary conditions cannot be scoped to the edges of line bodies such that all of their nodes in combination are collinear.
- **Remote Force** and **Moment** applied to a rigid body is not supported.
- **Moment** is not supported for vertex scoping on 3D solid bodies because a beam entity is created for the load application. The beam entity changes the stiffness of the structural component shared and solved by the preceding modal analysis.
- Joint probes (p. 2118), **Energy** Probe, and Strain Energy (p. 2037) results are not supported when expanded from a Modal solution.
- The **Force Reaction** and **Moment Reaction** probes are not supported for a either a Harmonic Response analysis or a Transient structural analysis using the **Mode Superposition** method Solution Method when 1) the **Damping Ratio** property is defined with a non-zero value and 2) if the Analysis Settings property **Expand Result From** is set to **Harmonic Solution** or **Transient Solution**.
- Cyclic symmetry (p. 1134) models are not supported for a Transient Structural Analysis that is using a linked Modal Analysis System.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Spring probe (p. 2123) only supports Elastic force result when expanded from modal solution where as it supports both Elastic force and Elongation results when expanded from transient solution. The Elastic force results include the spring damping effect if the Reduced method is selected from Modal Solver controls (p. 1428), and Store Complex Solution is set to **No**.
- **Standard Earth Gravity** is not allowed in conjunction with the **Acceleration** load.
- Elemental Triads (p. 1936) results are not available on solid bodies.
- When the **Step Controls** are defined by **Substeps**, the time step value sent to the solver is based on the settings for the first load step. For the load **steps** greater than one, you may notice an inconsistent value of the number of sub-steps in the Details View or the Worksheet View.
- For the Samcef and ABAQUS solvers, Hydrostatic Pressure and Pipe Pressure are not supported.

Analysis Notes

- Remote Force and Moment loading combined with the Rigid contact behavior is allowed when the loading is scoped through a Remote Point Application (p. 1090).
- To obtain the most accurate results, it is recommended that you specify **Bonded** as the contact **Type** and set the contact **Formulation** to **MPC** in the **Details** for the Contact Region. See the Contact Definition (p. 1234) and Contact Advanced Category (p. 1240) for more detailed information about these settings.
- When the result is expanded from Modal Solution or when **Reaction Object** is scoped to a Contact Region, the Reaction Object requires both **Nodal Forces** and **Calculate Reactions** Output Controls settings to be turned **On**. If they are not set, the error message "A result is invalid with current output control settings" displays. For other cases, the **Reaction Object** requires only the **Calculate Reactions** Output Controls setting to be turned **On**.
- The default value of Numerical Damping is different for full and mode-superposition transient structural analyses. So, the results comparison of a model must be done by matching the Numerical Damping value settings in the Damping Controls (p. 1462) section.

File Management Options

When solving a **Transient** Structural analysis that includes an upstream linked **Modal** analysis, the application can reference the prerequisite files by specifying the full path of their location (refer to **RESUME** and **MODDIR** commands) instead of making copies. Referencing improves solution time and disk usage. The application will perform referencing once the following conditions and settings are performed:

- You must perform the solution locally (called "In Process (p. 1803)").

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The **On Demand Expansion Option** property of the Options (p. 1447) category of the **Analysis Settings** of the **Modal** analysis must be set to **Yes** manually or through the **Program Controlled** setting.

Important:

Make sure that **Modal** and the downstream **Transient** analysis request the same result data in the Output Controls (p. 1473). Otherwise, you may receive unexpected results.

Note:

If you specify material damping, the **Eqv. Damping Ratio From Modal** property (for the **Transient** Structural analysis) must also be set to **Yes**.

Limitations

Also note the following limitations for referencing. The **Transient** structural analysis cannot include:

- Cyclic Symmetry
- Line Pressure
- Pressure or Force applied using the surface effect elements created on the top of the scoped geometry (**Applied By** property set to **Surface Effect**)
- Remote Force or Moment not scoped to global remote points
- Commands (APDL) object

Transient Thermal Analysis

Introduction

Transient thermal analyses determine temperatures and other thermal quantities that vary over time. The variation of temperature distribution over time is of interest in many applications such as with cooling of electronic packages or a quenching analysis for heat treatment. Also of interest are the temperature distribution results in thermal stresses that can cause failure. In such cases the temperatures from a transient thermal analysis are used as inputs to a structural analysis for thermal stress evaluations. Transient thermal analyses can be performed using the Mechanical APDL, Samcef, ABAQUS, or LS-DYNA solvers.

Many heat transfer applications such as heat treatment problems, electronic package design, nozzles, engine blocks, pressure vessels, fluid-structure interaction problems, and so on involve transient thermal analyses.

Point to Remember

A transient thermal analysis can be either linear or nonlinear. Temperature dependent material properties (thermal conductivity, specific heat or density), or temperature dependent convection coefficients or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

radiation effects can result in nonlinear analyses that require an iterative procedure to achieve accurate solutions. The thermal properties of most materials do vary with temperature, so the analysis usually is nonlinear.

Preparing the Analysis

Create Analysis System

General *Basic general information about this topic (p. 323)*

Specifics ***... for this analysis type:***

From the **Toolbox**, drag the **Transient Thermal**, **Transient Thermal (Samcef)**, **Transient Thermal (ABAQUS)**, or LS-DYNA template to the **Project Schematic**.

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Thermal Conductivity, Density, and Specific Heat must be defined for a transient thermal analysis. Thermal Conductivity can be isotropic or orthotropic. All properties can be constant or temperature-dependent.

Attach Geometry

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

There are no special considerations for a transient thermal analysis.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

Mechanical does not support Rigid Bodies in thermal analyses. For more information, see the *Stiffness Behavior* documentation for Rigid Bodies (p. 852).

You can define a **Thermal Point Mass** (p. 905) for this analysis type.

Define Connections

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

In a thermal analysis only contact is valid. Any joints or springs are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

With contact the initial status is maintained throughout the thermal analysis, that is, any closed contact faces will remain closed and any open contact faces will remain open for the duration of the thermal analysis. Heat conduction across a closed contact face is set to a sufficiently high enough value (based on the thermal conductivities and the model size) to model perfect contact with minimal thermal resistance. If needed, you can model imperfect contact by manually inputting a **Thermal Conductance** (p. 1253) value.

By default, Contact Results (p. 2043) (accessible through User Defined Results (p. 2192) via CONTSTAT or CONTFLUX – see the User Defined Results for the Mechanical APDL Solver (p. 2203) section.) are not written to the result file in a thermal analysis.

Apply Mesh Controls/Preview Mesh

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

There are no specific considerations for transient thermal analysis itself. However if the temperatures from this analysis are to be used in a subsequent structural analysis the mesh must be identical. Therefore in this case you may want to make sure the mesh is fine enough for a structural analysis.

Establish Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

For a Transient Thermal analysis, the basic **Analysis Settings** controls include:

Step Controls for Static and Transient Analyses (p. 1418)

Step Controls are used to: i) specify the end time of the transient analysis, ii) control the time step size, and iii) create multiple steps when needed.

The rate of loading could be important in a transient thermal analysis if the material properties vary rapidly with temperature. When such nonlinearities are present it may be necessary to apply the loads in small increments and perform solutions at these intermediate loads to achieve convergence. Multiple steps are needed if you want to change the solution settings, for example, the time step size or the solution output frequency over specific time spans in the transient analysis.

Output Controls (p. 1473)

Solver Controls enable you to specify the time points at which results should be available for postprocessing. A transient analysis involves calculating solutions at several time points in the load history. However: i) you may not be interested in all the intermediate results and ii) writing all the results can make the results file size unwieldy. In this case you can restrict the amount of output by requesting results only at certain time points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Nonlinear Controls (p. 1468)

Nonlinear Controls enable you to modify convergence criteria and other specialized solution controls. Typically you will not need to change the default values for this control.

Analysis Data Management (p. 1484)

Analysis Data Management settings enable you to save specific solution files from the transient thermal analysis for use in other analyses.

Define Initial Conditions

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

A transient thermal analysis involves loads that are functions of time. The first step in applying transient thermal loads is to establish initial temperature distribution at Time = 0.

The default initial condition for a transient thermal analysis is a uniform temperature of 22°C or 71.6°F. You can change this to an appropriate value for your analysis. An example might be modeling the cooling of an object taken out of a furnace and plunged into water.

You can also use the temperature results from a steady-state analysis of the same model for the initial temperature distribution. A casting solidification study might start with different initial temperatures for the mold and the metal. In this case a steady-state analysis of the hot molten metal inside the mold can serve as the starting point for the solidification analysis.

In the first iteration of a transient thermal analysis, this initial temperature is used as the starting temperature value for the model except where temperatures are explicitly specified. In addition this temperature is also used to evaluate temperature-dependent material property values for the first iteration.

If the **Initial Temperature** field is set to **Non-Uniform Temperature**, a **Time** field is displayed where you can specify a time at which the temperature result of the steady-state thermal analysis (selected in **Initial Condition Environment** field) will be used as the initial temperature in the transient analysis. A zero value will be translated as the end time (of the steady-state thermal analysis) and this value can not be greater than the end time.

Apply Loads and Supports

General *Basic general information about this topic (p. 354)*

Specifics ***... for this analysis type:***

The following loads are supported in a transient thermal analysis:

- Temperature (p. 1598)
- Internal Heat Generation (p. 1620)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Convection (p. 1602)
- Radiation (p. 1609)
- Heat Flow (p. 1614)
- Perfectly Insulated (p. 1614)
- Heat Flux (p. 1617)
- Imported Heat Generation (p. 1748)
- Imported Temperature (p. 1756)
- Imported Convection Coefficient (p. 1744)
- System Coupling Region (p. 1651)

In this analysis, the load's magnitude could be a constant value or could vary with time as defined in a table or via a function. Details of how to apply a tabular or function load are described in Defining Boundary Condition Magnitude (p. 1771). In addition, see the Apply Loads and Supports (p. 354) section for more information about time stepping and ramped loads.

Solve

General *Basic general information about this topic (p. 355)*

Specifics ***... for this analysis type:***

The **Solution Information** (p. 1833) object provides some tools to monitor solution progress.

Solution Output continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Any convergence data output in this printout can be graphically displayed as explained in the **Solution Information** (p. 1833) section.

You can also insert a **Result Tracker** (p. 1844) object under **Solution Information**. This tool enables you to monitor temperature at a vertex as the solution progresses. **Result Tracker** is not available to the Samcef or ABAQUS solver.

Review Results

General *Basic general information about this topic (p. 356)*

Specifics ***... for this analysis type:***

Applicable results are all thermal result types (p. 2153).

Once a solution is available you can contour the results (p. 62) or animate the results (p. 2260) to review the response of the structure.

As a result of a nonlinear analysis you may have a solution at several time points. You can use probes (p. 1946) to display the variation of a result item over the load history. Also of interest is the ability to plot one result quantity (for example, maximum temperature on a face) against another results item (for example, applied heat generation rate). You can use the Charts (p. 1929) feature to develop such charts.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Note that Charts (p. 1929) are also useful to compare results between two analyses of the same model.

Special Analysis Topics

This section includes special topics available the Mechanical application for particular applications. The following topics are included:

Additive Manufacturing Process Simulation
Composite Analysis
ECAD Analysis using Trace Mapping
Electromagnetics (EM) - Mechanical Data Transfer
External Data
External Study Import
Fluid-Structure Interaction (FSI)
Icepak to Mechanical Data Transfer
Inverse Solving for Nonlinear Static Structural Analyses
Mechanical-Electronics Interaction (Mechatronics) Data Transfer
Polyflow to Mechanical Data Transfer
Reinforcement Specification Using Mesh-Independent Method
Rotordynamics Analysis
Substructure Analysis
Static Analysis From Rigid Dynamics Analysis
Submodeling
System Coupling
Thermal-Stress Analysis
Transferring Hydrodynamic Loads to a Structural System
Twin Builder/Rigid Dynamics Co-Simulation

Additive Manufacturing Process Simulation

An additive manufacturing process constructs 3D objects by building one layer at a time until the object is complete. Of the various 3D print processes available in additive manufacturing, Ansys Mechanical simulates the Laser Powder Bed Fusion (LPBF) and Directed Energy Deposition (DED) processes for metals. These processes use a laser to melt and fuse each individual layer of metal powder to the layers below it to create a part.

For detailed, step-by-step instructions and background information on simulating these additive manufacturing methods, consult the LPBF Simulation Guide and the DED Simulation Guide.

Composite Analysis

Layered Composites: You can create a layered composite model using the Ansys Composite PrepPost (ACP) application. You can then import the model into Mechanical and perform a composite analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Often, you employ the Composite Failure Tool (p. 2072) to perform post-processing on your composite structures that you modeled using ACP. To better understand how to perform the steps for this analysis, see the Workbench Workflow Examples in the *ACP User's Guide*.

Short Fiber Reinforced Composites: You can simulate short fiber reinforced composites by importing injection molding simulation results in Mechanical through the **Injection Molding Data** system. This process involves importing and mapping fiber orientation tensor data from other software tools onto the mesh defined in Mechanical. When combined with a homogenized material computed with Material Designer, this workflow allows you to perform a Mechanical analysis of parts made of short fiber reinforced composites. To better understand how to perform the steps for this analysis, see the Short Fiber Composites Guide.

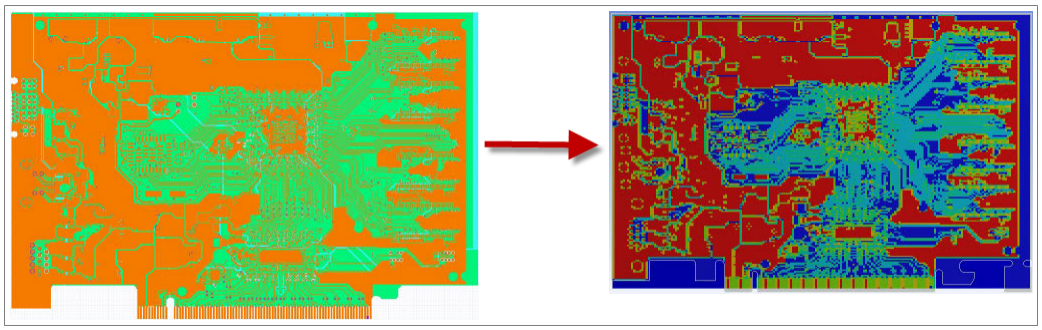
ECAD Analysis using Trace Mapping

Mechanical's trace mapping feature provides a fast and efficient method to model Printed Circuit Boards (PCBs) and packages that would otherwise require an inordinate amount of time to process a geometry and mesh. Typically, the geometry that you produce in your electronic computer-aided design (ECAD) system contains a large number of bodies and therefore large amounts of data. Given the complexity of the system, producing a mesh can be very difficult. The trace mapping feature simplifies your model by modeling the geometry as dielectric layers and includes the effect of traces by mapping the metal fraction onto the dielectric layers.

Mapping takes place in two stages. As illustrated below, during the first stage, a representation of the layout is built upon a rectangular grid using the data from a specified ECAD layout. The cell size of the grid is governed by the smallest features in the layout that have to be resolved. This size can be controlled by the user and should be specified based on the resolution required. A metal fraction value is assigned to each cell depending on the contribution of metal to that cell. The metal fraction value ranges from 0 to 1, where the 0 value represents a pure dielectric material and 1 a pure metal material.

The conduction paths that connect the metal traces between the different layers, that is, the vias, can be specified as either hollow or solid (default).

Building a Representation Upon a Rectangular Grid

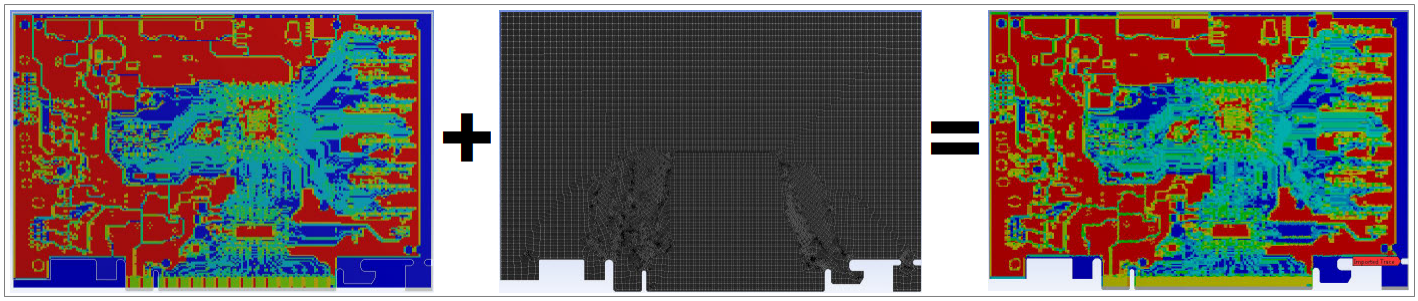


During the second stage, the metal fraction values are mapped from the source grid to the target mesh. Once the mesh is created, Mechanical then generates the mapped metal fractions. The sequence of this construction is illustrated below.

Mapping Metal Fractions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General Workflow from Workbench

The following workflow describes the general steps to perform an analysis of a Printed Circuit Board that includes trace mapping.

1. Import supported ECAD files into External Data and update your project.
2. Add the supported Mechanical system to the project.
3. Add appropriate materials using Engineering Data.
4. You can open an ECAD layout geometry in an analysis system in Workbench once you have the proper file format. You can use SpaceClaim or SCDM.

   Note:

   You can open Ansys EDB, ODB++, and IPC2581 files in Mechanical using the **Geometry** cell of the analysis system. An External Data system is not required until you import the trace mapping. See the ECAD Import (p. 703) section for more information.

5. Connect the **Setup** cell of the External Data system to the analysis system's **Model** cell.
6. Launch Mechanical. See the next section Trace Mapping in Mechanical (p. 693) for the specifics of setting up the Imported Trace object.
7. Specify vias as required.
8. Assign (p. 345) Dielectric material to the bodies. As needed, modify additional **Geometry** object properties.

   Note:

   Reference temperatures specified on bodies are included in the solution calculation for any trace material contained in an element on the associated body.

9. Assign appropriate ECAD File to **Imported Trace** object.
10. Verify that the trace layout source is properly aligned (p. 698) with geometry.
11. Assign Metal material (p. 696) to imported trace layers.
12. Define and generate mesh.
13. Set discretization (p. 695).
14. Generate the trace mapping.
15. Define boundary conditions (p. 1500).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

16. Solve the analysis.

17. Define and evaluate results (p. 1927).

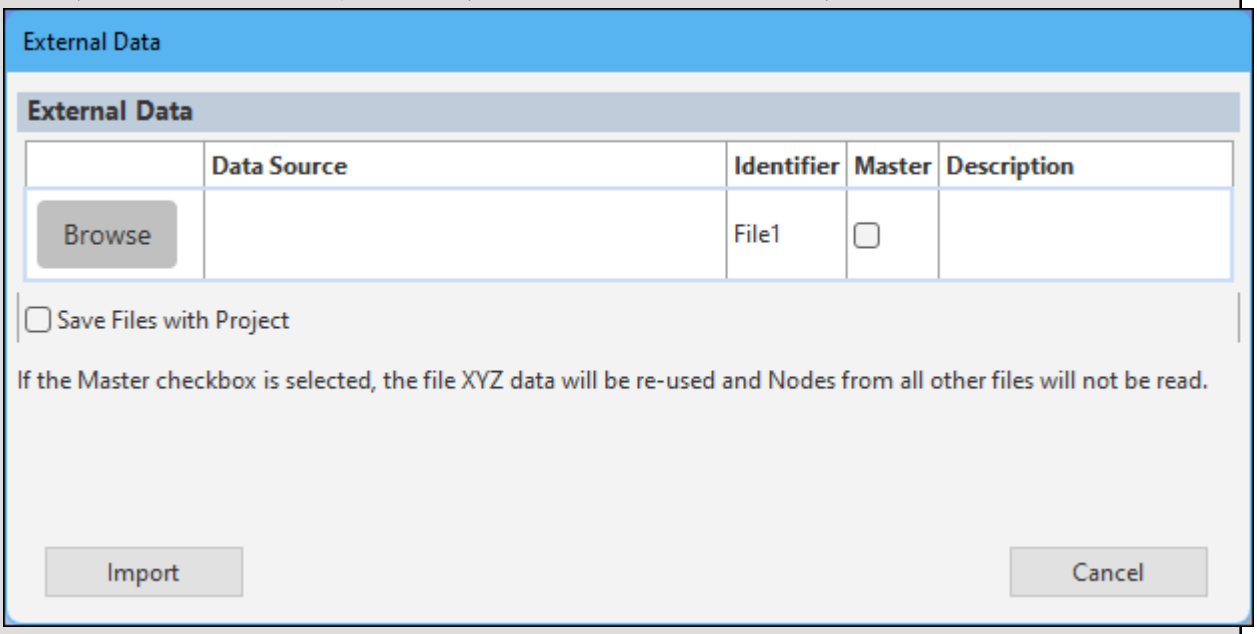
DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General Workflow from Mechanical

The following workflow describes the general steps to perform an analysis of a Printed Circuit Board that includes trace mapping.

1. Open the application from the **Start Menu** option **Ansys 2024 R1** and select **Mechanical 2024 R1**.
2. Insert an analysis system.
3. From the Materials object, select the **Imported Trace** option. The application automatically inserts the Imported Trace object into the tree and the External Data (p. 736) dialog displays. You use this dialog to specify the desired external file(s) you wish to import.

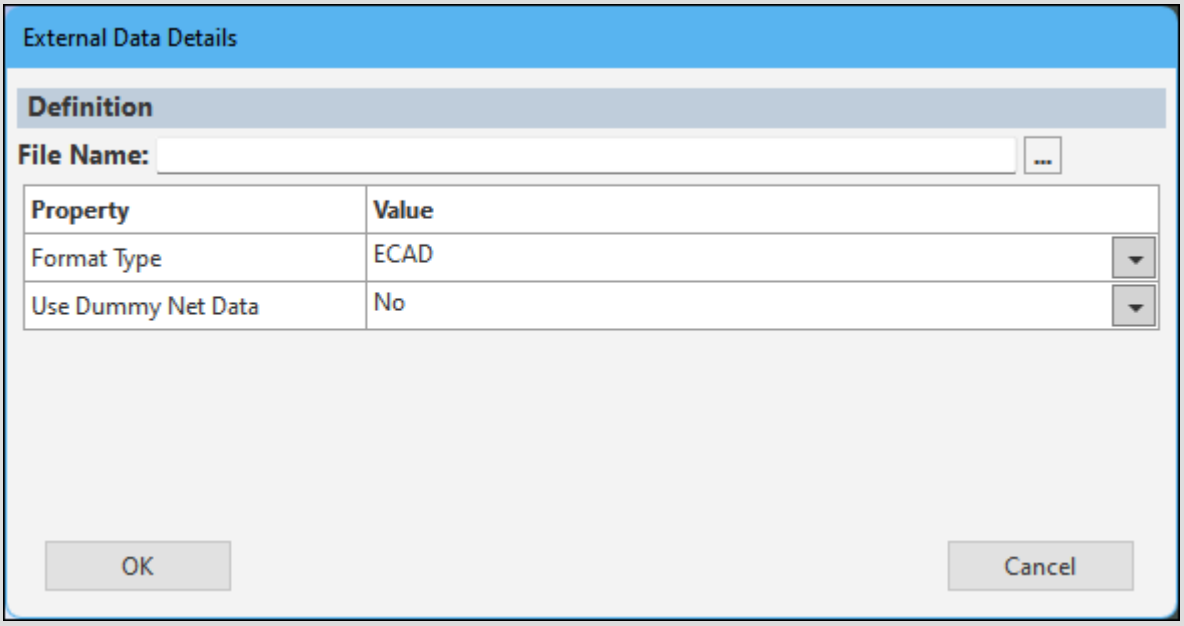


> **Note:**
>
> If you have already selected a file and displayed the dialog, there are two additional context (right-click) menu options: **Edit** and **Reload**. The **Edit** option re-opens the External Data dialog enabling you to modify the imported object's details and the **Reload** option reloads data from the source file(s).

4. Select the **Browse** button. The following dialog displays.
5. Select a desired file using the **File Name** option.
6. Specify a desired **File Format** and associated properties. Options include:
   - **ECAD**: Supported data files include ANSOFT ANF, ODB++ TGZ, IPC-2581 XML, or EDB DEF. The Use Dummy Net Data property specifies whether to import trace data belonging to the dummy net of the file.
   - **ICEPAK**: This format type supports ICEPAK BOOL or ICEPAK COND data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



7. Select the **OK** button and then the Import button. The data file is now associated with the object.

   See the next section Trace Mapping in Mechanical (p. 693) for the specifics of setting up the Imported Trace object.

8. Specify vias as required.

9. Assign (p. 345) Dielectric material to the bodies. As needed, modify additional **Geometry** object properties.

   Note:

   Reference temperatures specified on bodies are included in the solution calculation for any trace material contained in an element on the associated body.

10. Assign appropriate ECAD File to **Imported Trace** object.

11. Verify that the trace layout source is properly aligned (p. 698) with geometry.

12. Assign Metal material (p. 696) to imported trace layers.

13. Define and generate mesh.

14. Set discretization (p. 695).

15. Generate the trace mapping.

16. Define boundary conditions (p. 1500).

17. Solve the analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

18. Define and evaluate results (p. 1927).

Requirements

This workflow requires the following components or systems:

- Engineering Data: This component enables you to define materials required for the analysis. The materials defined the in the Engineering Data component will be available in the corresponding Mechanical model. For your convenience two materials (FR-4 and Copper Alloy) which commonly represent dielectric and metal in a PCB are available in the General Materials sample library.
- **Geometry**: This component enables you to create or import the geometry representing the board or the package layout. The SpaceClaim geometry editor enables you to directly import the supported ECAD formats (see below) and automatically create a (trace layout) **Geometry**. See the ECAD section in *Importing and exporting* in the SpaceClaim documentation for details.
- External Data: This component enables you to specify the ECAD file for import in Mechanical. The following ECAD File formats are supported for Trace Analysis (p. 693):

  In Mechanical

  - Ansys EDB
  - ODB++
  - IPC2581

  In External Data

  - Ansoft ANF (External Data)
  - ODB++ TGZ
  - Icepak BOOL+INFO

    Note:

    For this file type, when you are using the Linux platform, the metal fraction calculation is not supported when you start Workbench using remote access software.

  - Icepak COND+INFO

  Once an ECAD file is specified in **External Data**, additional **Rigid Transformation** controls are available in the component to align the trace data with the geometry.
- **Mechanical Systems**: The supported Mechanical systems enable you to import Trace Data, setup the analysis and solve. The following analysis are supported:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Supported Structural Analysis Systems

- Harmonic Response
- LS-DYNA
- LS-DYNA Restart
- Modal
- Static Structural
- Transient Structural

Supported Thermal Analysis Systems (Solid Bodies Only)

- Steady-State Thermal
- Transient Thermal

See Trace Mapping in Mechanical (p. 693) for the specifics of setting up an analysis.

Trace Mapping in Mechanical

This section examines the following trace mapping topics:

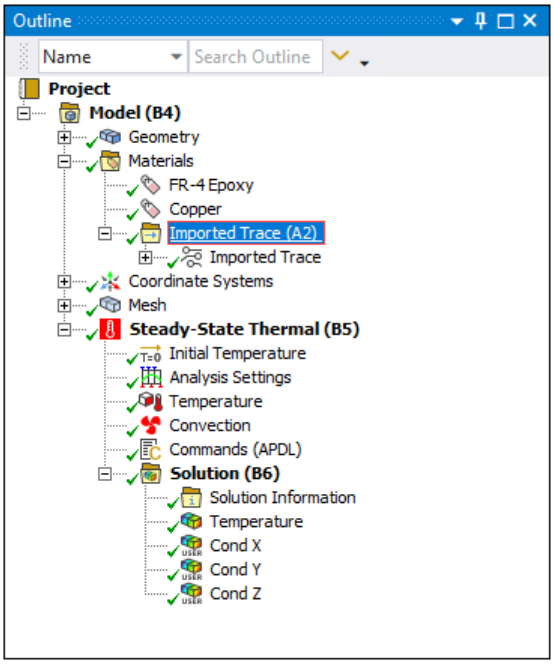
- Import Trace Mapping (p. 693)
- Specify External File (p. 694)
- Specify Properties and Settings (p. 695)
- Import Trace Layout Data (p. 699)
- Visualize Source Data (p. 701)
- Apply Boundary Conditions (p. 702)
- Solve the Analysis (p. 702)
- Evaluate Results (p. 703)

Import Trace Mapping

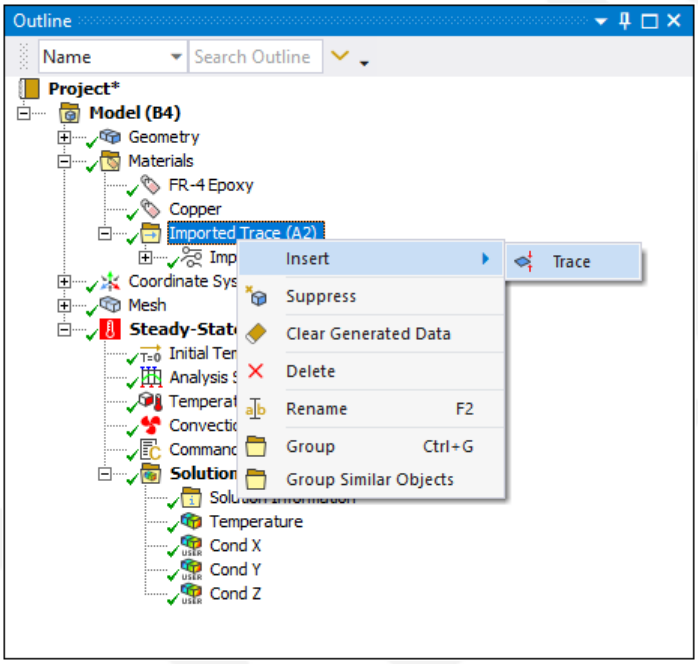
This step applies to the workflow for importing data from Workbench. In Mechanical, the application automatically inserts an Imported Trace folder and an Imported Trace object under the **Material** object.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Using the context menu (right-click) option **Insert** > **Trace** enables you to insert additional **Imported Trace** objects into the tree as needed.



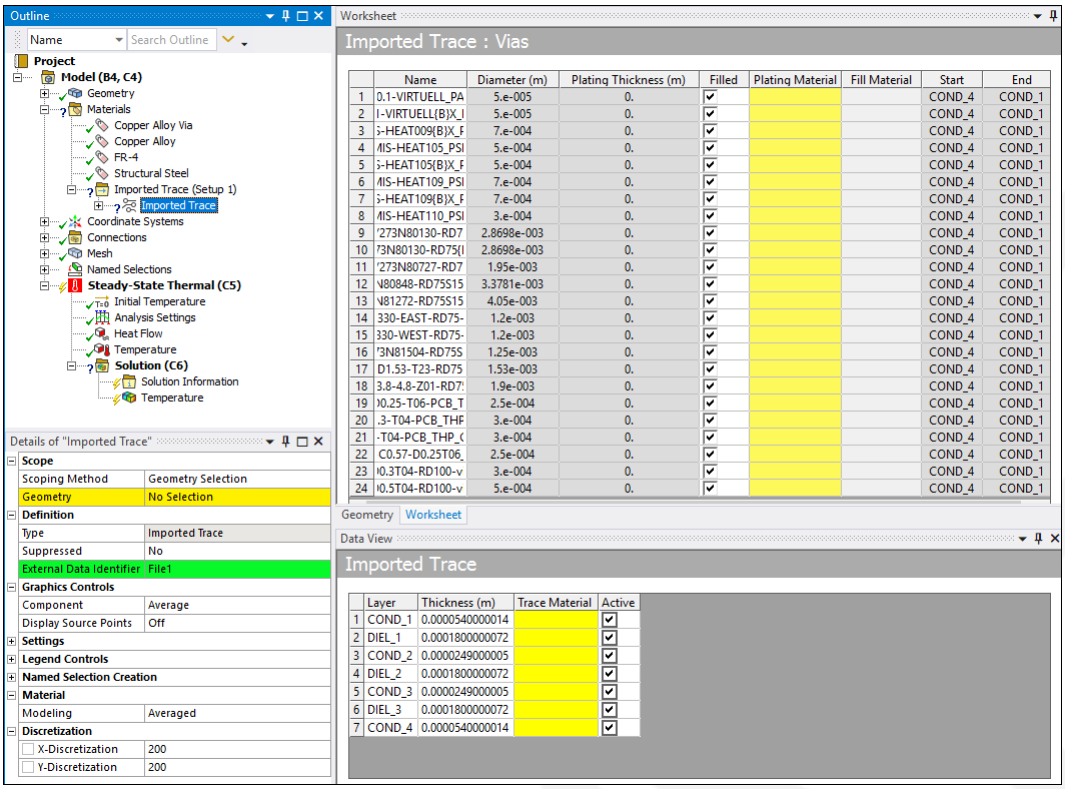
Specify External File

Once inserted, the **Imported Trace: Vias** Worksheet and the **Data View** window display. Once you specify the **External Data Identifier** property, source system data populates various application fields.

You can use the context (right-click) option on the **Worksheet** and **Data View** table to specify materials for the vias and imported traces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Specify Properties and Settings

The **Details** view and the **Data View** for the **Imported Trace** object enable you to control the specification of the imported trace file in Mechanical.

The following options are available in the Details pane to control the **Imported Trace** specification.

- **Scope**: The **Geometry** or the **Named Selection** properties, as specified by the **Scoping Method** property, support scoping to the bodies representing the layer geometry for the PCB or to elements contained on the layers of geometry.

  You can model the geometry of a PCB as a shell or solid. When modeled as solid, you should model each layer as a separate body. When modeling shells, the application models all of the layers as a single shell geometry. Mechanical uses layered shell elements to model the layers of a PCB.

  Note:

  Using the meshing Sweep Method Control, you can mesh a solid body with the Solid-Shell element (SOLSH190). This option is available from the **Element Option** property of the **Sweep Method**.

- **External Data Identifier**: This property provides a drop-down list of available ECAD files from the list of files specified in the **External Data** system.

- **X-/Y-Discretization**: Mechanical performs a two stage mapping to calculate metal fractions on the target mesh. First it computes a metal fraction distribution of the board from the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

source ECAD file to a regular grid, and then from the regular grid to the target mesh. The **X-/Y-Discretization** properties enable you to specify the size of the regular grid. The grid density count is **200** x **200** by default. Depending on the trace resolution and the computational costs desired, you can change the values for the rows and columns to receive optimum results. For accurate results, it is recommended that the X and Y discretization be specified such that the grid cell length be less than or equal to the minimum trace width. The Mechanical mesh size is recommended to be less than 4 times the grid cell length.

These fields are read-only when the ECAD File specified in **External Data Identifier** is of Icepak **COND+INFO** format and displays the discretization of the **COND** file.

Numerous additional properties are available in the Details. See the Data Transfer Mesh Mapping (p. 2389) section for more information about these various properties.

Solid Model

Details of "Imported Trace"

Scope	
Scoping Method	Geometry Selection
Geometry	7 Bodies
Definition	
Type	Imported Trace
Suppressed	No
External Data Identifier	File3
Graphics Controls	
Component	Average
Display Source Points	Off
Settings	
Mapping Control	Program Controlled
Mapping	Profile Preserving
Weighting	Volume Fraction
Transfer Type	Volumetric
Legend Controls	
Legend Range	Program Controlled
Minimum Source	Program Controlled
Maximum Source	Program Controlled
Named Selection Creation	
Unmapped Elements	Off
Mapped Elements	Off
Outside Elements	Off
Material	
Modeling	Averaged
Discretization	
X-Discretization	200
Y-Discretization	200

Shell Model

Details of "Imported Trace"

Scope	
Scoping Method	Named Selection
Named Selection	Outer_Body
Definition	
Type	Imported Trace
Suppressed	No
External Data Identifier	File1
Shell Offset	User Defined
Membrane Offset	-0.3 mm
Graphics Controls	
Layer	BOTTOM
Component	Average
Display Source Points	Off
Settings	
Mapping Control	Program Controlled
Mapping	Profile Preserving
Weighting	Volume Fraction
Transfer Type	Surface
Legend Controls	
Legend Range	Program Controlled
Minimum Source	Program Controlled
Maximum Source	Program Controlled
Named Selection Creation	
Unmapped Elements	Off
Mapped Elements	Off
Outside Elements	Off
Material	
Modeling	Averaged
Discretization	
X-Discretization	200
Y-Discretization	200

The **Data View** of the **Imported Trace** object enables you to see and/or control/override the following properties of the layers in the trace layout:

- **Layer**: The name of the **Layer**.
- **Thickness**: The thickness of the layers.
- **Trace Material**: This property specifies the material for the metal traces on each layer. This material is created in the Engineering Data workspace for the Mechanical system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Dielectric Material** (shell geometry only): This property specifies the dielectric material for each layer.

  This option is only available for traces scoped to shells. For imported traces scoped to solids, the base (dielectric) material is specified on the **Material Assignment** property of the selected bodies (under the Geometry object).

- **Active**: This option enables you to activate or deactivate one or more layers. This option is not available for shell geometries. All layers are sent to the solver for traces imported on shells.

- **Mechanical Layer No.**: Only available for traces scoped to shells. This read-only field displays the layer number by which this layer is identified. For example, if you want to post process results on the *L1* layer show below, you will need to specify *Layer 7* in the Details view of the result object.

Solid Model

Data View

Imported Trace

	Layer	Thickness (cm)	Trace Material	Active
1	TOP	0.00482599996	Copper	☑
2	UNNAMED_002	0.0067309998	Copper	☑
3	PWR	0.00330200009	Copper	☑
4	UNNAMED_004	0.12699999809	Copper	☑
5	LYR_1	0.00304799993	Copper	☑
6	UNNAMED_006	0.02031999975	Copper	☑
7	LYR_2	0.00304799993	Copper	☑
8	UNNAMED_008	0.02031999975	Copper	☑
9	GND	0.00330200009	Copper	☑
10	UNNAMED_010	0.0067309998	Copper	☑
11	BOTTOM	0.00482599996	Copper	☑

Shell Model

Data View

Imported Trace

	Layer	Thickness (mm)	Trace Material	Dielectric Material	Me
1	L1	0.0270000007	Copper	Fr4	7
2	D1-2	0.0399999991	Copper	Fr4	6
3	L2	0.0199999996	Copper	Fr4	5
4	mid-layer	0.150000006	Copper	Fr4	4
5	L3	0.0199999996	Copper	Fr4	3
6	D3-4	0.0399999991	Copper	Fr4	2
7	L4	0.0270000007	Copper	Fr4	1

Messages | Data View

The **Imported Trace: Vias** Worksheet enables you to view and/or control and override the following properties of the vias in the trace layout.

Worksheet

Imported Trace : Vias

	Name	Diameter (cm)	Plating Thickness (cm)	Filled	Plating Material	Fill Material	Start	End
1	C10N116	0.29464	0.	☑	Copper		BOTTOM	TOP
2	C10N116_flipped	0.29464	0.	☑	Copper		BOTTOM	TOP
3	C10N133	0.33782	2.5e-005	☐ 5	Copper	Fr4	BOTTOM	TOP
4	C10N25	6.35e-002	0.	☑	Copper		BOTTOM	TOP
5	C10N40	0.1016	0.	☑	Copper		BOTTOM	TOP
6	C10N43	0.10922	0.	☑	Copper		BOTTOM	TOP

Geometry | Worksheet

Column Heading	Column Description
Name	This field displays the name of the layer as defined in the source file.
Diameter	This read-only field displays the outer diameter value of the via.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Column Heading	Column Description
Plating Thickness	This field displays the thickness of the interior wall of the via that is not filled. Only applicable when via is hollow (not filled).
Filled	Check and/or uncheck this fill to indicate whether the vias is filled or hollow.
Plating Material	When active, this field displays the metal material of the via. It provides a drop-down menu that enables you to specify different materials for the via.
Fill Material	This field provides a drop-down menu when the corresponding **Fill** option is inactive, i.e. via is hollow (not filled). It enables you to specify a material for the hollow region of the via. You can assign Air (defined in Engineering Data) if the hollow region is empty.
Start/End	These read-only fields display the layer associated with where the via starts and ends.

In addition to these basic controls, the Details view of the **Imported Trace** object provides additional properties that enable you to control/visualize the source data in Mechanical:

- **Display Source Points**/**Interior Points**: These properties enable you to visualize the source points from the trace layout files. These settings can be used to verify the alignment of the source points with the target geometry. If misaligned, use the **Rigid Transformation** controls in the **External Data** system to align the source mesh with the target.
- **Mapping Control**: This property controls mapping settings for the import. Options include **Program Controlled** (default) and **Manual**. The **Program Controlled** option enables the application to determine the appropriate algorithm and settings based on the source and target mesh data, as well as the data type being transferred. See the Program Controlled Mapping (p. 2408) topic in the *Data Transfer Mesh Mapping* section for additional information.
- **Interpolation**: Options include **Nondirectional** (default) and **Directional**. The **Nondirectional** option calculates effective conductivity by averaging the trace data in each element. Using the **Directional** option, the application calculates the effective orthotropic conductivity for each element using the position and values of the trace data within each element.

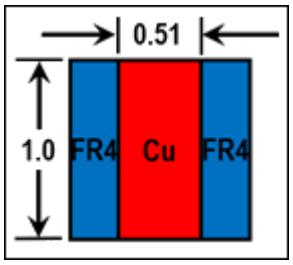
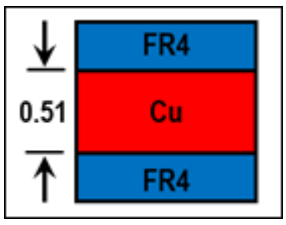
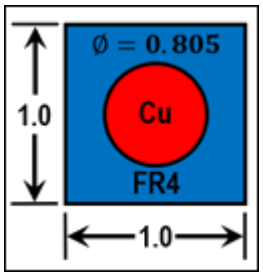
  Directional Trace Mapping

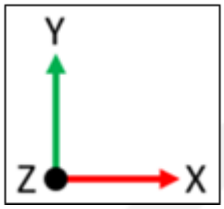
  This interpolation calculates orthotropic thermal conductivities for each element. The calculation is done by modeling the trace data in each element as a thermal resistance network. The examples illustrated below describe the calculated conductivity values based on the corresponding scenarios on the left. The dielectric in this example has a conductivity of 0.294 W/mK and the metal has a conductivity of 400 W/mK. Each element is 51% metal and 49% dielectric (Metal Fraction of 0.51).

	Metal Fraction	Kx (W/mK)	Kx (W/mK)	Kz (W/mK)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

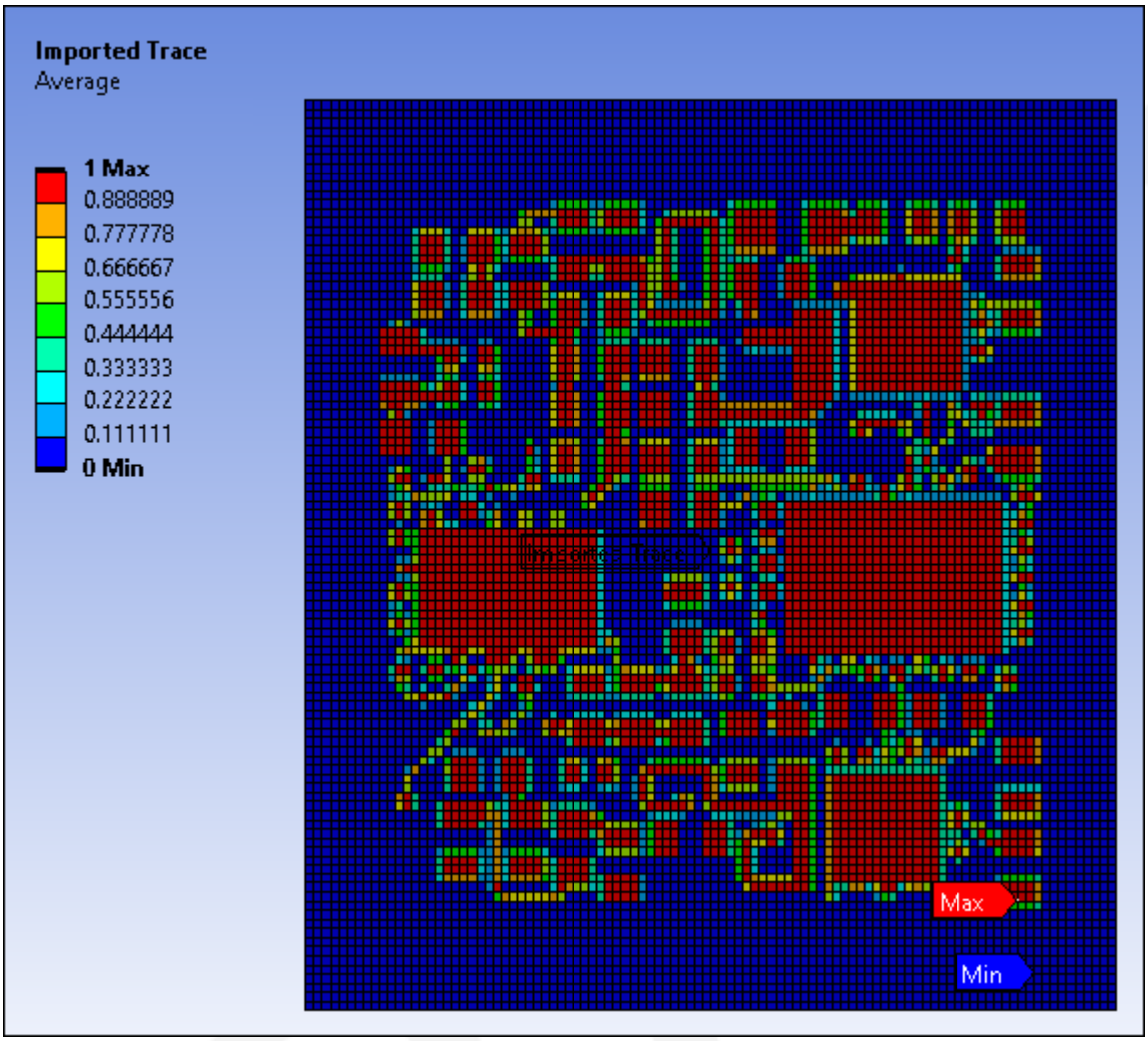
	0.51	0.60	204.1	204.1
	0.51	204.1	0.60	204.1
	0.51	~1	~1	204.1



Import Trace Layout Data

Once you have fully defined the object, right-click the **Imported Trace** object and select **Import Trace** to import the trace layout data onto the specified bodies. As illustrated below, the object displays the variable data, per element, of how much of the trace data is dielectric (metal) material. One (1) is 100% metal and zero (0) is 0% metal.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



When you import trace layout data, the application generates the Imported Data Transfer Summary object. This object presents a summary, in the Comment pane, of the mapped source data as compared to target data. More specifically, it presents an average metal fraction for target elements, including both averaged modeling and black/white modeling. This enables you to validate your target metal fractions against the external application, such as Ansys Sherlock, you use to provide this metal fraction. However, unless the source data aligns perfectly to the target mesh, Ansys does not recommended that you compare the source and target average metal fractions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Comment

Arial 1 (8 pt)

Using multiple cores: [Yes]
Number of cores requested: 64

Number of source elements: 38220
Number of target elements: 152880

Number of elements mapped : 152880
Number of elements not mapped : 0
Number of elements outside : 0

Percent nodes mapped: 100%
Weight calculation time: 0.689 (s)
Number of variables to interpolate: 4.
Interpolation time: 1.3e-002 (s)

Results:

Average Metal Fraction:

Row	Source			Target (Averaged)			Target (Black/White)		
	X	Y	Z	X	Y	Z	X	Y	Z
1	0.9969103	0.9944473	0.9949214	0.9969103	0.9944473	0.9949214	1.	1.	1.
2	0.1732562	0.1712296	0.1656506	0.1732562	0.1712296	0.1656506	0.1711931	0.1711931	0.171193
3	0.9969103	0.9944473	0.9949214	0.9969103	0.9944473	0.9949214	1.	1.	1.

Metal Volume (m³):

Row	Source			Target		
	X	Y	Z	X	Y	Z
1	4.17153e-010	4.161224e-010	4.163208e-010	4.166756e-010	4.156461e-010	4.158443e-010
2	9.712044e-010	9.598438e-010	9.285702e-010	9.700929e-010	9.587453e-010	9.275075e-010
3	4.17153e-010	4.161224e-010	4.163208e-010	4.166756e-010	4.156461e-010	4.158443e-010

In addition, the following properties in the **Graphics** category help to visualize the mapped data:

- **Component**: X, Y, or Z component or the average of the imported metal fraction data.
- **Layer**: Layer to display the data for (only applicable when scoped to shells)

Visualize Source Data

The source data can be visualized using the Validation object. To visualize the source data, use the context menu option **Insert**>**Validation** on the **Imported Trace** object to insert a **Validation** object in the tree. Once inserted, set the appropriate properties and use context menu option **Analyze**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

on the validation object. See Mapping Validation (p. 2412) in the *Ansys Mechanical User's Guide* for details.

Apply Boundary Conditions

You can freely apply any supported boundary condition (p. 1500) on solid bodies scoped to Imported Traces. Certain restrictions apply for boundary conditions on shell bodies scoped to Imported Traces. For example boundary conditions like Thermal Condition (p. 1594) and Imported Body Temperature (p. 1740) can only be applied to both the shell faces.

Solve the Analysis

The effects of **Imported Trace** data are included during the solution phase by computing the material properties of the materials assigned to the bodies based on the metal fraction. The **Modeling** property in the **Material** group controls how material properties are calculated based on computed metal fraction values. Two options are available:

- **Black/White**: Assign trace material to regions with average metal fraction greater than or equal to 0.5, and dielectric to the rest.
- **Averaged**: Calculate material properties based on calculated metal fraction. The supported material properties based on metal fraction are listed in the tables below:

  Thermal Analysis

  The supported thermal material properties include:

  – Thermal Conductivity (Kx/Ky/Kz)

  – Specific Heat

  Structural Analysis

  The supported structural material properties include:

  – Young's Modulus (Ex/Ey/Ez)

  – Poisson's Ratio (PRxy/PRyz/PRxz)

  – Coefficient of Thermal Expansion

  – Density

  Note:

  – Non-linear materials are not supported when **Material Modeling** is set to **Averaged**. If the application detects non-linear materials assigned either as trace or dielectric material when the material modeling is set to **Average**, then the **Imported Trace** object becomes invalid and the solution cannot proceed unless the conditions are made valid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- If any linear material properties other than the ones listed in the above table are present on either the trace or dielectric material, they are not sent to the solver.
- When using LS-DYNA trace mapping, thermal analysis is not supported, and the coefficient of thermal expansion is not a supported material property.

Material Property Calculation for Temperature Dependent Material Properties

For temperature dependent material properties, averaging takes place over two stages. First the material properties are calculated at the union of all the temperatures, and then the average metal properties are calculated based on the above table for each temperature point. For example, the Metal material has property **P** specified at temperatures **T1**, **T2** and **T3**, whereas the Dielectric material has property **P** specified at **T2** and **T4**, then the material property is first calculated for both Metal and Dielectric materials at temperatures **T1**, **T2**, **T3**, and **T4**, and then the effective material properties are calculated at **T1**, **T2**, **T3**, and **T4** using the table specified above.

Offsets (Only applicable for shells)

When scoped to shells, the **Shell Offset** property allows user to specify the shell offset for the scoped bodies. Available options are **Top/Middle/Bottom** and **User Defined**. The **User Defined, Membrane Offset** property allows you to specify the offset value.

Evaluate Results

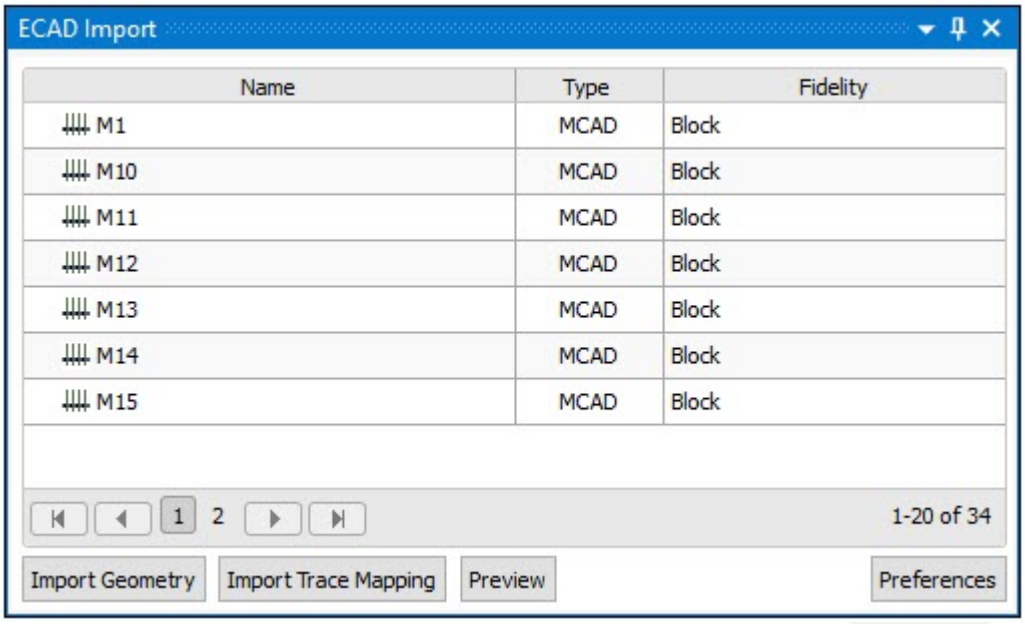
Once the solution is complete, user can insert appropriate results and evaluate them. Since the effect of metal and dielectric within an element is captured through material property averaging, stress results may deviate from full fidelity analysis. However it provides a qualitative description of stress distribution. User may perform a subsequent Submodeling analysis to get accurate stress distribution.

ECAD Import Pane (Windows Only)

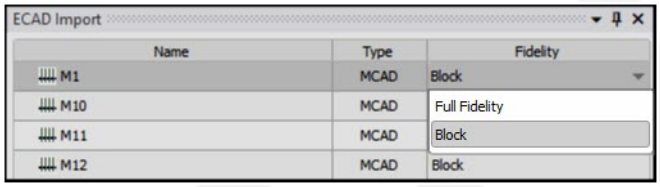
On the Windows platform, you can open Ansys EDB, ODB++, and IPC2851 ECAD files directly in Mechanical using the **Geometry** cell of the analysis system.

When you import ECAD files into Mechanical, the **ECAD Import** pane, illustrated below, displays automatically. Because printed circuit boards contain many detailed components, this feature enables you to either limit or expand the degree of detail for displaying a component and its bodies. Specifically, you can greatly simplify the display of complicated components that are not pertinent to your analysis and as a result speed up processes such as mesh and solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The first two columns of the pane list each part of your model by **Name** and **Type**. Parts can include PCBs, MCAD components (3D geometries representing board components like USB ports, power connectors), BGAs, etc. The **Fidelity** column enables you to specify whether to present a component as a **Block** (default) or to includes the detailed parts or bodies of the component using the **Full Fidelity** option.



ECAD Import Pane Features

The **ECAD Import** pane provides the following options for your analysis:

Import Geometry

This option imports your geometry into Mechanical. You can make fidelity changes before and after importing the geometry. You must select the **Import Geometry** button again in order changes to take effect.

Import Trace Mapping

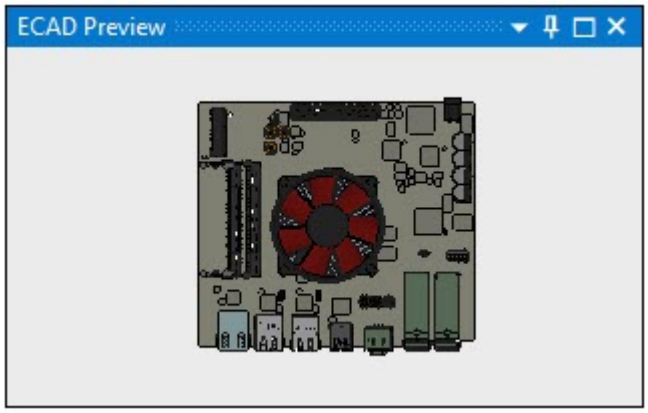
This option automatically links an upstream **External Data** system to the **Model** cell of your analysis on the Workbench project page and imports Trace Mapping data. An **Imported Trace (Group)** folder and its child object, Imported Trace, are automatically inserted under the Materials folder in the **Outline**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

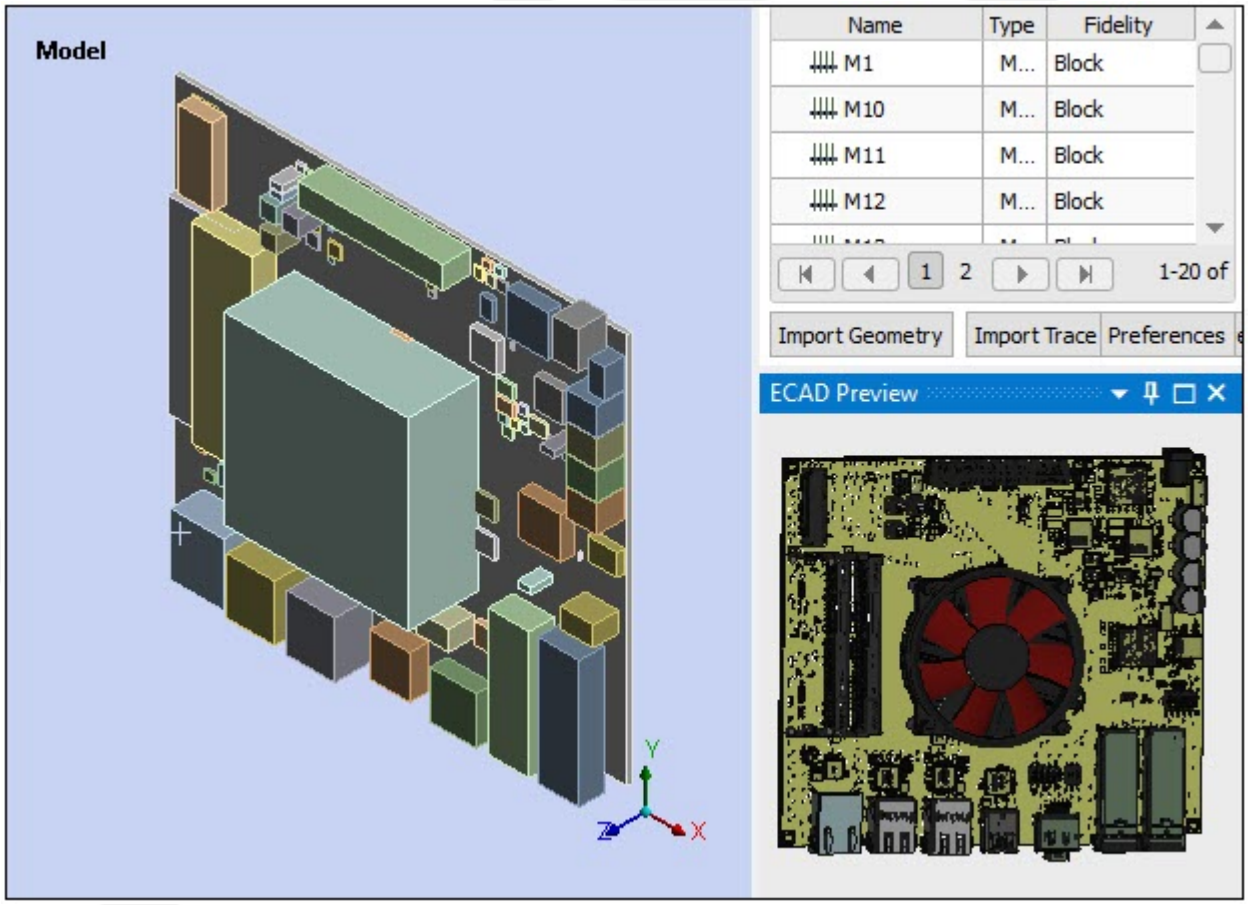
Published: 2023-10-23

Preview

The **Preview** option displays a rendering of your model. By default, before you import your geometry into Mechanical, the **ECAD Preview** pane renders it with all components in **Full Fidelity**.



Once you select the **Import Geometry** button, the model renders in the **Geometry** window based on the settings of the **ECAD Import** pane. An example is illustrated here with all components using **Block** fidelity.

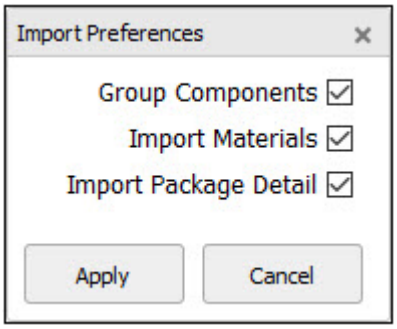


You can change the fidelity settings as desired and update the model rendering in the **Geometry** window using the **Import Geometry** button. Note that this action can invalidate scoping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Preferences

The **Preferences** button displays the dialog shown below. The options of the dialog are active by default. These options automatically take general import actions.

Dialog	Preference	Description
	**Group Components**	Automatically g under the **Geor**
	Import Materials	Automatically ir the materials fr to complete the
	Import Package Detail	Automatically ir

Defining Fidelity following Import

In addition to the drop-down menu of each row in the **ECAD Import** pane, you can change the **Fidelity** using any of the following methods. Once a fidelity change is made to a component, you must select the **Import Geometry** button in order for it to take effect. Also note that re-importing the model can affect the scoping of associated components.

- Select multiple rows simultaneously using the **[Shift]** key.
- Select multiple rows individually using the **[Ctrl]** key.
- Right-click on parts of the model in the **Geometry** window and select **Set Fidelity** > **Block** or **High Fidelity**. This action also automatically displays the ECAD Import pane if it is not already displayed.

Electromagnetics (EM) - Mechanical Data Transfer

You can import data generated by the HFSS, Maxwell, or Q3D Extractor applications and perform an analysis in Mechanical by applying the imported loads. In the case of loads originating from HFSS and Maxwell, you can also export the temperature or deformation results obtained from the Mechanical analysis so that they can be imported back into HFSS or Maxwell. Furthermore, you can import:

- Thermal loss data generated by the HFSS, Maxwell, or Q3D Extractor applications and use the imported load to perform a thermal analysis. The resulting temperature results then can be exported and applied during the subsequent solution of the upstream Maxwell analysis.
- Force densities generated by the Maxwell application and use the data to perform a static or transient structural analysis. The resulting deformation results can then be exported and applied during the subsequent solution of the upstream Maxwell analysis.
- Forces and Moments or Surface Force Densities generated by the Maxwell application to use as loading conditions in a Harmonic Response analysis using the load. If you perform a Maxwell Parametric Study, the application automatically transfers all Maxwell loads for all RPMs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Overall Workflow for an EM - Mechanical Analysis

1. Create and solve the electromagnetic application using HFSS, Maxwell, or Q3D Extractor.
2. Drag and drop a steady-state thermal, transient thermal, static structural, transient structural, or harmonic (Maxwell *only*) template on top of the HFSS, Maxwell, or Q3D Extractor systems solution cell to enable the data transfer.
3. Attach geometry to the Mechanical application, and then double-click **Setup** to open the Mechanical window. An **Imported Load** or an **Imported Remote Load** folder is added under the **Environment** folder, by default.

   Note:

   Note that a Maxwell system is not required for a Coupled Field Harmonic, Harmonic Acoustics, and Harmonic Response analysis to import Remote Force and Remote Moment boundary conditions. These boundary conditions can be imported directly from an external Maxwell file. See the Importing Forces and Moments (p. 711) section for more information.

4. As required, you can add or generate imported loads and set their options.
5. Perform all steps to set up a Steady-State Thermal (p. 657), Transient Thermal (p. 680), Static Structural (p. 652), Transient Structural (p. 665), Coupled Field Harmonic (p. 380), Harmonic Acoustics, or Harmonic Response (p. 496) analysis. Specify mesh controls, boundary conditions, and solution settings as you normally would.
6. Solve the Ansys analysis.
7. If applicable, export your results to make them available for import by the upstream applications.

See the following sections for more detailed procedures to import and/or export loads during Thermal, Structural, and Harmonic analyses.

- Importing Data into Thermal or Structural (Static or Transient) Analyses (p. 707)
- Importing Data into Harmonic Analyses (p. 710)
- Exporting Results from Thermal or Structural Analyses (p. 718)
- Electric Machines NVH Analyses (with Waterfall Diagram) (p. 720)

For additional information, refer to the **Maxwell Coupling** section in the **Maxwell Online Help**, available through the *Ansys customer site*.

Importing Data into Thermal or Structural (Static or Transient) Analyses

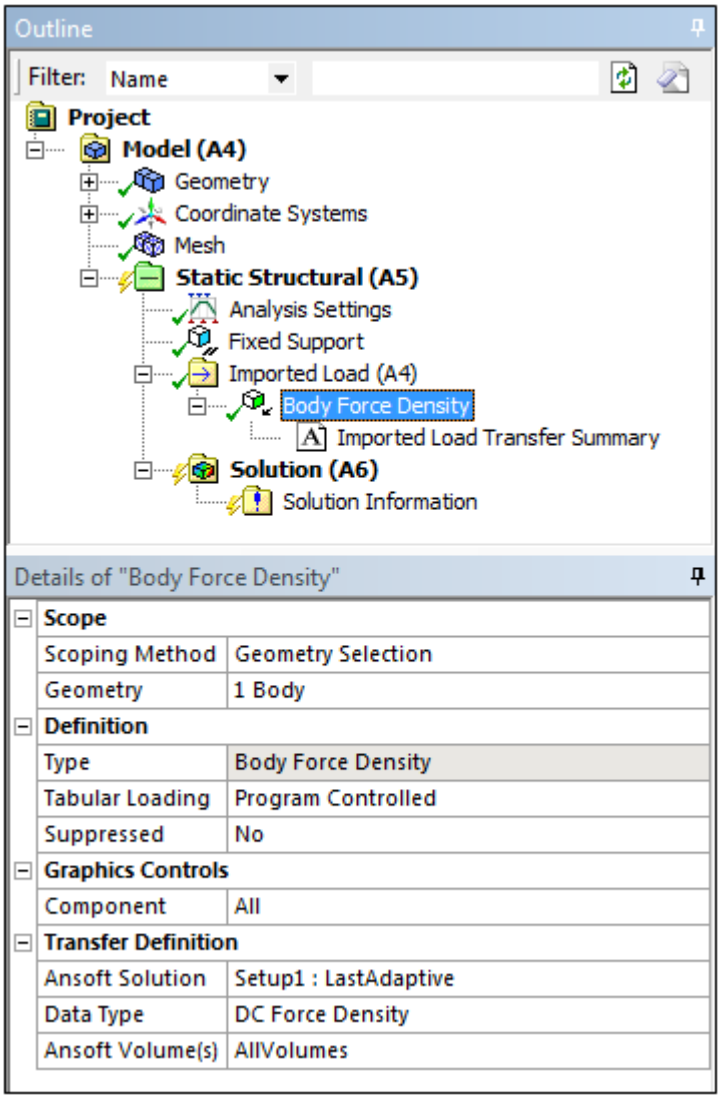
This feature enables you to perform a one-way Electromagnetics (EM) - Mechanical interaction problem by solving the electromagnetic analysis of the geometry in the HFSS, Maxwell, or Q3D Extractor applications, importing the thermal or structural results into the Ansys Mechanical applic-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

ation where the defined load is applied to a thermal or structural analysis which is then solved and post processed.

For a thermal analysis, you can import Imported Heat Generation (p. 1748) and Imported Heat Flux (p. 1748) load types.

For a structural analysis you can import Imported Body Force Density (p. 1739) (illustrated below) and Imported Surface Force Density (p. 1755) load types.



Add the Imported Load

Follow these steps to add an imported load and associate it with parts of the geometry.

1. Double-click the **Model** cell in your analysis system to open the Mechanical application.
2. Click the **Imported Load** group object. In the Details view, set the following field as needed:
   - If you want to suppress all of the loads under this **Imported Load** group, set the **Suppressed** field to **Yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. For the **Body Force Density** and the **Surface Force Density** loading types, you can choose to import the **Max Force Density** or the **DC Force Density**, if the source provides the data, using the **Data Type** property. By default, the application imports the **DC Force Density** values. The application combines AC and DC values to calculate the **Max Force Density** value. Because the AC force is a complex vector, the peak vector among all phases of the AC force is taken and added to the DC force to obtain the **Max Force Density**.

4. There are several ways to select an imported load and associate it with a part of your model.

   - Select an **Imported Load** group object in the tree, select a part of the model, then right-click **Imported Loads** and from the **Insert** menu, select the desired load type. The load will be applied to the object you selected on the model.

   - Click an **Imported Load** group object in the tree, then select the **Imported Loads** drop-down menu on the Context tab and select the desired load type from the allowed imported load types. In the Details view, click the **Geometry** field. Select the objects in the model to which you want to apply the load and click the **Apply** button in the Geometry field.

   - Right-click the **Imported Loads** group object and select **Insert** and the desired load type from the allowed imported load types. In the Details view, click the **Geometry** field. Select the objects in the model to which you want to apply the load and click the **Apply** button in the Geometry field.

   Note:

   Heat generation loads scoped to a surface body use the constant thickness value specified in the details view of the surface body object when data is imported. Surface body thickness defined using the thickness object is not accounted for when data is imported.

Set the Imported Load Options

1. Click the imported load object that you've added to the tree.

2. Select the desired Ansoft solution you would like to import the load from. Some of the properties in the Details view and **Data View** tab are filtered based on this selection.

3. Change any of the fields in the Details View as needed:

   - **Scoping Method**: Select the method of choosing objects to which the load is applied: Geometry Selection or Named Selection.

   - **Geometry or Named Selection**: Use these fields to choose the objects to which the load is applied, as appropriate from your Scoping Method choice.

   - **Suppressed**: Select **Yes** to suppress this load.

   - **Ansoft Surface(s)**: Select the Ansoft Surface(s) for a Heat Flux or Surface Force Density load.

     or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Ansoft Volume(s): Select the Ansoft Volume(s) for a Heat Generation or Body Force Density load.

Set the Imported Load Analysis Options

You can specify when the imported data should be applied and also modify the imported data, either by adding an offset or by using a scale factor.

To see the analysis setting for a load, click the object that you've added to the tree. The analysis options appear in the **Data View** tab of the window below the model. Make any changes to the load's analysis options as indicated below.

Change any of the columns in the **Data View** tab as needed:

- **Source Frequency** - Select from the drop-down list one of the frequencies supplied from the transfer file. The load values associated with this frequency will be imported.
- **Source Time** - Select from the drop-down list one of the Source Times supplied from the transfer file. The load values associated with this time will be imported.
- For thermal loads from Maxwell transient solutions, you must select from the drop-down list the desired **Source Start Time** and **Source Stop Time** to define the interval for integrating the power loss density distribution.
- **Analysis Time** - Choose the analysis time at which the load will be applied. This must coincide with the end time of a step defined in the Analysis Settings object in the tree.
- **Scale** - The amount by which the imported load values are scaled before applying them.
- **Offset** - An offset that is added to the imported load values before applying them.

You must re-solve after making any changes to the analysis options of a load.

You can define multiple rows in the **Data View** tab to import additional data from the selected Ansoft solution and apply the rows at different analysis times. If multiple rows are defined in the **Data View** tab, you can display imported values at different time steps by changing the **Active Row** option in the Details pane.

Right-click the **Imported Load** object and click **Import Load** to import the load. When the load has been imported successfully, a contour plot of the temperatures will be displayed in the Geometry window and a summary of the transfer is displayed as a comment in the particular load branch.

Importing Data into Harmonic Analyses

Mechanical enables you to import either object-based (**Imported Remote Loads** (p. 1736) - Forces and Moments) or element-based (**Surface Force Density** (p. 1755) or **Body Force Density** (p. 1739)) data from an upstream analysis. If there is a frequency sweep setup in Maxwell, the element-based data can be imported at any frequency specified in the upstream sweep. If a Parametric Setup is defined in Maxwell, loads are imported for all RPMs. Object-based data are always imported at the nominal frequency. Select a link below for additional information for each import type:

- Importing Forces and Moments (p. 711): Supported by Maxwell Transient and Eddy Current Solvers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Importing Surface Force Density (p. 716): Supported by Maxwell Eddy Current Solver and Transient Solvers.
- Importing Body Force Density (p. 1739): Supported by Maxwell Eddy Current Solver and Transient Solvers.

For additional information, refer to the **Maxwell Coupling** section in the **Maxwell Online Help**, available through the *Ansys customer site*.

Importing Forces and Moments

The following procedure assumes that you have properly defined and solved your Maxwell system.

Go to a section topic

- Import Remote Loads (p. 711)
- Define Imported Remote Loads (p. 711)

Import Remote Loads

To import data into Mechanical, either:

- Link your Maxwell system with a Harmonic Response analysis in the Workbench Project Schematic to automatically import the remote forces and moments or you can export your data from Maxwell and import it directly into a Harmonic Response system in Mechanical. For this configuration, the application only supports a Harmonic Response system.
- Open a Coupled Field Harmonic, Harmonic Acoustics, or Harmonic Response analysis and use the **Imported Remote Loads** option available in the **Loads** drop-down menu of the Environment tab, to manually insert an object of the same name that provides the **Transfer File** property. This property enables you to import Remote Force and Moment boundary conditions directly from an external Maxwell file.

Note:

When importing Maxwell files, note that there is an **Options** > Import (p. 222) category preference, **Improve Reading Performance**, that is active by default and that significantly improves the import process.

Define Imported Remote Loads

1. Select the **Imported Remote Loads** object in the **Outline**. In the Details pane, define the following properties, if necessary:

   Scoping Method

   This property defines the geometry on which the imported data is applied. Face/edge/vertex selections are supported for 3D analyses and edges/vertices for 2D analyses. Options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Geometry Selection**: default setting, indicating that the load is applied to a geometry or geometries.

  When the **Scoping Method** is set to **Geometry Selection**, the **Geometry** property becomes visible. Use this property to specify your desired geometry selections. Once specified, the field displays the type of geometry (Face or Edge) and the number of geometric entities (1 Face, 2 Edges) to which the load has been applied using the selection tools.

- **Named Selection**: indicates that the geometry selection is defined by a **Named Selection**.

  When the **Scoping Method** is set to **Named Selection** a **Named Selection** property becomes visible. This property provides a drop-down list of available user-defined **Named Selections**.

Transfer File

This property displays when you have manually inserted a **Imported Remote Loads** object from the **Loads** drop-down menu. You use this property to specify the external Maxwell file from which you want to import data. This property is not visible if you have an upstream Maxwell system linked to the analysis. Linking a Maxwell system in the Project Schematic automatically places an **Imported Remote Loads** object in the Outline.

Solution

Select the desired Maxwell solution you would like to import the load from.

Grouping

This property specifies that the loads are grouped by scoping location. That is, one load per location for all RPMs instead of one load per location and RPM. Options for this property include **No** (default) and **By Location**.

If you group remote loads, use the **RPM Selection** property of the individual loads to view the imported load data for each RPM step, presented in the **Tabular Data** window.

Data Type

This property enables you to select which remote loads to import. Options for this property include **Forces and Moments**, **Forces Only**, and **Moments Only**.

Remote Points

Select the appropriate option to generate **Internal** or **Globally Available** remote points. If you select the **Globally Available** option, the application automatically creates the Remote Points/Remote Point objects in the Outline. If you switch from the from the **Globally Available** setting to **Internal**, you need to select the **Generate Remote Loads** context menu option to remove the **Remote Points** objects from the tree.

On Data Refresh

This option is available when the **Remote Points** options is set to **Globally Available**. The **Regenerate** option deletes the remote points that were created during the previous

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

import and adds new remote points when data is imported. The **Reuse Remote Points** option reuses the previously added remote points and only updates the scoping and location, if necessary.

Number of Frequencies To Consider

With this property, you can specify the number of Maxwell frequencies to use in the analysis. The application automatically includes the first frequency, by default, and up to the value you specify. The default setting is **20**.

Import Status

This read-only property displays the status of the import. One of the following status conditions will exist:

- **Data Unavailable**: no data is available to perform the import.
- **Obsolete**: data is available to be imported, but no data has been imported or the data is obsolete and should be re-imported.
- **Uptodate**: all data has been imported.
- **Import Failed**: an error occurred during the import process and no data was imported
- **Number Of Frequencies To Consider**: The default value is **25**. Select the frequency row number used to synchronize the **Analysis Settings**. For each RPM, the application uses the **RPM Frequency Range Maximum** and **RPM Solution Intervals** of the selected **Number Of Frequency To Consider**.

Suppressed

If you want to suppress all of the loads under this **Imported Remote Loads** object, set this property to **Yes**.

Note:

Object-based data are always imported at the nominal frequency.

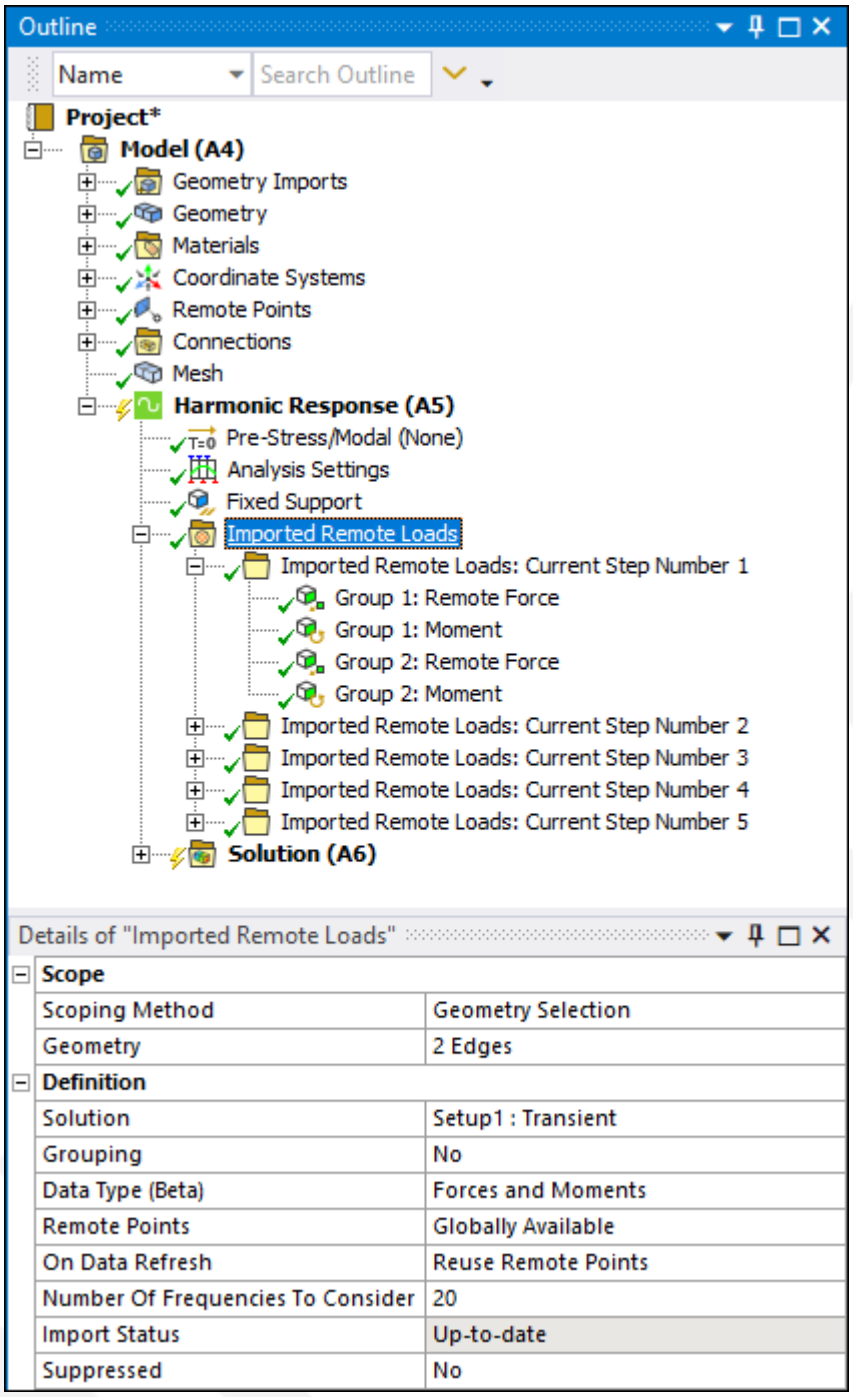
---

2. Once you have defined the necessary import options, right-click the **Imported Remote Loads** object and select **Generate Remote Loads**. This action imports the source data and associates it with the selected target geometry. Once executed, Mechanical adds objects to the tree based on the source data.

   The following items will be added into the tree based on the source data. For each location that Maxwell reports the calculated forces, Mechanical:

   - Adds a **Remote Force** object and a **Moment** object with the imported data. Each set of two loads are named with a Group ID number, as illustrated in the following example.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



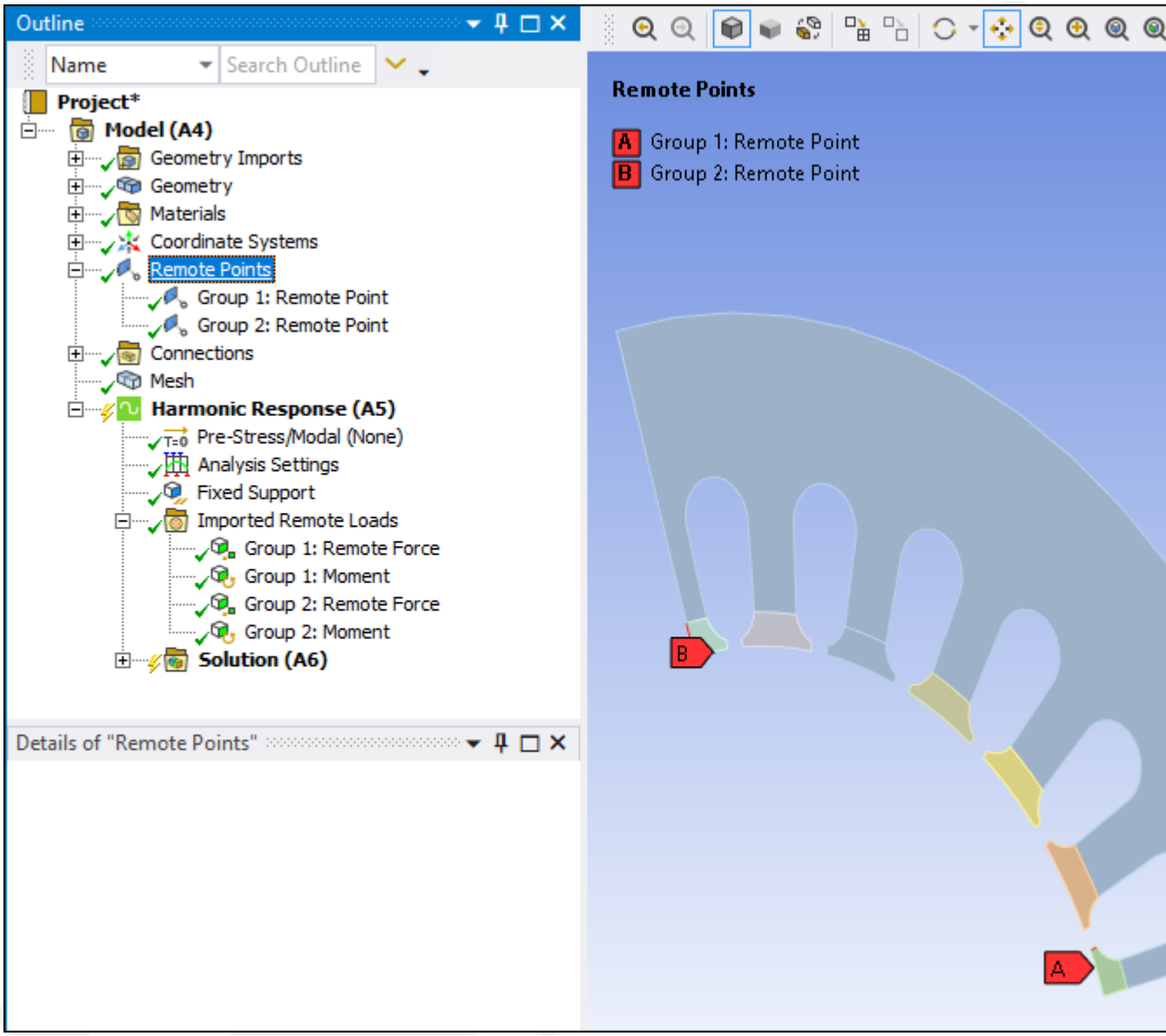
- Creates a **Remote Point** at that location and associates it with the group of two loads. The **Remote Point** is named with the same Group ID number as the load group, as illustrated below.

Note:

When using internal remote points, if you change the scoping or behavior of a load, all loads of the group automatically update because they share the same remote point.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



For each scoped reference (face or edge), applied using the **Scoping Method** property on the **Imported Remote Load** object, Mechanical finds the closest **Remote Point** and assigns the reference as scoping for that Remote Point.

Maxwell Parametric Setup

If your Maxwell analysis contains a Parametric Setup, remotes forces are created for each RPM and grouped into folders. In addition, RPM-varying Step Controls (p. 1426) (Analysis Settings) are automatically defined.

Note:

For a Mode Superposition Harmonic Response analysis, the application applies Remote Forces and Moments using **Tabular Data**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Specify mesh controls, boundary conditions, and solution settings.

4. Solve the analysis.

Importing Surface Force Density

The following procedure assumes that you have properly defined your model in Maxwell and that the source and target systems are connected on the Workbench Project Schematic. Given that, follow these steps to import data and associate it with parts of the geometry.

1. Double-click the **Model** cell of your harmonic analysis system to open the Mechanical application.

2. Open the **Imported Load** object to display the **Surface Force Density** object(s).

   If your Maxwell solution contains a Parametric Setup, you can use the context (right-click) menu option **Create Surface Force Densities and Sync Analysis Settings** to automatically create RPM varying analysis settings properties (Number of RPMs, RPM Value).

   Note:

   During a Mode Superposition Harmonic Response Analysis

   - For improved solution processing, set the **Load Application Type** property to either **Program Controlled** (default) or **Nodal Force**. The application uses the **Nodal Force** setting when the property is set to **Program Controlled**.
   - If you select **Element Pressure** optioin for **Load Application Type** property:
     - It is recommended that you expand the results from the Modal solution to improve performance.
     - If the **Surface Force Densities** loads are the only loads generating load vectors, the application issues the **MODCONT,ON,,,ON** command in the Modal restart phase to reduce the solution time.

3. Select the **Surface Force Density** object and change the default settings as necessary. The options are described below.

Category	Properties/Descriptions
Scope	**Scoping Method**: Options include Geometry Selection and Named Selection. Based on your selection, you need to specify the desired geometry or Named Selection using the associated property.
Definition	**Type**: This read-only property indicates the imported load type - **Surface Force Density**. **Tabular Loading**: This property provides the following options. • **Program Controlled** (default) • **Ramped**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Descriptions
	• **Stepped** • **Off** **Suppressed**: You use this property to include (**No**) or exclude (**Yes**) the load in your analysis. **Load Application Type**: This property is available for Mode Superposition Harmonic Response analyses Options for this property include: • **Program Controlled** (default) • **Element Pressure** • **Nodal Force** (used when **Programmed Controlled** is selected)
Graphics Controls	**Complex Component**: The options for this property include **Imaginary** (default) and **Real**. **Component**: This property provides the following options. • **All** • **Total** • **X Component** (default) • **Y Component** • **Z Component**
Transfer Definition	**Ansoft Solution** **Solution**: This property provides a drop-down menu of available upstream systems. **Variation** (Maxwell Parametric Setup only): This property provides a drop-down menu that includes RPM values for the selected **Solution**. **Data Type**: This is a read-only property. **AC Force Density** is the only supported loading type. **Ansoft Surface(s)**: Select the Ansoft surface(s) for the **Surface Force Density** load. The default setting is **All Surfaces**.

4. You can specify when the imported data should be applied and also modify the imported data, either by adding an offset or by using a scale factor. Change any of the columns in the **Data View** tab as needed:

   • **Source Frequency**: Select from the drop-down list one of the frequencies supplied from the transfer file. The load values associated with this frequency will be imported.

   • **Analysis Frequency**: Select the frequency at which the load will be applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Scale**: The amount by which the imported load values are scaled before applying them.
- **Offset**: An offset that is added to the imported load values before applying them.

Note:

- You can define multiple rows in the **Data View** tab to import additional data from the selected Ansoft solution and apply the load at different analysis frequencies. If multiple rows are defined in the **Data View** tab, you can display imported values at different frequency steps by changing the **Active Row** option in the Details pane.
- You need to re-solve the analysis if you make changes to the **Data View** options.

5. Solve the analysis.

Exporting Results from Thermal or Structural Analyses

If you have solved an analysis containing loads imported from Maxwell or HFSS, you can choose to export temperature or deformation results and apply them during the subsequent solve of the upstream analysis, if this option was previously set in the upstream analysis.

- Temperature results can be exported back to HFSS or Maxwell from a thermal analysis
- Deformation, Stress, and Strain results can be exported to Maxwell from a structural analysis.

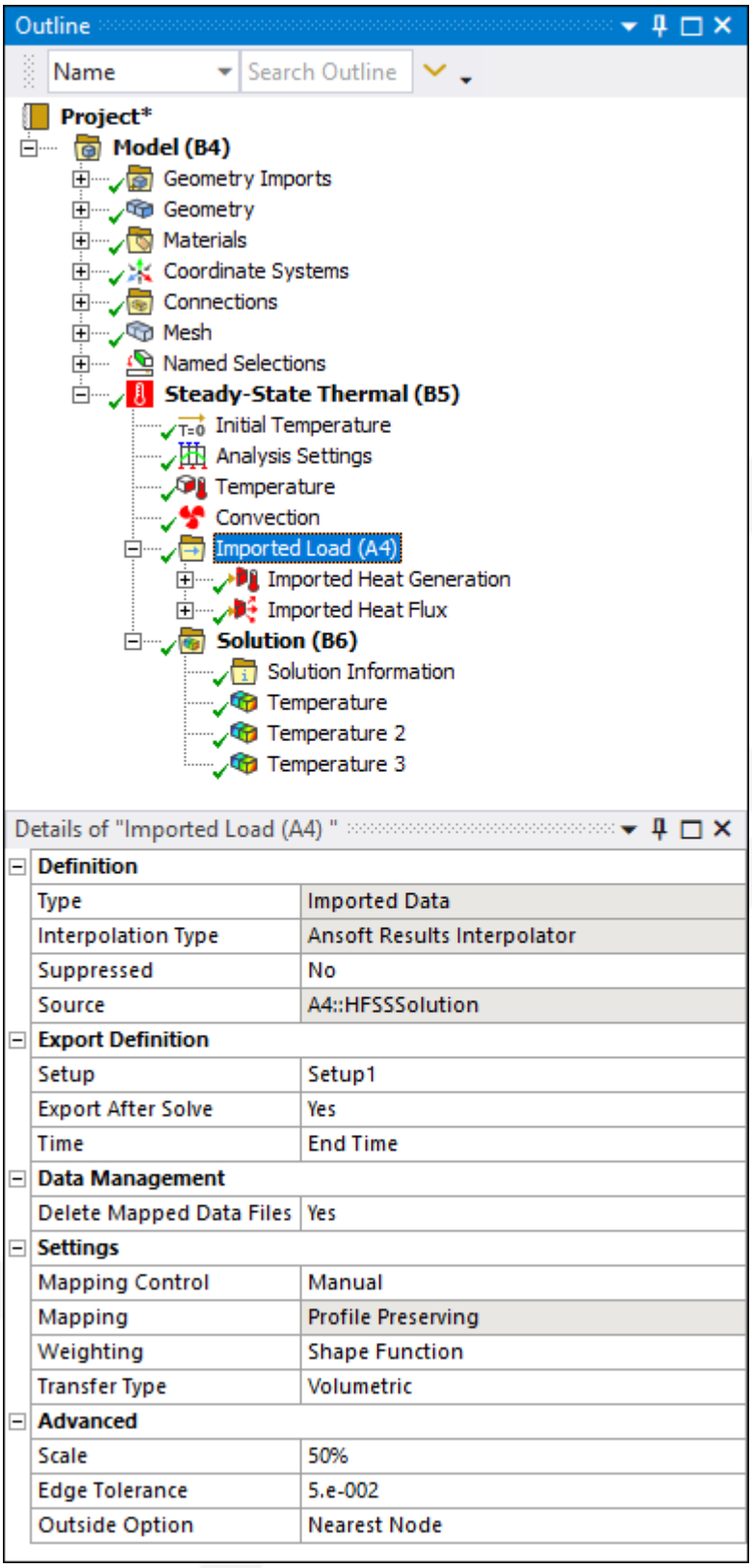
Note:

Mechanical only supports exporting results when the geometry is three-dimensional (3D) in both Mechanical as well as in Maxwell/HFSS.

Select the **Imported Load Group** object in the tree to view the **Details** pane. If the export option is set, you will see the **Export Definition** category. The **Setup** field enables you to specify the Ansoft Setup for which the exported results will be written. The **All** option for the **Setup** property exports results to all the setups requesting feedback.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



In the Details pane, you can also set:

- The **Export After Solve** property. Using this property, you can export the results automatically at the end of the analysis. Option include **Yes** and **No**. Setting the property to **Yes** writes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the results when the solution has finished. Setting the option to **No** exports the file after the solution.

- The **Time** property. This property specifies the analysis time at which results are exported. The default setting is **End Time**, which you select by entering 0. You must enter a value between 0 and the end time of the analysis.
- Properties of the Settings category. See the Data Transfer Mesh Mapping (p. 2389) section for more information about these properties.
- The **Delete Mapped Data Files** property of the **Data Management** category. This property enables you to delete the files generated by the application during the mapping process. Options include **Yes** and **No** (default).

Note:

- Refer to the Ansoft application documentation for more details on settings required to support the export from the Mechanical application to the Ansoft application.
- Results can only be exported to setups that have contributed to the current solution.
- This feature does not support the mapping of stress and strain results from shell bodies. Shell bodies and their corresponding results are not included in during the mapping process.

Electric Machines NVH Analyses (with Waterfall Diagram)

When you analyze electric machines and the effects of electromagnetic forces, noise and vibration is a primary consideration.

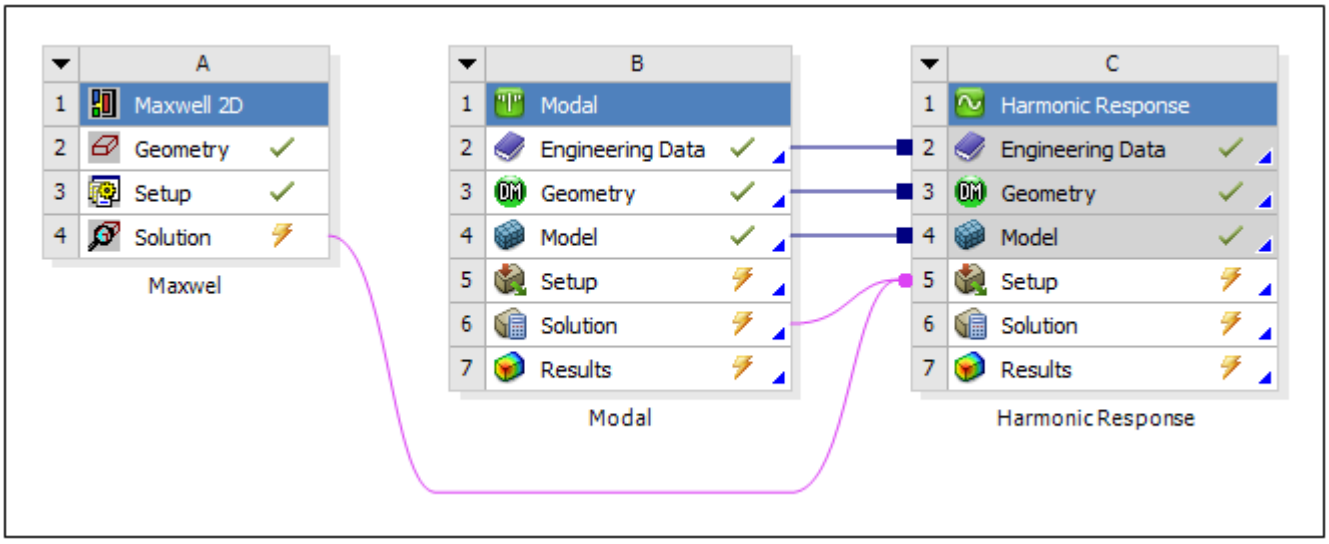
Ansys Workbench enables you to evaluate Equivalent Radiated Power (ERP) Waterfall diagrams. ERP diagrams provide efficient results that you can use to analyze structure vibration for a range of rotating conditions and frequencies.

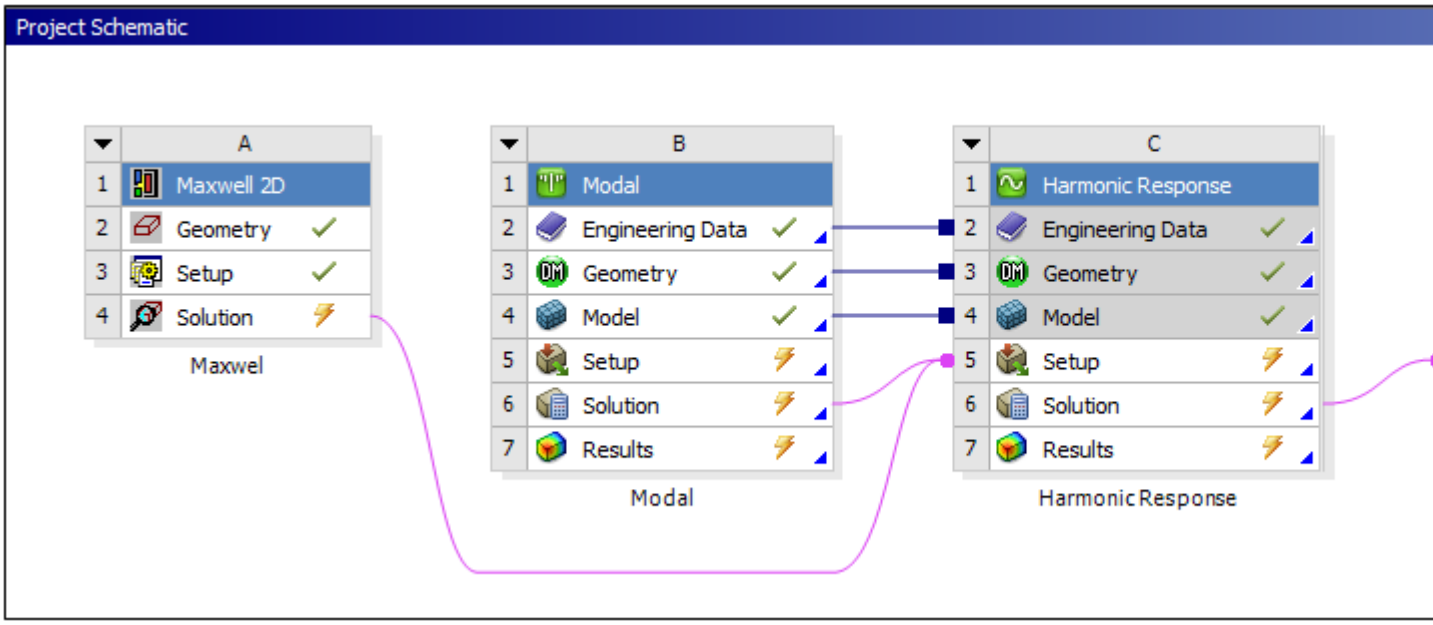
For a more in-depth acoustic analysis, you can generate waterfall diagrams for Far-field Sound Power Level and Far-field Sound Pressure Level Microphone results. See the Adding an Acoustics Analysis (p. 732) topic at the end of the section for the steps to incorporate this analysis type.

This section examines the workflows to simulate electromagnetic forces for rotational velocities as well as the steps to transfer the data to a downstream Ansys Mechanical (frequency domain) Harmonic Response and Harmonic Acoustics systems. Example system configurations are show below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



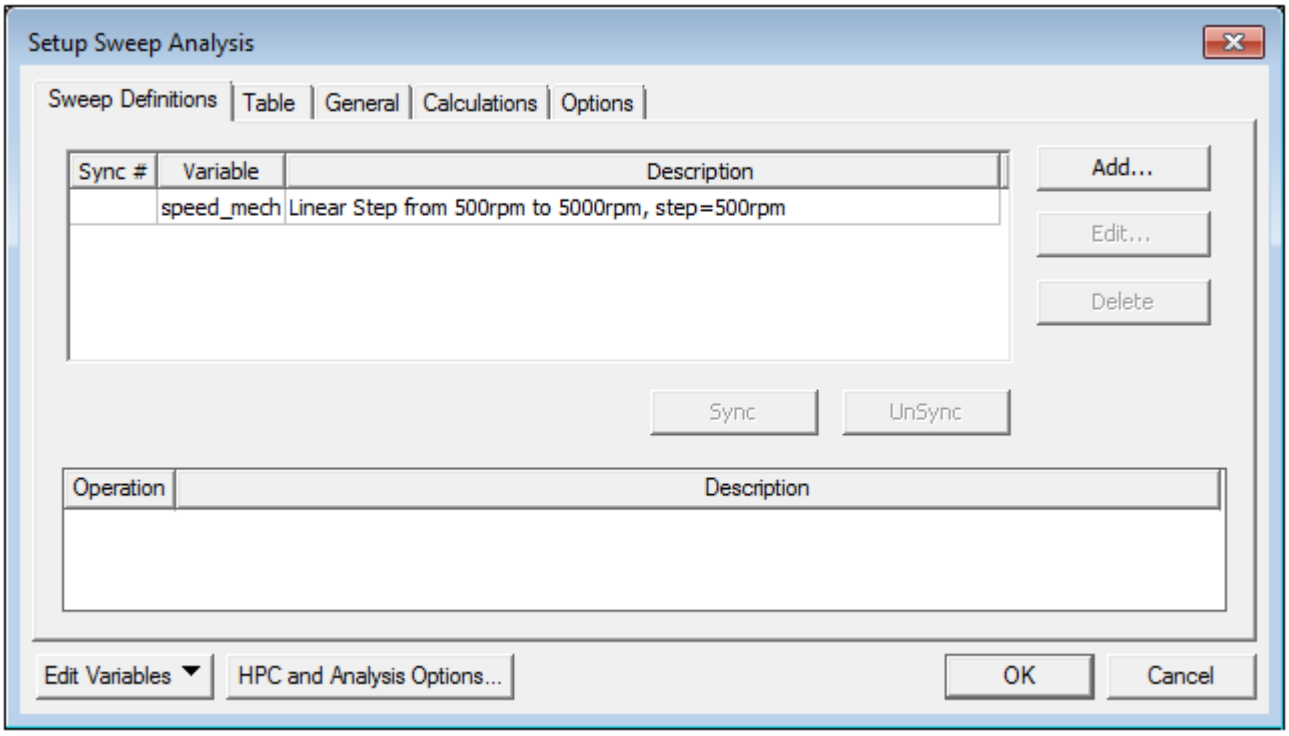


Also note that you can use the operating conditions from your Harmonic Response analysis for Acoustics analyses. This can include all or selected (more critical) operating conditions.

Maxwell Parametric Study

You can use a Maxwell analysis (2D or 3D) that includes several rotational velocities to analyze the overall operating conditions of a Mechanical system. You can use Maxwell to calculate the electro-magnetic forces for each of the RPM (Rotations Per Minute). As depicted in the following dialog, you can create a DesignXplorer setup in Maxwell using the rotational velocity as a variable for the Sweep Definitions. See the Maxwell Help for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



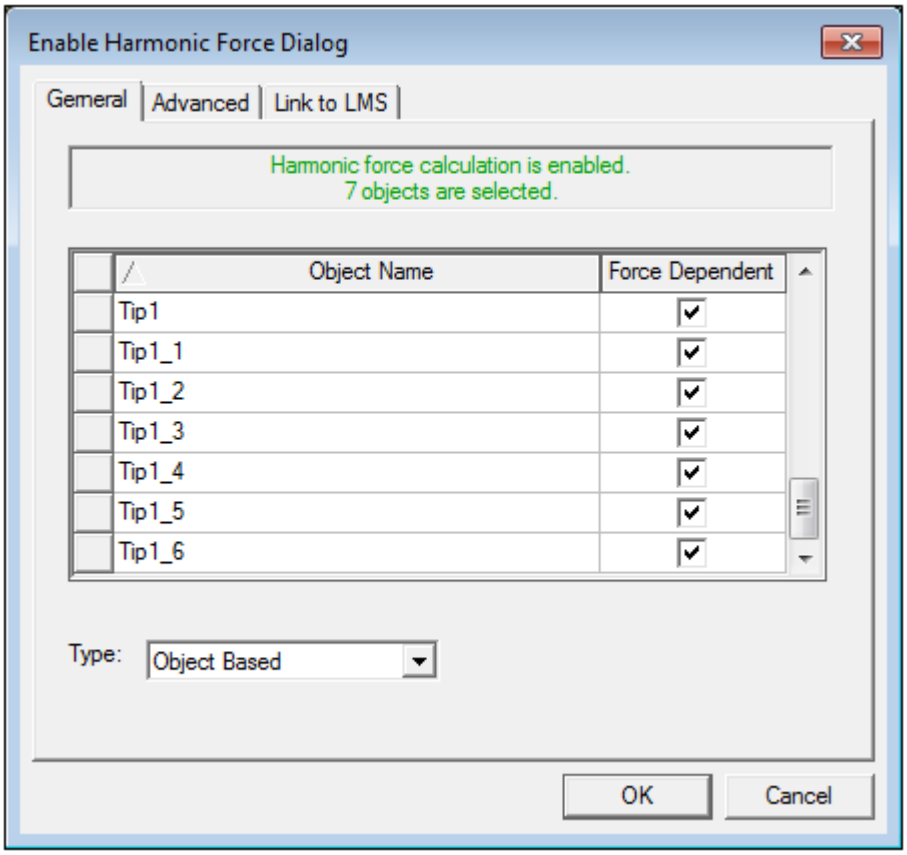
Import Loading

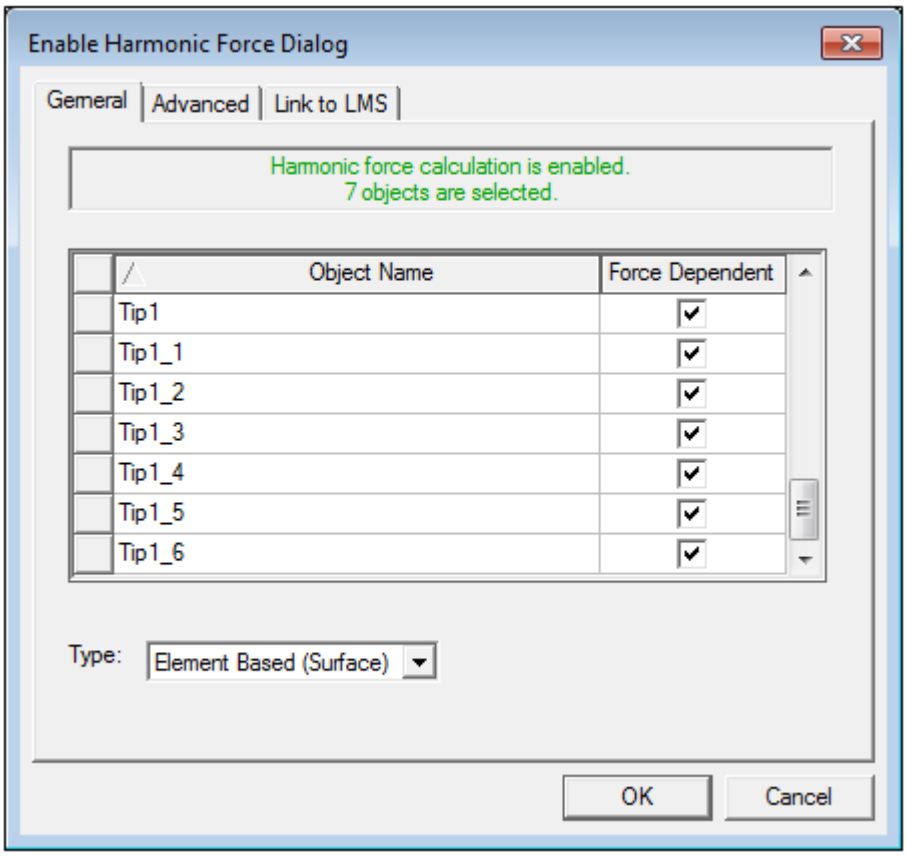
Based on the option **Type** (Object Based or Mesh Based) selected in **Enable Harmonic Force Calculation** dialog in Maxwell, you send either integrated force per object or spatially varying data are to Mechanical:

- **Object Based**: This type calculates remote forces (p. 1552) and moments (p. 1574) on each selected object and imports frequency varying Remote Forces and Moments (p. 711) to Mechanical.
- **Mesh Based (Surface)**: This type calculates forces on mesh entities for the selected objects, converted to frequency domain, and imports spatially varying Surface Force Density (p. 716) loads to Mechanical.
- **Mesh Based (Volume)**: It type calculates forces on mesh entities for the selected objects, converted to frequency domain, and imports spatially varying Body Force Density (p. 1739) loads to Mechanical.

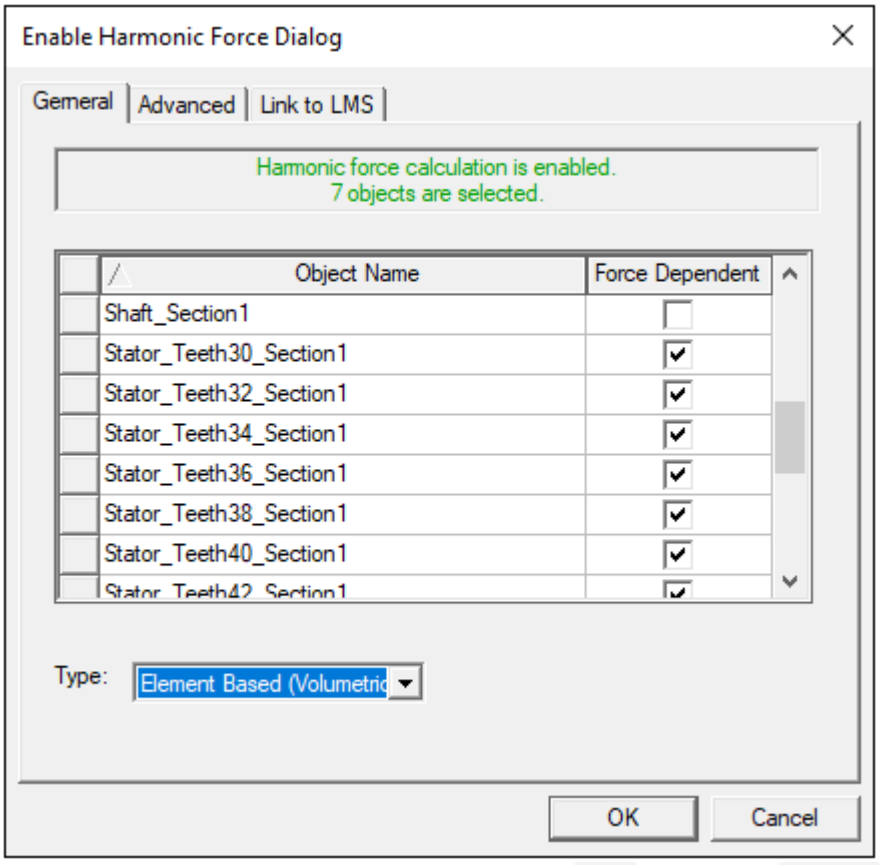
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

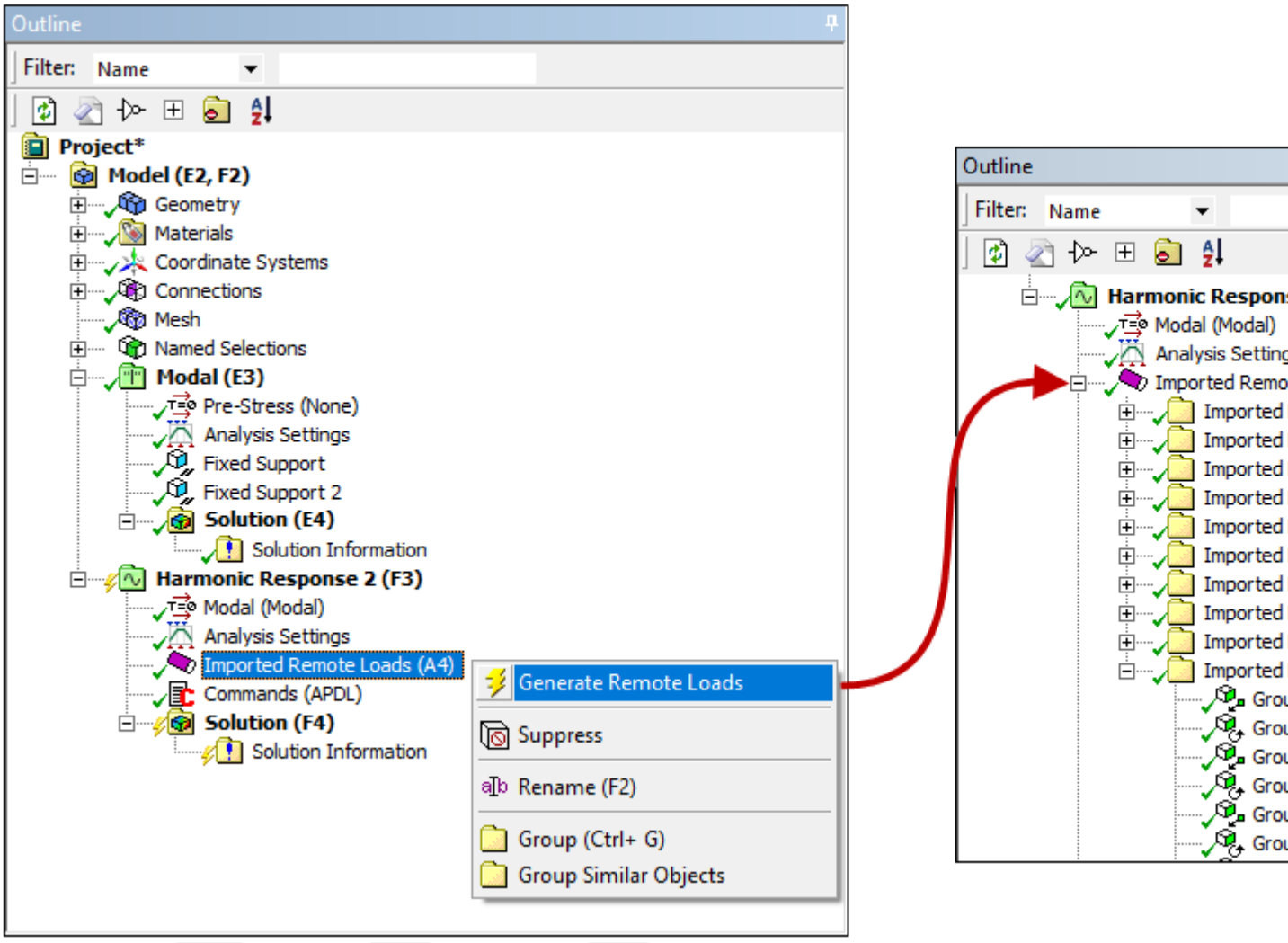


Object Based - Remote Force and Moment

For the **Object Based** option when you import your Maxwell analysis into Mechanical, the application automatically creates an **Imported Remote Loads** object. The context (right-click) menu for this object provides the option **Generate Remote Loads**. When you execute this option, the application automatically imports the remote forces (p. 1552) and moments (p. 1574) from Maxwell and maps the loads to the geometric entities on the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



For each RPM variation solved in Maxwell, Mechanical creates a folder containing the forces associated with the rotational velocity. Here are the Details properties for a selected (imported) Remote Force (p. 1552). You can see the load is specified as a varying load (**RPM Varying** property) and it includes a load value (**RPM Selection** property).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

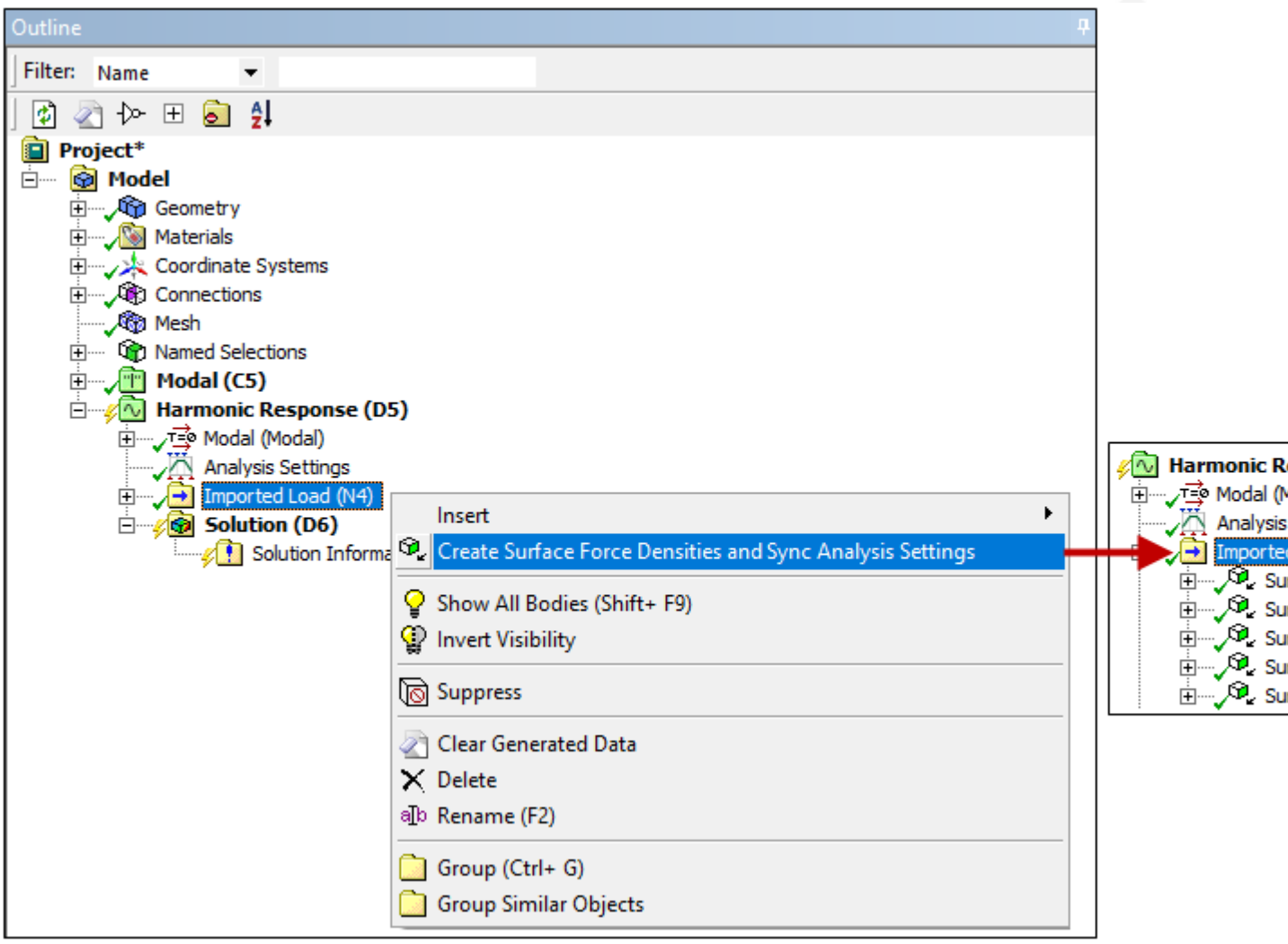
Details of "Group 1: Remote Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Coordinate System	Global Coordinate System
X Coordinate	2.6437e-003 m
Y Coordinate	8.0756e-002 m
Z Coordinate	0. m
Location	Click to Change
Definition	
ID (Beta)	8067
Type	Remote Force
Define By	Components : Real – Imaginary
X Component - Real	Tabular Data
X Component - Imag	Tabular Data
Y Component - Real	Tabular Data
Y Component - Imag	Tabular Data
Z Component - Real	Tabular Data
Z Component - Imag	Tabular Data
Suppressed	No
Behavior	Deformable
Step Controls	
RPM Varying	Yes
RPM Selection	500. RPM
Advanced	
Pinball Region	All
Loading Application	Table

Mesh Based - Surface Force Density

For the **Element Based (Surface)** option when you import your Maxwell analysis into Mechanical, the application automatically creates an **Imported Load** object. The context (right-click) menu for this object provides the option **Create Surface Force Densities and Sync Analysis Settings**. When you execute this option, the application automatically imports the **Surface Force Density** (p. 716) loads from Maxwell and maps the loads to the geometric entities on the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The application creates a **Surface Force Density** object for each RPM variation created in Maxwell.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Surface Force Density"	
Scope	
Scoping Method	Named Selection
Named Selection	AllTeethFace
Definition	
Type	Surface Force Density
Tabular Loading	Program Controlled
Suppressed	No
Source Frequency	All
Graphics Controls	
By	Active Row
Active Row	1
Complex Component	Real
Component	All
Transfer Definition	
Solution	Setup1 : Transient
Variation	1500 RPM
Data Type	AC Force Density

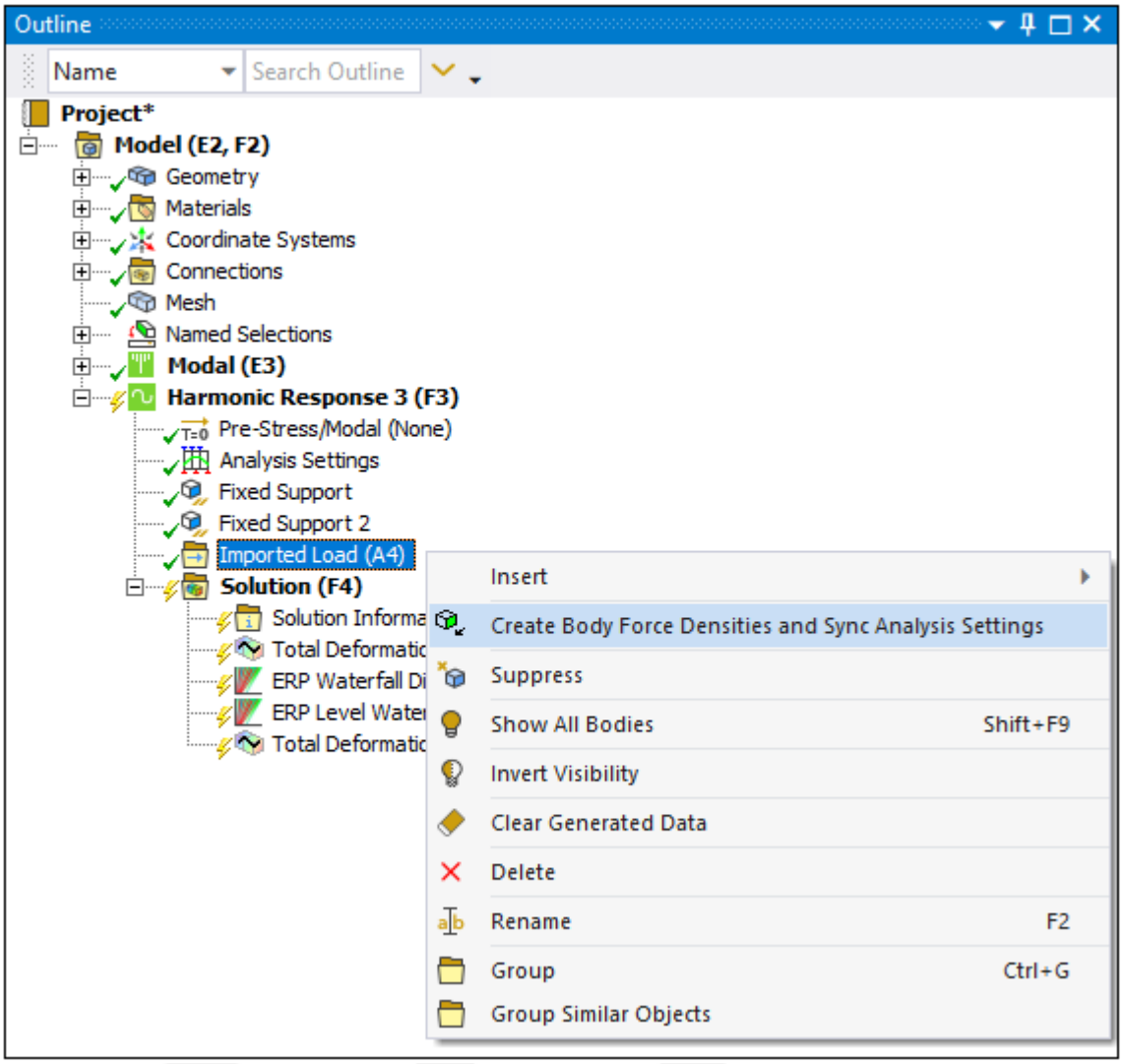
Mesh Based - Body Force Density

When you import your Maxwell analysis into Mechanical, the application automatically creates an **Imported Load** object. The context (right-click) menu for this object provides the option **Create Body Force Densities and Sync Analysis Settings**. When you execute this option, the application automatically imports the **Body Force Density** loads from Maxwell and maps the loads to the geometric entities on the model.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The application creates a **Body Force Density** object for each RPM variation created in Maxwell.

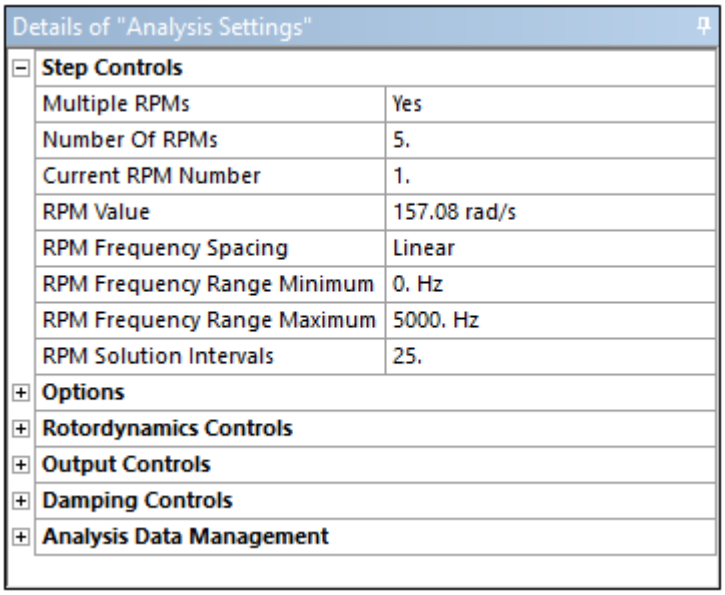
Details of "Body Force Density"

Scope	
Scoping Method	Geometry Selection
Geometry	4 Bodies
Definition	
Type	Body Force Density
Tabular Loading	Off
Suppressed	No
Source Frequency	All
Graphics Controls	
Complex Component	Real
Component	All
Transfer Definition	
Solution	Setup1 : Transient
Variation	500. RPM
Data Type	AC Force Density

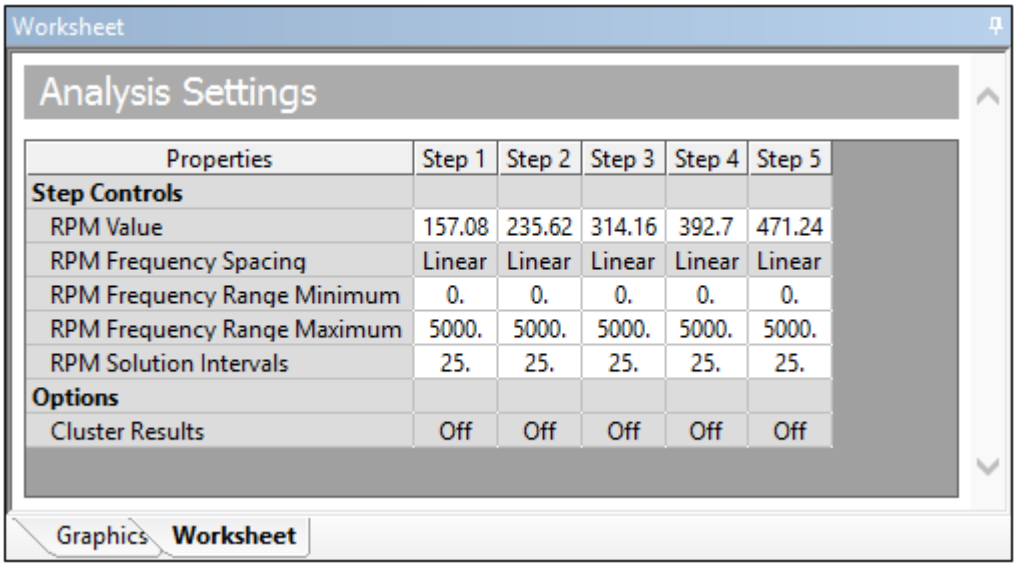
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Define Analysis Settings

When your Maxwell load data includes multiple RPM conditions, the application automatically reflects the RPM conditions as load steps in the **Step Controls** (p. 1426) category of the **Analysis Settings** of your downstream, Harmonic Response analysis. You can modify these RPMs (individually) using the **Step Controls** category.



You can also modify the RPM **Step Controls** using the **Worksheet**.



Harmonic Solution Method

The application supports the Maxwell import scenarios both the **Full** and **Mode Superposition** Solution Method in your Harmonic Response analysis. Each of these methods supports constant loading conditions that you may apply (on each step) in addition to the imported RPM varying loads, that the application applies on a per step basis. For example, you want to add loads in addition

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to the imported electromagnetic forces. You may wish to add loading conditions to consider other noise sources, such as those generated by bearings or aerodynamic forces.

For the **Mode Superposition** method, by default, the application defines the remote forces (p. 1552) using the **Table** option for the **Loading Application** property to improve solution performance.

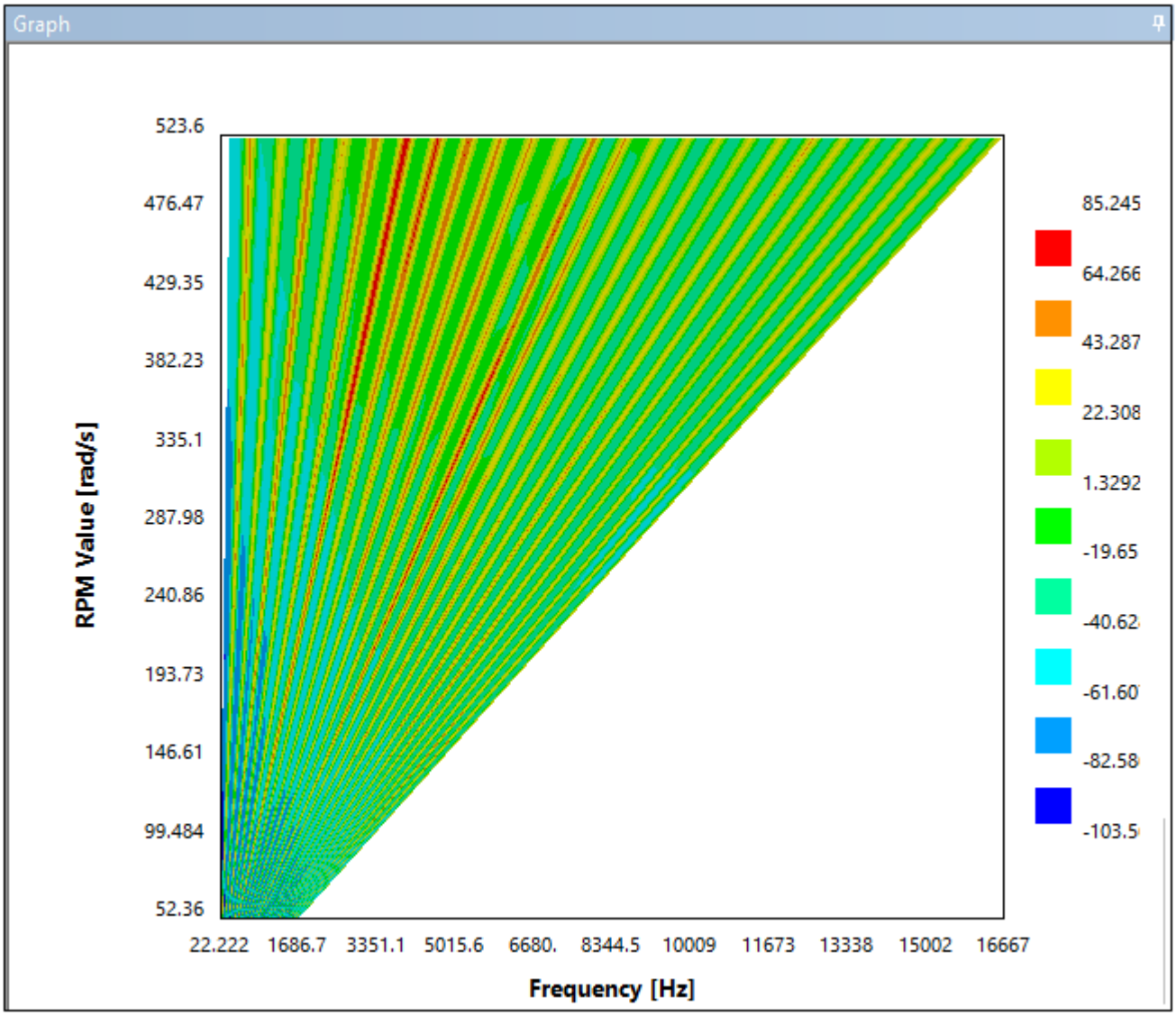
Details of "Group 1: Remote Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Coordinate System	Global Coordinate System
X Coordinate	2.6437e-003 m
Y Coordinate	8.0756e-002 m
Z Coordinate	0. m
Location	Click to Change
Definition	
ID (Beta)	8067
Type	Remote Force
Define By	Components : Real – Imaginary
X Component - Real	Tabular Data
X Component - Imag	Tabular Data
Y Component - Real	Tabular Data
Y Component - Imag	Tabular Data
Z Component - Real	Tabular Data
Z Component - Imag	Tabular Data
Suppressed	No
Behavior	Deformable
Step Controls	
RPM Varying	Yes
RPM Selection	500. RPM
Advanced	
Pinball Region	All
Loading Application	Table

Also note that when you import **Surface Force Densities**, you can improve solution performance by initiating the **Expand Results From** (p. 1477) property (**Output Controls** (p. 1473)) by setting either the **Stress**, **Strain**, or **Calculate Reactions** properties to Yes.

ERP Waterfall Diagram

The Equivalent Radiated Power (ERP) (p. 2047) waterfall diagram results provide a global acoustic signature based on the selected radiating faces. The application displays this result in two forms: (1) using polylines for each RPM case or (2) on a 2D color map, as illustrated here. This result enables you to detect the most critical operational conditions (RPM and Frequency) analyzed with additional detail in a downstream Harmonic Acoustics analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

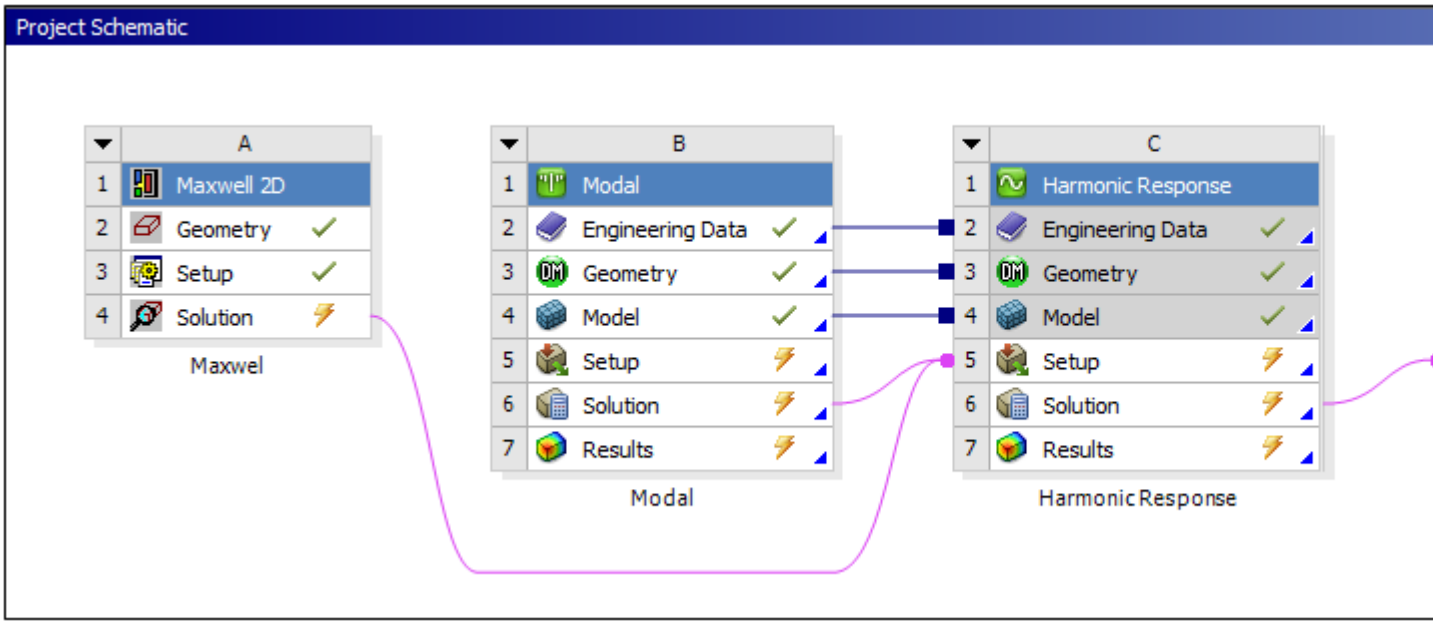


Adding an Acoustics Analysis

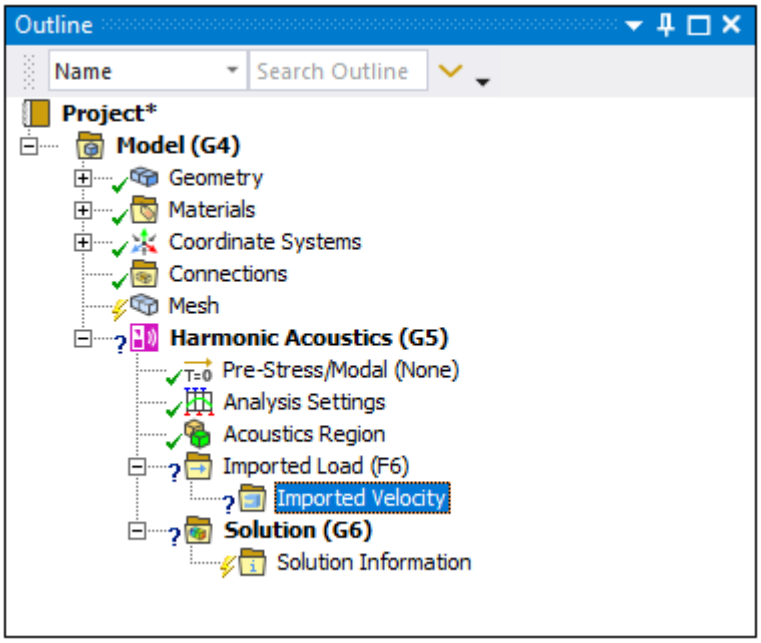
You can examine the acoustics of your structural vibrations by adding a Harmonic Acoustics analysis to your NVH simulation. As illustrated, you insert a Harmonic Acoustics into the project as a downstream system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

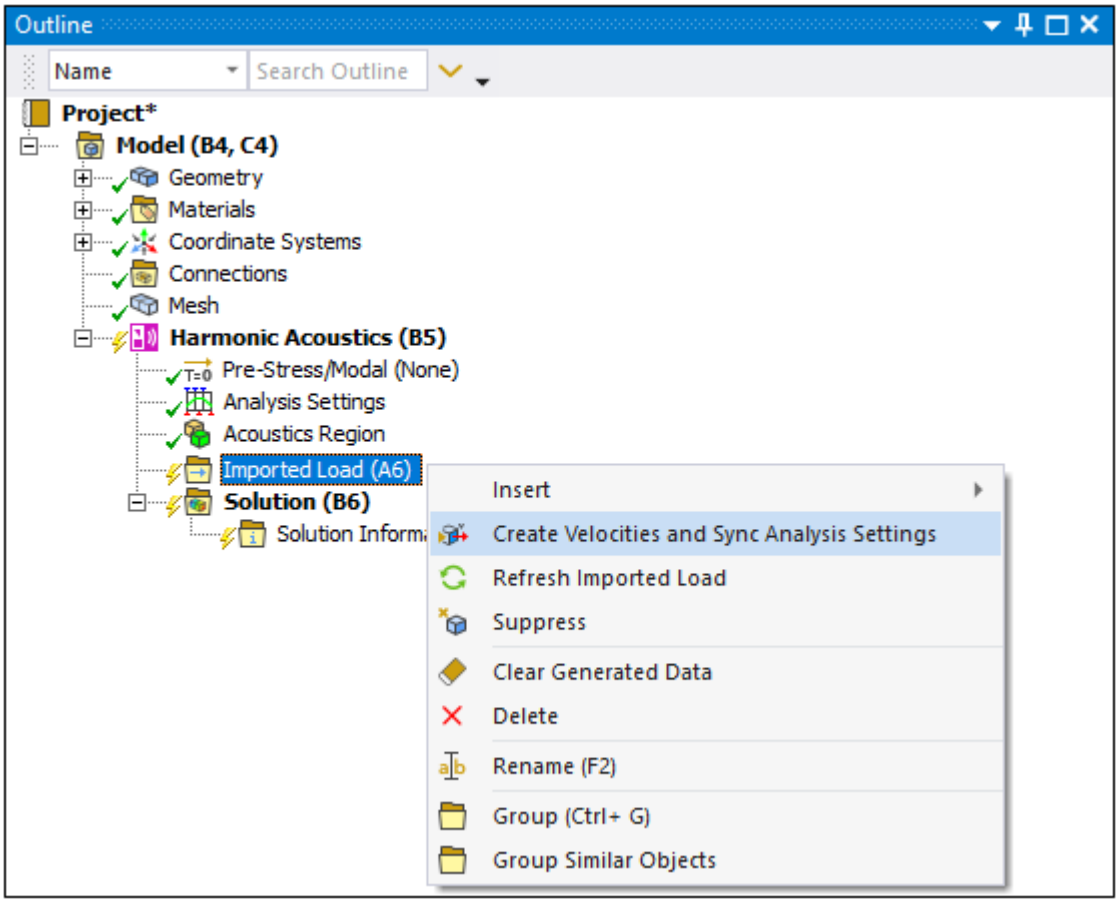


Once opened in Mechanical, the application automatically creates an **Imported Load** object. By default, the application inserts a **Imported Velocity** object.

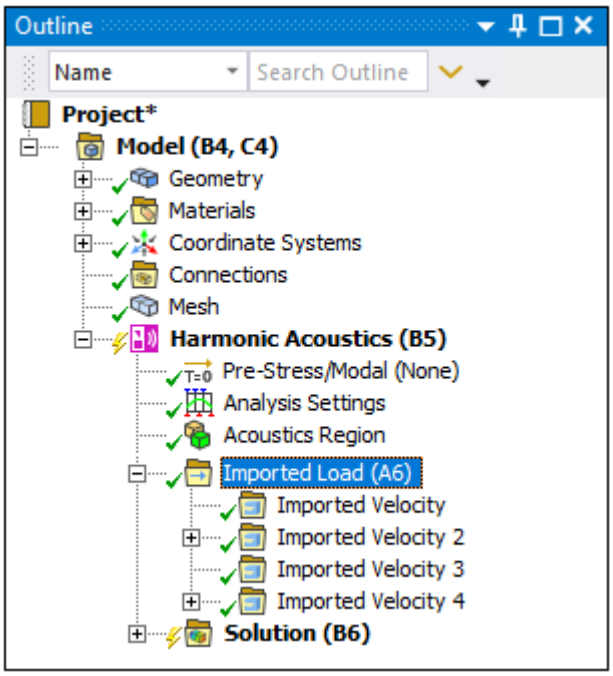


You can add **Imported Velocity** objects by right-clicking on the **Imported Load** folder and selecting **Insert** > **Velocity** or you can select the option **Create Velocities and Sync Analysis Settings** to import and map all of the velocity loads available in the upstream system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The application creates an **Imported Velocity** object for each RPM imported from the upstream Harmonic Response analysis.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details of "Imported Velocity"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Imported Velocity
Tabular Loading	Program Controlled
Suppressed	No
Source Bodies	All
RPM Selection	20.94 rad/s
Source Frequency	All
Graphics Controls	
Settings	
Rigid Transformation	
Legend Controls	
Named Selection Creation	

When you import multiple RPMs, the application automatically reflects the RPM conditions as load steps in the **Step Controls** category (that automatically displays in the **Worksheet**) of the **Analysis Settings** of your downstream Harmonic Acoustics analysis. This behavior is similar to the **Step Controls** of a Harmonic Response system. See the Step Controls for Harmonic Analysis Types (p. 1426) section for more information.

Worksheet

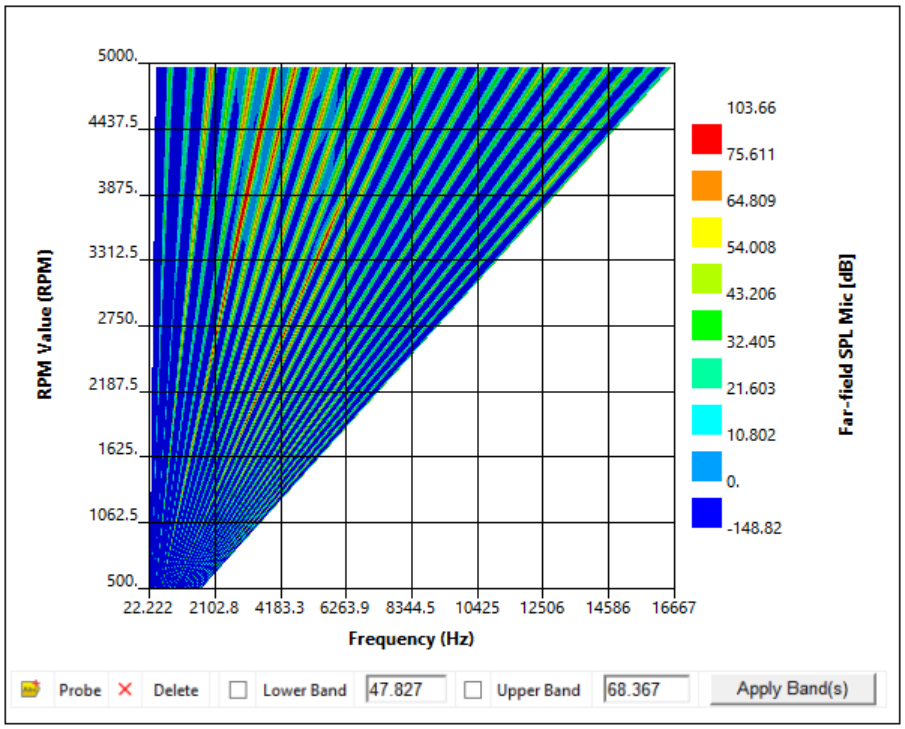
Analysis Settings

Properties	Step 1	Step 2	Step 3	Step 4
Step Controls				
RPM Value	20.944	41.888	62.832	83.776
RPM Frequency Spacing	Linear	Linear	Linear	Linear
RPM Frequency Range Minimum	0.	0.	0.	0.
RPM Frequency Range Maximum	15000	25000	35000	45000
RPM Solution Intervals	3.	3.	3.	3.

Result Diagrams

Once you complete your analysis, the result options Far-field Sound Power Level Waterfall Diagram and Far-field SPL Mic Waterfall Diagram enable you to analyze the acoustic behavior for the entire range of RPMs and frequencies so that you can detect the most critical situations. An example is illustrated below. See the Acoustic Waterfall Diagrams for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



External Data

Use **External Data** to import data from one or more text files and apply it in a Mechanical analysis. Supported analysis types include:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395): Supports Imported Body Temperature (p. 1740) (Thermal Condition) and Imported Temperature (p. 1756).
- Explicit Dynamics: Currently, the Explicit Dynamics system and the LS-DYNA ACT extension provide limited support for the Workbench **External Data** system.
- Modal (p. 515): Supports Imported Body Temperature (p. 1740).
- Static Structural (p. 652)/Transient Structural (p. 665): These analysis types do not support the use of the Rigid Dynamics solver for External Data import.
- Steady-State Thermal (p. 657)/Transient Thermal (p. 680)/Thermal-Electric (p. 661).

Note:

The surface pressures and line loads calculated in a Hydrodynamic Diffraction analysis or a time domain Hydrodynamic Response analysis can be transferred to panel and beam elements in a Static Structural analysis using the Hydrodynamic Pressure Mapping ACT extension, through a link on the Workbench Project Schematic page. See The Hydrodynamic Pressure Add-on for more information.

Go to a subsection topic based on how you want to use **External Data**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Importing Data from Workbench
Importing Data from Mechanical
Importing Thickness
Importing Loads
Exporting Data

Importing Data from Workbench

To import a supported file using the Workbench External Data system:

1. Insert an **External Data** system into the **Project Schematic** from the **Component Systems** group of the Toolbox.
2. Right-click the **Setup** cell and select **Edit**. This automatically opens the **External Data** tab
3. Browse to the location of the file to add.

   Note:

   Importing files from design point folders within the same project directory (with the exception of the Current design point) is not supported.

4. In the **Location** column of the **Outline of Schematic** pane, select the ellipsis button ( ), and select your data source files. The selected file name, location, and identifier are automatically displayed in the **Data Source** column. The identifier is a string that can be used to identify the file in the downstream Mechanical application.

   - When multiple files are added to the same system, each file is given a unique identifier (that is, File1, File2, and so on). These identifiers are used in conjunction with the data identifiers (Pressure1, Thickness1, and so on) to identify and apply the dataset(s) within Mechanical.
   - If your files contain data for the same nodal coordinates, or if only one of your files contains the nodal information, you can choose the **primary** option in the **External Data** system to designate a primary file. This option notifies the mapping utility that the group of files, defined in the **External Data** system, share the same nodal information. The nodal information is therefore processed and stored only from the primary file. This greatly reduces the memory usage by only allocating space for the nodes once, not once per file. It can also result in much faster import times as only one mapping operation will be required.
   - Mechanical APDL `.cdb` files can be added as a primary mesh in the **External Data** system; for details, see Importing a CDB File as Input in the *Workbench User's Guide*.

5. Return to the **Project** tab. Display the properties for the **Setup** cell of the **External Data** system by right-clicking on the cell and selecting **Properties**.
6. Specify the **Files Import Type** property as needed. Options include **By Copy** (default) and **By Reference**. The **By Reference** option instructs the application to directly reference the External

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Data file instead of transferring the file to the project folder. For large size files, this action can drastically reduce disk storage requirements and improve performance.

Note:

When you specify the **By Reference** option and wish to archive your project using the option **Imported files external to project directory** of the **Archive Options** dialog, the application automatically copies the External Data files to the archived project.

Important:

The **By Reference** option cannot be used with the **System Coupling** feature.

7. Specify additional Workbench properties as needed:

   - To transfer shell thickness data to Mechanical, either:

     – Drag the **Setup** cell of the **External Data** and drop it onto the **Model** cell of a Mechanical system to create the link.

     – Right-click the **Setup** cell of the **External Data** system and select **Transfer Data to New** > **[Mechanical_system]**. This creates a link to the **Model** cell of a new **Static Structural** system *and* a connection to the **Setup** cell. Remove the unwanted connection to the **Setup** cell.

   - To transfer load data to Mechanical, drag the **Setup** cell of the **External Data** system and drop it onto the **Setup** cell of an applicable Mechanical system.

     Alternatively, you can right-click the **Setup** cell of the **External Data** system and select **Transfer Data to New** > **[Mechanical_system]**. This creates a link to the **Setup** cell of a new **Static Structural** system *and* a connection to the **Model** cell. Remove the unwanted connection to the **Model** cell.

   - When an **External Data** system is connected to a system using the Samcef or ABAQUS solver, the following quantities cannot be used: Body Force Density, Stress, Strain, Heat Flux, and the Emag Condition.

   - When an **External Data** system is connected to a system using the ABAQUS solver, the ABAQUS system does not support the import of Forces.

8. Attach the geometry to the analysis system and open Mechanical. Right-click the **Setup** cell of the analysis system and select **Edit**. You can also double-click the **Setup** cell to open the application.

Importing Data from Mechanical

Mechanical enables you to import load data, thickness data, or element orientation coordinate data from external source files directly into the application. Go to a section topic:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Import Options (p. 739)
- External Data Dialog (p. 739)
- Definition and Preview Tabs (p. 742)

Import Options

Imported Loads

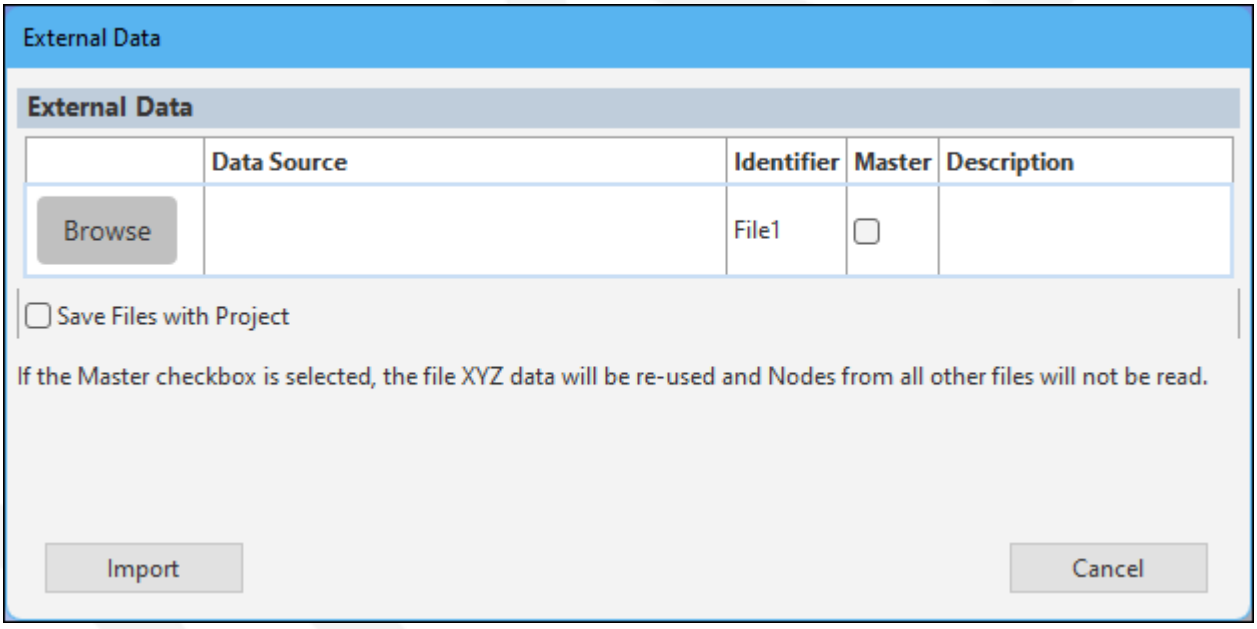
The **Imported Loads** menu of the **Environment** object context ribbon includes an **External Data** option to import load data. This option opens the **External Data** dialog.

Imported Thickness and Element Orientations

The **Imported Thickness** and **Imported Element Orientations** options of the **Geometry** object context ribbon enable you to import element orientation data or shell thickness data from an external file. Both of these options open the **External Data** dialog.

External Data Dialogs

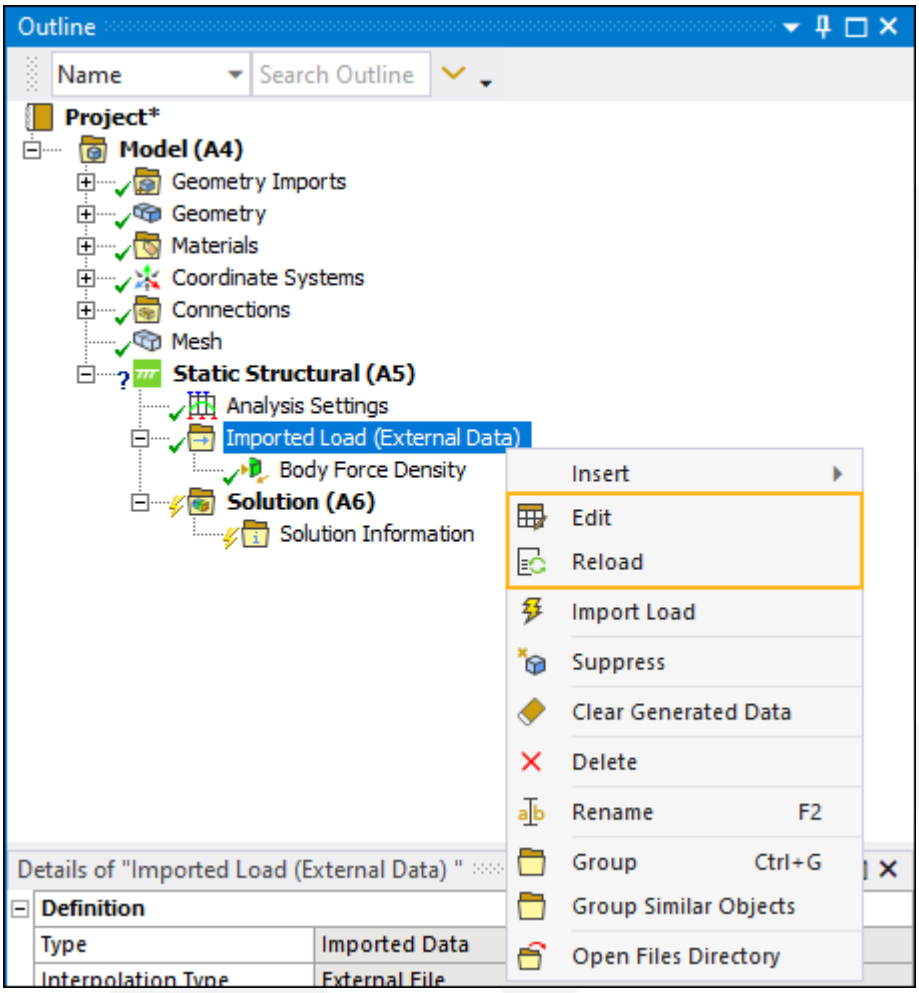
1. From the appropriate object, select the desired option. The application automatically inserts the corresponding object into the tree and the **External Data** dialog displays. You use this dialog to specify the desired external file(s) you wish to import.



Note:

- If you have already selected a file and displayed the dialog, there are two additional context (right-click) menu options: **Edit** and **Reload**. The **Edit** option re-opens the External Data dialog enabling you to modify the imported object's details and the **Reload** option reloads data from the source file(s).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- Although similar to using an **External Data** system in Workbench, there are some differences. You should review the External Data section of the *Workbench User's Guide* as well as the External Data System (p. 736) section in the *Mechanical User's Guide* for a description of the feature and its requirements.

Additional dialog options include:

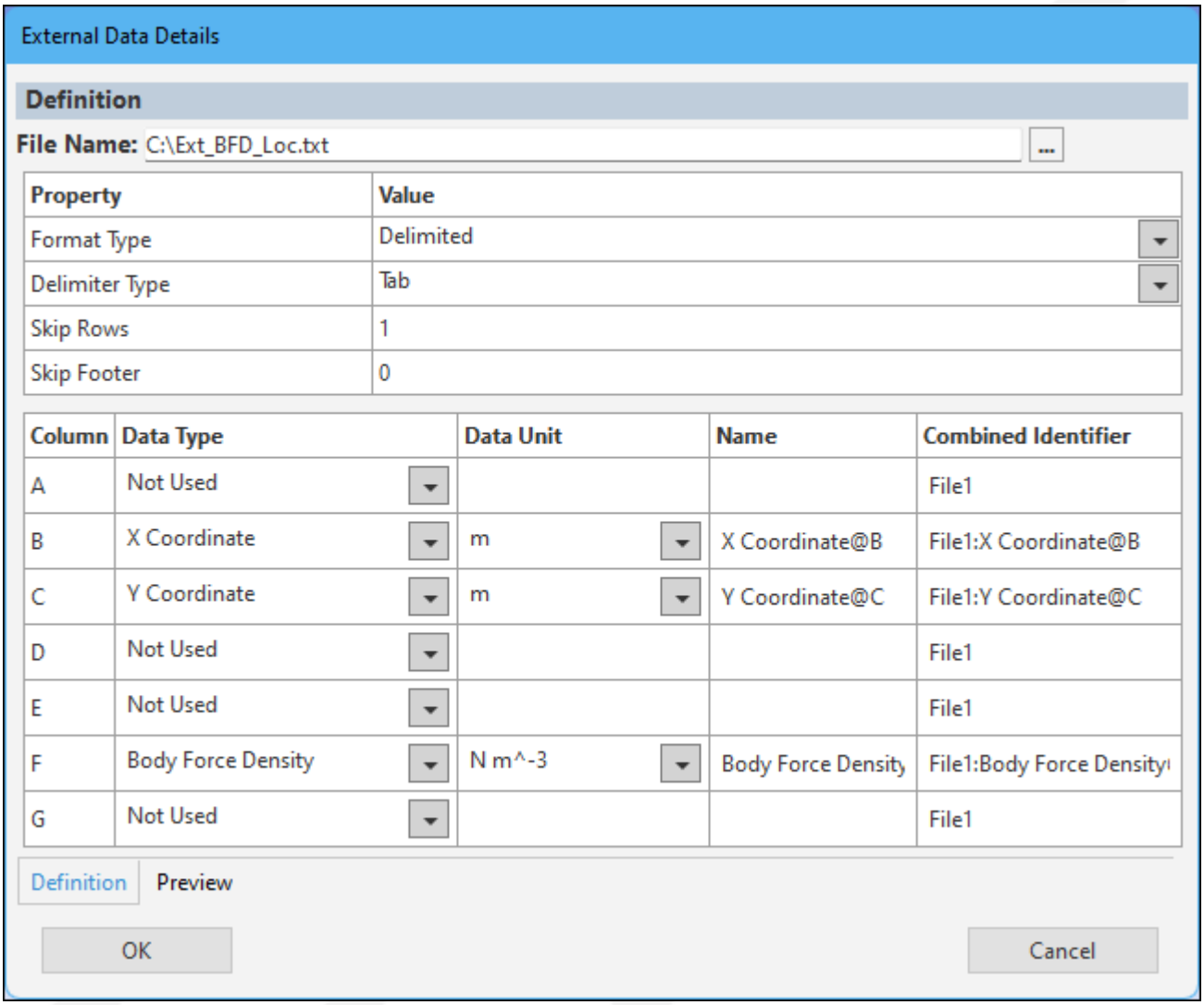
- **Save Files with Project**: Copy the external data files to the project directory. If left unchecked, the application references the source files directly using the File Path without making a copy. The default is unchecked.
- **Identifier**: The identifier is a string that can be used to identify the file in the downstream Mechanical application.
- **Master**: Specify the file as the master external data file. The nodal coordinate data for the selected master file is processed by the mapping utility. The application skips this nodal coordinate data in all other files. Only one file can be set as the master file. Mechanical APDL format type files should always be marked as the master file.

2. Select the **Browse** button. The **External Data Details** dialog displays. It includes two tabs: **Definition** and **Preview**. See the Definition and Preview Tabs topic below for a description of these dialogs and their options.

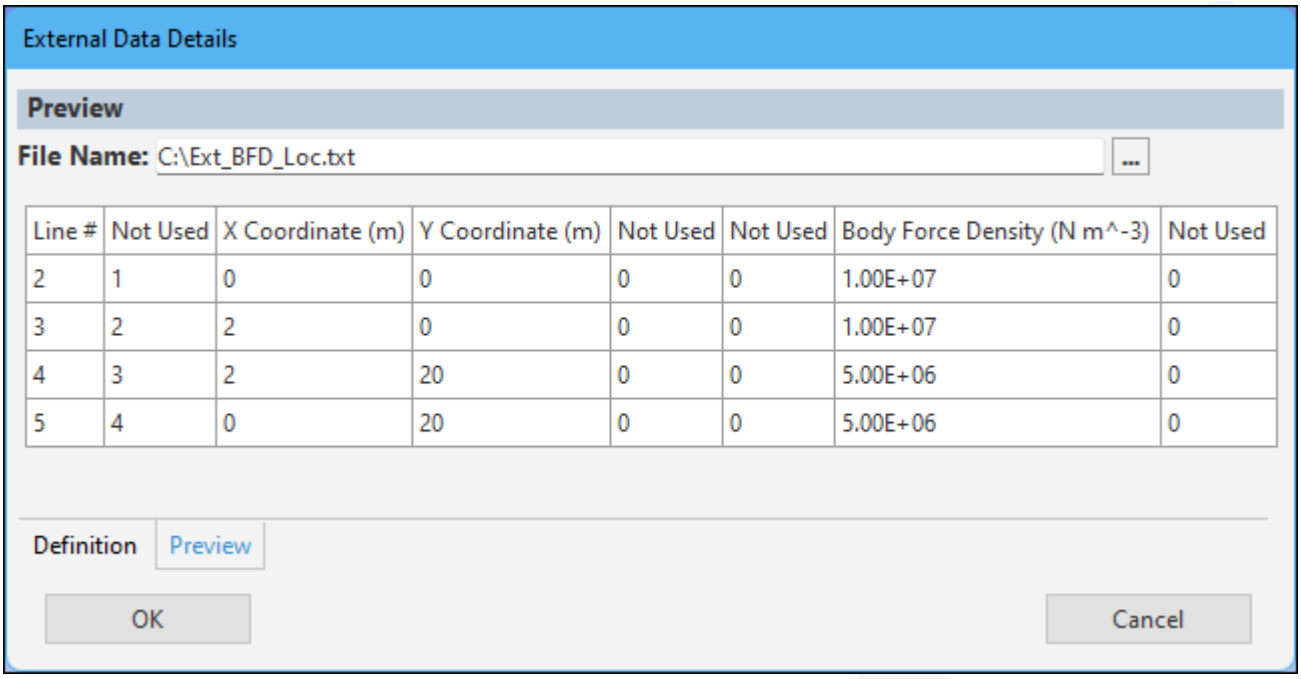
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. Select the **File Name** option to navigate to and select your file. Specify dialog properties as needed and select **OK**.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



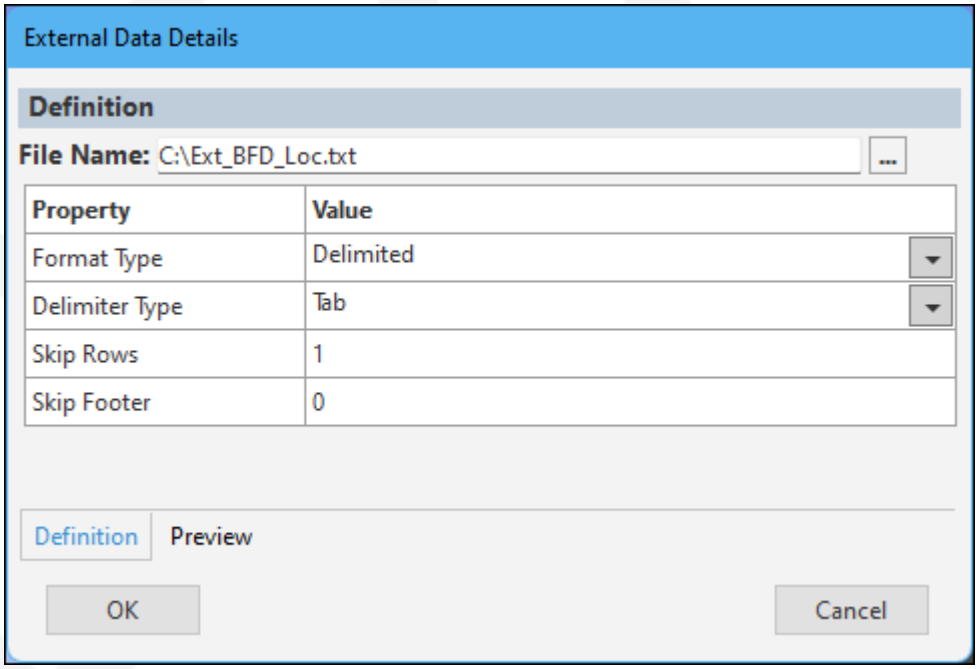
4. Select the **Import** button. The data file is now associated with the object.

Definition and Preview Tabs

Descriptions of the properties of the **Definition** and **Preview** tabs for **External Data Details** dialog are shown below.

Definition Tab

The **Definition** tab specifies the details of the external data file.



Properties for the **Definition** dialog include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	Description/Value
Format Type	This property sets the corresponding format for the selected file. Different format type results in different properties. Options for this property include: • Delimited (default): This format type represents a file containing data with each column delimited by a specific character. When selected, the Delimiter Type property also displays. This property specifies the delimiter used to separate the columns in data file. Options include **Comma**, **Semicolon**, **Space**, **Tab**, and **User Defined Separator**. If you select **User Defined Separator**, the additional property User Defined Separator displays. You use this property to specify any character as the data delimiter. • **Fixed Width**[a]: This format type represents a file that contains data in columns that have a fixed width. Based on the width, the column specifies a string using Fortran-like fixed-format specifications. This is equivalent to the User-Defined option of the Format Type property as described in the External Data section of the Workbench User's Guide. Review the description in that section for the supported format specifiers, such as Decimal floating point, Scientific notation, etc. • **MAPDL**: This format type represents a Mechanical APDL CDB data file. When selected, the following additional properties display: – **Dimension**: Specifies the source data dimension. Options include **2D** and **3D** (default). – **Unit Lengths**: Specifies the length unit used for source point locations. The default value is Meters. **Note:** If you have multiple external data files that include the MAPDL import format, the property **Average Corner Nodes To Midside Nodes** displays under the **Delimited** or **Fixed Width** file definition. If you set this property to **Yes**, the application calculates data at the midside nodes as an average of the values contained at the corner nodes.
Skip Rows	Specifies the number of rows to skip at the start of the data source. The default is **0**.
Skip Footer	Specifies the number of rows to skip at the end of the data source. The default is **0**.

[a] **Fixed Width Example**

In the following example, the line has one integer followed by four exponential data types:

385 8.333333333E-003 9.375000000E-003 3.375000000E-001 0.000000000E+000

The corresponding format is 1i3,4e17.9.

Where:

– The first entry is the number of occurrences.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

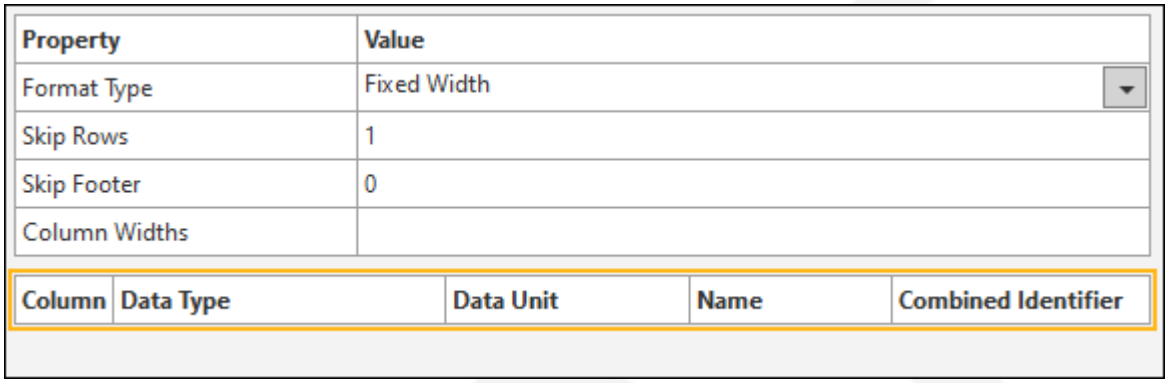
– The second entry is the format specifier.

– The third entry is the number of characters of data in the definition (including numeric values (0-9), the letter "e" (for instances of scientific notation), and any white space and + or - signs).

– The fourth entry is the number of digits after the decimal point.

Column Definition

When you select the **Delimited** or **Fixed Width** format types, the following table displays in the dialog.



Column: Index field.

Data Type: Select a desired data type from the drop-down menu.

Name: String used to identify a specific column.

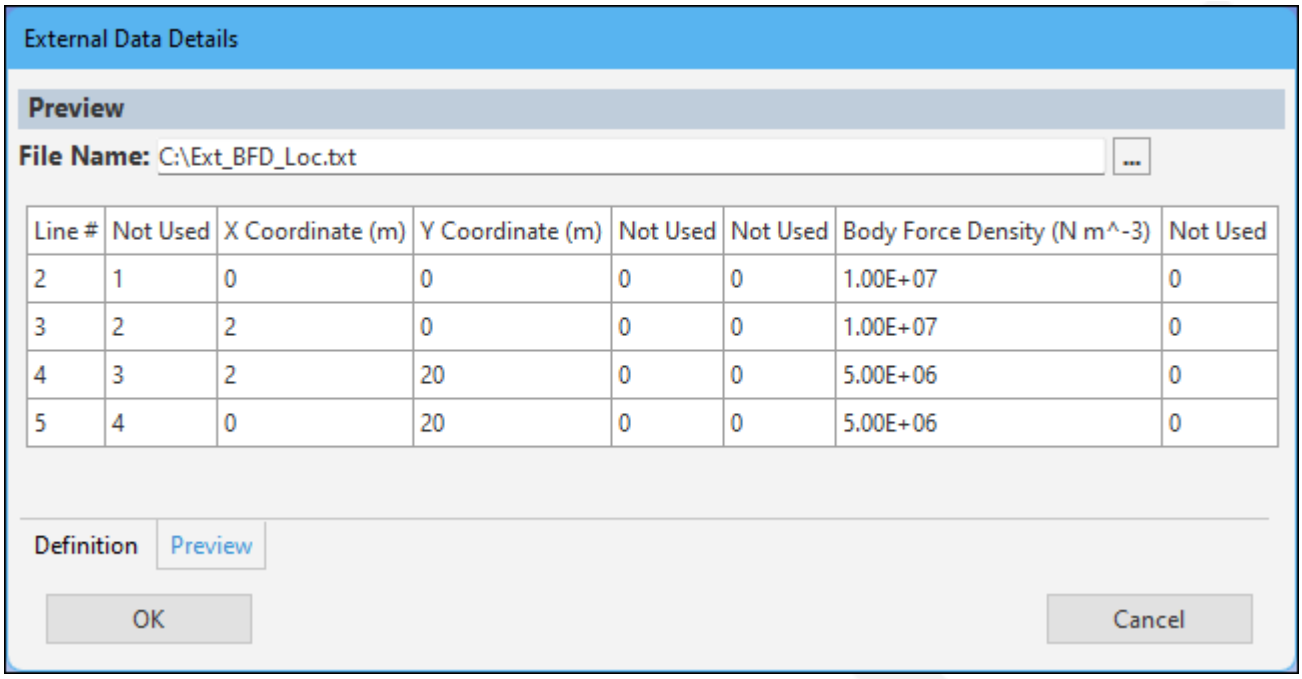
Combined Identifier: String that combines the **File Identifier** and **Column** name, it can be used to identify the column in the child imported load object.

Preview Tab

The **Preview** tab displays a preview of the imported lines from the file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

- The application only displays the first 10 rows of data for preview.
- The **Line #** column refers to the line number in the source data file.

Importing Thickness

If your simulation has a shell thickness defined from **External Data**, an **Imported Thickness** folder is automatically added under the **Geometry** folder.

Application

To specify imported thickness:

1. Select appropriate geometry or mesh entity.
2. Specify properties as needed. You can modify the mapping settings to achieve the desired mapping accuracy.

   You can specify a thickness value for the unmapped target nodes using the **Unmapped Data Value** property. By default, a zero thickness value is assigned to the unmapped nodes.

Important:

For the Mechanical APDL solver, the thickness value at each node must be greater than zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Right-click the **Imported Thickness**, and then click **Import Thickness** to import the thickness. When the thickness has been imported successfully, a contour plot will be displayed in the **Geometry** window and any mesh display will be based upon the mapped thickness of the elements.

Importing Loads

If your simulation has load data defined from **External Data**, an **Imported Load** folder is added under the **Environment** folder.

1. To add an imported load, select the **Imported Load** folder and select a desired load from the **Imported Loads** drop-down menu on the **Environment** Context tab or right-click the **Imported Load** folder and select the appropriate load from the **Insert** context menu.

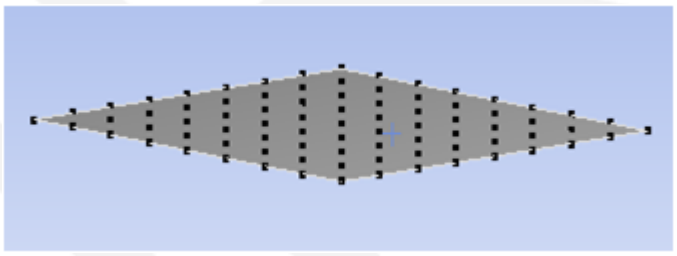
2. Select appropriate geometry or mesh entity.

3. In a 3D structural analysis, if the Imported Body Temperature load is scoped to one or more surface bodies, the **Shell Face** option in the Details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.

4. When mapping data to surface bodies, you can control the effective offset and thickness value at each target node, and consequently the location used during mapping, by using the **Shell Thickness Factor** property. By default, the thickness value at each target node is ignored when data is mapped.

   You can choose to enter a positive or negative value for the **Shell Thickness Factor**. This value is multiplied by each target node's physical thickness and is used along with the node's offset to determine the top and bottom location of each target node. A positive value for the **Shell Thickness Factor** uses the top location of each node during mapping, while a negative value uses the bottom location of each node. For example:

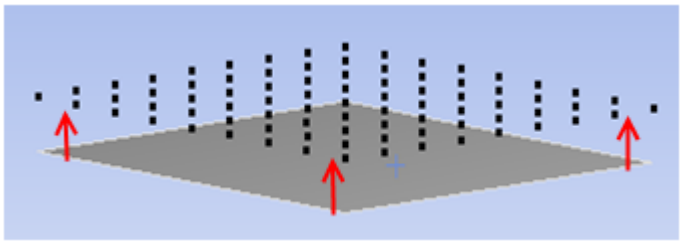
   - A value of 0.0 means that the physical thickness and offset of the surface body nodes will be ignored; all target nodes are mapped at default surface body locations.

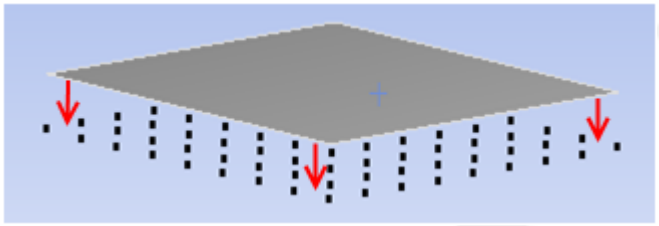
   - A value of 1.0 means that the thickness used for a target node will be equal to the physical thickness value specified for that node. The top location of the node will be used during the mapping process. The image below depicts a change to the position of the nodes, but this is not a reflection of what you will see in the application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

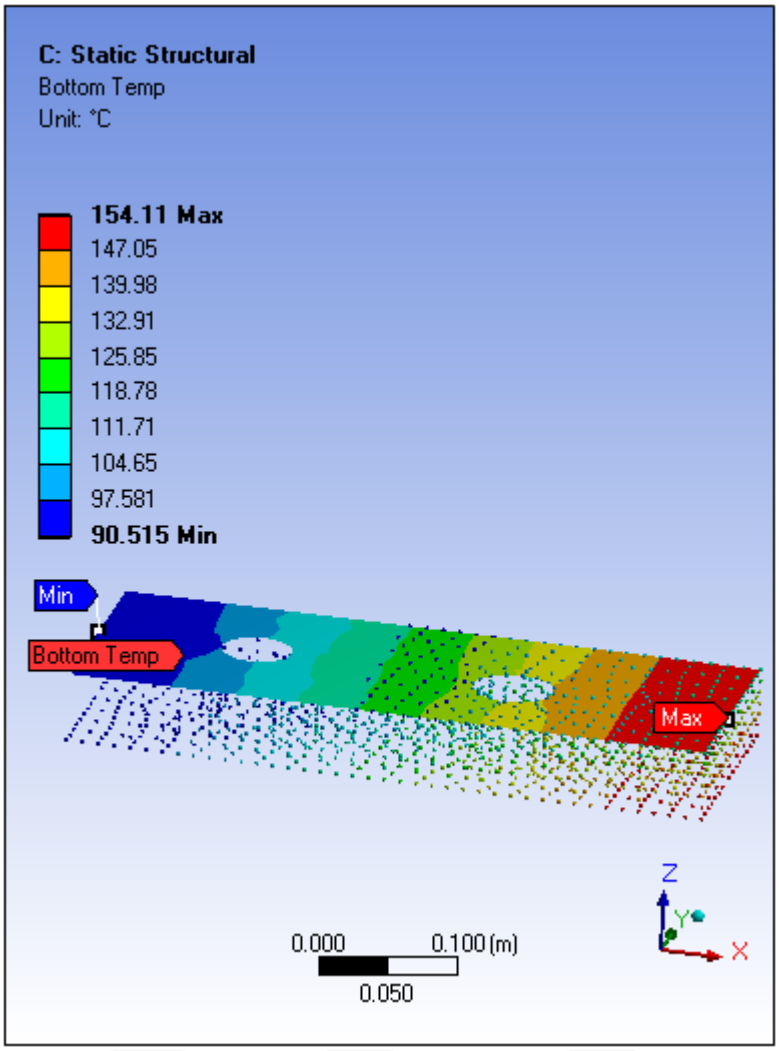


- A value of -2.0 means that the thickness used for a target node will be equal to twice the physical thickness value specified for that node. The bottom location of the node will be used during the mapping process. The image below depicts a change to the position of the nodes, but this is not a reflection of what you will see in the application.



The view will look similar to the following for a value of –1.0. The colored dots represent the location and corresponding values of the source nodes. In this case, each target node will be projected using its physical thickness value to its bottom location and then mapped.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



5. Select appropriate options in the **Details** view. You can modify the Settings (p. 2389) category properties to achieve the desired mapping accuracy. And, mapping can be validated by using mapping Validation (p. 2412) for an object.

   - For pressure loads, you can apply the load in the direction normal to the face or by specifying a direction. Setting **Define By** to **Components** enables you to define the direction by specifying the x, y, and z magnitude components of the load. The z component is not applicable for 2-D analyses. For pressure loads in Harmonic Response, you can apply both real and imaginary components of the loads.

   - In a 3D analysis, if the **Triangulation** mapping algorithm is used, the **Transfer Type** mapping option defaults to **Surface** when an Imported Temperature or Imported Body Temperature load scoping is only on shell bodies. If the scoping is on shell bodies and other geometry types, the **Transfer Type** mapping option will default to **Volumetric**. In such cases, to obtain a more accurate mapping, you should create a separate imported load for geometry selections on shell bodies, and use the **Surface** option for **Transfer Type**. See **Transfer Type** under Mapping Settings (p. 2389) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For Imported Pressure loads, you can apply the load onto centroids or corner nodes using the **Applied to** property in the **Details** view. See Imported Pressure (p. 1752) for additional information.
- For imported force loads, both conservative and profile preserving algorithms are available using the **Mapping** property. See Imported Force (p. 1747) and Mapping Settings (p. 2389) for additional information.

6. For each load step, if an **Imported Displacement** and other support/displacement constraints are applied on common geometry selections, you can choose to override the specified constraints by using the **Override Constraints** option in the details of the Imported Displacement object. By default, the specified constraints are respected and imported displacements are applied only to the free degrees of freedom of a node.

7. For **Vector**[1]and **Tensor**[2]loads, the **Coordinate System** property can be used to associate the component identifiers, defined in the worksheet, to a particular coordinate system. This option is useful when the source data is defined, or needs to be defined, with respect to a coordinate system that is not aligned with the Global coordinate system. If a cylindrical coordinate system is chosen, the data is interpreted to be in the radial, tangential, and axial directions. By default, the Source coordinate system is used.

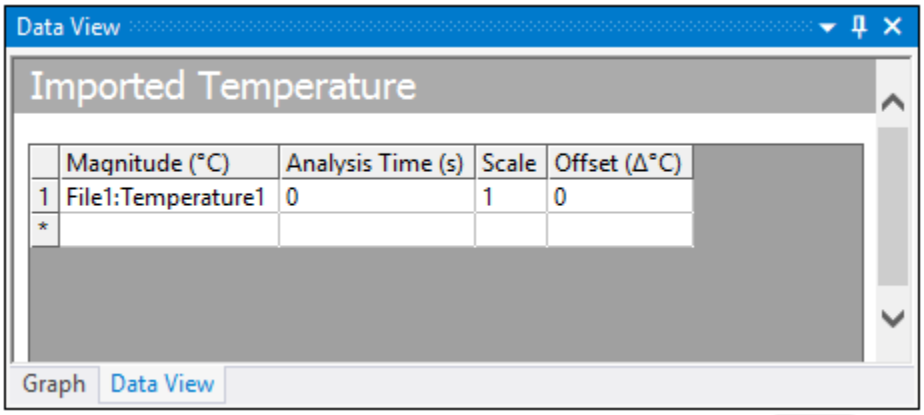
   Note:

   The **Source Coordinate System** option of the **Coordinate System** property is an internal coordinate system that is not visible in the Outline. This coordinate system represents source points transformed first by Rigid Transformation process in the upstream External Data system and then by the Rigid Transformation process in the Mechanical application. If there are no rigid transformations from the upstream **External Data** system or in Mechanical, then the **Source Coordinate System** is the same as the **Global Coordinate System**.

8. Under **Data View**, select the desired **data Identifier**, for the imported load. The data identifier (File Identifier: Data Identifier) strings are specified in **External Data**. You can also change the **Analysis Time/Frequency** and specify **Scale** and **Offset** values for the imported loads.

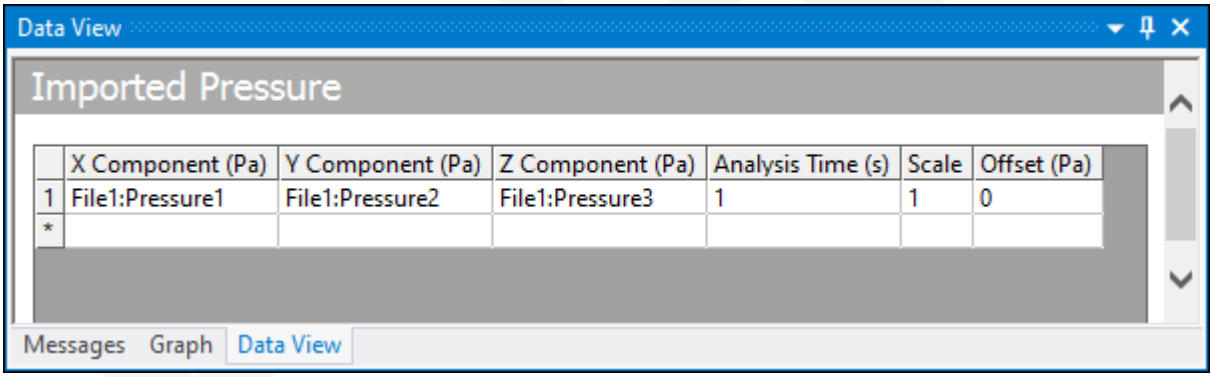
[1] Supported loads include Imported Displacement, Imported Force, Imported Pressure, and Imported Velocity.
[2] Supported loads include Imported Stress and Imported Strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

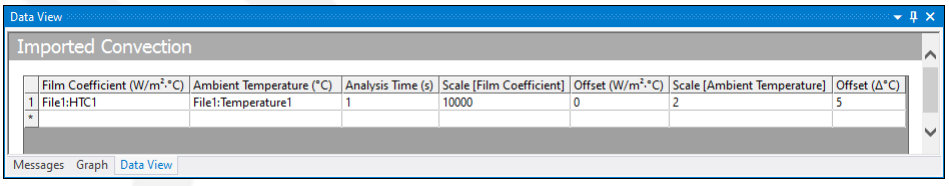


- For **Vector**[1] and **Tensor**[2] loads, if the **Define By** property is set to **Components** you should select data identifiers that represent the x/radial, y/tangential, and z/axial magnitude components of the load. For **Vector**[1] and **Tensor**[2], the components are applied in the **Coordinate System** specified in the **Details** view. The z component is not applicable for 2-D analyses. For **Imported Displacement** load, you can choose to keep a component free, or fixed *(displacement = 0.0)* by selecting the **Free** or **Fixed** option from the list of data identifiers. For all other loads, you can choose to ignore a component if you do not have data for that direction by selecting the **Ignore** identifier from the drop-down list.

  – For **Imported Body Force Density**/**Imported Pressure**/**Imported Velocity** in Harmonic Response analyses, you should select data identifiers for both real and imaginary components. You can also specify **Scale** and **Offset** for both real and imaginary components.



- For **Imported Convections**, you should select data identifiers for film coefficient and ambient temperature. You can also specify **Scale** and **Offset** values for both film coefficient and ambient temperature.



9. Right-click in the **Data View** and select **Add row** to specify additional data for a different analysis time/frequency.

10. Change any of the columns in the **Data View** tab as needed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Magnitude/Film Coefficient/Ambient Temperature**

  Select the appropriate data identifier that represents the load values to be applied from the drop-down list.

- **X Component**

  Select the appropriate data identifier that represents the x component of the load values to be applied from the drop-down list.

- **Y Component**

  Select the appropriate data identifier that represents the y component of the load values to be applied from the drop-down list.

- **Z Component**

  Select the appropriate data identifier that represents the z component of the load values to be applied from the drop-down list.

Note:

If you do not have data for a direction you can choose to ignore that component by selecting **Ignore** from the appropriate drop-down box. Select the **Fixed** option from the drop-down list to make the component constant with a value of zero or the **Free** option for the component to be without any constraints.

If multiple files have been used in External Data, the data identifiers for component-based vector or convection loads must come from the same file or from files that have a primary file association. For example, you can select File1:PressureX, File1:PressureY, and File1:PressureZ, but you cannot select File1:PressureX, File2:PressureY, File3.PressureZ (assuming that File1, File2, and File3 do not have a primary file association).

- **XX**, **YY**, **ZZ**, **XY**, **YZ**, and **ZXComponent**

  Select the appropriate data identifiers to represent the components of the symmetric tensor to be applied from the drop-down list.

- **Analysis Time/Frequency**

  Choose the analysis time at which the load will be applied.

- **Scale**

  The amount by which the imported load values are scaled before applying them.

- **Offset**

  An offset that is added to the imported load values before applying them.

11. In the Outline, right-click the **Imported Load**, and then click **Import Load** to import the load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. When the load has been imported successfully, a contour or vector plot will be displayed in the **Geometry** window.

    - For **Vector**[1] loads, contours plots of the magnitude (**Total**) or X/Y/Z component can be viewed by changing the **Data** option in the details pane. Defaults to a vector plot (**All**).
    - For **Tensor**[2] (p. ?) loads, contours plots of the **Equivalent (von-Mises)** or XX, YY, ZZ, XY, YZ and ZX components can be viewed by changing the **Data** option in the details pane. Defaults to a Vector Principal plot.
    - For **Imported Convections** loads, contours plots of film coefficient or ambient temperature can be viewed by changing the **Data** option in the details pane.
    - For complex load types, such as Pressure/Velocity in Harmonic Response, the real/imaginary component of the data can be viewed by changing the **Complex Data Component** option in the details pane.

    Note:

    - Many imported loads as well as Element Orientations support Vector Display (p. 74) options.
    - The range of data displayed in the graphics window can be controlled using the **Legend controls** options. See Imported Boundary Conditions (p. 1736) for additional information.

13. If multiple rows are defined in the **Data View**, imported values at different time steps can be displayed by changing the **Active Row** option in the details pane.
14. To activate or deactivate the load at a step, highlight the specific step in the **Graph** or **Tabular Data** window, and choose **Activate/Deactivate***at this step!* See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

Important:

- For **Vector**[1] and **Tensor**[2] loads, when the **Define By** property is set to **Components**:
  - Any transformations specified in **External Data** and/or in Mechanical, are applied to the mapped data if the **Coordinate System** property is set to **Source Coordinate System**. If another Coordinate System is specified, then the components are applied in that Coordinate System.
  - Rotations produced by a cylindrical projection coordinate system, for 2D to 3D mapping, are also applied to the mapped data. Rotations resulting from analytical transformations specified in **External Data**, do not get applied to the mapped data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For **Imported Displacements**, selecting the **Free** identifier for a source component will result in the corresponding target component being left unconstrained and free to deform in that direction, whereas **Fixed** identifier results in a value of zero being applied. For other load types, a value of zero is applied on selecting the **Ignore** component.

Exporting Data

You can export results data to one or more text files and use them in External Data in order to apply the data as boundary conditions.

To export result data, right-click a result and select Export (p. 249) > **Export Text File**. Be sure to include the Node Locations which are off by default as described in the External Data section.

For thermal analyses, you can export temperatures and heat flows on any surface with a **Fluid Solid Interface** boundary condition. See Fluid Solid Interface (p. 1648) for more information.

External Study Import

The **External Study Importer** is an **ACT App**. It assists the designer-to-analyst workflow by enabling you to import files from simulation applications such as Discovery, Autodesk Fusion 360, and PTC Creo Ansys Simulation into Mechanical for further analysis. Imported files can include a full definition of geometry, materials, loads, and constraints from the source simulation.

Note:

The **External Study Importer** is not supported on Linux operating systems.

This section examines how to:

- Create a/an:
  - Discovery Simulation Data File (p. 753)
  - Autodesk Fusion 360 Simulation Data File (p. 754)
- Enable the External Study Importer (p. 754)
- Import the following simulations into Mechanical:
  - Discovery (p. 754)
  - Autodesk Fusion 360 (p. 755)
  - PTC Creo Ansys Simulation (p. 755)

Create a Discovery Simulation Data File

To transfer a Discovery file to another application, you need to transfer the Discovery file using the following steps:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. From the **Transfer** menu in the ribbon of the **Simulation** tab, choose a product from the list.

2. In the **Save Discovery Document** dialog box, type a name for the Discovery file in the **File name** field and click **Save**.

Create an Autodesk Fusion 360 Simulation Data File

Perform these steps from within Autodesk Fusion 360 to create a Fusion 360 Simulation file:

1. Open the Simulation workspace for a Generative Design outcome.

2. Select **Send Study to Ansys** from the Solve group in the ribbon.

   - If Mechanical 2020 R1 is installed, this will automatically populate a Workbench project with the study and launch Mechanical.

   - If Mechanical 2020 R1 is not installed, you can save a Fusion 360 Simulation (`.sdz`) file, which you can later import via the External Study Importer.

Enable the External Study Importer

The External Study Importer is provided as an ACT App that is installed by default with Workbench.

To load the External Study Importer:

1. From the **Project** tab in Workbench, click **ACT Start Page** to open the ACT Home page.

2. Click **Manage Extensions**.

3. On the Extension Manager page, click **ExternalStudyImporter** and select **Load extension** from the context menu.

   If you want the External Study Importer to be loaded automatically for future Workbench sessions, click **ExternalStudyImporter** and select **Load as default** from the context menu.

4. Return to the ACT Home page and click **Launch Wizards**.

5. On the Wizards page, click **Execute wizard** to launch the External Study Importer.

Import a Discovery Simulation into Mechanical

After creating a Discovery Simulation Data file, you can import it into Mechanical. You can import it into a new project or an existing project.

To import a Discovery simulation into a new project in Mechanical: in Windows File Explorer, double-click a `.dscodat` file to launch Workbench and import the simulation into a new project.

To import a Discovery simulation into an existing project in Mechanical:

1. On the Wizards page (which you can access as described above), click **Execute wizard** to launch the External Study Importer Wizard.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Ensure **Study Type** is set to **Discovery Simulation**.
3. In **File To Import**, browse to the Discovery Simulation Data file you want to add to the project and click **Open**.
4. In the External Study Importer Wizard, click **Import**.
5. The import process creates a fully-populated Simulation System on the Workbench **Project Schematic** page, which you can subsequently open in Mechanical.
6. In the External Study Importer Wizard, click **Finish**.

Note:

Because of the way that Discovery manages the scoping of the Default Convection load in certain instances, you could see differences between your model in Discovery compared to Mechanical when the load is transferred between the results of the two applications.

Import an Autodesk Fusion 360 Simulation into Mechanical

You can import an Autodesk Fusion 360 Generative Design simulation into Mechanical via a Fusion 360 Simulation file (`.sdz`). In release 2020 R1, the External Study Importer supports Autodesk Fusion 360 Simulation files that have been created from Fusion 360 Generative Design outcomes. You can import the .sdz file into a new project or an existing project.

To import an Autodesk Fusion 360 Generative Design simulation into a new project in Mechanical: in Windows File Explorer, double-click a `.sdz` file to launch Workbench and import the simulation into a new project.

To import an Autodesk Fusion 360 simulation into an existing project in Mechanical:

1. On the Wizards page (which you can access as described above), click **Execute wizard** to launch the External Study Importer Wizard.
2. Ensure **Study Type** is set to **Fusion 360 Simulation**.
3. In **File To Import**, browse to the Fusion 360 Simulation Data file you want to add to the project and click **Open**.
4. In the External Study Importer Wizard, click **Import**.

   The import process creates a fully-populated Simulation System for each load case in the Fusion 360 simulation, which you can subsequently open in Mechanical.
5. In the External Study Importer Wizard, click **Finish**.

Import a PTC Creo Ansys Simulation into Mechanical

As described below, you can open a PTC Creo Ansys Simulation Data (`.casdat`) file as a new Mechanical system or you can open it in an active Mechanical system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Create a New System

Navigate to the file folder that contains the file and double-click the file. This 1) opens Workbench and places a new system in the **Project Schematic** and 2) opens the system Mechanical.

Import File into Existing System

1. As described above, open the **ACT Start Page**, select the **Launch Wizards** option, and select the **External Study** option to launch the **External Study Importer** wizard.
2. Set the **Study Type** field to **PTC Creo Ansys Simulation**.
3. Use the **Browse** option to specify your file in **File To Import** field.
4. Select the **Import** button. The import process creates a fully-populated simulation system in the Workbench Project Schematic.
5. Select the **Finish** option.
6. Open the new system in Mechanical.

Fluid-Structure Interaction (FSI)

Fluid-Structure Interaction (FSI) analysis is an example of a multiphysics problem where the interaction between two different physics phenomena, done in separate analyses, is taken into account. From the perspective of Mechanical, an FSI analysis consists of performing a structural or thermal analysis in the application, with some of the loads (forces or temperatures, for example) coming from a corresponding fluid analysis or previous CFD analysis. In turn, the results of the Mechanical analysis may be used as loads in a fluids analysis. The interaction between the two analyses typically takes place at the boundaries that the Mechanical model shares with the fluids model. These boundaries of interaction are collectively called the *fluid-structure* interface. It is at this interface where the results of one analysis are passed to the other analysis as loads.

A general way of tying two otherwise independent analyses together is described in System Coupling (p. 820). The specific use of System Coupling as one way to perform certain FSI analyses is mentioned where applicable in the following sections.

For one specific multiphysics problem, the structural thermal-stress analysis, an FSI analysis is not always required. If the thermal capabilities of the Mechanical application are sufficient to determine a proper thermal solution, an FSI approach (using separate applications for separate analyses) is not required and the thermal-stress analysis can be done entirely within Mechanical. In the case where the thermal solution requires the specialized capabilities of a CFD analysis, the structural thermal-stress analysis is done using the FSI approach. The CFD analysis is done first, then the calculated temperatures at the fluid-structure interface are applied as loads in the subsequent Mechanical analysis.

Typical applications of FSI include:

- Biomedical: drug delivery pumps, intravenous catheters, elastic artery modeling for stent design.
- Aerospace: airfoil flutter and turbine engines.
- Automotive: under-the-hood cooling, HVAC heating/cooling, and heat exchangers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Fluid handling: valves, fuel injection components, and pressure regulators.
- Civil engineering: wind and fluid loading of structures.
- Electronics: component cooling.

The Mechanical application supports two types of Fluid-Structure Interaction: one-way transfer (p. 757) and two-way transfer (p. 757). In one-way FSI, CFD results are applied as loads in the Mechanical analysis, but the results of the Mechanical analysis are not passed back to a fluids analysis. In two-way FSI, the results of the Mechanical analysis *are* passed back as loads to the fluids model. Two-way FSI is important when the Mechanical analysis could produce results that, when applied as loads in the fluids analysis, would significantly affect the fluids analysis.

One-Way Transfer FSI

In a one-way transfer FSI analysis, the CFD analysis results (forces, pressures, temperatures, convection loads, or heat flows) at the fluid-structure interface are transferred to the Mechanical model and applied as loads. The subsequently calculated displacements or temperatures at the interface are not transferred back to the CFD analysis. One-way transfer is appropriate when displacements and temperatures differentials calculated in the Mechanical application are not large enough to have a significant effect on the fluid analysis.

There are four supported applications of a one-way FSI analysis:

1. Based on the setting of the **Interpolation Type** property:
   - **Mechanical-Based Mapping** (default): Pressure results from a CFD analysis are input as normal pressure in a structural analysis at the fluid-structure interface.
   - **CFD Results Interpolator**: Pressure results from a CFD analysis are input as applied forces in a structural analysis at the fluid-structure interface.
2. Temperature results from a heat transfer CFD analysis are input as body loads in a structural or LS-DYNA analyses to determine the thermally induced displacement and stresses (thermal-stress analysis).
3. Convections from a heat transfer CFD analysis are input as convection boundary conditions (film coefficients and bulk temperatures) in a thermal analysis at the fluid-structure interface.
4. Temperatures or heat flows from a heat transfer CFD analysis are input as temperature or heat flow boundary conditions in a thermal analysis at the fluid-structure interface.

There are two methods available for performing a one-way FSI analysis: importing loads and System Coupling. See Using Imported Loads for One-Way FSI (p. 758) and One-Way FSI Transfers Using System Coupling in Workbench (p. 828), respectively.

Two-Way Transfer FSI

In a two-way transfer FSI analysis, the CFD analysis results (forces, temperatures, heat flows, or heat transfer coefficients and near wall temperatures) at the fluid-structure interface are transferred to the Mechanical model and applied as loads. Within the same analysis, the subsequently calculated displacements, temperatures, or heat flows at the fluid-structure interface are transferred back to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the CFD analysis. Two-way transfer is appropriate when displacements and temperature differentials calculated in Mechanical are large enough to have a significant effect on the fluid analysis.

Because of the two-way interaction between the two analyses, the analyses are looped through repeatedly until overall equilibrium is reached between the Mechanical application solution and CFD solution. Two-way FSI is supported between Mechanical and Fluent and Mechanical and CFX. In either case, you set up the static or transient structural portion of the analysis in the Mechanical application, including defining one or more fluid-structure interface boundary conditions. You continue the analysis in Fluent or CFX, and view the structural results in the Mechanical application.

For general information on two-way FSI, see System Coupling (p. 820). For more information on two-way FSI using Mechanical and Fluent, see Performing System Coupling Simulations Using Fluent in the *Fluent User's Guide*. For more information on two-way FSI using Mechanical and CFX, see Coupling CFX to an External Solver: System Coupling Simulations in the *CFX-Solver Modeling Guide*.

Note:

In a System Coupling setup, if you apply an external force or external heat flow on the same region as a Fluid-Structure Interaction load, this external variable will not be acknowledged by the Mechanical APDL solver.

Using Imported Loads for One-Way FSI

This feature enables you to import fluid forces, pressures, temperatures, and convections from a steady-state or transient CFD analysis to a Mechanical application analysis.

This one-way transfer of face forces (tractions) or pressures at a fluid-structure interface enables you to investigate the effects of fluid flow in a static or transient structural analysis. Similarly, the one-way transfer of temperatures or convection information from a CFD analysis can be used in determining the temperature distribution on a structure in a steady-state or transient thermal analysis or to determine the induced stresses in a structural or LS-DYNA analysis.

To import loads from a CFD analysis:

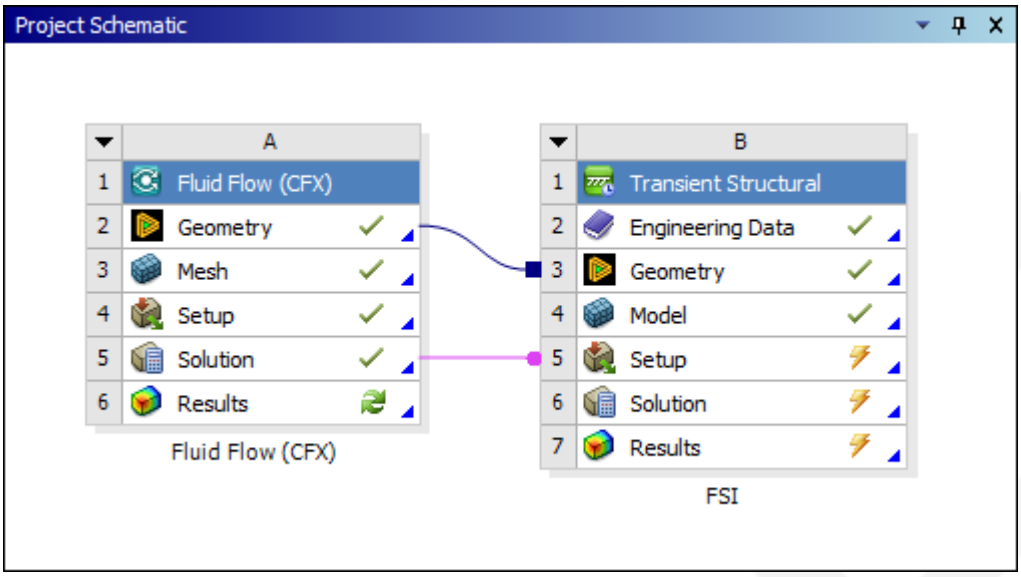
1. Complete your CFD analysis. From the **Project Schematic**, add an appropriate Mechanical analysis and create links between:

   a. The **Solution** cell of your CFD analysis and the **Setup** cell of the newly added Mechanical or LS-DYNA analysis.

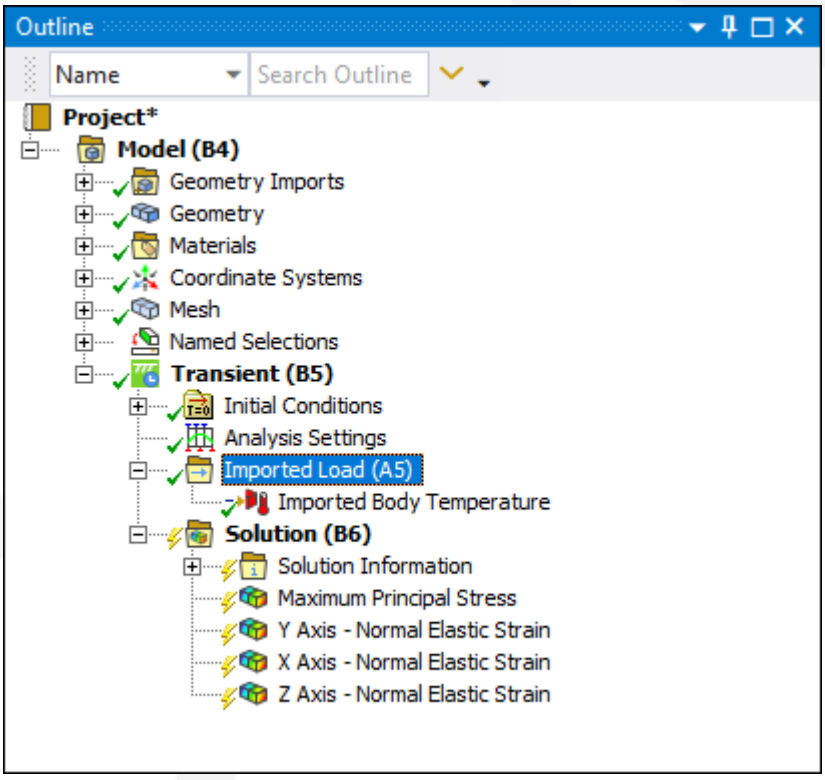
   Your system should appear as illustrated here.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



2. Double-click **Setup** cell to open Mechanical. By default, an **Imported Load** folder is added under the **Environment** folder. Alternatively, select the Fluids Results File option of the **Imported Loads** drop-down menu on the **Environment** ribbon to manually insert an **Imported Load (Fluids Results File)** object to select an existing fluids results file from the disk.



3. Select the **Imported Load (Group)** object and:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

a. Specify the **Interpolation Type** property as either: **CFD Results Interpolator** or **Mechanical-Based Mapping** (default).

Note:

The **Mechanical-Based Mapping** option is not available for LS-DYNA analyses.

b. Right-click, select **Insert**, and then select the desired load type you wish to add. Loads can also be added via the **Environment** Context Tab **Imported Loads** drop-down menu.

4. For your newly inserted load object, you can modify the following properties in the **Definition** category:

- **Tabular Loading**: This property controls the creation and content of the **Data View** table. **Data View** values are applied at each load step. The options for this property include **Program Controlled** (default), **Ramped**, **Stepped**, **Off**.
- **Source Time**: This property enables you to manage how the data from your source analysis is imported into the **Data View** table. The options for this property include **Worksheet** (default), **All**, and **Range**. Based on your selection, the time-based data contained in the **Data View** table displays accordingly.

Note:

– While defining source times and time step data, at any time you can right-click the imported load object and select the option **Sync Analysis Times**. This option synchronizes the data of the **Data View** table with your **Step Controls** (p. 1418) in the **Analysis Settings** object.

– As needed, you can also change the values of the **Source Time** column entries in the **Data View** table. To do so, select a **Source Time** cell for the imported load in **Data View** table and specify a different value. Press **Enter**. The **Source Time Step** value changes based on the source time you select. If the selected source time corresponds to more than one source time step, you will also need to select the desired time step value. You can also change the **Analysis Time** and specify **Scale** and **Offset** values for the imported loads.

– **Source Time** is not available for LS-DYNA analyses.

Note:

For the **Mechanical-Based Mapping** setting, the **Source Time** field does not support non-zero values that are less than or equal to 1e-9s.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. For your newly inserted load object, you can modify the following properties in the **Transfer Definition** category based on the setting of the **Interpolation Type** property. Note the following when making your specifications:

   - If you specify the **Interpolation Type** property as **Mechanical-Based Mapping** (default), refer to the Data Transfer Mesh Mapping (p. 2389) section for detailed descriptions of the properties contained in the **Settings**, **Graphics Controls**, and **Legend Controls** categories.

   - Under the **Transfer Definition** category:

     – For surface transfer, open the drop-down menu for the **CFD Surface** property and select the corresponding CFD surface. If you have specified the **Interpolation Type** property as **Mechanical-Based Mapping**, you can use the **Ctrl** key to select multiple options from this menu.

     – For volumetric transfer, open the drop-down menu for the **CFD Domain** property and select the corresponding CFD Domain. If you have specified the **Interpolation Type** property as **Mechanical-Based Mapping**, you can use the **Ctrl** key to select multiple options from this menu.

   - For **CFD Convection** loads only: select the appropriate **Ambient Temperature Type**.

     Note:

     – **CFD Near-Wall Ambient (bulk) Temperature** (default): This option uses the fluid temperature in the near-wall region as the ambient temperature for the film coefficient calculation. This value will vary along the face. When you specify **Mechanical-Based Mapping** as the **Interpolation Type**, Mechanical calculates the **Wall Adjacent Heat Transfer Coefficient** as the film coefficient. If you wish to use the Fluent derived heat transfer coefficients, export the solution data from Fluent and import it via External Data (p. 736) import.

     – **Constant Ambient Temperature**: This constant value applies to the entire scoped face(s). The film coefficient will be computed based on this constant ambient temperature value. Use of a constant ambient temperature value in rare cases may produce a negative film coefficient if the ambient temperature is less than the local face temperature. If this is the case, you can define a Supplemental Film Coefficient. This value will be used in place of the negative computed film coefficient and the ambient temperature adjusted to maintain the proper heat flow.

   - In a structural analysis, if the **Imported Body Temperature** load is scoped to one or more surface bodies, the **Shell Face** option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

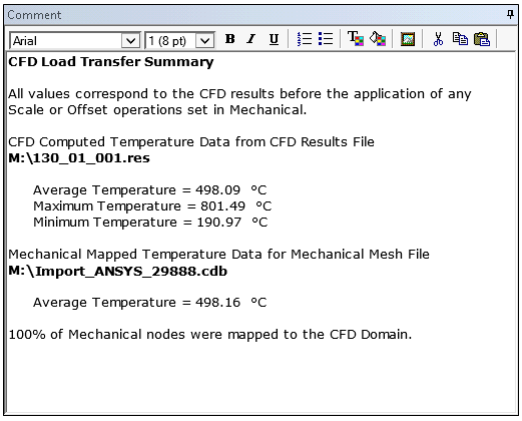
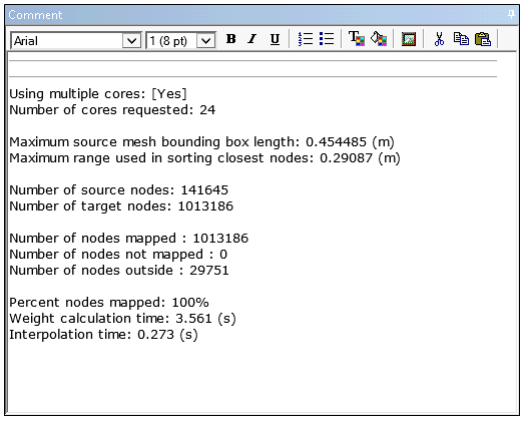
6. In the Project **Outline**, right-click the imported load object and then select the **Import Load** option to import the load. When the load has been imported successfully, a contour plot will be displayed in the **Geometry** window.

Once the solution is complete for the **Interpolation Type** property set to **CFD Results Interpolator**, a CFD Load Transfer Summary is displayed as a Comment in the particular CFD load branch. The summary contains the following information:

- For a CFD Pressure load: the net force, due to shear stress and normal pressure, on the face computed in CFD and the net force transferred to the Mechanical application faces. Note that Mechanical-Based Mapping only imports normal pressure.
- For a CFD Temperature load: For surface transfers - the average computed temperature on the CFD boundary and the corresponding average mapped temperature on the Mechanical application faces.

  For volumetric transfers – the average, maximum, and minimum temperature of the CFD domain and the corresponding Mechanical Application body selection(s).
- For a CFD Convection load: the total heat flow across the face, and the average film coefficient and ambient temperature on the face.

The computed and mapped face data may be compared in order to get a qualitative assessment of the accuracy of the mapped data. Examples of the **Imported Load Transfer Summary** for the Interpolation Types are illustrated below.

CFD Results Interpolator	Mechanical-Based Mapping
	

The following additional topics are covered in subsequent sections:

Face Forces at Fluid-Structure Interface

Face Temperatures and Convections at Fluid-Structure Interface

Volumetric Temperature Transfer

CFD Results Mapping

Face Forces at Fluid-Structure Interface

You can use results at a fluid-structure interface from a CFD analysis as face forces (from the vector sum of the normal pressures and shear stresses) on corresponding faces in the Mechanical

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

application. The import process involves interpolating a CFD solution onto the Mechanical application face mesh. This requires that the following conditions are met:

- The fluid-structure interface must be a defined *boundary* in CFD.
- The location of the CFD boundary (with respect to the global Cartesian coordinate system) must be the same as the corresponding face(s) in the Mechanical application model.

Refer to the Imported Boundary Conditions (p. 1736) section for more information.

Face Temperatures and Convections at Fluid-Structure Interface

This feature allows the transfer of either of the following thermal solutions from a CFD solution boundary to a corresponding face in the Mechanical application model:

- Temperatures at the fluid-structure interface.
- Film coefficients and bulk temperature values at the fluid-structure interface.

The import process involves interpolating a CFD solution onto the Mechanical application face mesh. This requires that the following conditions are met:

- The fluid-structure interface must be a defined *boundary* in CFD.
- The location of the CFD boundary (with respect to the global Cartesian coordinate system) must be the same as the corresponding face(s) in the Mechanical application model.

Refer to the Imported Loads (p. 1736) section for more information.

Volumetric Temperature Transfer

You can transfer temperature results from a CFD analysis and apply them as body loads in the Mechanical application. The import process involves interpolating a CFD solution onto the mesh for the bodies selected in the Mechanical application. This requires that the following condition is met:

- The location of the bodies in the Mechanical application model (with respect to the global Cartesian coordinate system) must be the same as the corresponding CFD domains.

CFD Results Mapping

When mapping CFD results onto the Mechanical application face(s) the Mechanical nodes are projected on to the CFD face. All the Mechanical application face nodes will map to the CFD face according to the following rules:

a. Project normal to the CFD mesh faces.

b. If rule a fails, project to the closest edge.

c. If rule b. fails, project to the closest node on the CFD face.

Rule c. will always work, so in the end every node will get some kind of mapping. However the most accurate load mapping occurs for nodes projected normal to the mesh face. The percentage of the Mechanical application nodes that mapped successfully using rule a. above is reported in

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the diagnostics. When the Mechanical application mesh is very coarse, there can be some misses near the edges of the CFD boundary. However all nodes become mapped eventually. The accuracy of force transfer improves as the Mechanical application mesh is refined.

When mapping CFD domain results onto the corresponding Mechanical Application body selection(s), all the Mechanical Application nodes that cannot be mapped to the CFD domain will be set to the average temperature.

Icepak to Mechanical Data Transfer

The Mechanical application enables you to transfer temperature data from Icepak into Mechanical. This process involves the import of temperature data from the solid objects defined in Icepak onto the geometry defined in Mechanical. As the meshes used in Icepak and Mechanical could be quite different, mapping the temperatures involves an interpolation method between the two. Once the mapping is completed, it is possible to view the temperatures and utilize them to perform a Mechanical analysis. The workflow is outlined below.

Workflow for Icepak Data Transfer

1. In Icepak, perform all steps for an Icepak analysis by creating the Icepak model, meshing and solving the model. After the solution has finished, Icepak writes out the temperature data for each of the solid objects to a file with the extension loads. In addition, a summary file with the extension load summary is written out.

   Note:

   The **CFD Post/Mechanical data** option must be enabled in the **Solve** panel to transfer data to CFD-Post/Mechanical. If this option was not enabled prior to solving, you also have the option of exporting data using the **Post** > **Workflow data** menu in Ansys Icepak.

2. Drag and drop a Mechanical cell, which could be one of Static Structural, Steady-State Thermal, Transient Structural, Transient Thermal, or Thermal-Electric analysis on top of the Icepak Solution cell.

3. Import the geometry or transfer the geometry into the Mechanical application. Double click the Setup cell to display the Mechanical application.

4. In the Details section of **Imported Temperature** or **Imported Body Temperature** under **Imported Loads**, you will first select the Scoping method. Select **Geometry Selection** as the Scoping method unless you have created a **Named Selection**. See Scoping Analysis Objects to Named Selections (p. 1083) for a detailed description.

5. If **Geometry Selection** is selected as the **Scoping method**, pick the geometry using **Single select** or **Box select** and click **Apply** or select a **Named Selection** object in the drop-down list.

6. In a structural analysis, if the Imported Body Temperature load is scoped to one or more surface bodies, the Shell Face option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

7. To suppress this load, select **Yes**. Otherwise, retain the default setting.

8. In the drop-down field next to **Icepak Body**, select one body at a time, **All** or a **Named Selection**. If selecting an individual body, make sure your selection corresponds to the volume selected in step 5. If **All** bodies were selected, select **All**.

9. The Icepak Data Solution Source field displays the Icepak temperature source data file.

10. You can modify the Mapper Settings (p. 2389) to achieve the desired mapping accuracy.

11. Click the imported load object, then right-click and select **Import Load**. This process first generates a mesh, if one doesn't already exist, and then interpolates the temperatures from the Icepak mesh onto the Mechanical mesh. This process might take long if the mesh size or the number of bodies is large. Improving the quality of the mesh will improve the interpolation results but the computation time may be higher.

    Note:

    If the import is successful, you can see the temperature plot in the graphics display window.

    If multiple time steps refer to the same time, an error will be displayed in the Mechanical message window.

12. You can apply other boundary conditions and click **Solve** to solve the analysis.

How to Set up a Transient Problem

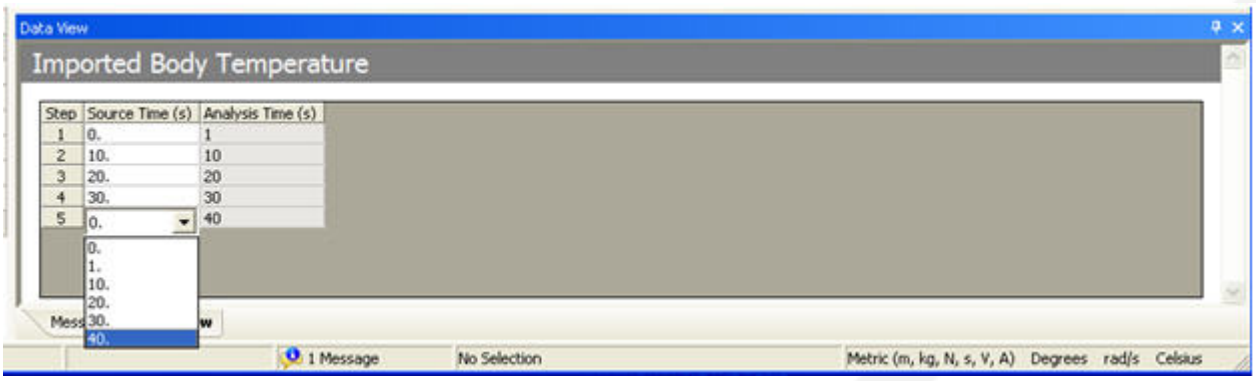
1. In Icepak, perform all steps for a transient Icepak analysis and solve the model.

2. Perform steps 2 – 9 as described above.

3. Click the **Analysis Settings** object in the tree. Begin adding each step's **End Time** values for the various steps to the tabular data window. You can enter the data in any order but the step end time points will be sorted into ascending order. The time span between the consecutive step end times will form a step. You can also select a row(s) corresponding to a step end time, click the right mouse button and choose **Delete Rows** from the context menu to delete the corresponding steps. See Establish Analysis Settings (p. 349) for further information. Whenever a new row is added or deleted, the imported body temperature data view will be updated to match the number of rows in the **Analysis Settings**.

4. Click the imported load object and the Data View tab with updated **Analysis Times** is displayed. If the **Analysis Time** is different, the **Source Time** will display the original time, matching to the closest available **Source Time** coming from Icepak. If the match is not satisfactory, you can select a **Source Time(s)** from the drop-down list and Mechanical will calculate the source node and temperature values at that particular time. This combo box will display the union of source time and analysis time values. The values displayed in the combo box will always be between the upper and lower bound values of the source time. If the user modifies the source time value, the selection will be preserved until the user modifies the value even if the step's end time gets changed on the analysis settings object. If a new end time value is added/deleted, Source Time will get the value closest to the newly added Analysis time value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



5. Click the imported load object, then right-click and select **Import Load**. This will interpolate the value at all the selected time steps.

6. User can display interpolated temperature values at different time steps by changing the **Active Row** option in the detail pane.

7. Apply required boundary conditions, continue with any further analysis and solve.

Inverse Solving for Nonlinear Static Structural Analyses

Introduction

The basic difference between a traditional (forward) analysis and the use of the Inverse Option, is the state of the initial geometry. For a forward solution process, the initial geometry is deformed under loading conditions and results are evaluated on the deformed geometry. While using the Inverse Option for your solution, you begin with a known input geometry that is already deformed under a set of loads that produced the deformation. Therefore, the solution output generated by the solver is the geometry (also referred to as solution geometry or reference geometry) that would have existed without the application of the loads. However, the result values are always calculated on the input geometry during the inverse stage of the solution.

For both solution processes, the basic concepts, solution approaches, and steps are the same.

Important:

Inverse solving is only supported for Static Structural (p. 652) nonlinear analyses with the **Large Deflection** property set to **On**, that is, when the deformations may be large enough to affect the solutions results.

Application

To perform inverse solving, simply set the **Inverse Option** property to **Yes** in the **Advanced** category (p. 1461) of the Analysis Settings. Setting the property to **Yes** also displays the **End Step** property. The **End Step** property specifies the step number in the analysis when the inverse solving routine ends. The default value of **End Step** is **1** and is read-only (unless Beta Options options are enabled). The steps in the analysis until the **End Step** of Inverse Option are referred to as inverse steps. The steps in the analysis that occur after the **End Step** of the Inverse Option are referred as forward steps.

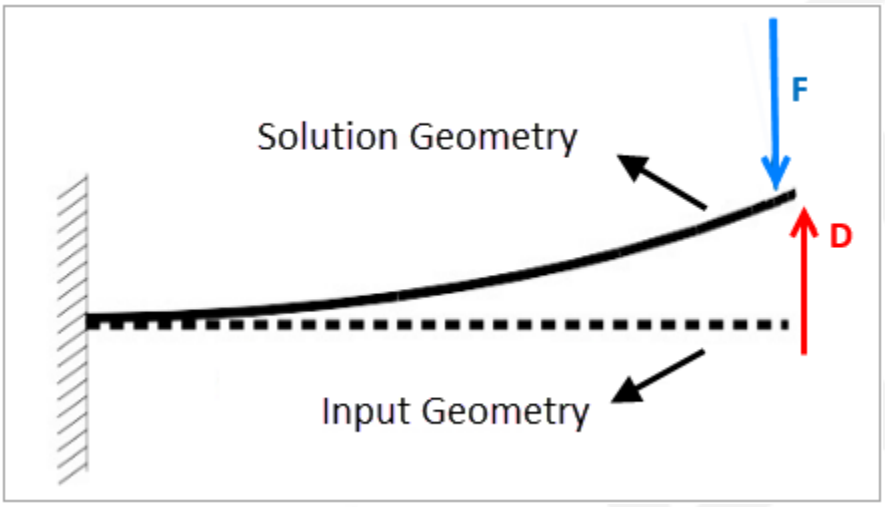
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When set to **Yes**, the **Inverse Option** will always perform an inverse solving on at least the first load step. This assumes that the input geometry for the analysis is a geometry that is already deformed under the specified loading during the inverse steps.

Proper specification of the **End Step** and **Number of Steps** properties enables you to perform:

- Inverse solving with the intent to obtain the solution/reference geometry only. You can perform purely inverse solving by setting the same value for **End Step** and the **Number of Steps** properties. The direction of force-based loading, such as Pressure, Force, etc., should be considered such that it brings the Solution Geometry to the Input Geometry. However, for displacement-based loads, such as Displacement, Remote Displacement, etc., the prescribed displacement should be defined such that it brings the Input Geometry to the Solution Geometry.



- Inverse solving followed by a forward solving. This is useful when you need to examine the structural response under additional loading. You perform this type of analysis by setting the **End Step** property to a value that is less than the value provided for the **Number of Steps** property. The analysis begins with an input geometry that is already deformed under the specified loading conditions during the inverse steps. During the start of the forward solution step, the solver resets the displacements to zero and start from the Input geometry. As a result, you may see a sudden jump in displacement from the end of inverse step to the beginning of the forward step because of this reset.

- Loop test to verify the modelling details used with **Inverse Option**. To perform a loop test, 1) enter an **End Step** property value that is less than the setting of the **Number of Steps** property, 2) make sure the applied loads remain the same between the inverse and forward steps. Because the applied load is same for the Inverse and forward step, the same Input geometry, which exists at time = 0, will be recovered at the end of the forward step.

Note:

To perform a loop test, it is recommended that you export the solution geometry (**Export** > **STL**) following the inverse step in order to have a separate analysis and apply the same loads that were applied for the inverse steps.

Be sure to review the Requirements and Limitations (p. 768) outlined below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Requirements and Limitations

Review the categorized requirements and limitations listed below. In order to the inverse capability, you must make sure these requisites are met.

Analysis Settings

When defining Analysis Settings, the **Inverse** option requires the:

- **Large Deflection** property to be set to **On**.
- The **Inverse Option** requires the Unsymmetric solver. If the **Inverse Option** property is set to **Yes**, the application automatically selects this solver.

Elements

The Inverse Option only supports a mesh with higher order elements.

Contact

The **Inverse** option only supports the **MPC** setting for the **Formulation** property for contact conditions and does not support Bearings, Springs, Joints, Beams, and Spot Welds connection types.

Loading and Boundary Conditions

Supported Load and Boundary Conditions include:

- Inertial (p. 1500): Acceleration, Standard Earth Gravity, Rotation Velocity, and Rotational Acceleration.
- Loads (p. 1520): Pressure (direct and normal pressures only), Remote Force, and Moment.
- Supports (p. 1663): Fixed Support, Displacement, Remote Displacement, Frictionless Support, and Cylindrical Support.
- Conditions (p. 1697): Coupling and Constraint Equation.
- Direct FE (p. 1719): Nodal Force, Nodal Pressure, Nodal Displacement, and Nodal Orientation.

Important:

Because displacement type loads like Displacement and Remote Displacement brings the displacement from Input Geometry to Solution Geometry, it is important to reverse the direction of these prescribed displacements during the forward step if you wish to perform the loop test or continue with additional loads/displacements in the forward step.

Important:

The following loading conditions are **not supported**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Remote loads (Remote Forces/Remote Displacements/Moments) specified using beam behavior.
- Loads that rely on the internal generation of elements (surface effect elements, beam elements, pretension elements, etc.).

Materials

The **Inverse** option supports the following materials:

- Linear Elastic
- Hyperelastic with a material incompressibility factor $d \neq 0$.

Any nonlinear material effects, such as plasticity and creep, are not supported.

Linked Environments

If you use an inversely solved Static Structural analysis as a prestress environment for a downstream Modal or Harmonic Response analysis:

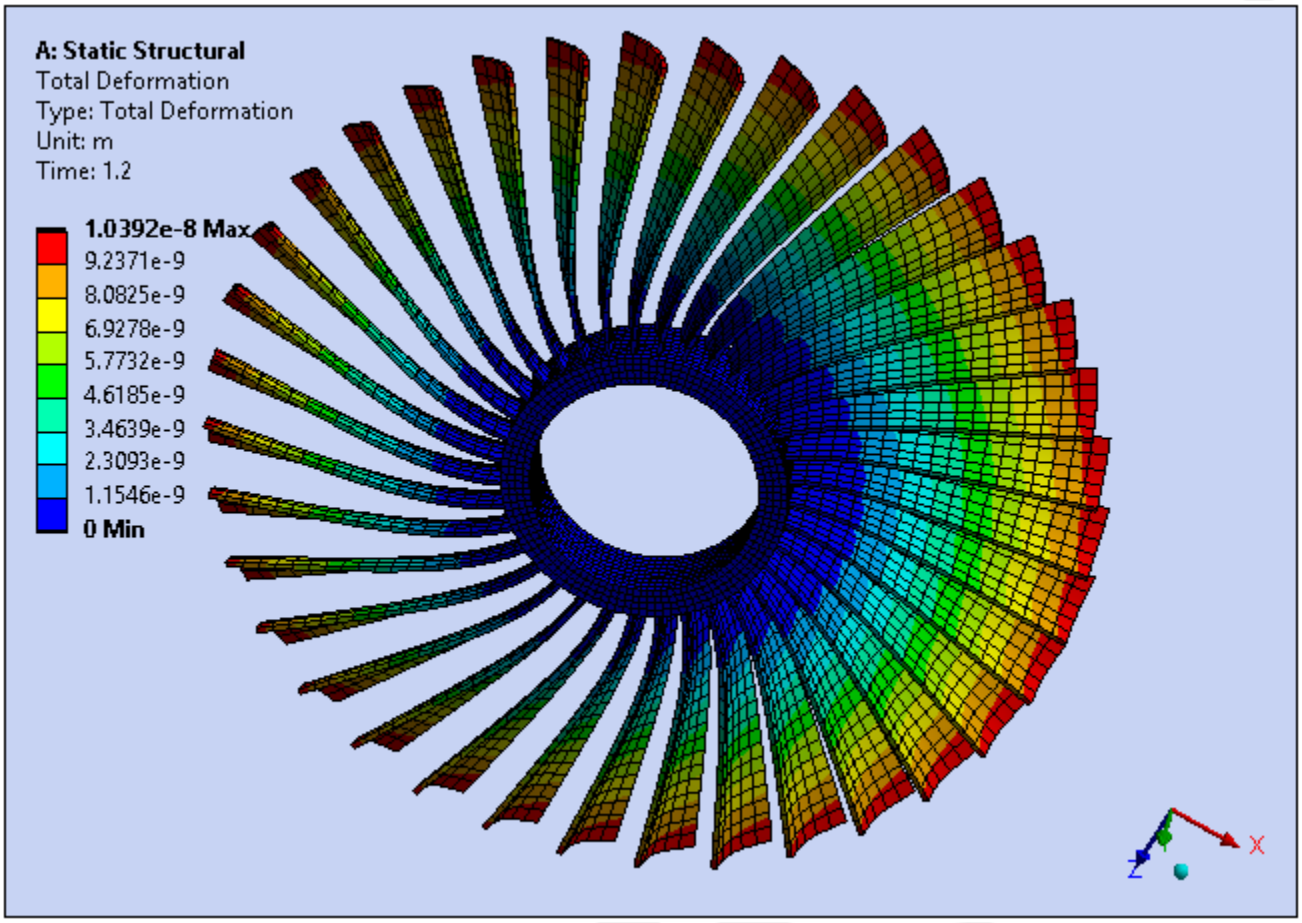
- Only forward solved steps can be selected as prestress steps for linked Modal/Harmonic Response analyses.
- For undamped Modal analyses, the application uses the **Unsymmetric** setting for the **Solver Type** property (via the **Program Controlled** setting). The other solver types (Direct, Iterative, Subspace, and Supernode) are not supported for an undamped Modal environment.

Post-Processing of Results

For result entities plotted at **Time** or **Result Set** corresponding to an inverse step, please note the following:

- Deformation results retrieved for inverse step, represent the deformation of the model with respect to the input geometry. Deformation results are plotted on the solution geometry.

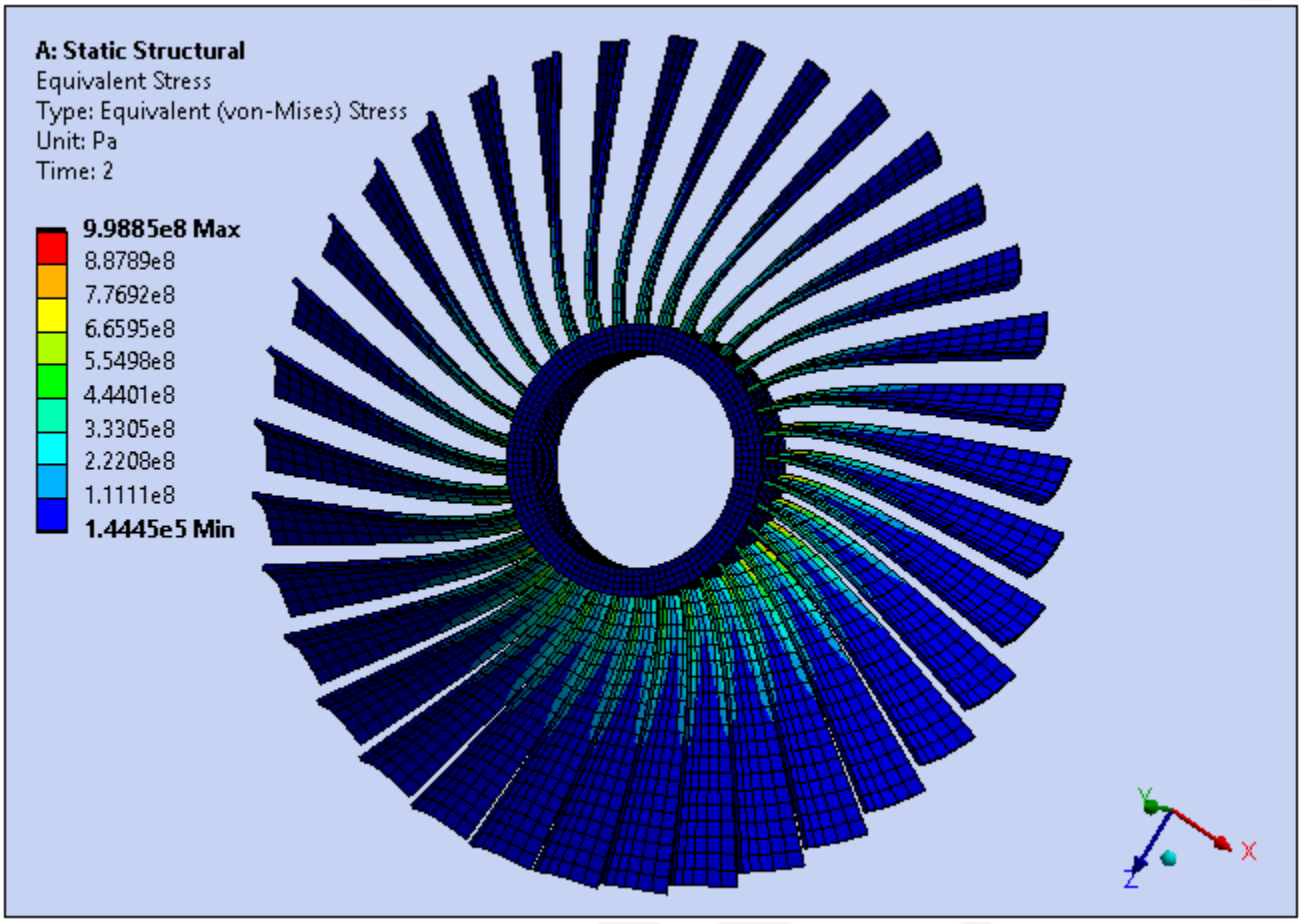
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



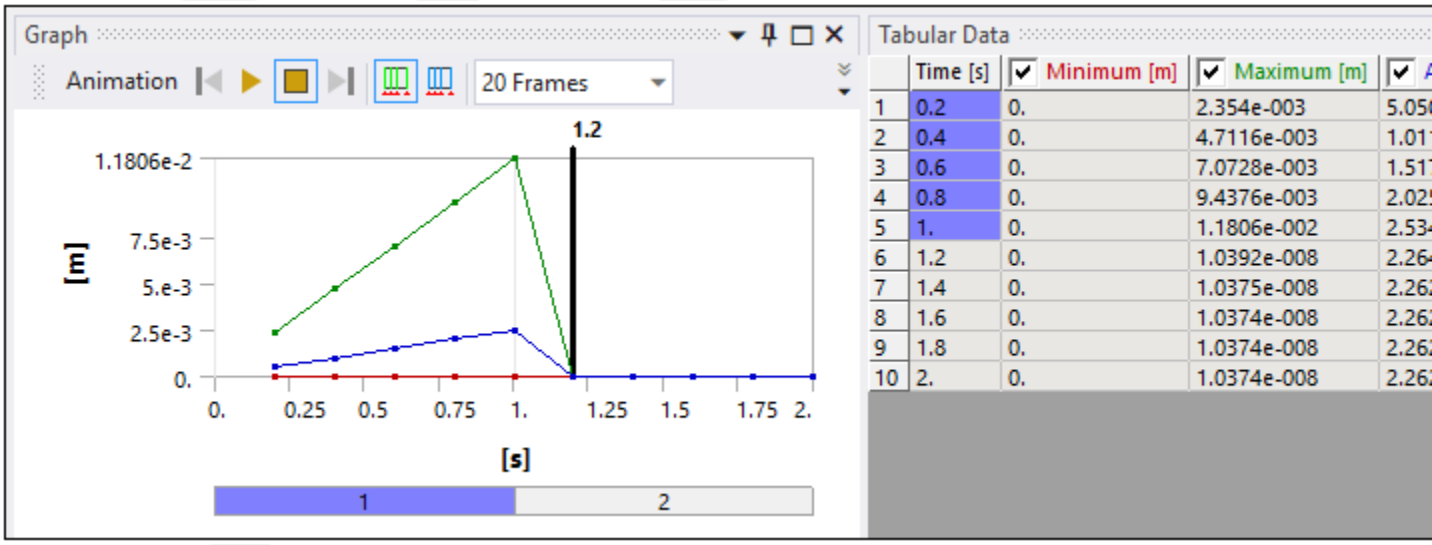
- All other result items (e.g. stress, strains, probes etc.) are plotted on the input geometry of the analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- Results produced during the Inverse solution are shown with a colored highlight in the **Graph** and **Tabular Data** window.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- You can save the solution/reference geometry using the context (right-click) menu option **Insert** > **Export** > **STL** on a deformation result created for the **Display Time**/**Load Step** corresponding to the **End Step** of the inverse solution.

Note:

Be sure to use the Zoom to Fit Animation (p. 2263) option in the **Graph** window to properly animate your Inverse results. When the option is active, Mechanical loops through all the time steps to compute a scale factor that accommodates a full range of time steps and makes sure that the animation fits properly within the **Geometry** window.

MAPDL Reference

For an additional information, review the Nonlinear Static Analysis with Inverse Solving section of the Mechanical APDL Structural Analysis Guide.

Mechanical-Electronics Interaction (Mechatronics) Data Transfer

You can export a reduced model that can be imported into Twin Builder.

Overall Workflow for Mechatronics Analysis

1. Create a modal analysis system.
2. Define the inputs using **Remote Points** and/or **Named Selections**. The names of the entities created must include the prefix `input_` and the degree of freedom in the trailing suffix, signified by an underbar (e.g. "`input_MyName_ux`").

   Note:

   The **Named Selection** can only be scoped to a vertex.

   ---

3. Define the outputs using **Named Selection**. The names of the entities created must include the prefix `output_` and the degree of freedom in the suffix (e.g. "`output_MyName2_rotx`").

   Note:

   The **Named Selection** can only be scoped to a vertex.

   ---

4. Specify the modal damping in a Commands Object under an Environment, for example:

```
dmprat,.02          ! 2% damping on all modes
mdamp,1,.05         ! 5% damping on mode 1
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. At Solution level, add a Commands Object and import the macro `ExportStateSpaceMatrices.mac` to export the reduced model. It is located at the installation folder under: `ANSYS Inc\v241\aisol\DesignSpace\DSPages\macros`

   Note:

   The macro is based on the Mechanical APDL command **SPMWRITE**.

6. Solve the Modal Analysis.

7. The reduced model file (`file.spm`) and the graphics file (`file_spm.png`) will exist in the solver files directory and can then be imported into Twin Builder. (See Saving and Managing Project Files in *Workbench User's Guide* for more information on solver files directories.)

Set up the Mechanical Application for Export to Twin Builder

To set up the Mechanical application to retrieve the inputs and outputs defined so they can be used in the reduced model exported to Twin Builder:

1. From the **File** tab, select the **Variable Manager** option.

2. In the **Variable Manager** window, add/activate the variable **ExportToTwinBuilder** and set it to 1.

Polyflow to Mechanical Data Transfer

This feature enables you to import data from a Polyflow system and apply it in a Mechanical application analysis. Temperature data can be imported into a static structural, transient structural[3] steady-state thermal, transient thermal or thermal-electric analysis.

To import data from a **Polyflow** system:

- In the **Project Schematic**, right-click the **Solution** cell of the **Polyflow** system and select **Transfer Data to New**><*Mechanical system*>, a link is created to the **Model** cell of the selected Mechanical system. If you select **Transfer Data to New** > ***<Mechanical system>***, this operation automatically creates a link to the **Model** cell of the Mechanical system. Alternatively, you can drag the **Solution** cell of the **Polyflow** system and drop it onto the **Model** cell of a Mechanical system to create the link.
- To transfer temperature data to Mechanical, drag the **Solution** cell of the **Polyflow** system and drop it onto the **Setup** cell of an applicable Mechanical system.
- To transfer thickness data to Mechanical, drag the **Solution** cell of the **Polyflow** system and drop it onto the **Model** cell of an applicable Mechanical system.

If your simulation has thickness defined from a Polyflow system, an **Imported Thickness** folder is added under the **Geometry** folder.

[3] The rigid dynamics solver is not supported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Select appropriate geometry in the **Details** view, and then click **Apply**.
2. Select appropriate options in the **Details** view. You can modify the mapping settings to achieve the desired mapping accuracy.
3. You can specify a thickness value for the unmapped target nodes using the **Unmapped Data Value** property. By default, a zero thickness value is assigned to the unmapped nodes.

   Important:

   For the Mechanical APDL solver, the thickness value at each node must be greater than zero.

4. Right-click the **Imported Thickness** object, and then click **Import Thickness** to import the thickness. When the thickness has been imported successfully, a contour plot will be displayed in the **Geometry** window and any mesh display will be based upon the mapped thickness of the elements.

If your simulation has temperature data defined from a Polyflow system, an Imported Load folder is added under the Environment folder.

1. Select an imported temperature load to add from the **Imported Loads** drop-down menu on the **Environment** Context tab or right-click the **Imported Load** folder and select the appropriate load from the **Insert** context menu.
2. Select appropriate geometry in the **Details** view, and then click **Apply**.
3. In a 3D structural analysis, if the Imported Body Temperature load is scoped to one or more surface bodies, the **Shell Face** option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.
4. Select appropriate options in the **Details** view. You can modify the mapper settings (p. 2389) to achieve the desired mapping accuracy.

   - In a 3D analysis, if the **Triangulation** mapping algorithm is used, the **Transfer Type** mapping option defaults to **Surface** when an Imported Temperature or Imported Body Temperature load scoping is only on shell bodies. If the scoping is on shell bodies and other geometry types, the **Transfer Type** mapping option will default to **Volumetric**. In such cases, to obtain a more accurate mapping, you should create a separate imported load for geometry selections on shell bodies, and use the **Surface** option for **Transfer Type**.

5. Under **Data View**, select the desired **data Identifier**, for the imported load. The data identifier (File Identifier: Data Identifier) strings are specified by the upstream **Polyflow** system. You can also change the **Analysis Time** and specify **Scale** and **Offset** values for the imported loads.
6. Right-click in the **Data View** and select **Add row** to specify additional data for a different analysis time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

7. In the project tree, right-click the **Imported Load** object, and then click **Import Load** to import the load. When the load has been imported successfully, a contour plot will be displayed in the **Geometry** window.

8. If multiple rows are defined in the **Data View**, imported values at different time steps can be displayed by changing the **Active Row** option in the details pane.

9. Change any of the columns in the **Data View** tab as needed:

   - **Magnitude**

     Select the appropriate data identifier that represents the load values to be applied from the drop-down list.

   - **Analysis Time**

     Choose the analysis time at which the load will be applied. For the Mechanical APDL solver, this must coincide with the end time of a step defined in the **Analysis Settings** object in the tree.

   - **Scale**

     The amount by which the imported load values are scaled before applying them.

   - **Offset**

     An offset that is added to the imported load values before applying them.

10. To activate or deactivate the load at a step, highlight the specific step in the **Graph** or **Tabular Data** window, and choose **Activate/Deactivate***at this step!* See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

Reinforcement Specification Using Mesh-Independent Method

Introduction

For an appropriately modeled geometry, Mechanical provides options to specify reinforcements within a structure. Mechanical can model these reinforcing geometries with specialized reinforcing elements. These reinforcing elements provide extra reinforcing by interacting with the standard structural or thermal elements, referred to as base elements, via the shared nodes.

This section examines:

- Feature Overview (p. 776)
- Application (p. 777)
- Loading Condition Specification (p. 779)
- Post Processing (p. 779)
- Requirements and Limitations (p. 779)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Feature Overview

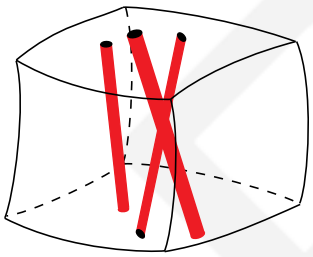
Reinforcement is supported for the following three-dimensional analysis types:

- Static Structural
- Steady-State Thermal
- Thermal-Stress (Steady-State Thermal linked to Static Structural)
- Modal (Standalone and Pre-Stressed)
- Harmonic Analysis (Standalone, Pre-Stressed, and MSUP)
- Random Vibration
- Response Spectrum

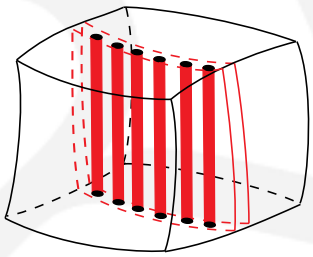
These analyses require that your imported model already include the line or surface bodies you wish to specify as reinforcements as well as an appropriate material to assign to these reinforcement bodies.

As illustrated, Mechanical provides reinforcement specification for line bodies (discrete reinforcing) and surface bodies (smeared reinforcing). Each line body specified as reinforcement basically represents a reinforcing fiber arbitrarily oriented in space. Each surface body specified as reinforcement basically represents a reinforcing layer. This reinforcing layer can be either a homogeneous reinforcing layer (membrane) or reinforcing layer with evenly spaced fibers.

Discrete (Line Bodies Only)



Smeared (Surface Bodies Only)



Mechanical uses the Mechanical APDL-based Mesh-Independent Method for Defining Reinforcing to create reinforcing elements. The mesh-independent method uses MESH200 elements to represent the reinforcing member locations inside the generated reinforcing elements.

When the solution is initiated, the application temporarily defines the reinforcement locations using MESH200 elements along with the base elements. During the solution process, the application internally creates the element REINF264 for line bodies or the element REINF265 for surface bodies based on the intersection of corresponding MESH200 and base elements.

Important:

- The line and surface bodies defined as reinforcements are *reinforcement representations*, not actual reinforcing elements or members created by the MAPDL solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Different solver units might affect the number of reinforcing elements or members generated during the solve due to tolerances. This could lead to small differences in the result values.

Application

The steps to specify line or surface bodies as reinforcements include:

Line Bodies

For the Line Body object:

1. Select the **Reinforcement** option for the **Model Type** property.
2. As needed, assign a material to the Line Body reinforcement.
3. Verify that the desired cross section is specified in the **Cross Section** property. The area value of this cross section is sent to the solver (**Area** property of the corresponding cross section object).

   Note:

   - The solver does not consider the shape of the cross section. The area of the cross section is the only required value. See the *Reinforcing* topic in the **SECDATA** command section of the *Mechanical APDL Command Reference* for more information.
   - During the solution process, by default, the application 1) turns on Tension and Compression and 2) removes the base material from the space occupied by the reinforcing fibers.

4. Repeat the above steps for all line bodies to be used as a reinforcement.

Surface Bodies

For Surface Body objects, the default setting for the **Model Type** property is **Shell**. The **Shell** setting treats the body as a surface body - there is no reinforcement treatment.

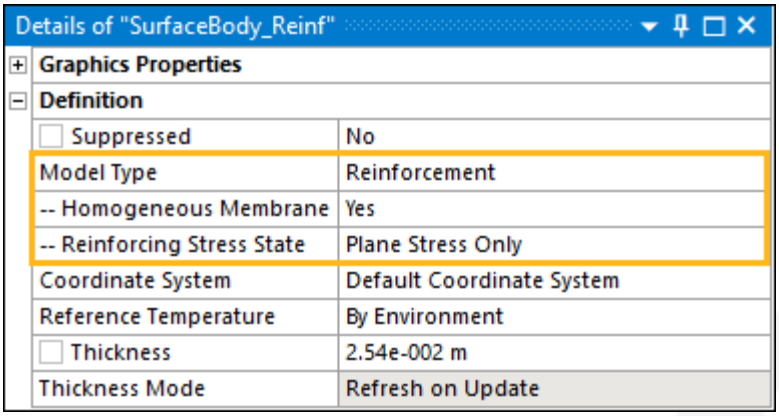
To specify the Surface Body as a reinforcement:

1. Select the **Reinforcement** option for the **Model Type** property.
2. As needed, assign a material to the Surface Body reinforcement.
3. When you select the **Reinforcement** option, an additional property, **Homogeneous Membrane**, displays.

   If the **Homogeneous Membrane** property is set to **Yes** (default), then the Surface body is treated as a homogeneous reinforcing layer (membrane) in a plane-stress state. Verify

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

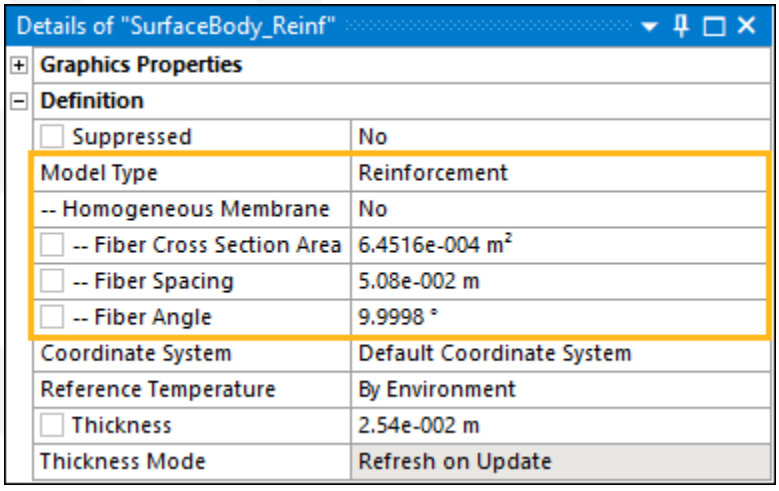
the value in the **Thickness** property since this value is used for defining the cross section area of the reinforcing membrane inside the reinforcing element.



For 3D Structural analyses, when the **Homogeneous Membrane** property is set to **Yes**, the additional property **Reinforcing Stress State** displays. This property enables you to specify the reinforcing stress state and stiffness behavior of the homogeneous reinforcing layer (membrane). Options include **Plane Stress Only** (default), **Plane Stress with Transverse Shear**, and **Plane Stress with Transverse Shear and Bending**. See the Reinforcing topic in the **SECCONTROL** section of the *Mechanical APDL Command Reference* for more information.

If **Homogeneous Membrane** property is set to **No**, then the Surface Body is treated as a reinforcing layer with a cluster of parallel and equidistant fibers having the same cross section. This means that the layer has a uniaxial-stress state in the direction of the fibers. The value in the **Thickness** property is ignored. The following additional properties display and require specification:

- **Fiber Cross Section Area**: Cross section area of the fiber.
- **Fiber Spacing**: Distance between fibers.
- **Fiber Angle**: Orientation of the associated fibers in the specified coordinate system.



In addition, select the appropriate Coordinate System to define the specific orientation of the fibers. See **Element Coordinate System** description in the MESH200 element section for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Repeat the above steps for all surface bodies to be used as a reinforcement.

Note:

During the solution process, by default, the application 1) turns on Tension and Compression when the Homogeneous Membrane property is set to No (uni-axial stress state) and 2) removes the base material from the space occupied by the reinforcing layer.

Loading Condition Specification

Only an Internal Heat Generation (p. 1620) load or an Imported Heat Generation (p. 1748) load can be applied to the bodies specified as Reinforcements.

Post Processing

You can create results for reinforcement bodies through **Geometry Selection** or using **Named Selections**. You can select specific reinforcement bodies or use the default setting of **All Bodies** and hide appropriate base bodies to view the results on the reinforcements.

Because the solver generates reinforcing elements, only body-based results are available on the reinforcement bodies. Mechanical does not allow Face/Edge/Vertex/Node/Element/Element Face based results on Reinforcement bodies.

Also, if you mix reinforcement body selections with non-reinforcement body selections for your result scoping, the application displays results for the bodies that are not specified as Reinforcements and excludes the evaluation of results on the Reinforcement bodies.

Note:

If you set the **Scoping Method** property to **Result File Item**, the **Global ID** property displays for results on a specified set of reinforcing members. This corresponds to the global identifier (**EGID**) in Mechanical APDL.

Requirements and Limitations

Note the following when performing this analysis type:

- Reinforcement is supported for 3D Static Structural and Steady-State Thermal analyses.
- If you are using **Material Assignment** (p. 1049) objects, Ansys recommends that you use separate **Material Assignment** objects or Line bodies and Surface bodies modeled as Reinforcements as well as for any non-reinforcement bodies. This helps ensure the application properly plots result data.
- If you have assigned a material to the base body that does not support a 1D stress state, such as Drucker Prager, the application does not remove the base material in the space occupied by reinforcing members that have uniaxial stress state.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you have a result object selected, certain mesh-based features are not available for the reinforcement elements of the geometry, including mesh selection filters (nodes (p. 269), elements, and element faces (p. 276)), as well as the ability to display a Node ID in the probe label (p. 206), and Selecting Nodes and Elements by ID (p. 279).
- Reinforcements do not support bending for nonhomogeneous membranes.
- Thermal analyses do not support surface and line bodies as base bodies for creating reinforcing elements.
- Reinforcement bodies do not support Remote Points (p. 1089), Symmetry (p. 1127), Fracture (p. 406), or Connections (p. 1207). These features are ignored when scoped to reinforcement bodies.
- Condensed Parts are not supported when a model contains reinforcements. If present, you can suppress the Condensed Part(s) or Reinforcement bodies in order to proceed with the solution.
- Only Body-based results are supported for Reinforcement Bodies. (See the Post Processing (p. 779) topic above)
- Reinforcements do not support **Elemental Triads** and **Elemental Euler Angle** results.
- For nodal results, such as nodal Reactions (p. 2107), that are scoped to reinforcement elements, the application displays values that are extrapolated, via shape functions, to element intersection points (II, JJ, etc.). This is also the case if you export result values. Exported result values may contain node numbers, but the result values are the interpolated intersection values. In addition, the summation of Mechanical nodal reaction results may not match those of the Mechanical APDL application.
- For element nodal Reactions (p. 2107) force results on reinforcement elements, the application only displays a subset of nodes on the result as compared to Mechanical APDL. For example, MAPDL lists 20 nodes for a smeared reinforcement element whereas Mechanical only lists four nodes. This also applies to exported result values.

Rotordynamics Analysis

Rotordynamics is a specialized branch of applied mechanics that studies the behaviors of rotating structures. This rotating structure, or "rotor system," is typically composed of rotors, stators, and bearings. For a simple rotor system, the rotor component rotates about an axis that is stabilized by a bearing that is supported by a stator. This structure can be as simple as computer disk or as complicated as a jet engine.

The Mechanical Rotordynamics Analysis helps to direct you when selecting properties such as rotor stiffness and geometry, bearing stiffness, damping, and stator properties for a rotor system based on a given rotating speed. For example, to effectively study a system's vibratory characteristics, you can use a Campbell diagram. A Campbell diagram enables you to determine critical speeds (for different rotating modes), such as the rate at which the rotating structure experiences resonance (peak response) to avoid possible catastrophic failure. Or, a Rotordynamic Analysis can be used to determine safe operational ranges for a rotor system.

In the Mechanical documentation, see the Rotordynamics Controls (p. 1490) section for more information, and in the Mechanical APDL documentation, the *Rotordynamic Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Refer to the following areas of the documentation for additional and associated information for Rotordynamics:

- Import Shaft Geometry
- Bearings (p. 1413)
- Campbell Diagram Chart Results (p. 2132)

Substructure Analysis

Substructuring is a procedure that condenses a group of finite elements into one element represented as a matrix. The single-matrix element is called a superelement. You can use a superelement in an analysis as you would any other element type. By summarizing the mechanical behavior of the bodies, they can be assembled to synthesize the response of a more complex structure, or to model flexible bodies in the Rigid Dynamics solver.

Mechanical provides the options to generate superelements. You use the Condensed Parts object to create superelements on a sub assembly or the Substructure Generation (p. 540) analysis to create a superelement of an entire assembly. When you use the **Substructure Generation** analysis, you can specify constraints and pre-stress effects and generate load vectors and use the elements in an analysis. When you use the Condensed Parts you can expand their solutions using the Expansion Settings object.

Supported Analysis Types

Substructuring currently supports the following analysis types:

- Harmonic Response (p. 496) (MSUP only)
- Modal (p. 515)
- Random Vibration (p. 524)
- Response Spectrum (p. 533)
- Rigid Dynamics (p. 550)
- Motion

Substructuring Types

Mechanical enables you to perform:

- **Top-down Substructuring**: Using top-down substructuring, the application generates superelements on parts that you specify using Condensed Part objects. Each **Condensed Part** object enables you to treat a set of bodies as a single superelement consisting of matrices and load vectors with far fewer degrees of freedom than the full finite element mesh. The use pass (described below) is then performed and includes all generated superelements. Using Expansion Settings, the solution expands the superelements for the specified parts. See the Top-Down

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Substructuring section in the *Mechanical APDL Substructuring Analysis Guide* for additional technical information.

Important:

You can use the Substructure Generation (p. 540) analysis to automatically generate superelements that include load vectors from the loading conditions and constraints that you apply. In addition, you can use an upstream Static Structural analysis to create pre-stress effects in the Substructure Generation analysis solution.

Or…

- **Bottom-up Substructuring**: Using bottom-up substructuring, you import superelements using the Imported Condensed Part (p. 790) object that were created in a different Mechanical session. These imported superelements represent subassemblies of the model on which the generation pass is already performed during a different Mechanical session and in the current Mechanical session these superelements are then combined with the rest of the model to perform the use pass on the entire model.

Important:

Bottom-up Substructuring does not support the Expansion pass.

Condensed Parts Introduction

Condensed Parts are defined by three key pieces of information:

- A group of bodies whose elements are to be reduced to a superelement.
- A set of interfaces defining the primary nodes that should be retained in the generated superelement.
- A list of solution settings that control the reduction process.

Substructuring Process

The top-down substructuring of condensed geometry involves the following operations:

1. **Generation**: The preliminary computation, the "Generation Pass," reduces the Condensed Part bodies into a single superelement and its primary nodes, located on the defined interfaces. The remaining interior nodes become hidden from the analyses that use the Condensed Part. For top-down substructuring, the generation pass is done in the same mechanical session as the use pass. For bottom-up substructuring, the generation pass is done in a different mechanical session and superelements are imported in the current session on which no generation pass is required.
2. **Use**: Once your Condensed Parts are properly defined and generated and Imported Condensed parts are imported into the Mechanical session, they can be used in the solution (the "Use Pass"). By hiding the interior nodes, flexible bodies can be included in a Rigid Dynamics

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

analysis when they are included in a Condensed Part. They can also be assembled together in Modal analyses to compose the overall vibration properties of a more complex structure.

3. **Expansion** (Top-down substructuring only): Following the Use Pass, you can obtain results on elements inside the Condensed Geometry using an "Expansion Pass."

Rigid Dynamics Only

For a Rigid Dynamics analysis, additional results can be produced by the Generation Pass to allow a faster expansion, which does not need to go back to the finite element model to compute stresses and deformation on a condensed part (see Expansion Pass (p. 800)).

Note:

Material and System Damping

- The solver ignores any Material Dependent Damping (**Damping Ratio** and **Constant Structural Damping Coefficient** material properties) specified in the material assigned to the bodies included in Condensed Part(s). Material dependent damping is defined in Engineering Data.
- If you specify the property **Constant Structural Damping Coefficient** (p. 1465) as global system damping using the Damping Controls (p. 1462) of the Analysis Settings, the solver ignores this value during a Modal that includes Condensed Parts or Imported Condensed Parts.

Importing Condensed Parts (Bottom Up Substructuring)

Ansys refers to the substructuring procedure described above as “top-down” substructuring. There is also a procedure to perform “bottom-up” substructuring. This procedure imports superelements that are created separately, in different Mechanical sessions. All superelements are then assembled during the use pass.

Bottom-up substructuring is performed using the Imported Condensed Part object. This object enables you to import previously generated superelements that you can then use for the Use Pass in the current system. Supported file types include:

- **Exported Condensed Part (.cpa)** (default): This file format is for superelements generated in Mechanical using the Condensed Part feature.
- **Generation Pass Output (.sub)**: This file format is for superelements generated in Mechanical APDL.
- **Super Element Matrix (.dmig)**: This file format is for superelements generated in NASTRAN.

Note:

No load vectors are imported in to Mechanical for the **Imported Condensed Part** and therefore cannot be applied to Use pass.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

See the Imported Condensed Part (p. 790) section for the steps to use this option.

Substructure Generation Analysis

Note that you can employ the Substructure Generation (p. 540) analysis to perform a generation pass on your model that condenses all the finite elements into one superelement and creates load vectors for all loading conditions. In addition, the analysis supports the use of an upstream Static Structural system to incorporate pre-stress effects in the simulation.

Analysis File Management

To improve solution processing times and memory usage when solving the Use Pass and Expansion Pass, the Mechanical APDL Solver (required) refers to the prerequisite files generated by the upstream condensed parts using the entire path to their location. See the **SEOPT** and **MODDIR** commands in the *Mechanical APDL Command Reference* for more information.

Feature Description and Application

See the following sections that describe the use of the substructuring/Condensed Part features.

Condensed Part Overview
Condensed Part Application
Condensed Part Worksheet
Exporting Condensed Parts
Imported Condensed Parts
Expansion Pass
Limitations
Best Practices

Condensed Part Overview

You insert a Condensed Geometry object via the **Model** object in the **Outline** and then insert and define Condensed Part objects. To properly define a **Condensed Part**, you need to configure the following:

- A group of bodies
- A set of interfaces
- Solution settings

Condensed Part Bodies and Interconnections

You create a Condensed Part from one or more interconnected bodies.

Important:

Because the resulting superelement is linear, any nonlinearities in the Condensed Part elements, materials, and interior connections are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When defining a **Condensed Part** using multiple bodies, the following connections may be included:

- Beam Connections
- Bonded (p. 1234) Contact
- Distributed Mass
- Fixed Joint
- Point Mass
- Spring
- Shared topology

General Restrictions for Part Bodies and Interior Connections

- Cyclic Symmetry is not supported.
- Rigid bodies may be used, but it is best for at least one body to be flexible.
- Elements cannot use Lagrange multipliers. For example, interior contact regions cannot use the **Normal Lagrange** Formulation (p. 1241).

Part Bodies and Interconnection Solver Restrictions (Ansys Rigid Dynamics Solver)

- Multi-body parts must be fully contained in a single **Condensed Part**, that is, partial selections are invalid. You may however use several multi-body parts in a single **Condensed Part**.
- The single connected component must produce six rigid modes. An insufficient number of modes will cause the Use Pass not to converge. An excess of modes can usually be remedied by breaking the **Condensed Part** into smaller rigidly connected components.

Interfaces

An interface defines the primary nodes for the resulting superelement and therefore suggests how a Condensed Part could connect to the rest of the model. In Mechanical, an interface table lists each of these connections, detailing their Type, Side, and Name. Supported interface types are based on:

- **Geometry**: Geometry interfaces directly expose all nodes on a particular topology, for example, on a vertex, face or edge. These primary nodes are therefore suitable for the application of boundary conditions such as contacts and supports.
- **Remote Points**. These entries expose remote points as primary nodes, while hiding interior nodes on the underlying topology that attach to them. Common examples include the Mobile or Reference side of Joints and Springs and the scoping of Point Masses and Remote Points. This is the only type of interface supported for the Ansys Rigid Dynamics Solver.
- **Named Selections**: Named Selections can be used to expose any node as a primary node, with the help of criterion-based Named Selection. For example, selected internal nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

within a condensed part volume could be relegated to the interface in order to produce a higher resolution of the structure's inertial or flexibility behavior

- **Connections**: Contact Region, Joint, etc.

  Note:

  For a **Contact Region** interface of a **Condensed Part**, when the **Trim Contact** (p. 1237) property is set to **On**, the generation of the **Condensed Part** trims the contact nodes on the interface and uses the trimmed nodes as the Master Degrees of Freedom for the generation pass. The reduction of the number of nodes can significantly improve the performance of the generation pass. When you select the Condensed Part object after the generation pass, the application displays the trimmed nodes in the **Geometry** window for the **Contact Region** Interface instead of the actual **Contact Region** scoping.

- **Loads and Supports**: Fixed Supports, Displacements, etc.
- **Point Masses**

General interfaces each contribute the number of nodes on their topology and remote interfaces each contribute a single node.

Important:

When a Condensed Part Interface includes faces, edges, vertices, or nodes that overlap with other interfaces where constraint equations are applied, such as remote points, boundary conditions scoped to remote points, MPC-based contact (p. 1243), etc.), the solver may remove some master node DOFs in the generation pass. This may interfere with the use pass solve as well as postprocessing.

Note:

When duplicate DOFs are disqualified and eliminated during the Generation Pass, the Rigid Dynamics solver issues an error and aborts the Use Pass.

Interface Solver Restrictions (Ansys Rigid Dynamics and Motion Solvers)

Only interfaces that are based on Remote Points are presently recognized by the Rigid Dynamics and Motion solvers (Joints, Springs, Point Masses, and Remote Points). In addition, a Remote Point must have all six DOFs with an interface treatment of either **Rigid** or **Deformable**.

Condensed Part Application

This section describes the steps to define Condensed Geometry objects. You should review all associated documentation so that you understand the application-driven actions that take place as well as the inherent limitations of the feature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The operation of condensing parts in Mechanical is largely automated. For example, Mechanical provides facilities that analyze the connectivity of the bodies in the model and group them into Condensed Parts. Connections are classified as interior or exterior (interfaces).

Application

To create a **Condensed Part**:

1. From the Workbench **Project Schematic**, create a supported analysis system.
2. Select a geometry and open Mechanical
3. Create and define the applicable boundary conditions and connections, that is Joints, Springs, Point Mass, Remote Points, Supports and/or Contacts.

   For Rigid Dynamics analyses, expand the **Geometry** folder and specify the desired connected parts as **Flexible**. These parts will be converted to Condensed Parts following the completion of the steps below.
4. Select the **Model** object and insert a **Condensed Geometry** object.

   Note:

   The application automatically inserts the **Expansion Settings** object under the **Solution** folder when a **Condensed Geometry** object is created.
5. From the **Condensed Geometry** object, you can create your condensed parts using the context (right-click) menu option **Create Automatic Condensed Parts** or you can insert individual **Condensed Part** objects in to the tree.

   The application provides two automatic methods for defining interfaces for a **Condensed Part** using the context menu options:

   - **Create Automatic Condensed Part**: This option selects the bodies and detects the interfaces and automatically creates new **Condensed Part** objects. For example, in Rigid Dynamics analyses, flexible bodies may be grouped into condensed parts and linked to other bodies through joints at the interfaces.
   - **Detect Condensed Part Interface**: This option automatically suggests interfaces. This is especially useful if you would like to manually select bodies.

   Note:

   - The **Condensed Part** object displays the number of superelement master nodes resulting from interfaces in the **Details** view.
   - The application does not consider objects imported from External Model systems for the automatic interface detection. For the imported objects that are associated with condensed parts, use the promote feature in the worksheet of the imported objects in order to be able to perform interface detection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the object reference pages for the Condensed Geometry and Condensed Part objects for descriptions of all associated **Details** view properties.

6. Insert desired results. This feature supports Deformation, Stress, and Strain results.
7. Solve the analysis.
8. As desired, you can obtain results in the original elements within the **Condensed Part** by selecting the **Expansion Settings** object and displaying the Worksheet. From there, select the desired result for your Condensed Part(s).

   It is possible to add or remove interfaces directly on the Interfaces **Worksheet** using its context menus. This is especially useful for specifying new remote interfaces not already automatically detected. Mechanical attempts to verify all interface selections during the Generation and Use Passes and issues a warning should any be missing. Mechanical also tracks operations that remove and suppress upstream objects affecting the interface and reacts accordingly.

Note:

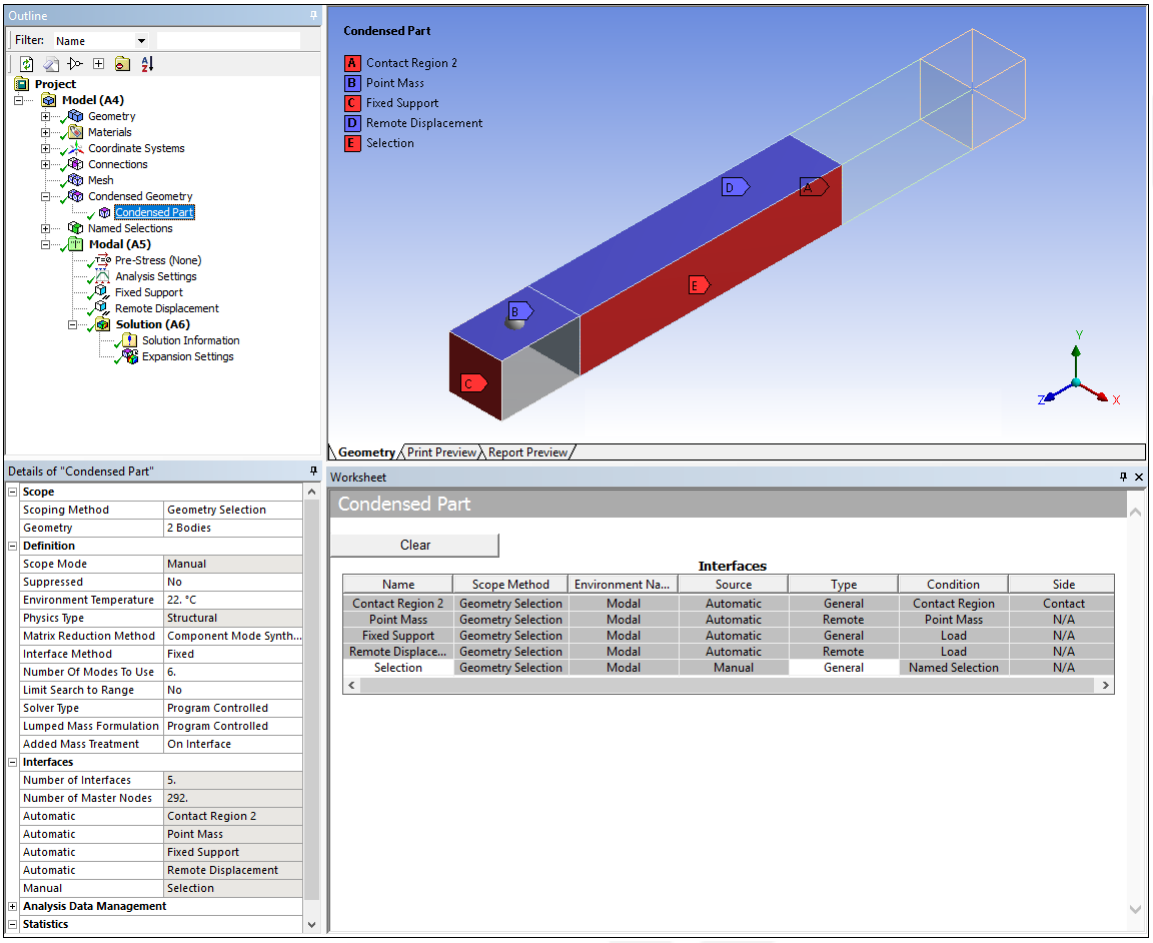
To delete a **Condensed Geometry** object, it is necessary to clean and remove any generated Condensed Parts. Similarly, to delete an **Expansion Settings** object, it is necessary to clean any generated data.

Condensed Part Worksheet

The **Worksheet** for the **Condensed Part** object is illustrated below. The **Worksheet** populates when you generate interfaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

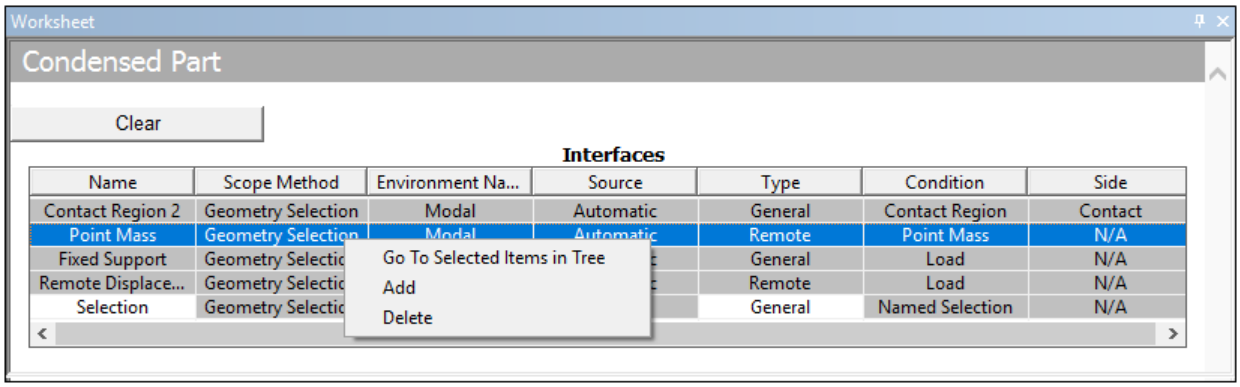
Published: 2023-10-23



Column	Description
Name	This column displays the Name of the tree object.
Scope Method	This column displays the scoping method used for the interface: Geometry Selection, **Named Selection** (geometry- or node-based), **Worksheet**, **Remote Point**, and **Free Standing**.
Environment Name	This column displays analysis system associated with the interface.
Source	This column displays how the interface is added. The field is set to **Automatic** when the interface is inserted using the **Condensed Part** object option, **Detect Condensed Part Interface** or when you manually enter an interface using the **Add** option.
Type	This column identifies the interface as either a **General** interface or a **Remote** interface. The **Remote** option is assigned for remote boundary conditions (p. 1735). In addition, the **Remote** option is the only option available for the Ansys Rigid Dynamics Solver.
Condition	This column displays the contact condition for the interface. Whether it is a **Contact Region** or a loading condition, Named Selection, **Point Mass**, etc.
Side	This column displays the topology with which the interface connects with. As such, the options include **Contact**, **Target**, **Reference**, **Mobile**, **Source Geometry**, or **Target Geometry**. See the Connections folder section for more information.

Once populated, the **Worksheet** provides the context (right-click) menu options illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



These common **Worksheet** options include:

- **Go To Selected items**: This option automatically selects and displays the corresponding object in the tree **Outline** based on the Name of the selected row in the table. You can select multiple rows using either the **Ctrl** or **Shift** keys to select and display multiple objects in the tree. Selected **Outline** objects are also highlighted in the **Geometry** window by annotations, if you switch to that view.
- **Add**: This option adds a row to the **Worksheet** that you can then manually define.
- **Delete**: This option deletes the selected row. You can select multiple rows for deletion using either the **Ctrl** or **Shift** keys. The **Delete** key can also be used to delete table rows.

Exporting Condensed Parts

The ability to export an existing solved Condensed Part enable you to use the exported file in a different simulation as an Imported Condensed Part (p. 790).

To export a Condensed Part:

1. Right-click a solved **Condensed Part** object and select **Export Condensed Part**. A progress bar displays during the export operation and offers an option to stop (red square icon) the process. The Mechanical file format for generated superelements is **Exported Condensed Part (.cpa)**.
2. Specify a file name in the **Save As** dialog and click **Save**

Note:

You can also export the solution of a Substructure Generation (p. 540) analysis as a **Exported Condensed Part (.cpa)** file.

Imported Condensed Parts

This section describes how to use the **Imported Condensed Part** object. Go to a section topic:

- Overview (p. 791)
- Application (p. 791)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Worksheet Examples (p. 793)

Overview

Using an **Imported Condensed Part** object, you can import superelements from another system into your current system. This enables you to use the superelements in the current solution ("Use Pass"). **Imported Condensed Part** objects are defined by the following information:

- A set of master nodes that represents the reduced superelement being imported.
- A set of interfaces defining the master nodes that are retained when the application generates the superelement.
- A list of controls that define the location and node numbering for the imported superelement.

Application

The following steps assume that you have already created a **Condensed Geometry** object and that you are prepared to import a supported file type.

To specify an **Imported Condensed Part** object:

1. Right-click the **Condensed Geometry** object and select **Insert** > **Imported Condensed Part**. You can also select the **Imported Condensed Part** option from the **Condensed Geometry** Context tab. Once inserted, the **Worksheet** automatically displays.
2. Select the desired file type from the drop-down list of the **Import File Format** property. Supported file formats include:
   - **Exported Condensed Part (.cpa)** (default): You create this file by either exporting Condensed Parts (p. 790) or exporting a Substructure Generation (p. 540) solution.
   - **Generation Pass Output (.sub)**
   - **Super Element Matrix (.dmig)**
3. Using the **Condensed Part File** property, define the path to the import file.
4. Right-click the new object and select **Import Condensed Part**. A progress bar displays during the import operation and offers an option to stop (red square icon) the process. Once imported, the **View Transformed Data** option/display activates. If you need to make manual **Worksheet** entries, see the steps provided in the Worksheet Options and Actions (p. 792) topic below.
5. As needed, modify the following properties:
   - **Unit System**
   - **Locating System**: Specify a desired Coordinate System to orient the imported super element. The default setting is **Global Coordinate System**.
   - **Node Offset**: Specify an offset number to be used on the imported nodes. This property has a default setting of **Program Controlled** that equates to the value zero. Using this

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

setting the application automatically offsets the node number based on the Mesh. Any value that is greater than zero offsets the Node Ids by that value.

Note:

- Remote Points need to be connected to the interfaces of the Imported Condensed parts. To avoid rigid body modes, set the **Behavior** property of the **Remote Point** to **Coupled**.
- The outline of the **Imported Condensed Part** is only displayed on the part if the information is available in the selected source file.

Worksheet Options and Actions

There may be times that you need to import or copy and paste data into the **Worksheet** using the available interface options. The **Worksheet** presents interface options and interface data based upon file format.

Exported Condensed Part

For the **Exported Condensed Part (.cpa)** format, interface data includes **ID**, **Name**, **Node Connectivity**, and **Number of Nodes**. Only the **Clear** option is available for this file format.

Note:

If a Master node of the interface is reduced during the Generation Pass, the Worksheet displays a column listing the **Reduced Number of Nodes**.

Generation Pass Output and Super Element Matrix

For the **Generation Pass Output (.sub)** and **Super Element Matrix (.dmig)** formats, interface data includes **Node ID**, **Location** and **Rotation** data. In addition, the following radio buttons display and enable you to display either the original or transformed state of the interface data.

Important:

For the **Super Element Matrix (.dmig)** file format, Mechanical reads the file's matrices in the order shown below. Make sure that the source file uses this order because the application only processes the first three matrices.

1st: Stiffness Matrix

2nd: Mass Matrix

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3rd: Damping Matrix

Tip:

If you have multiple **Super Element Matrix (.dmig)** files for each matrix, use the following API method on **Project** object to generate a single file that uses the required order of matrices.

```
CombineTextFiles(primaryFilePath, secondaryFilePath)
```

Worksheet Option	Description
View Original Data	Radio button that displays imported data when you use the **Read Interface Data** option. It displays **Original Interface Data**.
View Transformed Data	Radio button that activates automatically once the import process is complete. It displays **Transformed Interface Data**.
Read Interface Data	This option is available when the **View Original Data** option is selected, and no data has been imported. When selected, the application reads file data and populates the table with read-only Node ID and Location data from the selected file. This enables you to validate the file data. If the file contains no location data, the Location fields in the table will contain zeros (0). To complete the import process once you have read a file, right-click the object and select **Import Condensed Part**.
Edit	Select this option to enter data manually. Manual data entry can be done using copy and paste. This includes copying data from a spreadsheet. Properly select all location data from your spreadsheet and highlight the first data cell, Location X, and then paste all data. See the Worksheet Examples (p. 793) topic below for more information about manually entering data.
Save Changes	Save table data.
Clear	Clear all table data.

Worksheet Examples

The following examples describe the scenarios when importing data. Go to a topic:

- Overview (p. 794)
- Load Vectors (p. 795)
- View Original Data (p. 796)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

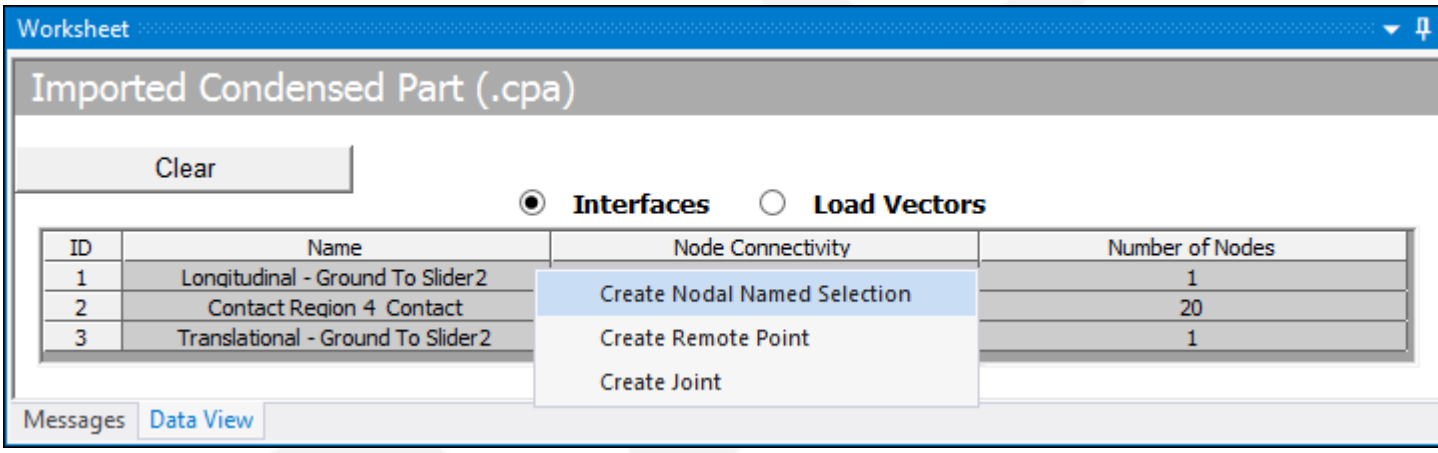
- View Transformed Data (p. 797)
- Manually Importing Spreadsheet Data (p. 798)

Overview

A successful import for an **Exported Condensed Part (.cpa)** file is shown here. The worksheet for this file type has two display options: **Interfaces** and **Load Vectors**, as described below.

Interfaces

Whether user-defined or system generated, the condensed part interfaces are listed in the worksheet by default. As illustrated, you can right-click the data presented in the **Worksheet** to create Nodal Named Selections, Remote Points, and Joints for each interface that can then be used to define connections and/or boundary conditions.



Note:

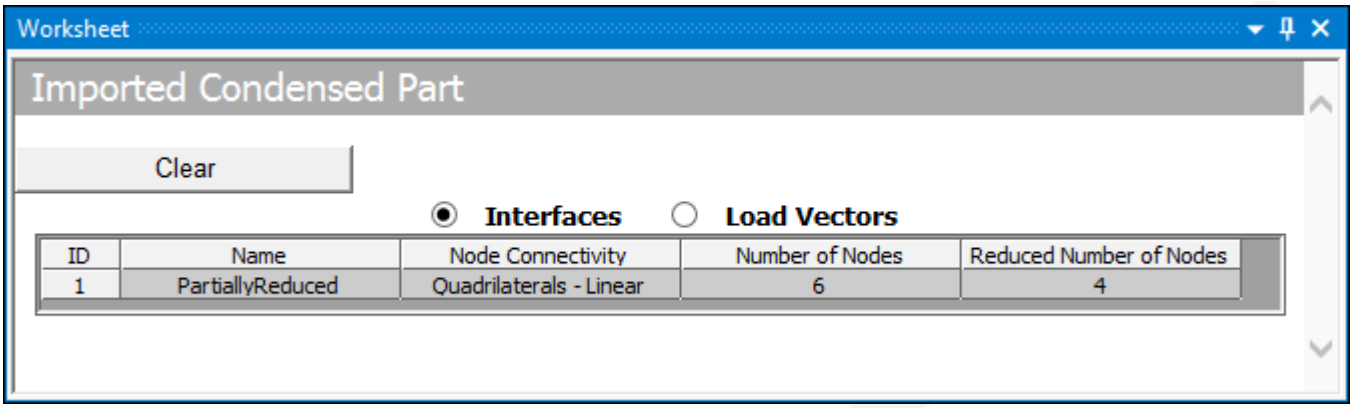
If you select the **Clear Generated Data** option on the **Imported condensed Part**:

- Named Selections or Remote Points you created will lose their scoping. When you re-import the Imported Condensed Part, the Named Selections or Remote Points get reassociated with their original scoping.
- Contact Region scoped to a Named Selection which is created from the Interfaces of the **Imported Condensed Part** remains properly defined.

During the solution Generation Pass, the application could reduce the Master degree of freedom (MDOF) as a result of an internal constraint equation. This could result in the reduction of the Master node (see **M** Command). For example, if you have an interface with six nodes and due to an application generated internal constraint equation, the solver ignores two Master nodes. This results in an interface of only four Master nodes. If the application reduces a Master node, the **Worksheet** displays an additional column, as illustrated below, that shows the exact number of nodes on that interface. This node value is supported by the options to create Nodal Named Selections, Remote Points, and Joints.

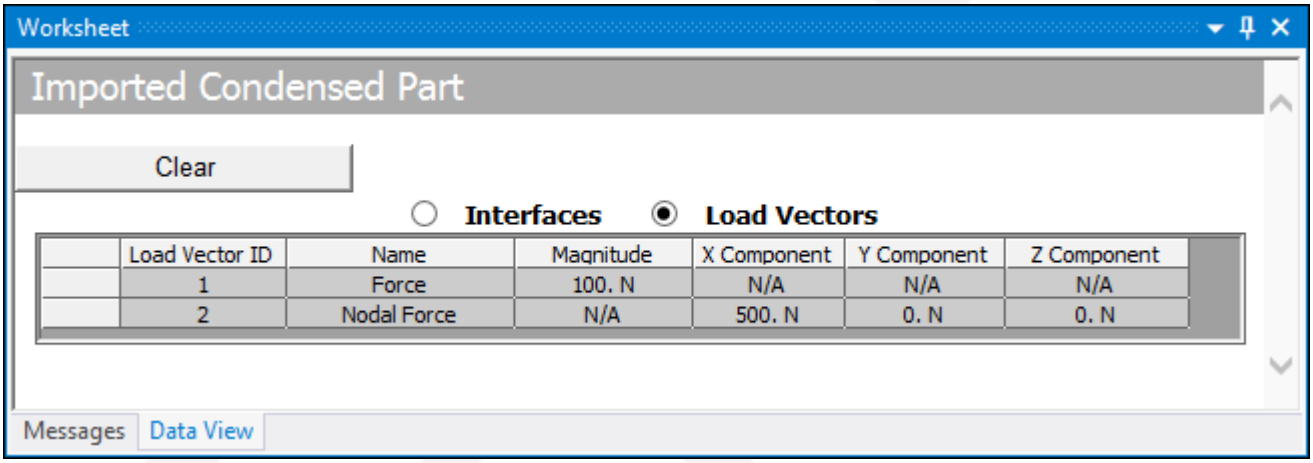
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

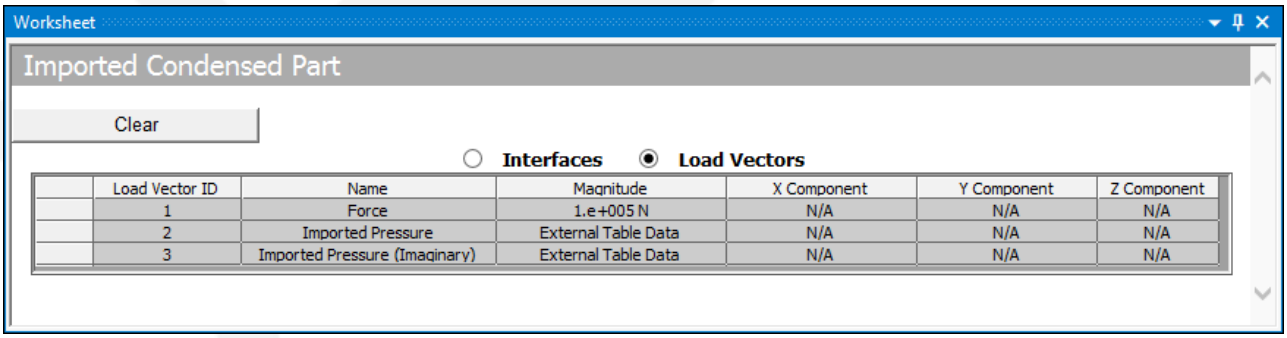


Load Vectors

Load Vectors, as well as prestressed load vector values, are exported from a solved Substructure Generation (p. 540) analysis as a `.cpa` file, and imported and displayed in the worksheet, as shown in the example below. If no load vectors are available, the table is blank.

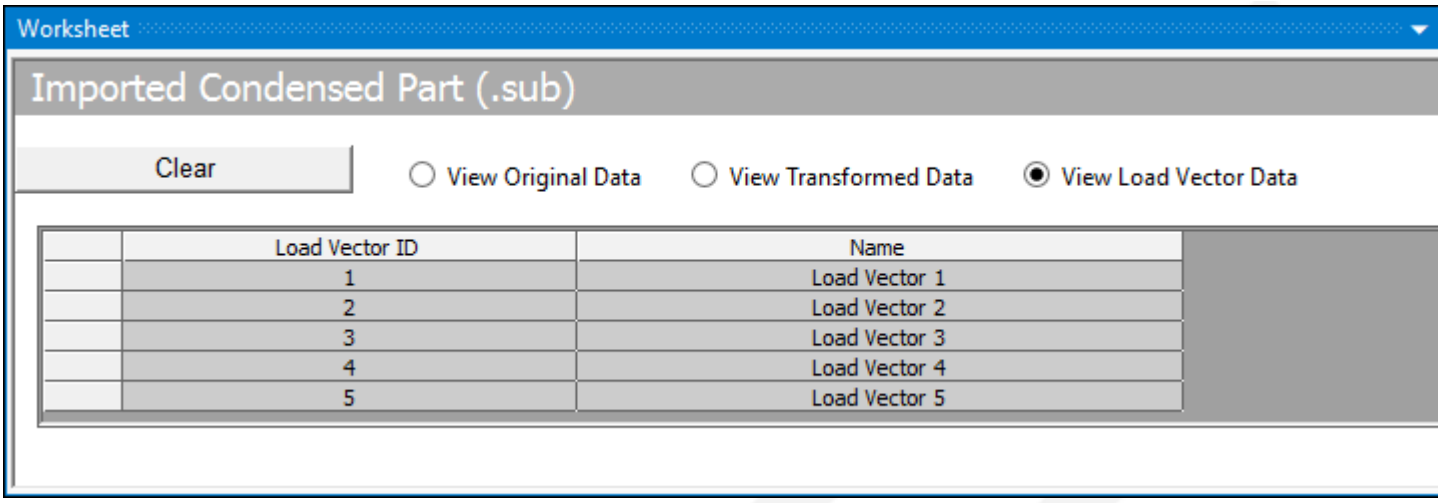


Load vectors are also generated for imported loading conditions.



Load vectors can also be imported from a **Generation Pass Output (.sub)** file that was created in the Mechanical APDL application, as illustrated here.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



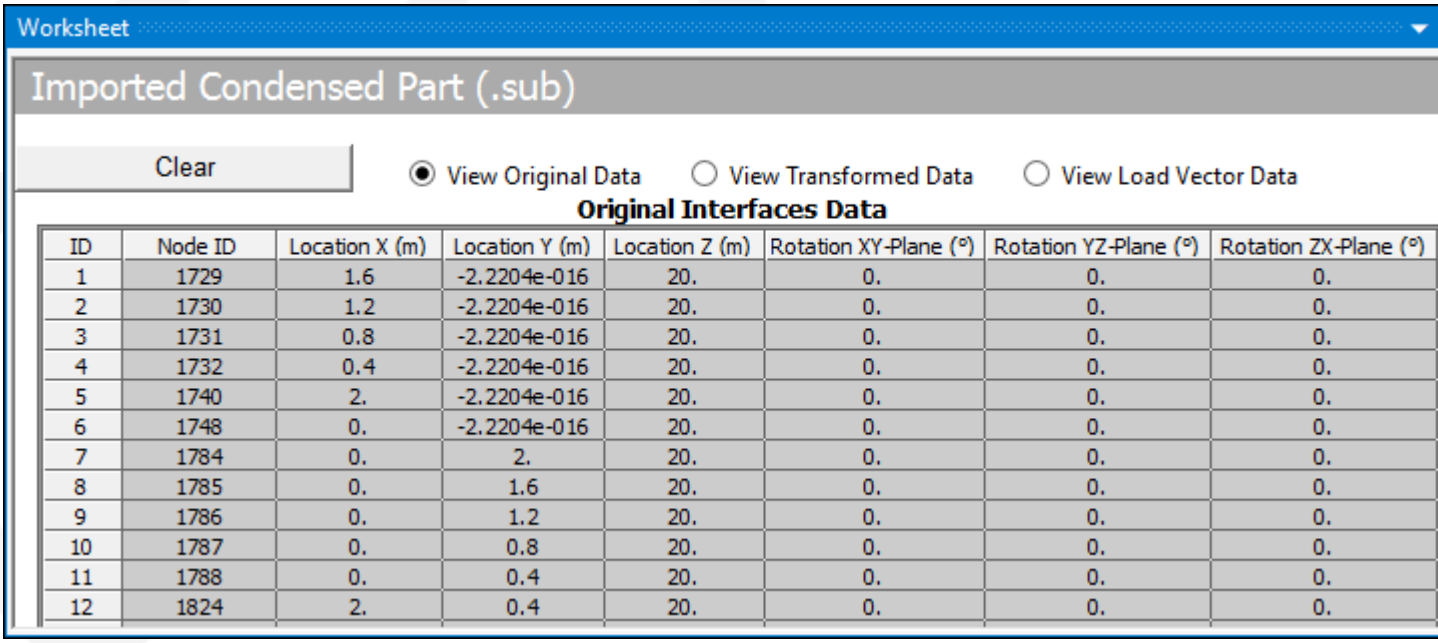
Note:

Depending upon the specified loading, the following analysis types automatically create load vectors during the solution process. If you include an **Imported Condensed Parts** object in either analysis, all of the load vectors will be contained in the solution output file.

- Harmonic Response Analysis (Mode Superposition) (p. 496)
- Harmonic Response Analysis using Linked Modal Analysis (p. 509)

View Original Data

Here is an example of the **View Original Data** option for the data read from a **Generation Pass Output (.sub)** file.

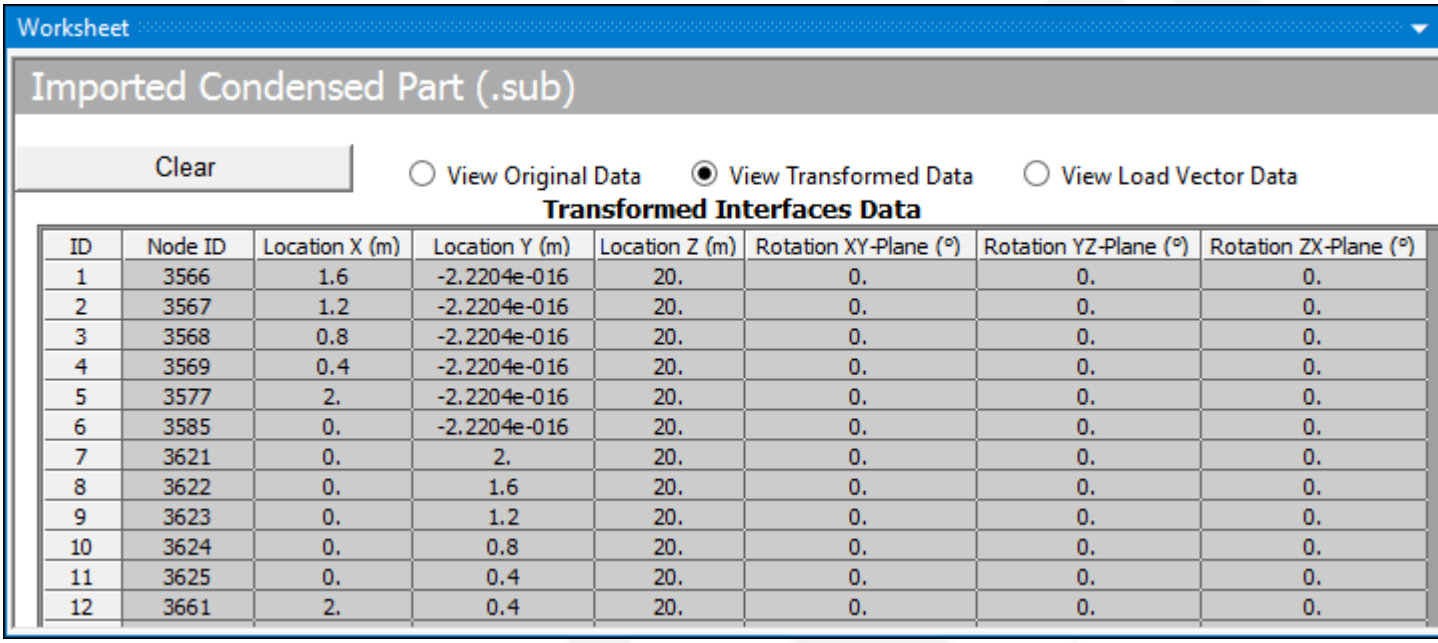


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

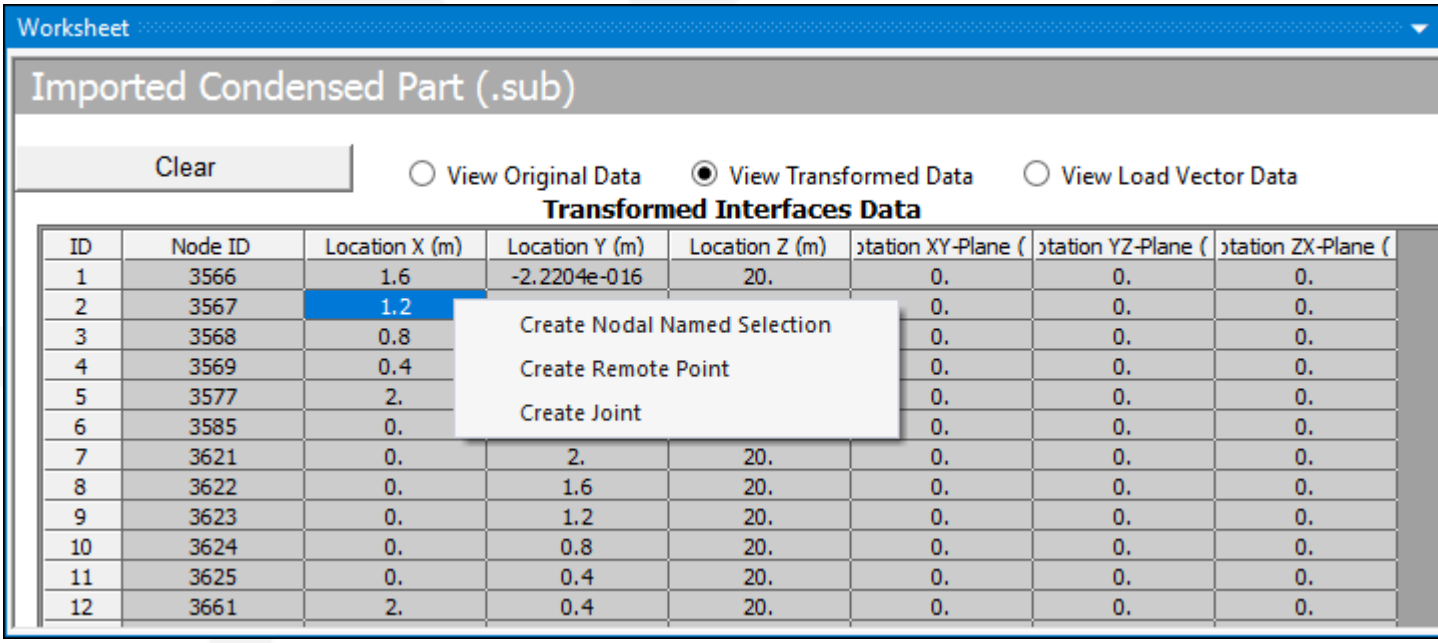
View Transformed Data

Once imported, the **View Transformed Data** option automatically displays. Here you can see transformed data. Note the Node ID values for this example versus the above example. These values are based on the **Actual Node Offset** value which for this example is 3261.



ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	Rotation XY-Plane (°)	Rotation YZ-Plane (°)	Rotation ZX-Plane (°)
1	3566	1.6	-2.2204e-016	20.	0.	0.	0.
2	3567	1.2	-2.2204e-016	20.	0.	0.	0.
3	3568	0.8	-2.2204e-016	20.	0.	0.	0.
4	3569	0.4	-2.2204e-016	20.	0.	0.	0.
5	3577	2.	-2.2204e-016	20.	0.	0.	0.
6	3585	0.	-2.2204e-016	20.	0.	0.	0.
7	3621	0.	2.	20.	0.	0.	0.
8	3622	0.	1.6	20.	0.	0.	0.
9	3623	0.	1.2	20.	0.	0.	0.
10	3624	0.	0.8	20.	0.	0.	0.
11	3625	0.	0.4	20.	0.	0.	0.
12	3661	2.	0.4	20.	0.	0.	0.

In addition, for the **View Transformed Data** option, you can create Nodal Named Selections/Remote Points/Joints for each interface.

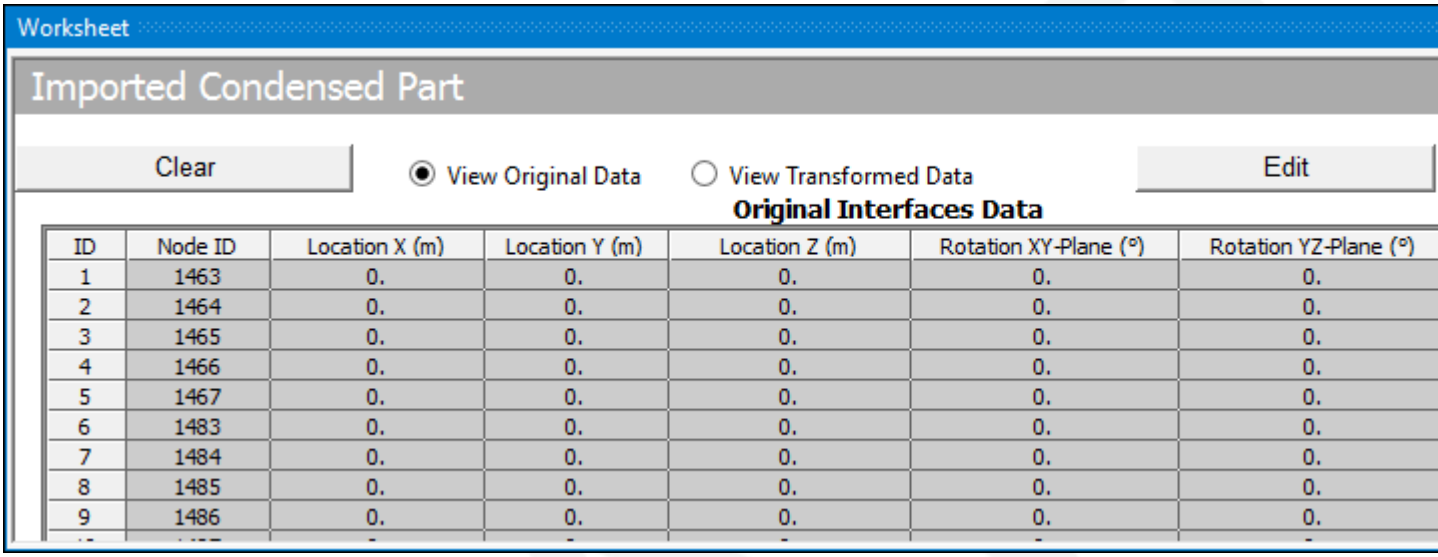


ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	otation XY-Plane (	otation YZ-Plane (	otation ZX-Plane (
1	3566	1.6	-2.2204e-016	20.	0.	0.	0.
2	3567	1.2			0.	0.	0.
3	3568	0.8			0.	0.	0.
4	3569	0.4			0.	0.	0.
5	3577	2.			0.	0.	0.
6	3585	0.			0.	0.	0.
7	3621	0.	2.	20.	0.	0.	0.
8	3622	0.	1.6	20.	0.	0.	0.
9	3623	0.	1.2	20.	0.	0.	0.
10	3624	0.	0.8	20.	0.	0.	0.
11	3625	0.	0.4	20.	0.	0.	0.
12	3661	2.	0.4	20.	0.	0.	0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Manually Importing Spreadsheet Data

For the **Super Element Matrix (.dmig)** file format, there may be times that you need to manually copy data into the worksheet table. For the **Worksheet** example shown here, no location data was found in the file. To make manual entries, select the **Edit** button.



ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	Rotation XY-Plane (°)	Rotation YZ-Plane (°)
1	1463	0.	0.	0.	0.	0.
2	1464	0.	0.	0.	0.	0.
3	1465	0.	0.	0.	0.	0.
4	1466	0.	0.	0.	0.	0.
5	1467	0.	0.	0.	0.	0.
6	1483	0.	0.	0.	0.	0.
7	1484	0.	0.	0.	0.	0.
8	1485	0.	0.	0.	0.	0.
9	1486	0.	0.	0.	0.	0.

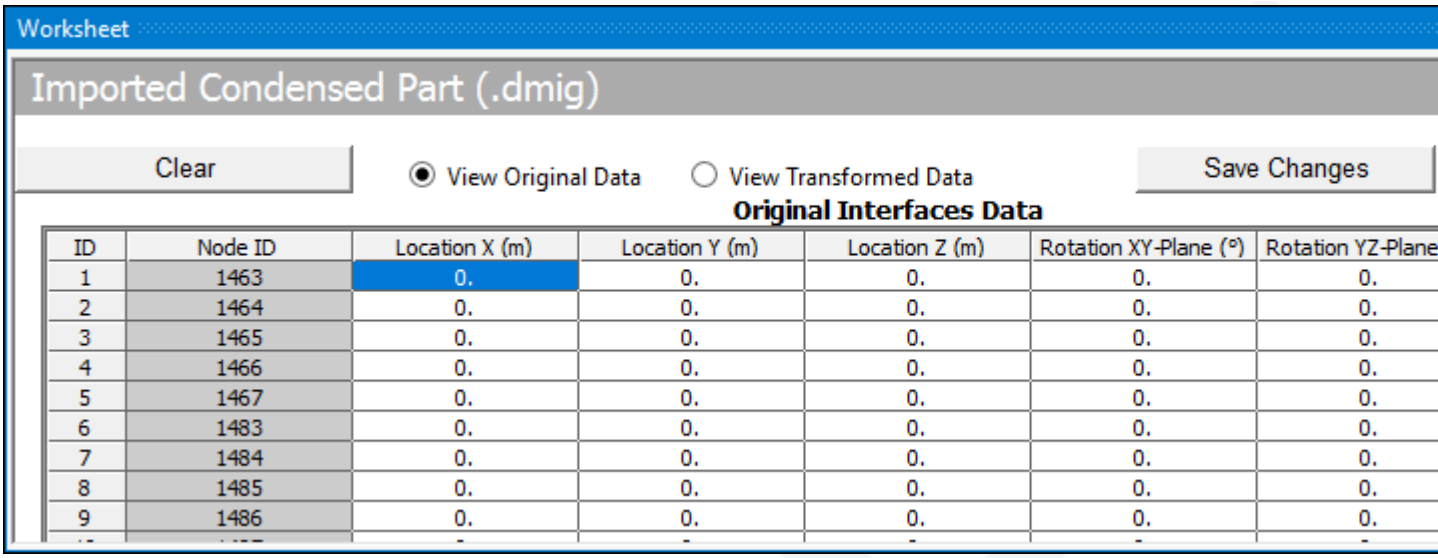
Properly select all location data (only) from your spreadsheet.

C	D	E	F
Node ID	Loc X	Loc Y	Loc Z
1463	1.67E-02	0.165	3.00E-02
1464	3.33E-02	0.165	3.00E-02
1465	5.00E-02	0.165	3.00E-02
1466	6.67E-02	0.165	3.00E-02
1467	8.33E-02	0.165	3.00E-02
1483	0.1	0.165	3.00E-02
1484	0.1	0.165	0.33
1485	8.33E-02	0.165	0.33

Highlight the **Location X** field as illustrated below. This ensures that the data will be properly distributed throughout the table when you paste.

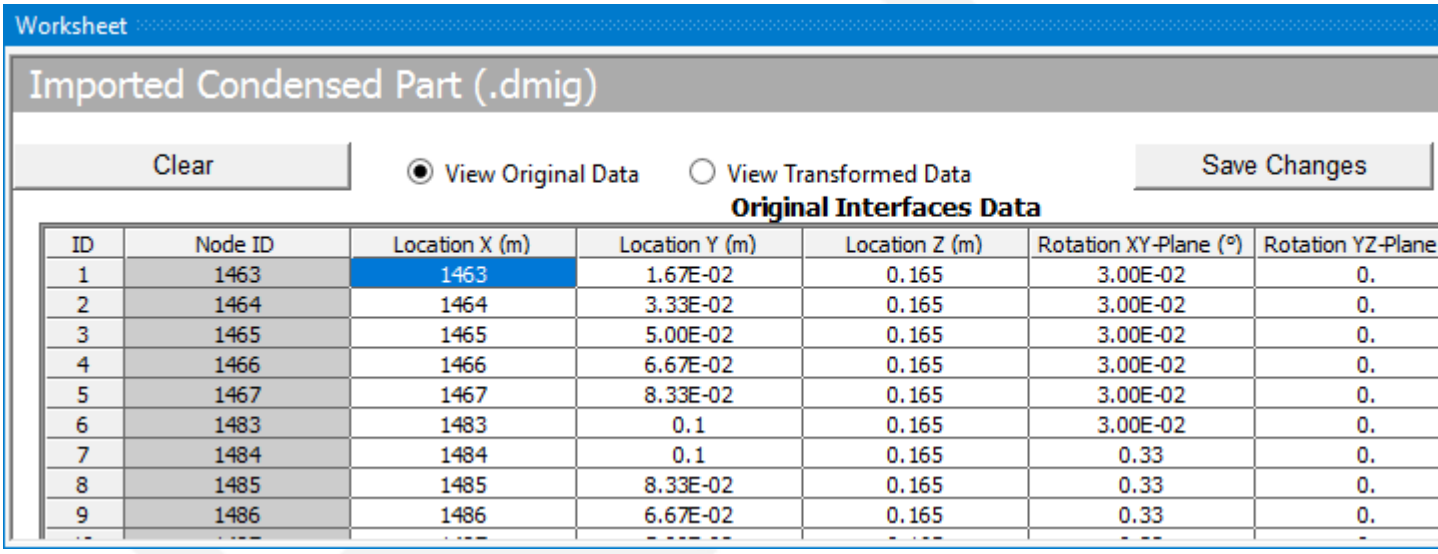
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	Rotation XY-Plane (°)	Rotation YZ-Plane
1	1463	0.	0.	0.	0.	0.
2	1464	0.	0.	0.	0.	0.
3	1465	0.	0.	0.	0.	0.
4	1466	0.	0.	0.	0.	0.
5	1467	0.	0.	0.	0.	0.
6	1483	0.	0.	0.	0.	0.
7	1484	0.	0.	0.	0.	0.
8	1485	0.	0.	0.	0.	0.
9	1486	0.	0.	0.	0.	0.

Review the pasted data and then select the **Save Changes** button.



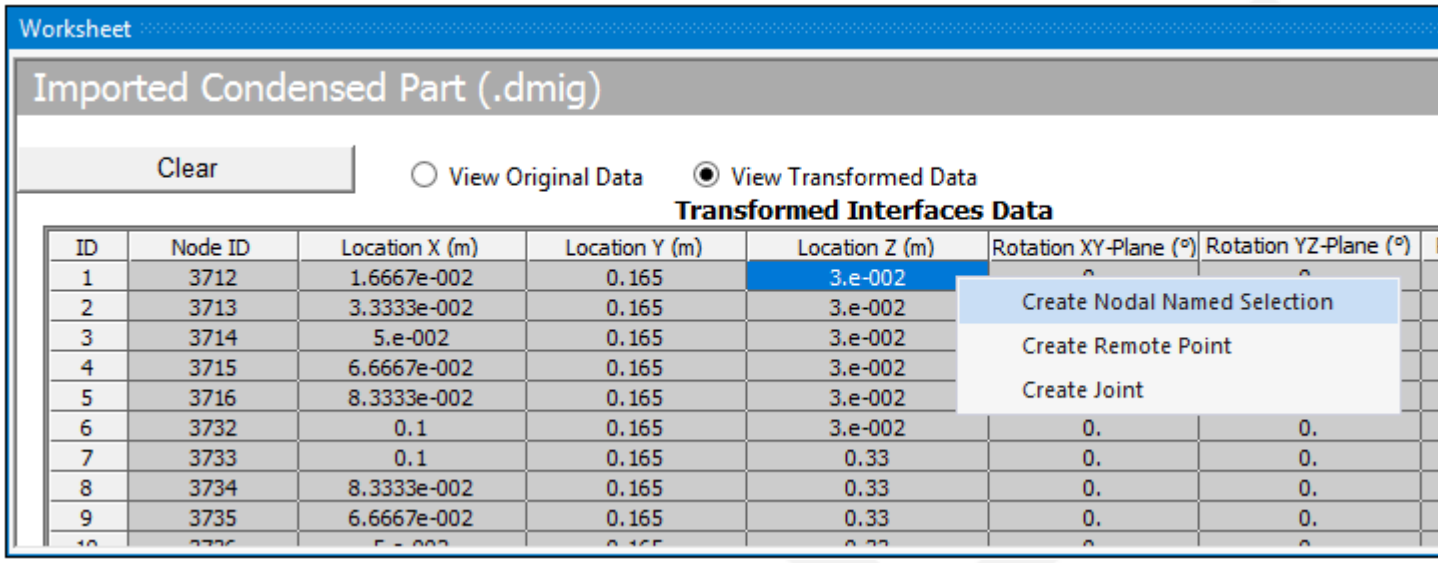
ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	Rotation XY-Plane (°)	Rotation YZ-Plane
1	1463	1463	1.67E-02	0.165	3.00E-02	0.
2	1464	1464	3.33E-02	0.165	3.00E-02	0.
3	1465	1465	5.00E-02	0.165	3.00E-02	0.
4	1466	1466	6.67E-02	0.165	3.00E-02	0.
5	1467	1467	8.33E-02	0.165	3.00E-02	0.
6	1483	1483	0.1	0.165	3.00E-02	0.
7	1484	1484	0.1	0.165	0.33	0.
8	1485	1485	8.33E-02	0.165	0.33	0.
9	1486	1486	6.67E-02	0.165	0.33	0.

Once saved, right-click the object and select **Import Condensed Part**. This action imports the data, and automatically displays **Transformed Interface Data** that you can use to options to create Nodal Named Selections, Remote Points, or Joints for the selected interface.

Note:

- When you create a Nodal Named Selections or a Remote Point, the application automatically scopes to the object for the selected interface nodes. For a generated Remote Point, the **Behavior** property is automatically set to **Coupled**.
- When you create a Joint, the application creates a Joint whose **Reference Side** is automatically scoped to the selected interface nodes with the **Direct Attachment** option specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



ID	Node ID	Location X (m)	Location Y (m)	Location Z (m)	Rotation XY-Plane (°)	Rotation YZ-Plane (°)
1	3712	1.6667e-002	0.165	3.e-002		
2	3713	3.3333e-002	0.165	3.e-002		
3	3714	5.e-002	0.165	3.e-002		
4	3715	6.6667e-002	0.165	3.e-002		
5	3716	8.3333e-002	0.165	3.e-002		
6	3732	0.1	0.165	3.e-002	0.	0.
7	3733	0.1	0.165	0.33	0.	0.
8	3734	8.3333e-002	0.165	0.33	0.	0.
9	3735	6.6667e-002	0.165	0.33	0.	0.

Expansion Pass

The solution of an analysis containing a **Condensed Part** immediately produces the results at the Condensed Part interface, that is, at the master nodes. In order to obtain results on the original bodies in the **Condensed Part** an optional calculation called an "Expansion Pass" is necessary. Mechanical furnishes an Expansion Settings object for this purpose.

Important:

To perform an Expansion Pass, you must set the **Keep Files For** property to **MAPDL Expansion** for the **Condensed Part** object.

When you expand results on selected **Condensed Parts** in the **Expansion Settings Worksheet**, the application uses the settings of the **Distribute Solution** and the **Max number of utilized cores** options defined in the **Advanced Settings** of the Solve Process Settings (p. 1804) dialog.

Note:

- You cannot currently run expansions for Random Vibration (p. 524) and Response Spectrum (p. 533) analyses.
- For a Rigid Dynamics analysis, additional results can be produced by the Generation Pass to allow a faster expansion. To enable this faster expansion, set the **Condensed Part Expansion** field in the **Details** pane of the **Expansion Settings** object to On Demand. Then insert Stress or Displacement results under the **Solution** object and evaluate them.

Using the **Worksheet** on the **Expansion Settings** object you can request that results be expanded within each **Condensed Part** separately and incrementally. Keep in mind that any unexpanded Condensed Parts will not display in results or animations. Once an expansion is requested, it will be automatically performed upon evaluating an affected result. It is possible to run expansions by themselves, which will obsolete any affected results. It is possible to run expansions separately using

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Run Expansions Only, and to clean their generated data, both of which will obsolete any affected results.

Note:

Do not use the **Worksheet** if you set the **Condensed Part Expansion** field in the details panel of the **Expansion Settings** object to **On Demand** in a Rigid Dynamics analysis. Making any selections in the Worksheet will cancel the on demand expansion.

The **Expansion Settings** object is automatically inserted when the **Condensed Geometry** object is inserted into the **Outline**.

Note:

- Unless you do an expansion, you may not detect the minimum or maximum results since they may reside within the condensed part. Review results carefully.
- Commands object Snippets targeting the Mechanical APDL solver are not supported.
- Velocity and Acceleration results are not presently supported for flexible bodies.
- Motion analyses do not require an Expansion Pass, therefore Expansion Settings are not created for a Motion system.

Limitations

General

Note the following general limitations associated with the definition of Condensed Parts:

- The Master Degrees of Freedom Nodes (MDOFs) may be reduced by the solver during the:
  - Generation of a Condensed Part.
  - Solution of a Substructure Generation (p. 540) analysis

  This could be the result of a constraint defined on the MDOF nodes. The application reports this with a warning. In addition, in this case, there could be a discrepancy between Mechanical MDOFs and solver MDOFs. A warning message will also display while eliminating the redundant degrees of freedom.
- The application does not write Named Selections scoped to the interface nodes of a Condensed Part to Use Pass input file unless the Named Selection is used in the scoping of a nodal boundary conditions (Nodal Force, Nodal Pressure, Nodal Displacement and Nodal Orientation) in the Use Pass.
- For **Imported Condensed Parts**:
  - Do not support Rigid Dynamics or Motion analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - The application does not import load vectors from **Super Element Matrix** (`.dmig`) files. As a result, they cannot be applied to the Use pass.

  - When you import superelements from another system into your current system using an Imported Condensed Part object, and you change the Unit System of the Imported Condensed Part, the application does not automatically resize Interface Mesh nodes. They can be resized using **Rescale** option from **Annotation** (p. 79) group of **Display** tab.

  - Note the following when you are working with Model Assembly (p. 991):

    → If the **assembled system** includes imported condensed parts and you change the transfer property, **Number of Copies**, and update, the application may not import the exact copies of the geometry and/or source mesh.

    → The application does not currently support importing condensed parts that are included in the **source system**.

  - If you use the **Clear Generated Data** option on the mesh or an **Imported Condensed Part** object, the application also clears the scoping for joints created using the Imported Condensed Part interface. You must recreate the joints in this case.

  - The application only shows the outline of the superelement when the active object is an **Imported Condensed Part** or **Condensed Geometry**.

- If your analysis includes only Imported Condensed Parts and no geometry file, parameters and design points are not supported. However, you can import a model and suppress the bodies to use the Parameter (p. 2337) options..

- Reinforcements using the **Mesh Independent Method** are not supported with **Condensed Parts** and **Imported Condensed Parts**.

- **Remote Solve Manager (RSM)**

  You can employ RSM for the Generation and Use passes only. Solution processing for the Expansion Pass using the RSM is not supported. The application always performs the Expansion Pass process on your local machine.

  The Generation Pass uses the default queue selected in the Solve Process Settings (p. 1804) dialog. And, the application always performs the Generation Pass in synchronous mode regardless of the settings of the Advanced properties in the Solve Process Settings.

- You cannot scope result Probes to objects which are fully contained inside any **Condensed Part**. For example, a spring can be fully contained inside a **Condensed Part** if the topologies of both the reference and the mobile scoping are on bodies that belong to the **Geometry Selection** scoping of the same **Condensed Part**.

- When you have 1) active Expansion Settings in the Worksheet **and** 2) you set the **Scoping Method** property for a result to **Result File Item** (p. 2002), the only supported option for the **Item Type** property is **Component Name**.

  Also for this specification, **Material IDs** and **Element Name IDs** are not available for the **Material and Element Type Information** selection on the **Solution Quantities and Result Summary** (p. 1961) page of the Worksheet.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For Harmonic Response analysis or a Transient analysis linked to an upstream Modal analysis, the **On Demand Expansion Options** property is available (see Options for Analyses (p. 1447)). If you set this property to **Yes**, either manually or from the **Program Controlled** setting, the condensed part results are always expanded. As a result, **Expansion Settings** specifications become redundant.
- Stress and strain cannot be evaluated on **Condensed Parts**.
- Contact and Target contact may get flipped internally to meet Condensed Part requirements.
- The edge-edge contact between a shell body and beam body is not supported when the **Target** side of the contact pair is on the shell body that belongs to a Condensed Part.
- Elements cannot use Lagrange multipliers, such as interior contact regions cannot using the Normal Lagrange Formulation.
- The following are not supported:
  - Force Reaction and Moment Reaction probes (Expansion Pass only)
  - Pipe Idealization
  - Spot Weld connections
  - Symmetry Region scoped to a Condensed Part
  - Cyclic Symmetry
  - Linear Periodic Symmetry
  - The **Detect Condensed Part Interface** option does not detect imported loads or connections, however, they can be added manually. For the imported objects that are associated with condensed parts, use the promote feature in the worksheet of the imported objects in order to be able to perform interface detection.
- For Joints internal to a Condensed Part, the application only supports Fixed (p. 1298) joints and Bushing (p. 1303) joints (with **Formulation** property set to **Bushing**).
- The Rotational Velocity and Thermal Condition loads are only supported on a non-condensed part.
- Once you have scoped and generated Condensed Parts, any mesh modifications made to the parts of the model that **do not** belong to the generated Condensed Parts, do not require you to regenerate the Condensed Parts. That is, Condensed Parts maintain an up-to-date status following mesh changes to other model parts. However, there are a few exceptions. If you have imported your model using external systems, **External Model** and/or **Mechanical Model** systems, any geometry modifications to geometries not specified as a Condensed Part, do cause the Condensed Parts to become obsolete and require regeneration. In addition, the use of features such as **Mesh Edit**, **Element Orientation**, or **Mesh Numbering** objects cause up-to-date Condensed Parts to become obsolete and require regeneration.
- Condensed Parts require regeneration if you make any geometry modifications in the CAD application. This applies even if you have the **Smart CAD Update** option (SpaceClaim only) of the **Advanced Geometry Options** in **Geometry** cell properties of the Project Schematic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

See the Geometry Preferences section in the CAD Integration documentation for more information.

- When importing `.mechdat` files, the **Solution** object displays with an unsolved state.
- The application does not display finite element connection lines (p. 1840) for any object scoped to the bodies included in the definition of a **Condensed Part**.

Note:

For additional restriction information, see the MATRIX50 Element Description in the *Mechanical APDL Element Reference* as well as the Assumptions and Restrictions (within Superelement) topic in the *Substructuring Analysis* section of the *Mechanical APDL Theory Reference*.

Rigid Dynamics Analyses

Note the following specific limitations for Rigid Dynamics analyses:

- You can apply loading conditions only to the interfaces of Condensed Parts. Remote Displacements and Remote Forces applied to the Condensed Parts using geometry selection, must match the scoping of an interface of the Condensed Part, or be applied directly via Remote Point selection. Inertial loads are automatically applied on all Condensed Parts by the Rigid Dynamics Solver.
- Contact cannot be used to connect Condensed Parts with the rest of the model.
- For On Demand Expansion, the supported results are Total Deformation, Directional Deformation, and all Stress and Strain results **except** the vector Principal. The limitations on these supported results are:
  - Only global CS evaluation is supported.
  - For the stress and strain results, only the averaged results display is supported.

Modal Analyses

Note the following specific limitations for Modal analyses:

- Pre-stress analysis is not supported with Condensed Parts.
- Rigid bodies are supported but it is best that at least one body be Flexible.
- **Rotational Velocity** and **Thermal Condition** loads cannot be scoped to a Condensed Part.
- Fracture Analysis is not supported when Condensed Parts are defined in the model.

Random Vibration and Response Spectrum Analyses

You cannot currently run expansions for Random Vibration and Response Spectrum analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Best Practices

General

1. When a Condensed Part Interface includes faces, edges, vertices, or nodes that overlap with other interfaces where constraint equations are applied, such as remote points, boundary conditions scoped to remote points, MPC-based contact (p. 1243), etc.), the solver may remove some master node DOFs in the generation pass. This may interfere with the use pass solve as well as postprocessing.

2. If you define a joint between a rigid body and flexible body, you need to specify the Condensed Part on the Reference side in order to get accurate results.

Importing Condensed Parts

Use the right-click options of the Worksheet to automatically create node-based Named selections for use in the analysis to scope features such as Contact, Remote Points, Joints, etc.

Remote Points need to be connected to the interfaces of the Imported Condensed parts. To avoid rigid body modes, set the **Behavior** property of the **Remote Point** to **Coupled**. This applies to other connection types, such as a Joint that is also connected through remote points.

It is also recommended to use Imported Condensed part interface on the Reference/Source side while connecting the superelements to the rest of the model.

Rigid Dynamics Analyses

Review the following recommendations when you are defining Condensed Parts.

Analysis Settings

For the **Solver Controls** (p. 1428) property **Time Integration Type** of the Analysis Settings, the default option is Program Controlled. This option automatically uses the Implicit Stabilized Generalized Alpha setting. Ansys recommends the use of generalized implicit time integration. The Runge-Kutta explicit time integration option may lead to small time steps and consequently significant computation effort during the Use Pass. Older models (prior to R17.0) may be specified to use a Runge-Kutta option by default. If so, the processing requirements will be significant during the Use Pass. It is also recommended to set **Energy Accuracy Tolerance** property to **Program Controlled** for better performance during implicit time integration.

Note:

For additional information, see the **Analysis Settings** Help for the Nonlinear Controls for Rigid Dynamics Analyses (p. 1472).

The number of stored results points has a direct effect on the computational efforts required by Expansion Pass. It is therefore recommended that you specify the **Store Results At** property of the **Output Controls** (p. 1473), in the Analysis Settings, as **Equally Spaced Points** and set the **Value** property accordingly to limit the number of result time points and to reduce the processing requirements of the expansion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analyses Using the Mechanical APDL Solver

You should use the same setting for the number of cores for the generation process for all condensed parts as well as your solution and expansion(s) as defined by the **Distribute Solution** option in the **Advanced Settings** of the Solve Process Settings (p. 1804) dialog.

Static Analysis From Rigid Dynamics Analysis

You can perform a Rigid Dynamics Analysis (p. 550) and then change it to or link it to a downstream Static Structural Analysis (p. 652) analysis for the purpose of determining deformation, stresses, and strains (which are not available in the Rigid Dynamics analysis).

Note:

The Motion Loads (p. 622) ACT extension allows you to more easily apply the loads created in a Rigid Dynamics analysis on the flexible bodies in a Static Structural analysis especially when the model contains joints with translational joint degrees of freedom.

Creating an Analysis System

1. From the toolbox, drag and drop a **Rigid Dynamics** template onto the project schematic. Follow the procedure for creating a Rigid Dynamics Analysis (p. 550) analysis. Apply forces and/or drivers, and insert any valid solution result object(s).
2. Specify the time of interest in the tabular data table or in the Graph window.
3. Select a solution result object and click the right mouse to display the popup menu. Select **Export Motion Loads** and specify a load file name.
4. In the project schematic, duplicate the **Rigid Dynamics** analysis system. Replace the duplicated analysis system with a **Static Structural** analysis system.

   Note:

   If you do not need to keep the original **Rigid Dynamics** analysis, you can replace it with the **Static Structural** analysis system.

   ---

5. Edit the **Static Structural** analysis (using **Model**, **Edit**) by suppressing all parts except the desired part for the **Static Structural** analysis.
6. Change the **Stiffness Behavior** of the part to be analyzed from **Rigid** to **Flexible**.
7. Change mesh solver preference to be Ansys Mechanical instead of Ansys Rigid Dynamics.
8. Delete or suppress all loads used in the **Rigid Dynamics** analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

9. Import the motion loads that were exported from the **Rigid Dynamics** analysis. Highlight the **Static Structural** branch and then right mouse click, **Insert**> **Motion Loads...**.

   Note:

   Moments and forces created for the static structural analysis can be in an invalid state if all three components of the force/moment are almost equal to zero.

10. Delete the result objects and add new ones.

11. Solve the single part model with the static structural analysis and evaluate the results.

Point to Remember

It is important that you create the **Static Structural** analysis after the **Rigid Dynamics** analysis is finished and the export load is done.

Submodeling

The Submodeling feature in Mechanical enables you to import the solution output of an upstream (source) analysis and apply that output as boundary conditions in the downstream (target) system in order to more closely analyze a region of interest. The application uses the results for your entire model as the imported data. The application interpolates the data for the geometry region you specify as the submodel.

In order to use this feature, you need to prepare your submodel in the appropriate modeling application and understand where on the geometry the loading will be applied.

Note:

For nonlinear, load-history-dependent problems (for example, when plastic materials exist), you must cut boundary conditions from the coarse model at multiple substeps to simulate the load history dependency in the fine-mesh model analysis. The more boundary cutting you do, the more accurate are the results of the fine-mesh model analysis.

During the data transfer process, the application creates a Submodeling object in Mechanical. This object is essentially the same as an Imported Load (Group).

Note:

If you want to import results from an upstream system, Ansys recommends that you instead import data using the **Imported Load (Group)** object.

Mechanical allows submodeling for structural (stress) and thermal analyses. In a thermal analysis, the temperatures calculated on the cut boundary of the coarse model are specified as boundary conditions for the submodel.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Take the time to review the *Submodeling* section of the *Advanced Analysis Guide* for a more detailed technical description of the feature and the principles in use. Also see the Shell-to-Solid Submodels (p. 818) section for the specific requirements for submodeling shells.

Restrictions

The following restrictions apply to submodeling:

- Supported for the Mechanical APDL solver only.
- The principle behind submodeling assumes that the cut boundaries are far enough away from the stress concentration region. You must verify that this assumption is adequately satisfied.

Benefits

Aside from the obvious benefit of yielding more accurate results in a region of your model, the submodeling technique has other advantages:

- It reduces, or even eliminates, the need for complicated transition regions in solid finite element models.
- It enables you to experiment with different designs for the region of interest (different fillet radii, for example).
- It helps you in demonstrating the adequacy of mesh refinements.

Application

See the following topics for additional information and the steps to perform submodeling in structural and thermal analyses:

- Structural Submodeling Workflow (p. 808)
  - Beam-to-Solid/Shell Submodels (p. 813)
- Thermal Submodeling Workflow (p. 816)
- Shell-to-Solid Submodels (p. 818)

Structural Submodeling Workflow

Use the workflow below to perform a submodeling analysis using linked structural systems. You should review the Data Transfer Mesh Mapping content in Appendix B: Data Transfer Mapping and Validation (p. 2389) of the Help. This section describes the various Details view properties associated with the loading types for the submodeling feature. Refer to this section as needed during your analysis.

Note:

You can add a template for the linked structural systems by creating your own template.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Supported Boundary Conditions

A structural analysis supports the following imported load options:

Body Temperature

This option inserts an **Imported Body Temperature** (p. 1740) object when you transfer temperatures from the coarse analysis.

Cut Boundary Constraint

This option inserts an **Imported Cut Boundary Constraint** (p. 1745) object when you transfer displacements for Solid- or Shell-Solid submodeling and displacements and rotations for Shell-Solid submodeling. Also see the **Imported Displacement** (p. 1746) section.

Cut Boundary Remote Force (Beam to Shell or Solid Only)

This option inserts an **Imported Cut Boundary Remote Force** (p. 1746) object in order to map forces and moments from beam-based coarse models to shell or solid submodels. These loading conditions are generated from the results defined on the nearest beam node of coarse model.

Important:

In order to add beam forces to the results file and make them available for your submodel, you need to set the **Nodal Forces** property (Analysis Settings>Output Controls (p. 1473)) to **On** in the coarse model.

Cut Boundary Remote Constraint (Beam to Shell or Solid Only)

This option inserts an **Imported Cut Boundary Remote Constraint** (p. 1744) object in order to map remote displacements and rotational forces from beam-based coarse models to shell or solid submodels. These loading conditions are generated from the results defined on the nearest beam node of coarse model.

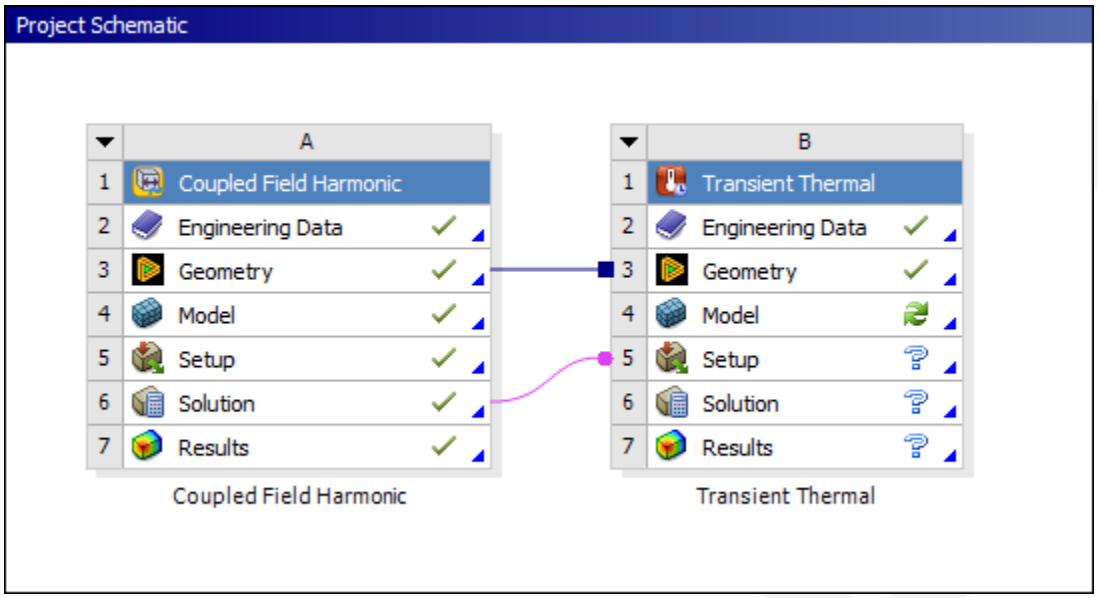
Structural Analysis Submodeling Workflow

1. On the Workbench **Project** page, create and complete (solve) a static or transient structural analysis. Perform all of the steps to set up and analyze the model. Specify mesh controls, boundary conditions, and solution settings as you normally would and solve the analysis.

   To easily identify this initial model, this example uses the name "Coarse" to identity the upstream system. This does not mean that the mesh refinement is coarse, only that it is relatively coarse compared to the "Submodel."

2. Create a new **Static Structural** or **Transient Structural** analysis on the **Project** page. Link the **Solution** cell of the upstream onto the **Setup** cell of the downstream system. As required, you can also link the **Engineering Data** and **Geometry** cells.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Double-click the downstream system's **Setup** cell to open Mechanical. The application automatically adds a **Submodeling** object to the system's tree. This object references the upstream analysis' **Solution** object in parenthesis, for example, **Submodeling "(A6)"**.

4. Select the **Submodeling** object and add the desired imported conditions by:

   - Selecting the appropriate load from the **Imported Loads** drop-down menu on the **Environment** Context tab.

     Or...

   - Right-clicking on the **Submodeling** object and selecting **Insert** and then selecting the appropriate load from the context menu.

5. Now, you need to select the appropriate cut-boundaries from the geometry using either the **Geometry** or the **Named Selection** scoping option. The scoping options for the imported boundary conditions are described below.

Load Type	**3D Solid Analyses**	**2D Shell Analyses**
Imported Cut Boundary Constraint	Face, Edge, or Nodes	Face, Edge, or Node
Imported Cut Remote Force	Face or Edge	Edge
Imported Boundary Remote Constraint	Face or Edge	Edge

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Thermal Condition	Body or Nodes	Body or Nodes

Note:

- You cannot mix the scoping of surface bodies with other geometry types.
- Node-based Named Selections are not valid for transferring temperatures for Shell-Shell submodeling.

6. The application automatically populates the **Transfer Key** property. Options include:

Shell-Shell

The application selects this option for a shell-based geometry in the upstream system.

Note the following for **Shell-Shell** submodeling:

- You have the option to import **Displacements**, **Rotations**, or **Both** using the **Sub Type** property.
- You have the option to import temperatures from the **Top/Bottom** face or the **Middle** shell plane using the **Shell Face** option. The **Top/Bottom** option uses the temperature on both the top and bottom shell face to calculate the temperature on a target node, whereas the **Middle** option only uses the temperature at the middle shell plan.

Solid-Solid

The application selects this option for a solid body geometry in the upstream system.

Shell-Solid

You may select this **Transfer Key** option for Shell-to-Solid (p. 818) submodels.

For **Shell-Solid** submodeling, you have the option to import temperatures on **Top/Bottom** face or the **Middle** shell plane using the **Shell Face** option. The **Top/Bottom** option calculates and applies the temperatures on the top and bottom face independently, whereas the **Middle** option calculates the temperature at the middle shell plane and applies it across the thickness of the shell.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Beam-Shell/Solid

The application selects this option for a beam-based geometry in the upstream system.

Important:

If your submodel includes only forces and moments and has no other constraints, you must set the **Weak Springs** (p. 1433) property (Analysis Settings>Solver Settings (p. 1428)) to **On**.

Note:

- When you set the **Transfer Key** property to **Shell-Shell** or **Shell-Solid**, only shell bodies are selected from the upstream analysis.
- When the **Transfer Key** is set to **Beam-Shell/Solid**, only beam bodies are selected from the upstream analysis.
- When you set the **Transfer Key** property to **Solid-Solid**, the values on the middle shell plane of shell bodies are used for mapping.
- Mapping Validation is not supported when the **Transfer Key** property is set to **Shell-Solid** or **Beam-Shell/Solid**.

---

7. As needed, modify **Details** view properties. See Appendix B: Data Transfer Mapping and Validation (p. 2389) for additional information.
8. Select your defined imported load, right-click the object, and click **Import Load**. When the load successfully imports, a plot of the mapped values displays in the **Geometry** window.

   For displacement loads, the following data is available for viewing:

   - Displacement
   - Rotations (**Transfer Key** setting of **Shell-Shell** only)

   Note:

   - When multiple data types are available for viewing, the appropriate data type can be chosen in the **Data** field the **Graphics Controls** category.
   - Contours plots of the magnitude (**Total**) or X/Y/Z component can be viewed by changing the **Vector Component** option in the details pane. Defaults to a vector plot (**All**).
   - The range of data displayed in the **Geometry** window can be controlled using the properties of the **Legend Controls** category. See Imported Boundary Conditions (p. 1736) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For temperature loads on bodies when the **Transfer Key** property is set to **Shell-Shell**, the **Shell Face** option becomes available under **Graphics Controls**. This property enables you to view the data on top, middle or the bottom face of the shell.
- The data displayed on the middle face is calculated by averaging the interpolated data on the top and bottom face.

9. To activate or deactivate the load at a step, highlight the specific step in the **Graph** or **Tabular Data** window, and select **Activate/Deactivate**.

   See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

10. Define any other loads and boundary conditions, specify load step options, and obtain the submodel solution.

11. The final step is to verify that the cut boundaries of the submodel are far enough away from the concentration. You can do this by comparing results (stresses and so on) along the cut boundaries with those along the corresponding locations of the coarse model. If the results are in good agreement, it indicates that proper cut boundaries have been chosen; otherwise, you will need to recreate and reanalyze the submodel with different cut boundaries further away from the region of interest.

Note:

If the upstream (coarse) system is modified and re-solved after importing the load, a refresh operation on the Submodel system's Setup cell is required to notify Mechanical that source data has changed and re-import is required. Alternatively, the source data can be refreshed using the right-click operation on the Submodeling folder and choosing the **Refresh Imported Load** option.

Beam-to-Solid/Shell Submodels

For the Beam-to-Shell or -Solid submodeling technique, the coarse model is a beam model, and the submodel is either a 3D solid model or 3D shell model.

During the process of beam submodeling:

- The application determines the beam node that is nearest to each cut face (for beam-solid) or edge (for beam-shell) on the submodel. Then, depending upon the desired submodeling method, the application calculates either forces and moments or displacements and rotations from the coarse analysis's beam solution. If the scoped cut boundaries include connected faces or edges, the application uses the result from the nearest common beam node for each group of connected boundaries.
- The application applies the extracted forces and moments or displacements and rotations, from the beam nodes to the submodel by generating remote loads. The application posi-

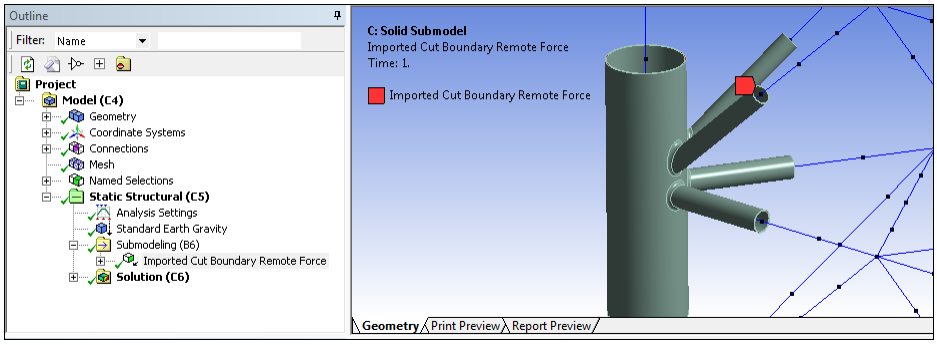
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions these remote loads at the nearest beam node, but scoped to the appropriate cut face or edge on the submodel.

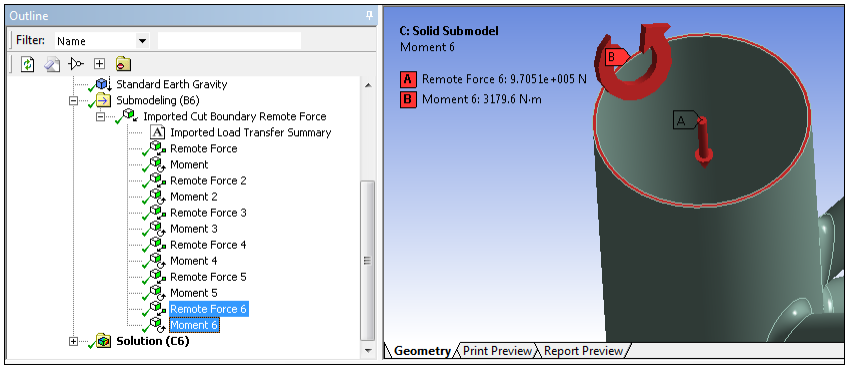
- Beam to shell or solid submodeling transfers can transfer the following from a beam model:
  - Forces and moments using the Cut Boundary Remote Force option. Using this method, each remote force and moment pair share a remote point that is by default, deformable. See examples below.
  - Displacements and rotations using the Cut Boundary Remote Constraint option. Using this method, the application applies displacements and rotations using remote displacement objects that by default, are rigid. See examples below.
- The generated remote loads are by default, read-only, however, you can change this setting using the Read Only property for generated load.

Loading Condition Examples

Here is an example of imported forces and moments using the **Cut Boundary Remote Force** import option. Note the beam nodes at the cut face locations (as well as all beam node locations). The remote points are located at the nearest beam node.



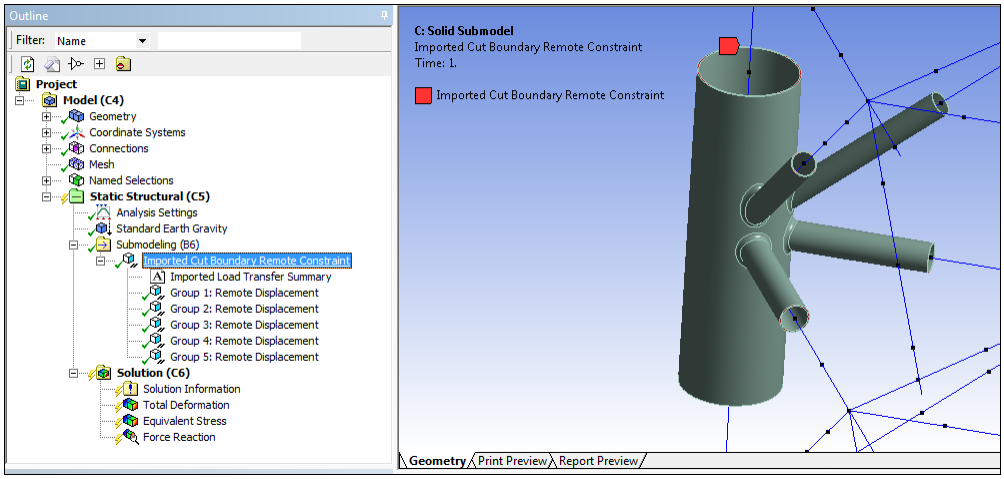
This example illustrates the force and moment loads for a specific cut face.



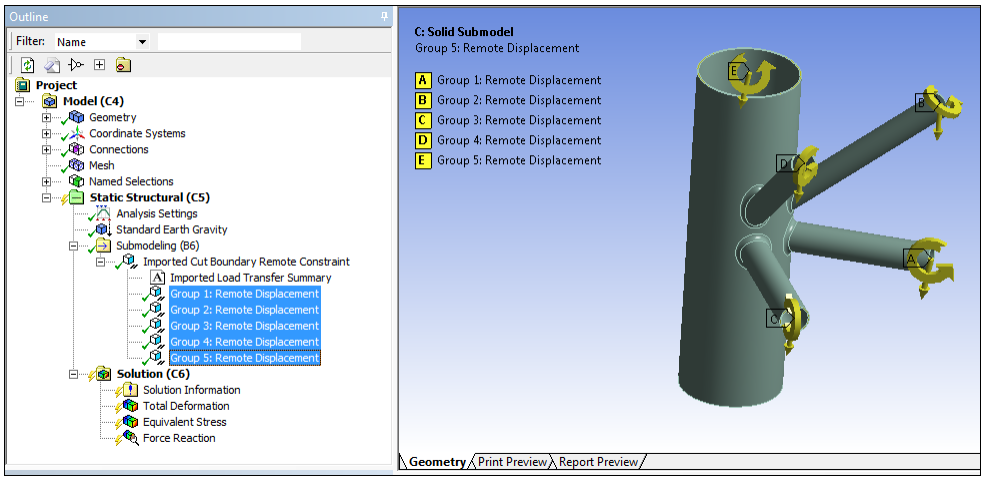
This example illustrates the **Cut Boundary Remote Constraint** import option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

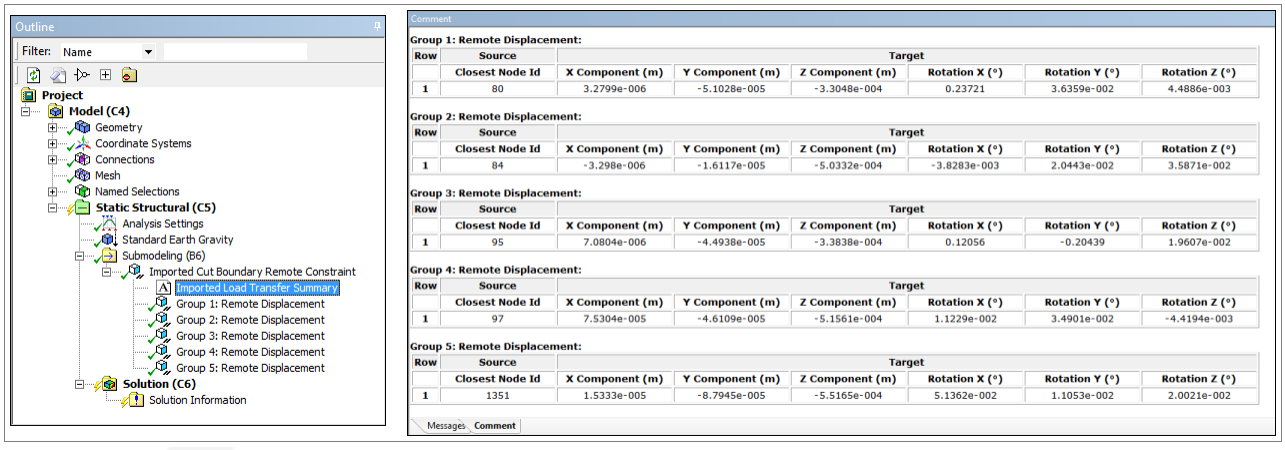
Published: 2023-10-23



Here you can see the remote displacements for the imported loading for the specific cut faces.



In addition, note the information provided by the **Imported Load Transfer Summary** object. This important tabular information enables validation of the data transfer. In particular, the nodes used to map the imported loads as well as the target coordinates.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Thermal Submodeling Workflow

Use the workflow below to perform a submodeling analysis using linked thermal systems. You should review the *Data Transfer Mesh Mapping* content in Appendix C (p. 2389) of the Help. This section describes the various Details view properties associated with the loading types for the submodeling feature. Refer to this section as needed during your analysis.

Supported Boundary Conditions

Thermal submodeling analyses support the Imported Temperature (p. 1756) only.

Thermal Analysis Submodeling Workflow

1. On the Workbench **Project** page, create and complete (solve) a steady-state thermal or transient thermal analysis. Perform all of the steps to set up and analyze the model. Specify mesh controls, boundary conditions, and solution settings as you normally would and solve the analysis.

   To easily identify this initial model, this example uses the name "Coarse" to identity the upstream system. This does not mean that the mesh refinement is coarse, only that it is relatively coarse compared to the "Submodel."

2. Create a new **Steady-State Thermal** or **Transient Thermal** analysis on the **Project** page. Link the **Solution** cell of the upstream onto the **Setup** cell of the downstream system. As required, you can also link the **Engineering Data** and **Geometry** cells.

Note:

- You can add a template for the linked thermal systems by creating your own template.
- Data can be transferred from a 2D coarse model to a 3D submodel. The settings for 2D projection of target mesh nodes can be specified in Appendix B: Data Transfer Mapping and Validation (p. 2389).

3. Double-click the downstream system's **Setup** cell to open Mechanical. The application automatically adds a **Submodeling** object to the system's tree. This object references the upstream analysis' **Solution** object in parenthesis, for example, **Submodeling "(A6)"**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

An imported temperature object is automatically inserted under the **Submodeling** folder to represent the transfer. To add additional Imported Temperature objects, right-click the **Submodeling** folder and select the appropriate load from the **Insert** context menu.

4. Now, you need to select the appropriate cut-boundaries from the geometry using either the **Geometry** or the **Named Selection** scoping option.

   The Imported Temperature (p. 1756) boundary condition supports Face, Edge, and Node selections for 3D solids and Edge and Node selections for 2D shells.

   Note:

   You cannot mix the scoping of surface bodies with other geometry types.

5. The application automatically populates the **Transfer Key** property. Options include:

   Shell-Shell

   The application selects this option for a shell-based geometry in the upstream system.

   Solid-Solid

   The application selects this option for a solid body geometry in the upstream system.

   Shell-Solid

   You may select this **Transfer Key** option for Shell-to-Solid (p. 818) submodels.

   Beam-Shell/Solid

   The application selects this option for a beam-based geometry in the upstream system.

   Note:

   - When you set the **Transfer Key** property to **Shell-Shell** or **Shell-Solid**, only shell bodies are selected from the upstream analysis.
   - When the **Transfer Key** is set to **Beam-Shell/Solid**, only beam bodies are selected from the upstream analysis.
   - Mapping Validation is not supported when the **Transfer Key** property is set to **Shell-Solid** or **Beam-Shell/Solid**.
   - If you are using the Material Assignment (p. 1049) feature on source bodies that are different (shell and beam), you could experience mapping errors. The application may skip a source body during the mapping process. To address this issue, use the feature on the bodies individually – do not mix body types.
   - The application only considers beam and shell section type elements from the source data. It ignores all other section types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The range of data displayed in the graphics window can be controlled using the **Legend controls** options. See Imported Boundary Conditions (p. 1736) for additional information.

6. As needed, modify **Details** view properties. See Appendix C (p. 2389) for additional information.
7. Right-click the **Imported Load** object and click Import Load to import the load. When the load has been imported successfully, a plot of the mapped values will be displayed in the Geometry window.
8. To activate or deactivate the load at a step, highlight the specific step in the **Graph** or **Tabular Data** window, and choose **Activate/Deactivate***at this step*!

   See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.
9. Define any other loads and boundary conditions, specify load step options, and obtain the submodel solution.
10. The final step is to verify that the cut boundaries of the submodel are far enough away from the concentration. You can do this by comparing results (stresses and so on) along the cut boundaries with those along the corresponding locations of the coarse model. If the results are in good agreement, it indicates that proper cut boundaries have been chosen; otherwise, you will need to recreate and reanalyze the submodel with different cut boundaries further away from the region of interest.

Note:

If the upstream (Coarse) system is modified and re-solved after importing the load, a refresh operation on the Submodel system's Setup cell is required to notify Mechanical that source data has changed and re-import is required. Alternatively, the source data can be refreshed using the right-click operation on the Submodeling folder and choosing the **Refresh Imported Load** option.

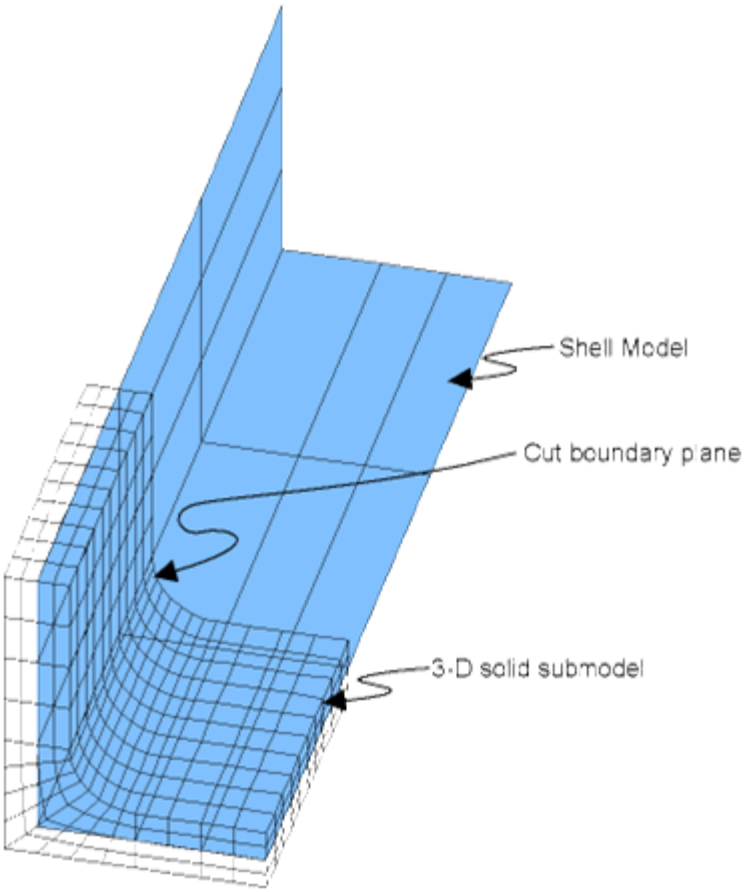
Shell-to-Solid Submodels

In the shell-to-solid submodeling technique, the coarse model is a shell model, and the submodel is a 3D solid model, as shown in this example:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Figure 17: 3D Solid Submodel Superimposed on Coarse Shell Model

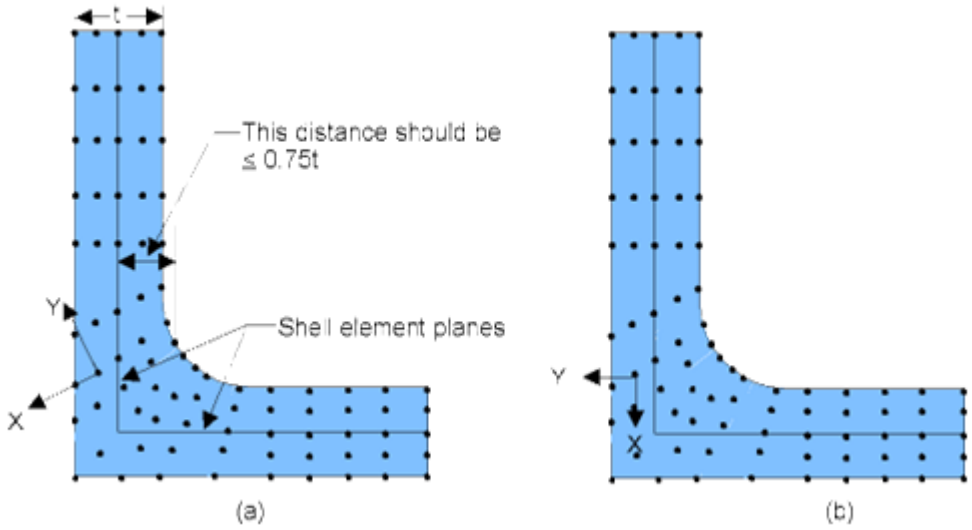


The procedure for shell-to-solid submodeling is essentially the same as that for solid-to-solid submodeling, with these exceptions:

- Shell-to-solid submodeling is activated by setting the **Transfer Key** to Shell-Solid in the Imported Load details view.
- Cut boundaries on the submodel are the end planes that are normal to the shell plane (see Figure 18: Node rotations (a) before mapping command, (b) after mapping command (p. 820)).
- To determine the degree-of-freedom values at a cut-boundary node, the program first projects the node onto the nearest element in the shell plane. The degree-of-freedom values of this projected point are then calculated by interpolation and assigned to the corresponding node.
- In a structural analysis, only translational displacements are calculated for the cut-boundary nodes, but their values are based on both the translations and rotations of the projected point. Also, the node is rotated such that the nodal UY direction is always perpendicular to the shell plane, as shown in Figure 18: Node rotations (a) before mapping command, (b) after mapping command (p. 820). A UY constraint is calculated only for nodes that are within 10 percent of the average shell element thickness from the shell plane, preventing overconstraint of the submodel in the transverse direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 18: Node rotations (a) before mapping command, (b) after mapping command



Note:

- If you are using the Material Assignment (p. 1049) feature on source bodies that are different (shell and beam), you could experience mapping errors. The application may skip a source body during the mapping process. To address this issue, use the feature on the bodies individually – do not mix body types.
- The application only considers beam and shell section type elements from the source data. It ignores all other section types.

System Coupling

You can use System Coupling to perform coupled simulations between multiple physics solvers, coupling active co-simulation participants and/or importing static data from a external data source. For example, you can run Mechanical and CFX in a coupled analysis or import data from an output file into a Fluent analysis. Once the physics and coupling setups are complete, the coupled analysis is executed and managed by System Coupling.

System Coupling can be used in the following contexts:

System Coupling's User Interfaces:

System Coupling is run from its graphical user interface (GUI) or its command-line interface (CLI). System Coupling's interfaces provide enhanced control over coupled simulation processes, including automatic starts and restarts for participants, the ability to manipulate System Coupling's data model, and an interactive solution workflow.

In these contexts, you still set up participant physics in the participant's user interface, but you'll perform the coupled analysis — starting System Coupling, loading participants, specifying values for coupling-related analysis settings, and automatically starting participants —

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

using System Coupling's GUI or CLI. Alternatively, if a coupled analysis setup was exported from Workbench, you can open it and execute it in the GUI or CLI.

A list of supported coupling participants is given in Supported Coupling Participants in the *System Coupling User's Guide*.

For more information, see Using System Coupling's User Interfaces in the *System Coupling User's Guide*.

System Coupling in Workbench:

System Coupling is run from the Workbench interface.

In this context, you'll connect the **Setup** cell from a Workbench analysis system to the **Setup** cell for the System Coupling component system, signaling that the Mechanical solver will act as a co-simulation participant in a coupled analysis. Most of the coupling-related analysis settings are made using the System Coupling system's **Setup** cell.

Once the physics and coupling setups are completed in Workbench, you can either execute the coupled analysis in Workbench, or export the setup for execution in one of System Coupling's user interfaces.

A list of supported coupling participants is given in Supported Coupling Participants for System Coupling in Workbench.

For more information, see Using System Coupling in Workbench in the *System Coupling User's Guide*.

The few coupling-related settings that are required in the setup are described below in System Coupling Related Settings in Mechanical (p. 825).

Once the physics and coupling setups are complete, the coupled analysis is executed and managed by System Coupling.

Additional information can be found in the following sections:

Supported Capabilities and Limitations

Variables Available for System Coupling

System Coupling Related Settings in Mechanical

Using Higher-Order Meshes for Coupled Analyses

One-Way FSI Transfers Using System Coupling in Workbench

Coupled Field Co-Simulation Using System Coupling

Coupling Thin Surfaces in CFX

Restarting Structural Mechanical Analyses as Part of System Coupling

Running Mechanical as a Coupling Participant in System Coupling's GUI or CLI

Troubleshooting Two-Way Coupling Analysis Problems

Product Licensing Considerations when using System Coupling

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Supported Capabilities and Limitations

Mechanical supports the following capabilities when used in a System Coupling analysis:

- Data exchange across **System Coupling Region** (p. 1651) interfaces defined on regions in the Mechanical model. This boundary condition defines the interface between the fluid in the coupled participant system (for example, Fluent) and the solid in the Mechanical system.
- Data transfer regions are the regions upon which the **System Coupling Region** (p. 1651) condition is applied. In a coupled analysis, at each data transfer region, the following variables can be sent and received:
  - In a **coupled structural analysis**, Force and Displacement can be transferred at data transfer regions.
    - → Both Force and Force Density transfers are available. They are very similar, except that the Force Density variable uses follower effects within the Mechanical solution: as the structural geometry deforms within a time step, the Force rotates along with it. This can lead to a more robust solution, especially for soft materials such as hyperelastic solids.
    - → When the solver receiving the motion (such as Fluent) solves before or simultaneously with the solver sending the motion (such as Mechanical), then the Incremental Displacement transferred during the first coupling iteration of each coupling step is identically zero.
  - In a **coupled thermal analysis**, Heat Transfer Coefficients and Near Wall Temperatures, Temperatures, and Heat Flows can be transferred at data transfer regions.
    - → Heat Transfer Coefficient is also known as "convection coefficient."
    - → Near Wall Temperature is also known as "bulk temperature," or "ambient temperature."
  - In a **coupled thermal-structural analysis**, Mechanical can send Displacements along with only one Temperature or Heat Flow condition, and can in turn receive a Force load and either one Temperature or Heat Flow, or a combined Heat Transfer Coefficient and Reference Temperature.
  - In a **coupled thermal-structural-fluid analysis** (performed using a coupled field system (p. 830)), both Displacement and thermal data can be transferred on data transfer regions.

  For a listing of Mechanical's data transfer variables, see Variables Available for System Coupling (p. 824).
- Surface and volume meshes that include higher-order elements.
- Shared memory parallel mode. Note that convergence and therefore results will change between repeated runs of Mechanical in shared memory parallel mode. These changes will occur even if no setup changes were applied. The changes in the coupled analysis' convergence and results are due to the segregated solution algorithm used and the inherent sensitivity of the coupled physics problems being solved.
- Distributed parallel mode. Note that to run Mechanical in distributed parallel mode from within the Workbench interface, the working directory must be a shared network directory with the same path for all computer servers. Alternatively, the analysis can run in different working direct-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

ories on all servers if Mechanical is run as a coupling participant from one of System Coupling's user interfaces. For more information, see Running Mechanical as a Coupling Participant in System Coupling's GUI or CLI (p. 835).

- **SOLID** and **SHELL** elements. For a complete list of elements, see Load-Transfer Coupled Analysis -- Workbench: System Coupling in the *Coupled-Field Analysis Guide*.

- Structural convergence information and Result Tracker information are provided to System Coupling for display in the charts generated by System Coupling in Workbench.

  When using the Result Tracker in a System Coupling in Workbench analysis, note that **Kinetic Energy** and **Stiffness Energy** values are only computed at the end of a coupling step, and values of zero are reported for the intermediate coupling iterations. The **Kinetic Energy** and **Stiffness Energy** values reported in System Coupling are lagged, so the value reported at the start of a coupling step is actually the value corresponding to the end of the previous coupling step. The value corresponding to the last coupling step will not be reported in System Coupling.

- Custom restart files are available when using System Coupling's GUI or CLI. These files allow for the insertion of pre-solve command snippets into Mechanical's restart file. Mechanical APDL commands that are supported for coupled analysis restarts (such as **TIMINT**) may be used.

Note the following limitations when using Mechanical in a System Coupling analysis:

- System Coupling requires participants to use 3D meshes, with data transfer regions consisting of element faces within the 3D mesh. Data transfer regions cannot exist in 2D meshes (where the data transfer would be a line/curve). Line elements such as **BEAM** elements in Mechanical cannot form data transfer regions but may be included elsewhere in the Mechanical model.

- In a System Coupling setup, if you apply an external force or external heat flow on the same region as a **System Coupling Region** interface, this external variable will be ignored by the Mechanical APDL solver.

- Transfers of Force Density are not supported for:

  – Cases where Mechanical uses shell elements and Force Density is transferred to both sides of any shell.

  – Execution using System Coupling in Workbench. To make the FDNS input variable available in a coupled analysis created in Workbench, export the coupling setup so the analysis can be executed in System Coupling's GUI or CLI. For details, see Exporting a System Coupling Setup and Running an Exported System Coupling Setup in the *System Coupling User's Guide*.

- When Mechanical participates in a System Coupling analysis, only one load step can be defined in Mechanical. Loads can still vary as a function of time within this load step. Other operations that would normally require multiple load steps will require a System Coupling restart to be performed. For example, a pre-stressed analysis can be performed by executing a System Coupling simulation using the pre-stressing load conditions in Mechanical, then continuing the analysis by restarting System Coupling after making the necessary changes in Mechanical.

- Mechanical restarts are not supported for the transfer of thermal variables.

- Note that the internally computed contact damping is a function of the total number of substeps. The internal damping is reduced in subsequent substeps within a load step, and very little

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

damping is applied in the last substep. Therefore, the solution convergence pattern is different when solving a contact analysis that has only one substep (or a few substeps) per load step compared to an analysis having multiple substeps per load step. In some cases, the solution may fail to converge if a small number of substeps is used per load step. You can specify absolute damping coefficients to overwrite internal damping values by inserting a Command object (p. 2303) under the **Contact Region** and setting appropriate values for the real constants, FDMN and FDMT.

- If you are using a System Coupling in Workbench component system in combination with your Mechanical analysis, the **Save Project Before Solution** and **Save Project After Solution** properties of the Project object are not supported.
- When transferring data to or from a wall boundary in a sliding mesh zone, you must make sure that Mechanical does not rotate the mesh. You can accomplish this by using a Rotational Velocity (p. 1511). For information about sliding meshes in Ansys Fluent, see the Using Sliding Meshes documentation in the *Fluent User's Guide*.

Variables Available for System Coupling

The following variables are available on all data transfer regions.

Table 1: Variables On Boundary Wall Regions

Display Name / Internal Name	**Transfer Direction**	**Tensor Type**	**Extensive/Intensive**	**Quantity Type**
Force / FORC	Input	Vector	Extensive	Force
Force Density / FDNS[1]	Input	Vector	Intensive	Force
Displacement / INCD	Output	Vector	Intensive	Length
Heat Flow / HFLW	Input and Output	Scalar	Intensive	Heat Rate
Heat Transfer Coefficient / HCOE	Input	Scalar	Intensive	Heat Transfer Coefficient
Temperature / TEMP	Input and Output	Scalar	Extensive	Temperature
Near Wall Temperature / TBULK	Input	Scalar	Intensive	Temperature
1: FDNS is not supported for System Coupling in Workbench or for cases where Mechanical uses shell elements and Force Density is applied to both sides of any shell.				

Table 2: Variables On Body System Coupling Regions

Display Name / Internal Name	**Transfer Direction**	**Tensor Type**	**Extensive/Intensive**	**Quantity Type**
Heat Rate Density / HGEN	Input	Scalar	Intensive	Heat Rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Display Name / Internal Name	Transfer Direction	Tensor Type	Extensive/Intensive	Quantity Type
Temperature / TEMP	Output	Scalar	Intensive	Temperature
Temperature / TPLD	Input	Scalar	Intensive	Temperature

Note:

When using System Coupling in Workbench, the data plotted in System Coupling's chart monitors are provided by the coupling participants. For non-linear analyses, the structural convergence quantities from Mechanical are plotted in terms of the activated degrees of freedom in the structural solver. For linear analyses, the structural convergence quantities from Mechanical are plotted only for thermal analyses with the temperature degrees of freedom.

System Coupling Related Settings in Mechanical

End Time Specification

For transient analyses, Mechanical requires the end time specified in the setup to be respected. When coupling participants require their end time to be respected, the maximum allowable end time for the coupled analysis is the minimum of the end time specified by such participants.

Other participant systems, such as Fluent, can run past the end time specified in the setup. These participant systems have no effect on the allowable end time of the coupled analysis.

Ramping of Data-Transfer Loads

Mechanical has two types of ramping that can modify the loads obtained through data transfers in a coupled analysis. The two types of ramping are ramping over substeps, and ramping over coupling steps. The ramping on your load will be determined by the interaction of ramping settings you have set between Mechanical as well as System Coupling. Both Mechanical ramping behaviors are controlled by the Solution Command Object **KBC**, which can be set to **1** or **0**. In a steady-state analysis, the default setting is **KBC = 0** (ramping on). In a transient analysis, the default setting is **KBC = 1** (ramping off).

In System Coupling, substeps are unique to Mechanical, and are steps within a coupling iteration. Loads can be ramped over these substeps. The ramping factor applied to these loads is based on the number of substeps, N_{SS}. At the i^{th} substep, the ramping factor of i/N_{SS} is applied. This ramping is based on the initial value of the load at the end of the last step. When KBC = 0 and $N_{SS}>1$, ramping over substeps occurs. If KBC = 1 or $N_{SS}=1$, ramping over substeps does not occur.

In System Coupling, Mechanical has a second ramping option which ramps loads over the coupling steps. Regardless of what other ramping settings are on, System Coupling always transfers the full load at the end of the coupling step, and then Mechanical applies a ramping factor to this full value at each coupling step. The ramping factor applied to the full load at the coupling step is based on the number of coupling steps, N_{CS}. At the i^{th} coupling step, the ramping factor of i/N_{CS} is applied. When **KBC = 0** and $N_{CS}>1$, ramping over coupling steps occurs. If **KBC = 1** or $N_{CS}=1$, ramping over

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

coupling steps does not occur. Note that if you set ramping over coupling steps to occur in a transient analysis, loads received from System Coupling will be ramped over all coupling steps, and so the full load will only be applied at the last coupling step. This situation is not physical, but may still be useful when using a Transient Structural system to get steady-state results, for example when pre-stressing the structure for a further transient analysis.

Ramping of Loads Within Mechanical

Loads within the Mechanical system (that is, loads that are not transferred to Mechanical through the coupled analysis) are ramped linearly using the **Step End Time** specified in Mechanical. In a steady-state coupled analysis, ramping of these loads is controlled by the relationship between the **Step End Time** specified in Mechanical, and the number of coupling steps specified in System Coupling. For a steady-state analysis, each coupling step in System Coupling corresponds to 1 s of time in Mechanical.

- When the number of seconds set for the **Step End Time** in Mechanical equals the number of coupling steps set in System Coupling, the load is ramped linearly across all steps in the coupled analysis.
- When the number of seconds set for the **Step End Time** in Mechanical is less than the number of coupling steps set in System Coupling, the load is ramped linearly to the coupling step that matches the end time, and then the full load is applied for the remaining steps.
- When the number of seconds set for the **Step End Time** in Mechanical is more than the number of coupling steps set in System Coupling, the load is ramped linearly, but it will not reach its full value. The final value applied will be the ramped value that corresponds to the last coupling step, which may cause inaccuracies in your simulation.
- If Mechanical's **Step End Time** is set to 1s, this ramping will not occur.

Note that Mechanical's computational end time and its load-based end time are independent. The computational end time is equal to the number of coupling steps. The load-based end time controls the ramping behavior, and is set by the **Step End Time** option in the Mechanical interface.

Output Controls

When Mechanical is connected to System Coupling, behavior of the **Output Controls** is changed. For a normal Mechanical run, the **Store Results At** settings are applied per step and **Specified Recurrence Rate** is defined as the output frequency within a step at some substep frequency. When the Mechanical application is connected to System Coupling, these settings are applied across all steps, not within one step. This means that you cannot output results within a step. System Coupling simulations may run many steps, so these settings should be used to reduce the results frequency as needed.

For more information about how to use Mechanical for FSI analyses, see Fluid-Structure Interaction (FSI) (p. 756).

Using Higher-Order Meshes for Coupled Analyses

System Coupling supports both surface and volume meshes with mid-side nodes, providing improved interpolation for Mechanical cases with quadratic element meshes. Previously, Mechanical participants dropped mid-side mesh nodes and sent only corner nodes to System Coupling. Beginning in the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2020 R2 release, Mechanical sends mid-side nodes to System Coupling by default for the following types of cases:

- New coupled analyses created with the 2020 R2 or later release
- Unsolved existing coupled analyses created with a pre-2020 R2 release

For instructions on other scenarios, see:

- Applying Mid-Side Nodes to Existing Solved Cases (p. 827)
- Reverting to the Use of Corner Nodes (p. 827)

Applying Mid-Side Nodes to Existing Solved Cases

To apply the new feature to existing cases that were created with an earlier release and that have already been solved using only the corner nodes, perform the following steps:

1. Clear coupled analysis solution data.
2. Save the Mechanical project.
3. Regenerate Mechanical's solver file.
4. Re-solve the coupled analysis.

Reverting to the Use of Corner Nodes

To revert to Mechanical's previous behavior of sending only corner nodes to System Coupling, perform the following steps:

1. Load the case into Mechanical.

   Note:

   If the case has already been solved using the new functionality, perform the steps described in above topic, Applying Mid-Side Nodes to Existing Solved Cases (p. 827).

2. Open the Commands (APDL) object, using either of the following methods:

   a. On the **Home** tab, **Insert** group, select **Commands**.

   b. In the **Outline**, right-click the **A5** analysis environment object and select **Insert** > **Commands**.

   The **Commands (APDL)** object is added to the **Outline** tree and the corresponding Worksheet opens. The text displayed indicates that the command will be placed in the /SOLU section of the .dat file, so will be executed just prior to the **SOLVE** command.

3. In the **Worksheet**, enter the following command snippet:

```
SCOPT,,NO
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. From the **File** tab, select **Save Project**.

When the solution is run, Mechanical will run the command snippet and revert to sending only linear meshes and data to System Coupling, even when it is using quadratic meshes in its own solution.

One-Way FSI Transfers Using System Coupling in Workbench

One-way data transfers using System Coupling in Workbench are particularly useful as a first step in a sequence of coupled analyses that may advance to co-simulation involving two-way transfers. In some cases, one-way coupled analyses may also be an alternative to transferring data using imported loads.

Coupled analyses may be used to transfer force into, structural data out of, or thermal data into or out of the Mechanical analyses. Either co-simulation data from a coupling participant system or static data from an External Data system may be transferred through System Coupling. (Transferring data from an External Data system may be useful when users with different licenses are working on the same project.)

For all cases (whether Mechanical is the source or target participant, and whether co-simulation or static data are being transferred):

- Start by defining a System Coupling Region (p. 1651) boundary condition at the location corresponding to the fluid-structure interface.
- Connections between coupling participant systems and a System Coupling system are made by linking their **Setup** cells.

For more information, see:

- Transferring Data Into a Mechanical Analysis (p. 828)
- Transferring Data Out of a Mechanical Analysis (p. 829)

Transferring Data Into Mechanical Analyses

In a one-way coupled analysis with Mechanical as the target participant, a System Coupling system can transfer steady-state or transient thermal data (temperature, heat flow, or heat transfer coefficient and near wall temperature), or force (from CFD pressures and viscous forces) to a Mechanical system. Either co-simulation data or static data may be transferred into Mechanical.

Transfer co-simulation data into Mechanical:

When a co-simulation coupling participant, such as a Fluent system, is connected to a Mechanical system via System Coupling, Mechanical is given access to data directly from the other participant.

1. Connect a Mechanical system to a System Coupling system.
2. Connect another co-simulation participant system to the System Coupling system.
3. Edit the System Coupling system to define data transfers from the other co-simulation participant system to the Mechanical system.

Transfer static data into Mechanical:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When the External Data system is connected to a Mechanical system via System Coupling, Mechanical is given access to static data from Ansys External Data (.axdt) files.

1. Connect a Mechanical system to a System Coupling system.
2. Connect an External Data system to the System Coupling system.
3. Edit the External Data system to select one or more .axdt files.
4. Edit the System Coupling system to define data transfers from the External Data system to the Mechanical system.

The Ansys External Data text-formatted files can be generated by the CFD-Post component system from another participant's (such as Fluent's) analysis results.

Transferring Data Out of Mechanical Analyses

In a one-way coupled analysis with Mechanical as the source participant, a System Coupling system can transfer displacement or thermal data (temperature or heat flow) from Mechanical to other coupling participants. Either co-simulation or static data may be transferred out of Mechanical.

Transfer co-simulation data out of Mechanical:

When a Mechanical system is connected via System Coupling to another co-simulation coupling participant, such as a Fluent system, the other participant is given access to data directly from Mechanical.

1. Connect a Mechanical system to a System Coupling system.
2. Connect another co-simulation participant system to the System Coupling system.
3. Edit the System Coupling system to define data transfers from the Mechanical system to the other co-simulation participant system.

Transfer static data out of Mechanical:

To transfer static data out of Mechanical, export Ansys External Data files (`.axdt`) from Mechanical and use the files to transfer the data into an External Data system.

1. In Mechanical's **System Coupling Region**, set the **Export Results** property to **Yes**.
2. It you are running Mechanical from the command line for a thermal coupled analysis, add the macro `dumpFSIHeats.mac` to your **Mechanical APDL** directory before running Mechanical.

   The macro is available in the **C:\Program Files\ANSYS Inc\v241\aisol\DesignSpace\DSPages\macros** directory.
3. Add an External Data system to the **Project Schematic**.
4. Edit the External Data system to select one or more `.axdt` files.

   When the coupled analysis is run, Mechanical writes one `.axdt` file per **System Coupling Region** to its solver files directory. Each file may contain temperatures and heat flows, with

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

only corner node values recorded. (The mid-side node values for heat flows, if present, are summed to the corresponding corner nodes.) Heat flow data includes the sum of heat flows through surfaces with applied temperatures, convection, and radiation.

5. Connect the External Data system to the System Coupling system.

6. Edit the System Coupling system to define data transfers from the External Data system to the other co-simulation participant system.

Coupled Field Co-Simulation Using System Coupling

System Coupling supports static and transient coupled field co-simulations involving thermal-fluid-structural couplings between Mechanical and another coupling participant (such as Fluent). The co-simulation may be run using either System Coupling in Workbench or one of System Coupling's user interfaces (GUI or CLI). Setup of Mechanical's coupled field system is the same in all scenarios.

To set up and run a coupled field co-simulation, perform the following steps:

1. Set up the coupled field system in Mechanical.

   a. Add a **Coupled Field Static** or **Coupled Field Transient** analysis system to the **Project Schematic**.

   b. Set up the system in Mechanical as usual (as described in Coupled Field Static (p. 391) analysis or Coupled Field Transient (p. 395) analysis), with the following qualifications:

      - Structural and/or thermal variables must be defined.
      - Make sure the initial temperature is set correctly. (This is defined by the **Initial Temperature** and **Initial Temperature Value** fields under **Coupled Field Transient**/**Initial Physics Options**.)
      - A **System Coupling Region** must be used to create the coupling interface and to send both structural and thermal Degrees of Freedom (DOFs) to the co-simulation.

2. Finish setting up CFD participant and System Coupling according to where you want to complete the setup and run the co-simulation, as follows:

   - To complete the setup and solve the co-simulation in Workbench, follow the steps outlined in Setting Up a Coupled Analysis in Workbench and Running a Coupled Analysis in Workbench.
   - To complete the setup outside Workbench and solve the co-simulation in one of System Coupling's user interfaces, follow the steps outlined in Preparing for a Coupled Analysis, Creating a Coupled Analysis, and Running a Coupled Analysis.
   - To complete a full or partial setup in Workbench, export the setup, and solve the co-simulation in one of System Coupling's user interfaces, follow the steps outlined in Exporting a System Coupling Setup and Running an Exported System Coupling Setup.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Coupling Thin Surfaces in CFX

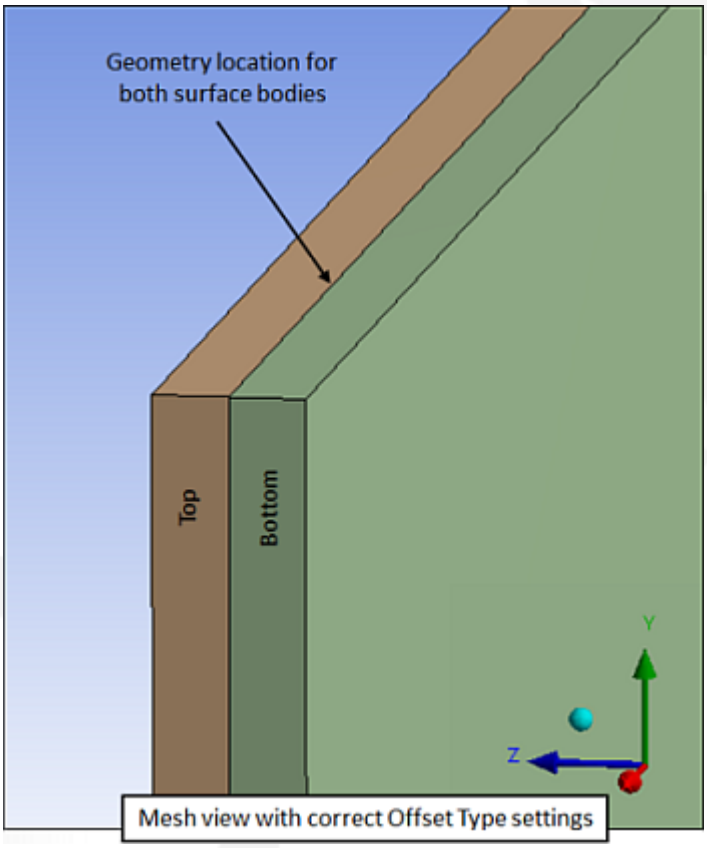
To transfer forces from a thin surface in CFX, you need to have a duplicate surface in Mechanical so that two surfaces exist in Mechanical Mechanicalto match the two surfaces in CFX.

Once you have two surfaces in Mechanical:

1. Use a bonded contact to connect the two surfaces in Mechanical.
2. Assign each surface half the true thickness of the thin surface.
3. Add an offset parameter to each of the Mechanical surfaces to ensure that the shell elements represents the true geometry. Typically, one surface uses **offset** = **top** and the second uses **offset** = **bottom**. Ensure that your elements are similar to those in the image below, where the mesh elements are not overlapping.
4. Create two fluid-solid interfaces in Mechanical, one for each surface body.

In this case, you cannot use a single set of shell elements in Mechanical.

Figure 19: Two Surfaces in Mechanical with the Correct Offset Parameter for Coupling with a Thin Surface



Restarting Structural Mechanical Analyses as Part of System Coupling

For instructions on performing restarts in a given coupling context, see the following sections in the *System Coupling User's Guide*:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Restarting a Coupled Analysis
- Restarting a Coupled Analysis in Workbench

To restart your coupled analysis, you will also need restart information specific to the participants connected to your System Coupling system. For details on other participant systems connected to your System Coupling system, review the participant's documentation on restarts.

For general information on performing restarts, see:

Generating Mechanical Restart Files

Specifying a Restart Point in Mechanical

Making Setup Changes Before Restarting

Recovering the Mechanical Restart Point after a Workbench Crash

Restarting a Thermal-Structural Coupled Analysis

Note:

Mechanical restarts are not supported for:

- Coupled analyses that include **Steady-State Thermal** or **Transient Thermal** systems.
- The transfer of thermal variables.

Generating Mechanical Restart Files

Restarts of a System Coupling analysis requires corresponding restart points to exist in the coupling service and in each of the solvers participating in the analysis.

In order to generate the restart files in Mechanical (`rdb/rXXX` files), perform the following steps:

1. In the Mechanical interface, select **Analysis Settings**.
2. In **Details of "Analysis Settings"**, ensure that **Restart Controls** > **Retain Files After Full Solve** setting is set to **Yes**.

Specifying a Restart Point in Mechanical

The restart point selected in the Mechanical solver must be consistent with the restart points selected for System Coupling and other coupling participants. A run-time error will be issued if an analysis is restarted with incompatible time points.

To specify a restart point in Mechanical, perform the following steps:

1. Double-click the Mechanical's **Solution** cell in Workbench.
2. In the **Outline** view tree, select **Analysis Settings**.
3. In the **Details of Analysis Settings**, under **Restart Analysis**, set **Restart Type** to **Manual** and select the correct restart point from the drop-down menu of **Current Restart Point**.
4. Close the Mechanical application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. In the **Project Schematic**, right-click the Mechanical system's **Setup** cell and select **Update**.

Making Setup Changes Before Restarting

In some cases, setup changes are desired or are required to avoid failure of the coupled analysis. You may make these changes either by modifying settings in Mechanical or (if working in System Coupling's GUI or CLI) by using a custom restart file.

For instructions, see:

Modifying Settings in Mechanical

Using Custom Restart Files in System Coupling

Modifying Settings in Mechanical

Open Mechanical and save your project before modifying any settings so that, if needed, you can restore the saved project and any deleted restart points.

Make the required changes to the setup, then save the project and close the Mechanical application. The changed settings will be applied for subsequent coupled analyses.

Note:

The modification of some settings in Mechanical may invalidate and cause the deletion of all restart points. This deletion of restart points can cause the runtime error which warns of incompatible restart points. For a list of these settings, see Modifications Affecting Restart Points (p. 1818).

Using Custom Restart Files in System Coupling

Custom restart files are controlled by the Mechanical participant's **Execution Control | Additional Restart Input File** setting in the System Coupling data model. This setting accepts a Mechanical solver input file (`.dat`) and is blank by default.

To use a custom restart file, set Additional Restart Input File to the path of the custom restart script you wish to use.

Note:

Ensure that the specified restart file is valid. System Coupling does not validate restart files before the solution is restarted.

When a value is entered for this setting, System Coupling inserts a line into Mechanical's restart script, just before the **solve** command. The added line will include an **/input** command that references the specified restart file, as in the example below:

```
/batch
/solu
/gst,on,on
antype,4,rest,10,,continue
/input,myCommands.dat,,,,1
solve
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
save
finish
exit
```

The insertion of this line allows you to insert command snippets into the restart script. You may use Mechanical APDL commands that are supported restarting Mechanical in a coupled analysis — for example, you may use the **TIMINT** command to control time integration for Mechanical's restart.

System Coupling will use the custom script as the input for the restarted run, issuing the inserted command snippets before restarting the solution.

Recovering the Mechanical Restart Point after a Workbench Crash

Workbench or one of the components may crash such that restart files are available but they are not recognized or populated in the Workbench project.

See Recovering from a Workbench Crash for the steps needed to recover a coupled analysis after a Workbench crash. You will also need the information below about Mechanical, as well as information specific to the other participant systems connected to System Coupling. For other participant systems connected to your System Coupling system, see Supported Coupling Participants for System Coupling in Workbench for a list of supported systems and references to their corresponding documentation regarding restarts.

The result files from the run that crashed may reside in either the system's solution folder or in the `_ProjectScratch` directory. The `_ProjectScratch` directory is a temporary directory used by the Mechanical APDL solver. This directory contains the latest structural results and restart points written during the System Coupling run before Workbench crashed. Mechanical will need to read these file to recover the project using the steps below.

Note that the `.backup` directory contains the original version of any files which have been modified since the last save. These files are useful to recover the last saved state, but they are not useful for restarting your analysis.

To recover Mechanical's restart point after a workbench crash:

1. In the Project Schematic, double-click Mechanical's **Setup** cell. In the Mechanical interface, select the **Solution** entry from the tree.

2. From the **Solution** Context tab, select the **Read Result Files** option from the **Tools** group.

3. Browse to the location of the result files (which may be the systems solution directory or the `_ProjectScratch` directory) and select `file.rst`. Uncombined result files from a distributed-memory parallel solution can also be used instead of a single combined result file. In order to use the uncombined result files, the result file chosen must be `file0.rst`. Mechanical will now patch itself into a state consistent with the results files, with restarts points (if they were written) available for selection in Mechanical.

4. Select the restart point in Mechanical as in Specifying a Restart Point in Mechanical (p. 832) above.

5. Once you have selected Mechanical's restart point, in the Project Schematic, right-click Mechanical's **Setup** cell and select **Update**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Restarting a Thermal-Structural Coupled Analysis

When restarting coupled thermal-structural cases in which Mechanical APDL serves heat flow, use the default solve order, with Mechanical APDL solving first.

Mechanical APDL does not save the heat flows from the previous analysis, so initial heat flows sent to System Coupling are zero. Once Mechanical APDL has solved the first coupling iteration, the correct heat flows are sent to System Coupling.

If the default solve order is used, with Mechanical APDL solving first, then this will have no effect on the simulation because heat flows will have been calculated before Fluent solves the first coupling iteration. If the solution order is changed so that Fluent solves first, then Fluent will receive zero heat flow in the first coupling iteration after a restart, which could cause problems with the solution process.

Note:

The above information is for a coupled thermal-structural system. Mechanical does not support restarts for coupled analyses which include **Steady-State Thermal** and **Transient Thermal** systems.

Running Mechanical as a Coupling Participant in System Coupling's GUI or CLI

System Coupling analyses can be run using either of System Coupling's user interfaces. To run the Mechanical application as a coupling participant:

For coupled analyses set up in a System Coupling interface:

- To set up a coupled analysis in System Coupling's GUI or CLI, follow the steps outlined in Preparing for a Coupled Analysis, Creating a Coupled Analysis, and Modifying Coupled Analysis Settings.
- To run a coupled analysis in System Coupling's GUI or CLI, follow the steps outlined in Running a Coupled Analysis.

For coupled analyses that were set up using System Coupling in Workbench:

- To set up a coupled analysis using System Coupling in Workbench, follow the steps outlined in Setting Up a Coupled Analysis in Workbench.
- To export the coupled analysis setup, follow the steps outlined in Exporting a System Coupling Setup.
- To execute the exported setup in System Coupling's GUI or CLI, follow the steps outlined in Running an Exported System Coupling Setup.

Troubleshooting Two-Way Coupling Analysis Problems

The following files, found in the Mechanical run directory (`SYS/MECH` under a Workbench design point directory), may prove useful in troubleshooting coupled analysis problems:

- `file.err`: This file contains a summary of all of the errors that occurred during the run.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `solve.out` (or other output file): This file contains a complete summary of the current/latest run's evolution. This is one of the most useful files to determine why the coupled analysis failed. To generate extensive debug output during the analysis, enter the following command as a command snippet in the analysis branch when completing the Mechanical problem setup:

```
/debug,-1,,,,,2
```

Provide all of these files when submitting a request for service to Ansys personnel.

Product Licensing Considerations when using System Coupling

A distinct license is required for each coupling participant product, but no additional licenses are required for the System Coupling infrastructure itself.

Note, however, that Ansys EnSight is required for the postprocessing of System Coupling's interface results. If you are using a Multiphysics Licensing Bundle that does not include an EnSight license (e.g., the ***Ansys Mechanical Maxwell*** bundle), you will need a separate EnSight license to visualize co-simulation results.

For more information on applicable licenses, see Product Licensing Considerations in the *System Coupling User's Guide*.

Thermal-Stress Analysis

The Mechanical application enables you to apply temperatures from a thermal analysis as loads in a structural analysis for thermal stress evaluations. The load transfer is applicable for cases when the thermal and structural analyses share the mesh as well as for cases when the two analyses are solved using different meshes. For cases when the meshes are different, the temperature values are mapped and interpolated between the source and target meshes.

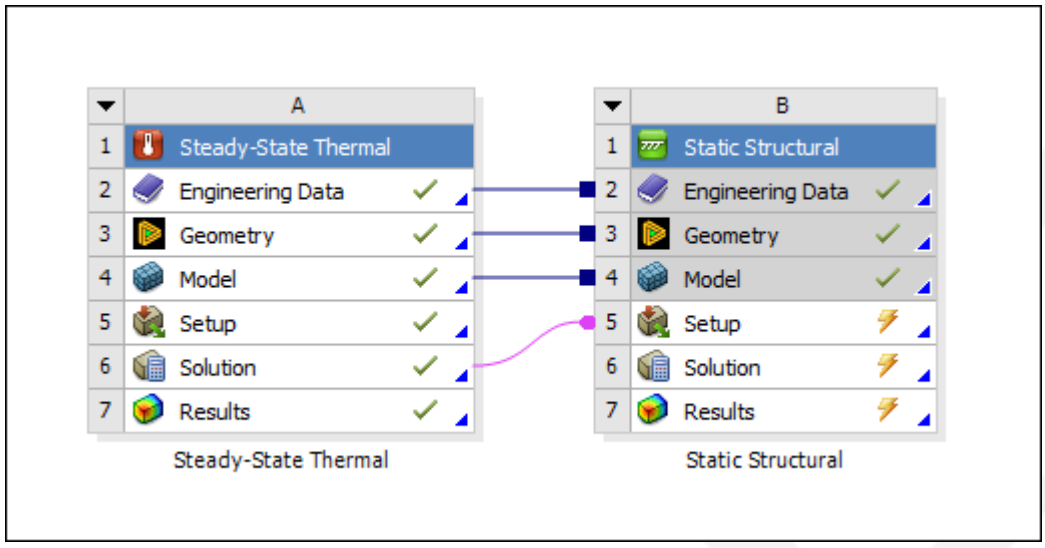
Workflow for performing a thermal stress analysis with:

Shared Model

1. From the toolbox, drag and drop a transient or steady-state thermal template onto the project schematic. Perform all steps to set up a Steady-State Thermal (p. 657) or Transient Thermal (p. 680). Specify mesh controls, boundary conditions, and solution settings as you normally would and solve the analysis.
2. Drag and drop a Static Structural, Transient Structural or LS-DYNA template on top of the thermal system's solution cell to enable the data transfer.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



3. Double-click the structural systems Setup cell. In the Mechanical application an Imported Body Temperature (p. 1740) load is automatically added into the structural system's tree under an Imported Load folder.

4. Select appropriate geometry in the **Details** view of the Imported Body Temperature object using the **Geometry** or **Named Selection** scoping option. If the load is scoped to one or more surface bodies, the **Shell Face** option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.

   Note:

   In a LS-DYNA system, temperatures cannot be scoped to beams.

5. Change any of the columns in the **Data View** tab as needed:

   - **Source Time**: The time at which the data will be imported from the thermal analysis.
   - **Analysis Time**: Select the analysis time at which the load will be applied.

   Note:

   The **Data View** can automatically be populated with the source and analysis times using **Source Time** property in the Details view. Use **All** to import data at all times in the source analysis, or **Range** to import data for a range specified by a **Minimum** and a **Maximum**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

6. Right-click the Imported Body Temperature object and click **Import Load** to import the load. When the load has been imported successfully, a contour plot of the temperatures will be displayed in the Geometry window.

   Note:

   The range of data displayed in the graphics window can be controlled using the **Legend controls** options. See Imported Boundary Conditions (p. 1736) for additional information.

7. You can define multiple rows in the **Data View** tab to import source data at multiple times and apply them at different analysis. If multiple rows are defined in the **Data View**, it is possible to preview imported load vectors/contour applied to a given row or analysis time in the **Data View**. Choose **Active Row** or **Analysis Time** using the **By** property under **Graphics Controls** in the details of the imported load and then specify the **Active Row**/**Analysis Time** to preview the data.

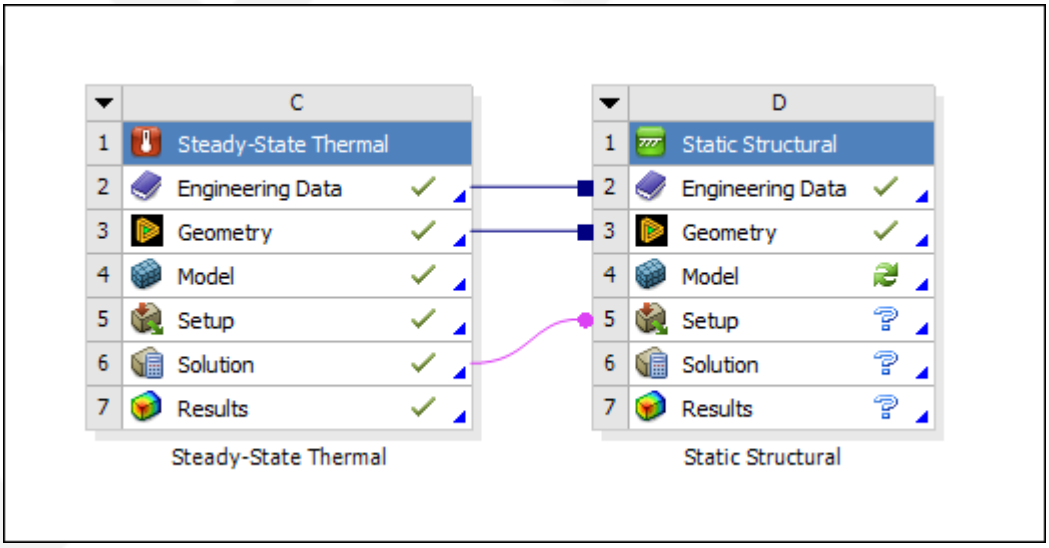
   Note:

   If the **Analysis Time** specified by the user does not match the list of analysis times in the **Data View**, the data is displayed at the analysis time closest to the specified time.

Unshared Model

1. From the toolbox, drag and drop a steady-state or transient thermal template onto the project schematic. Perform all steps to set up a Steady-State Thermal (p. 657) or Transient Thermal (p. 680). Specify mesh controls, boundary conditions, and solution settings as you normally would and solve the analysis.

2. Drag and drop a Static Structural, Transient Structural or LS-DYNA template onto the project schematic. Share the Engineering Data and Geometry cells if required and then drag the Solution cell of the thermal system onto the Setup cell of the structural system.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Double-click the structural systems Setup cell. In the Mechanical application, an Imported Body Temperature (p. 1740) load is automatically added into the structural system's tree under an Imported Load folder.

4. Select appropriate geometry in the **Details** view of the Imported Body Temperature object using the **Geometry** or **Named Selection** scoping option. If the load is scoped to one or more surface bodies, the **Shell Face** option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom** face(s) only. See Imported Body Temperature (p. 1740) for additional information.

   Note:

   In a 3D analysis, if the Triangulation mapping algorithm is used, the Transfer Type mapping option defaults to Surface when the load is scoped to shell bodies.

   Note:

   In a LS-DYNA system, temperatures cannot be scoped to beams.

5. The **Source Bodies** option in the **Details** view enables you to select the bodies, from the thermal analysis, that make up the source mesh for mapping the data. You can select one of the following options:

   - **Automatic**: The application uses heuristics based on the geometry to automatically match source and target bodies and map temperature values. A source body is matched with a target body if it satisfies the following criteria:

     – The percent volume difference is within the user-defined tolerance.

     – The distance between the centroid locations divided by the diagonal of the bounding box is within the user defined tolerance.

     The percent tolerance values can be specified in the **Tolerance** field. The default is set at 1%. The matching process is done in increments of 0.1 of the tolerance value, up to the defined tolerance. The process fails if multiple source bodies are found to match a target body or if no match is found for a target body. After the import is completed, a **Load Transfer Summary** is displayed as a comment object in the particular load branch. The summary shows the matched source and target bodies as well as the values that were used to determine the match. It is recommended that you verify the import using this information.

     Important:

     This option requires that:

     – Element volume results are present in the thermal results file. In the thermal analysis, make sure that the **Calculate Thermal Flux** or the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General Miscellaneous property under the **Analysis Settings** object is set to **Yes** so that this result is available.

– Each scoped body of the target has a matching source body with a unique material id. Therefore, this option may not work when **Imported Trace** or **Material Assignment** objects are present.

Note:

This option is not allowed when scoped to a node-based **Named Selection** as the heuristic is geometry based.

- **All**: The source mesh in this case will comprise all the bodies that were used in thermal analysis. For cases where the temperature values are significantly different at the boundaries across two or more bodies, this option could result in mapped target values that are generated by taking a weighted average of the source values across multiple bodies. Target regions can exist where the mapped temperatures differ significantly from the source.
- **Manual**: This option enables you to select one or more source bodies to make up the source mesh. The source body selections are made in the **Material IDs** field by entering the material IDs that correspond to the source bodies that you would like to use. Type material IDs and/or material ID ranges separated by commas to specify your selection. For example, type 1, 2, 5-10. The material IDs for the source bodies can be seen in Solution Information Object (p. 1833) of the source analysis. In the example below, text is taken from a solver output,

```
**********Elements for Body 1 "coil" **********
**********Elements for Body 2 "core" **********
**********Elements for Body 3 "bar" ***********
```

body 'coil' has material ID 1, body 'core' has material ID 2 and body 'bar' has material ID 3.

6. Change any of the columns in the Data View tab as needed:
   - **Source Time**: The time at which the data will be imported from the source analysis.
   - **Analysis time**: Choose the analysis time at which the load will be applied.

   Note:

   The **Data View** can automatically be populated with the source and analysis times using **Source Time** property in the Details view. Use **All** to import data at all times in the source analysis, or **Range** to import data for a range specified by a **Minimum** and a **Maximum**.

7. You can transform the source mesh used in the mapping process by using the **Rigid Transformation** properties. This option is useful if the source geometry was defined with respect to a coordinate system that is not aligned with the target geometry system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

8. You can modify the Mapper Settings (p. 2389) to achieve the desired mapping accuracy. Mapping can be validated by using Mapping Validation (p. 2412) objects.

9. Right-click the **Imported Body Temperature** object and click **Import Load** to import the load. When the load has been imported successfully, a contour plot of the temperatures will be displayed in the Geometry window.

10. You can define multiple rows in the **Data View** tab to import source data at multiple times and apply them at different analysis. If multiple rows are defined in the **Data View**, it is possible to preview imported load vectors/contour applied to a given row or analysis time in the **Data View**. Choose **Active Row** or **Analysis Time** using the **By** property under **Graphics Controls** in the details of the imported load and then specify the **Active Row**/**Analysis Time** to preview the data.

    Note:

    If the **Analysis Time** specified by the user does not match the list of analysis times in the **Data View**, the data is displayed at the analysis time closest to the specified time.

    Note:

    a. You can add a template for the linked thermal and structural systems by creating your own template.

    b. The transfer of temperatures is not allowed between a 2D analysis and 3D analysis or vice-versa.

Note:

If the upstream (thermal) system is modified and re-solved after importing the load, a refresh operation on the structural system's Setup cell is required to notify Mechanical that source data has changed and re-import is required. Alternatively, the source data can be refreshed using the Right-click operation on the Imported Load folder and choosing the **Refresh Imported Load** option.

Note:

When there is a shared model that includes a thermal-stress analysis and the structural system is duplicated using the Engineering Data, Geometry or Model cell context menu, the result is the Setup cell of the Thermal system linked to the Solution cell of the duplicated structural system. Temperature transfer to the duplicated structural system will require the data to be mapped and interpolated between the source and target meshes.

Transferring Hydrodynamic Loads to a Structural System

The Hydrodynamic Pressure Mapping ACT extension transfers the surface pressures and line loads calculated in a Hydrodynamic Diffraction analysis or a time domain Hydrodynamic Response analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

to panel and beam elements in a Static Structural analysis, through a link on the Workbench Project Schematic page. This removes the need to create, manipulate and run files external to Workbench. Multiple wave phase angles or time steps can be analyzed in a single Static Structural calculation, to provide a clear picture of finite element results due to hydrodynamic loading over the whole wave cycle or time series. To view the documentation explaining how to load and use the extension, click here.

The Offshore ACT extension exposes the family of OCEAN commands that are offered in Mechanical APDL. In this way the hydrodynamic loads on submerged beam and pipe elements – resulting from the motion of the structure through the fluid, or from the fluid motion around the structure due to current or waves – can be included in Static Structural, Transient Structural, Modal and Harmonic Response analyses. To view the documentation explaining how to load and use the extension, click here.

Twin Builder/Rigid Dynamics Co-Simulation

This feature is a co-simulation link (transient-transient) between Twin Builder and the Ansys Rigid Dynamics solver. This link enables you to combine detailed rigid mechanics models with system models such as complex electronic semiconductor device models used in controls. You can export a rigid dynamics sub-circuit and perform an analysis of the structure in Twin Builder.

Note:

You can also perform an FMU co-simulation with an external tool (co-simulation primary). For more details about the FMI standard and tools that support co-simulation with FMU, see fmi-standard.org. You set up an FMU co-simulation in the same manner as described here for a Twin Builder co-simulation.

- Twin Builder and rigid dynamics models are connected by Co-Simulation Pins (p. 844).
- Simulation is driven by Twin Builder.
- Results can be reviewed in Twin Builder, and then imported back to Ansys Mechanical.

Preparing the Analysis

Create a Rigid Dynamics Analysis System

General *Basic general information about this topic (p. 323)*

Define Engineering Data

General *Basic general information about this topic (p. 328)*

Specifics ***... for this analysis type:***

Density is the only material property utilized in a rigid dynamics analysis. Models that use zero or nearly zero density fail to solve using the Ansys Rigid Dynamics solver.

Attach Geometry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

General *Basic general information about this topic (p. 330)*

Specifics ***... for this analysis type:***

Only sheet and solid bodies are supported by the Ansys Rigid Dynamics solver. Plane bodies and line bodies cannot be used.

Define Part Behavior

General *Basic general information about this topic (p. 342)*

Specifics ***... for this analysis type:***

You can define a Point Mass (p. 901) for this analysis type. Part stiffness behavior is not required for the Ansys Rigid Dynamics solver in Ansys Workbench.

Define Joints and Springs

General *Basic general information about this topic (p. 348)*

Specifics ***... for this analysis type:***

Applicable connections for this type of analysis are joints (p. 1292) or springs (p. 1395).

When an assembly is imported from a CAD system, joints and constraints are not imported; however, joints can be created automatically or manually after the model has been imported.

Each joint is defined by its coordinate system of reference. The orientation of this coordinate system is essential, as free and fixed degrees of freedom are defined in this coordinate system.

Define Input and Output Pins

General *Basic general information about this topic (p. 844)*

Specifics ***... for this analysis type:***

The quantities that are driven by Twin Builder are defined as input pins. The quantities that are monitored by Twin Builder are defined as output pins.

Define Analysis Settings

General *Basic general information about this topic (p. 349)*

Specifics ***... for this analysis type:***

Some of the analysis settings might be overwritten by those defined in Twin Builder, because Twin Builder drives the co-simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Co-Simulation Pins

Note:

Pins apply for both Twin Builder and FMU co-simulations, but they are discussed here in the context of co-simulation with Twin Builder.

Co-simulation Pins are connection points that describe the interface between a rigid dynamics model and a Twin Builder model.

Pins have two distinct natures:

- **Input Pins** are used by Twin Builder to drive the rigid dynamics model.
- **Output Pins** are sensors used by Twin Builder to monitor the rigid dynamics model state.

Pins are defined by the degrees of freedom of joints (p. 1292). One pin can be attached to each degree of freedom of a joint. The type of joint *quantity* attached to pin depends on the nature of the degrees of freedom.

Translational degrees of freedom can have **Displacement**, **Velocity**, **Acceleration**, and **Force** pins.

Rotational degrees of freedom can have **Rotation**, **Angular Velocity**, **Angular Acceleration**, and **Moment** pins.

Note:

It is not recommended that you place additional joint conditions on degrees of freedom that are associated with pins.

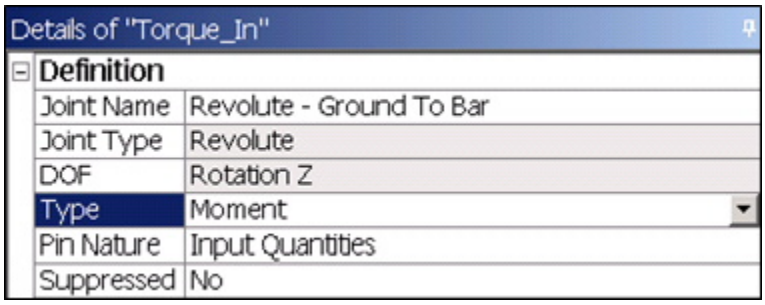
To create pins for a Rigid Dynamics analysis system:

1. Open a **Rigid Dynamics** analysis in Workbench, then double-click the **Model** field to open the model for editing in the Mechanical application.
2. Select the Rigid Dynamics (**Transient**) system in the Outline. From the **Co-simulation** group on the **Environment** Context tab, select the **Co-simulation Pin** option.

   If you select the **Co-simulation Pin** option while a joint is selected, the pin will automatically have joint information associated with it. If no joint is selected, you will need to associate the pin with a joint at a later time.
3. With the new pin selected in the Outline view, edit the **DOF**, **Type**, and **Pin Nature** fields in the Details view to complete the pin setup.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



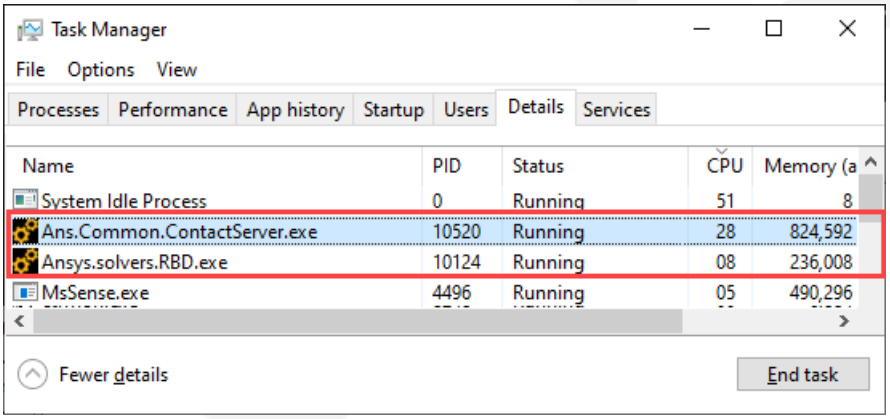
4. Rename the pin as it should appear in Twin Builder.

5. Repeat steps 2, 3, and 4 to add all pins of interest.

6. When finished adding pins, refer to Writing Ansys Rigid Dynamics Files (p. 1867) for more information.

Limitations

The Rigid Dynamics FMU uses a server-client architecture. The FMU is a thin client layer which uses the Rigid Dynamics solver in the background. Consequently, if the FMU primary abruptly aborts the simulation, the Rigid Dynamics solver executable may be left running. This orphan process prevents further solves. You must manually kill the solver process (`Ansys.Solvers.RBD.exe`), and the contact server (`Ans.common.ContactServer.exe`) using, for instance, the Windows Task Manager.



Typically, this problem occurs with Twin Builder when the simulation is aborted with the **Abort** option. Always use the **Clean Stop** as described here.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifying Geometry

The following topics are included in this section:

Geometry Introduction
Solid Bodies
Surface Bodies
Line Bodies
Simulation without Geometry
2D Analyses
Construction Geometry
Point Mass
Distributed Mass
Surface Coating
Thermal Point Mass
Models from External Meshes and Model Assemblies
Element Orientation
Part Transformations
Geometry from Deformation Results
Geometry From Rigid Dynamics Results
Bolt Tools Add-on

Geometry Introduction

While there is no limit to the number of parts in an assembly that can be treated, large assemblies may require unusually high computer time and resources to compute a solution. Contact boundaries can be automatically formed where parts meet. The application has the ability to transfer structural loads and heat flows across the contact boundaries and to "connect" the various parts.

The following topics are addressed in this section:

Parts and Bodies
Multibody Behavior and Associativity
Geometry Conditions and Requirements
Stiffness Behavior
Integration Schemes
Common Geometry Display Features

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Parts and Bodies

Parts imported into Mechanical are a grouping or a collection of bodies as defined by the CAD application you use to define the parts.

Multibody Parts

Imported parts are a grouping or a collection of bodies. Parts can include multiple bodies and are referred to as *multibody parts*. Parts may consist of:

- One or more solid bodies.
- One or more surface bodies.
- One or more line bodies.
- Combinations of line and surface bodies.
- Combinations of solid and shell bodies (for models imported from DesignModeler, SpaceClaim, or External Model).

All other combinations are not practically supported.

Note:

Body objects in the tree that represent a multibody part do not report centroids or moments of inertia in their respective Details view.

Meshing Multibody Parts

The mesh for multibody parts created in SpaceClaim or DesignModeler share nodes where the bodies touch one another. This facilitates the multiple parts having common nodes at the part interfaces.

Collapsing Line Bodies in SpaceClaim

For line body models in SpaceClaim, by default, the application imports one line body for every edge present in the body in SpaceClaim. This may create an undesirable number of bodies in Mechanical. To facilitate a more manageable model, SpaceClaim enables you to "collapse" all of the line bodies that have the same cross section, material, and that touch one another, into one body by setting the **Share Topology** property to either **Share** or **Group**. See the **Shared topology in Ansys** section in the *SpaceClaim Online Help* for additional information about the **Share Topology** property.

Working with Parts

There are several useful and important manipulations that can be performed with parts in an assembly.

- The part can include solid and shell geometries.
- Each part may be assigned a different material.
- Parts can be hidden for easier visibility.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Parts can be suppressed, which effectively eliminates the parts from treatment.
- The contact detection tolerance and the contact type between parts can be controlled.
- When a model contains a Coordinate Systems object, by default, the part and the associated bodies use the Global Coordinate System to align the elements. If desired, you can apply a local coordinate system (p. 1191) to the part or body. When a local coordinate system is assigned to a Part, by default, the bodies also assume this coordinate system but you may modify the system on the bodies individually as desired.
- When you click **Freeze Mesh on All Parts**, the generated mesh on all parts of the model is frozen. To access **Freeze Mesh on All Parts**, right-click **Geometry** in the **Tree** outline and click **Freeze Mesh on All Parts** to freeze the mesh. If you try to modify the mesh on the body on which the freeze mesh is applied, a warning message is displayed to unfreeze the mesh. To unfreeze the mesh, right-click **Geometry** in the **Tree** outline and click **Unfreeze Mesh on All Parts**.
- When you click **Freeze Mesh**, only mesh on the selected part is frozen. To access **Freeze Mesh**, right-click the part on which you want to freeze the mesh in the **Tree** outline and click **Freeze Mesh**. The mesh on the selected part will be frozen while the mesh for other parts can be cleared and generated. When you try to modify mesh on the part on which **Freeze Mesh** is applied, a warning message is generated to unfreeze the mesh on the part. To unfreeze the mesh on the part, right-click the part and click **Unfreeze Mesh** in the **Tree** outline.

Note:

– **Freeze Mesh** can be applied only if all bodies of the part are meshed.

– **Mesh Edit**, **Generate Mesh** and **Clear Generated Data** are disabled on frozen mesh parts. When you perform **Mesh Connection**, **Node merge**, **Contact Match**, **Node Move**, **Morph**, **Fracture**, **Generate Mesh**, **Clear Generated Data** on a frozen mesh part, a warning message stating "The mesh on the part is frozen. Please unfreeze the mesh and then mesh edit generate/clear should be allowed." is displayed.

Working with Bodies

There are several useful and important manipulations that can be performed with bodies in a part.

- Bodies grouped into a part result in connected geometry and shared nodes in a mesh.
- Each body may be assigned a different material.
- Bodies can be hidden for easier visibility.
- Bodies in a part group can be individually suppressed, which effectively eliminates these bodies from treatment. A suppressed body is *not* included in the statistics of the owning part or in the overall statistics of the model.
- Bodies can be assigned **Full** or **Reduced** integration schemes, as described above for parts (p. 857).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- When bodies in part groups touch they will share nodes where they touch. This will connect the bodies. If a body in a part group does not touch another body in that part group, it will not share any nodes. It will be free standing. Automatic contact detection is not performed between bodies in a part group. Automatic contact detection is performed only between part groups.
- Bodies that are not in a part group can be declared as rigid bodies (p. 852).
- When a model contains a Coordinate Systems object, by default, bodies use the Global Coordinate System. If desired, you can apply a local coordinate system (p. 1191).

Note:

Freeze Mesh cannot be applied on individual bodies.

Multibody Behavior and Associativity

Associativity that you apply to geometry attached from SpaceClaim or DesignModeler is maintained in the Mechanical and Meshing applications when updating the geometry despite any part groupings that you may subsequently change in SpaceClaim or DesignModeler. See Associativity (p. 850) for further information.

When transferring multibody parts from SpaceClaim or DesignModeler, the multibody part has the body group (part) and the prototypes (bodies) beneath it. When the part consists of just a single body the body group is hidden. If the part has ever been imported as a multibody part you will always see the body group for that component, regardless of the number of bodies present in any subsequent update.

Associativity

Associativity that you apply to geometry originating from SpaceClaim or DesignModeler is maintained in the Mechanical and Meshing applications when the geometry is updated despite any part groupings that you may subsequently change in SpaceClaim or DesignModeler. Types of associativity that you can apply include contact regions, mesh connections, loads, and supports. For example, consider the following scenario:

1. A model is created in SpaceClaim or DesignModeler and has six independent parts with one body per part.
2. The model is attached to Mechanical where loads and supports are applied to selected geometry.
3. In SpaceClaim or DesignModeler, the model is re-grouped into two multibody parts with each part including three bodies.
4. The geometry is updated in Mechanical. The loads and supports remain applied to the same selected geometry.

Note:

This feature does not hold true for instanced parts in SpaceClaim or DesignModeler.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The associativity is maintained only with geometry attached from SpaceClaim or DesignModeler and Mechanical systems created in release 13.0 or later. To ensure that the data necessary for retaining associativity is present in legacy dsdb/wbpj databases, you should perform the following:

1. Open the Mechanical session and open the SpaceClaim or DesignModeler session. This will ensure that both the Mechanical and SpaceClaim or DesignModeler files are migrated to the current version of the software.
2. Update the geometry model without making any changes to the model. This will ensure that the new data necessary for associativity is transferred from the migrated DesignModeler file into the migrated Mechanical file.
3. You can now modify and update the geometry as necessary.

Geometry Conditions and Requirements

Note the following for assemblies, parts, and bodies:

- Automatic contact detection will detect contact between bodies within a multibody part.
- To connect multiple bodies within a part using a shared node, the bodies must share a face or an edge. If they do not, the application may not achieve appropriate contact for the analysis and the analysis could experience rigid body motion.
- Removing (for example, suppressing) a body or a part from your source geometry and then refreshing and/or updating the geometry in Mechanical causes the application to remove the body or part from the tree Outline. If you subsequently return the body/part to the source and return it to Mechanical through a refresh or update, the body/part returns to default settings (Stiffness Behavior, material, etc.).

Stiffness Behavior

All geometry Bodies have an associated **Stiffness Behavior** property in Mechanical that you can modify during an analysis. This is most often performed when Defining Part Behaviors (p. 342) once you have imported your model into the application.

The **Stiffness Behavior** property is available for a body, a body group, surface bodies, and 2D models.

To set the **Stiffness Behavior** in the Mechanical application:

1. Select the desired Body in the **Geometry** folder.
2. In the Details view, set the **Stiffness Behavior** property to **Flexible**, **Rigid**, **Gasket**, or **Stiff Beam** (Line Bodies (p. 872) only).

See the following sections for more information about body stiffness behaviors:

Flexible Bodies

Rigid Bodies

Gasket Bodies

Stiff Beam

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Flexible Bodies

The **Flexible Stiffness Behavior** option is the default option.

Rigid Bodies

When you define a body's **Stiffness Behavior** as **Rigid**, you are telling to the application to not allow the body to deform during the solution process. This feature is useful if a mechanism has only rigid body motion or, if in an assembly, only some of the parts experience most of the strains. It is also useful if you are not concerned about the stress/strain of that component and wish to reduce CPU requirements during meshing or solve operations.

The application does not mesh a rigid body and the solver represents the body as a single mass element. However, the system maintains the mass element's mass and inertial properties. The **Mass**, **Centroid**, and **Moments Of Inertia** properties for the body are available in the Details view of the body object.

The following restrictions apply to rigid bodies:

- Rigid bodies are only valid in Static Structural (p. 652), Transient Structural (p. 665), Harmonic Response (p. 496), Modal (p. 515), Rigid Dynamics (p. 550), Random Vibration (p. 524), and Response Spectrum (p. 533) analyses for the objects listed below. Animated results are available for all analysis types *except* a Modal analysis.
  - Point Mass
  - Joint
  - Spring
  - Remote Displacement
  - Remote Force
  - Moment
  - Contact
- Rigid bodies are valid when scoped to solid bodies, surface bodies, or line bodies in an Explicit Dynamics analysis for the following objects:
  - Fixed Support
  - Displacement
  - Velocity
  - Spring
  - Remote Displacement

The following outputs are available for rigid bodies, and are reported at the centroid of the rigid body:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Results: Displacement, Velocity, and Acceleration.
- Probes: Deformation, Position, Rotation, Velocity, Acceleration, Angular Velocity, and Angular Acceleration.

Note:

- If you highlight **Deformation** results in the tree that are scoped to rigid bodies, the corresponding rigid bodies in the **Geometry** window are not highlighted.
- You cannot define a line body, 2D plane strain body, or 2D axisymmetric body as rigid, except that in an Explicit Dynamics analysis, 2D plane strain and 2D axisymmetric bodies may be defined as rigid.
- All bodies in a body group (of a multibody part) must have the same Stiffness Behavior. When **Stiffness Behavior** is **Rigid**, the body group acts as one rigid mass regardless of whether or not the underlying bodies are topologically connected (via shared topology).

Gasket Bodies

Gasket joints are essential components in most structural assemblies. Gaskets as sealing components between structural components are usually very thin and made of various materials, such as steel, rubber and composites. From a mechanics perspective, gaskets act to transfer force between components. The primary deformation of a gasket is usually confined to one direction, namely, through thickness. The stiffness contributions from membrane (in plane) and transverse shear are much smaller in general compared to the through thickness.

A typical example of a gasket joint is in engine assemblies. A thorough understanding of the gasket joint is critical in engine design and operation. This includes an understanding of the behavior of gasket joint components themselves in an engine operation, and the interaction of the gasket joint with other components.

Gaskets can be defined in one of two ways:

- By setting the **Stiffness Behavior** as **Gasket**. In this case, a Gasket Mesh Control will be added as a child of the gasket body in the model tree. You need to define the source face of the gasket in the Gasket Mesh Control to define the gasket material orientation.
- By setting the **Stiffness Behavior** as **Flexible**. In this case, you need to define a Gasket Mesh Control in the mesh folder. The gasket mesh control in the mesh folder can be applied to multiple bodies, so if there are many gasket bodies this option may be a more convenient approach to setting up the gaskets.

You must also specify a material with a valid gasket model in Engineering Data.

The following restrictions apply to Gasket bodies:

- Gasket bodies are valid only in Static Structural analyses.
- Gasket bodies are valid for 3D solids only, that is, 2D gasket bodies cannot be specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- A valid gasket material model must be specified.
- In addition to gasket bodies, a multibody part may also include flexible bodies but not rigid bodies.
- Gasket bodies are not supported for cyclic symmetry analyses.

The overall procedure for simulating gaskets in Ansys Workbench is to run a Static Structural analysis and perform the following specialized steps:

1. Specify a material with a valid gasket model in Engineering Data.
2. Define the gasket using one of two ways:
   a. **Body-based approach**:

      Set the **Stiffness Behavior** of the **Body** object to **Gasket**. This produces a **Gasket Mesh Control** object beneath the **Body** object.

      Adjust Details view settings for the **Gasket Mesh Control** object and generate the mesh. See the Gasket Meshing (p. 854) section for details.

   b. **Mesh control approach**:

      Set the **Stiffness Behavior** of the **Body** object to **Flexible**.

      Define a **Gasket** mesh control object in the **Mesh** folder. Adjust the Details view settings for the **Gasket** mesh control object and generate the mesh. See the Gasket Meshing (p. 856) section for details.
3. Specify the value for the **Gasket Initial Gap** property. The default value is **0**. Valid entries must be greater than or equal to **0**.
4. Solve and review the gasket result. See the Gasket Results (p. 2131) section for more information.

Gasket Meshing

A Gasket body is meshed with the INTER194 elements (if using **Element Order** = **Quadratic**, in such cases a single layer of elements are generated with midside nodes on top and bottom faces, but linear edges across thickness) or INTER195 elements (if using **Element Order** = **Linear**).

Define the gasket using one of two ways:

- Body-based approach (p. 854)
- Mesh control approach (p. 856)

Body-Based Gasket Meshing

Upon setting the **Stiffness Behavior** as **Gasket**, a **Gasket Mesh Control** object is added beneath the **Body** object in the tree. The **Mesh Method** property for the object is automatically set to **Sweep** and is read-only. By default, this property instructs the application to drop mid-side nodes on gasket element edges that are parallel (Normal To) to the scoped sweep direction. You may wish to refer

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to the Sweep Method Control section of the *Meshing User's Guide* for additional information about this property. The **Gasket Mesh Control** operates in the same fashion.

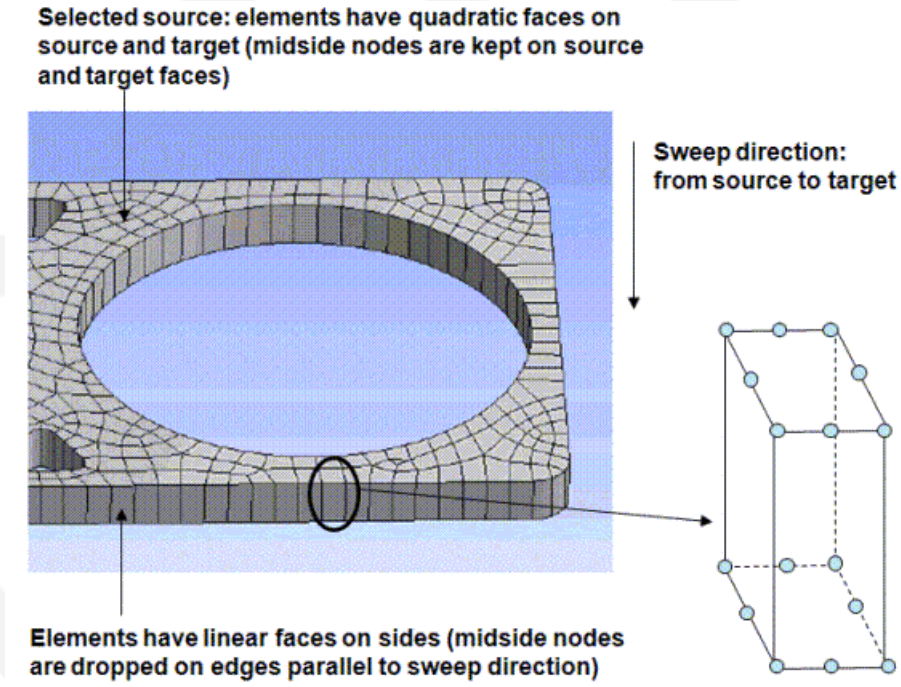
After setting the 3D **Body** object's **Stiffness Behavior** to **Gasket**:

1. If desired, you can change the setting of the **Free Face Mesh Type** property. Options include **All Quad**, **Quad/Tri** (default), or **All Tri**.
2. The **Mesh Method** is automatically set to **Sweep** and is read-only.
3. The **Element Order** property is set to **Use Global Setting** by default. This setting (as well as the **Quadratic** setting) results in quadratic elements with midside nodes to be dropped in the normal direction. When **Element Order** is set to **Linear**, the midside nodes are dropped, resulting in linear elements.
4. **Src/Trg Selection** is set to **Manual Source**. These are the default settings.
5. Select a **Source** face. The selected face must lie on the gasket body.
6. The **Target** selection is **Program Controlled** by default. If desired, you can set **Src/Trg Selection** to **Manual Source and Target**. You can then choose a **Target** face manually.

Note:

Mesh object property changes affect the state of the **Gasket Mesh Control** object.

When generating the gasket element mesh with **Element Order** set to **Quadratic**, the application drops the midside nodes on the edges that are parallel to the sweep direction. For example, consider the mesh shown below. To define the sweep method, **Src/Trg Selection** was set to **Manual Source**; one face (the "top" face) was selected for **Source**. In the resulting mesh, the gasket element faces on the source and target are quadratic, but the faces on the sides are linear.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mesh Control for Gasket Meshing

After setting the **Stiffness Behavior** as **Flexible**, define a **Gasket** mesh control object beneath the **Mesh** object. The **MultiZone** mesh method is used for this approach. Refer to the MultiZone Method Control section of the *Meshing User's Guide* for additional information about this method.

Adjust the Details view settings for the **Gasket** mesh control object as follows:

1. Select the **Scoping Method** for choosing the gasket bodies. The options available are **Geometry Selection** (default) or **Named Selection**. Based on your selection, you need to select the desired geometry or Named Selection.
2. The **Element Order** property is set to **Use Global Setting** by default. This setting (as well as the **Quadratic** setting) results in quadratic elements with midside nodes to be dropped in the normal direction. When **Element Order** is set to **Linear**, the midside nodes are dropped, resulting in linear elements.
3. Select the **Source Scoping Method** for choosing the source faces. The options available are **Geometry Selection** (default) or **Named Selection**. Based on your selection, you need to select the desired geometry or Named Selection. The selected faces must lie on the gasket body.

   The target faces are determined automatically.
4. If desired, you can change the setting of the **Free Face Mesh Type** property. Options include **Quad/Tri** (default), **All Quad**, or **All Tri**.

   As in the example (p. 855), when generating the gasket element mesh with **Element Order** set to **Quadratic**, the midside nodes are dropped on the edges that are parallel to the sweep direction.

Tip:

The gasket mesh control in the mesh folder can be applied to multiple bodies, so if there are many gasket bodies this option may be a more convenient approach to setting up the gaskets.

Note:

If a gasket is part of multibody part and the MultiZone mesh method is defined on the connected body/bodies, the MultiZone method generates linear elements even though the element order is set to quadratic when the mesh is generated for the whole part/assembly together. This does not occur for selective meshing or if the gasket body is meshed first.

Gasket Results

Specialized results are available for analyzing gaskets. See Gasket Results (p. 2131) for details.

Stiff Beam

Supported for line bodies only, when you define a line body's **Stiffness Behavior** as **Stiff Beam**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Only structural analyses support this feature. For example, thermal or electrical analyses are unaffected.
- The application approximates a rigid beam by making the Young's modulus *1e4* times higher than defined in the Engineering Data Workspace.
- The body must be assigned a material that is valid for Isotropic Young's Modulus.
- The application automatically specifies this option on a line body if the "RIGID" setting in SpaceClaim Direct Modeler is equal to True (this is triggered by the SpaceClaim Split Operation).
- A body specified with this option is meshed and will have results.

Integration Schemes

The **Brick Integration Scheme** property provides the options **Full** and **Reduced**. Use the **Full** setting for linear analyses or when the model has only one layer of elements in each direction. This method does not cause hourglass mode, but can cause volumetric locking in nearly incompressible cases.

In order to help prevent volumetric mesh locking in nearly incompressible cases, use the **Reduced** setting. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

Important:

This property supports hexahedral elements only.

Common Geometry Display Features

This section examines the following display features of the **Geometry** object.

- Hide or Suppress Bodies (p. 857)
- Hide or Show Faces (p. 858)
- Color Coding of Parts (p. 858)

Hide or Suppress Bodies

For a quick way to hide bodies (turn body viewing off) or suppress bodies (turn body viewing off and remove the bodies from further treatment in the analysis), select the desired body objects in the tree or select the bodies in the **Geometry** window (using the **Body** selection filter (p. 261)). Then right-click and choose **Hide Body** or **Suppress Body** from the context menu. Choose **Show Body**, **Show All Bodies**, **Unsuppress Body**, or **Unsuppress All Bodies** to reverse the states.

The following options are also available:

- **Hide All Other Bodies**: show only selected bodies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Hide** or **Show**: contains menu options to hide specific body types. Based on the body types of your model, options include: **Solid Bodies**, **Surface Bodies**, and **Line Bodies**.
- **Invert Visibility**: This option inverts the visibility of hidden bodies versus those that are visible. That is, all hidden bodies become visible and all visible bodies become hidden.
- **Suppress All Other Bodies**, allows you to unsuppress only selected bodies.

Note:

- If another model level object, such as a Remote Point, Joint, or Contact Region, is scoped to a Body that becomes Suppressed, that object also becomes suppressed until it is re-scoped or the body is Unsuppressed.
- Results from hidden bodies are used in the formulation of the maximum and minimum values in the contour legend and in the Details View.
- Results from suppressed bodies are suppressed and are **not** used in the formulation of maximum and minimum values.

Hide or Show Faces

You can hide selected faces on a model such that you are able to see inside the model. This feature is especially useful for bodies with interior cavities, such as engine blocks. To use the feature, first select faces on the model that you want to hide, then right-click anywhere in the **Geometry** window and choose **Hide Face(s)** in the context menu. This menu choice is only available if you have already selected faces.

Choose **Show Hidden Face(s)** from the context menu to restore the visibility of faces previously hidden using **Hide Face(s)**. The **Show Hidden Face(s)** menu choice is only available if there are hidden faces from choosing **Hide Face(s)**. It *cannot* be used to restore the visibility of faces previously hidden by setting **Visible** (p. 1078) to **No** in the Details view of a Named Selection object.

Note:

The selected faces will appear hidden only when you view the geometry. The feature is not applicable to mesh displays or result displays.

Color Coding of Parts

You can visually identify parts based on a property of that part. For example, if an assembly is made of parts of different materials, you can color the parts based on the material; that is, all structural steel parts have the same color, all aluminum parts have the same color and so on.

Select a color via the **Display Style** property of the **Details** view when the **Geometry** object is selected. You can specify colors based on:

- **Body Color** (default): Assigns different colors to the bodies within a part.
- **Part Color**: Assigns different colors to the different parts.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Shell Thickness** (surface bodies only): Assigns different colors based on specified body thicknesses. This coloring property does not apply to imported thicknesses or Thickness object based specifications. For those specifications, use the **Shell Thickness** display option of the Mesh object.
- **Assembly Color** (Model assembly only): Assigns a common color to the bodies of each source system (assembly).
- **Material**: The part colors are based on the material assignment. For example, in a model with five parts where three parts use structural steel and two parts use aluminum, you will see the three structural steel parts in one color and the two aluminum parts in another color. The legend will indicate the color used along with the name of the material. You can now assign colors to materials from the Outline Pane (p. 858) in the Engineering Data Workspace and have these colors display in Mechanical. For composite materials, the color assignment is program controlled.
- **Nonlinear Material Effects**: Indicates if a part includes nonlinear material effects during analysis. If you chose to exclude nonlinear material effects for some parts of a model, then the legend will indicate **Linear** for these parts and the parts will be colored accordingly.
- **Stiffness Behavior**: Identifies a part as **Flexible**, **Rigid**, or **Gasket** during analysis.
- **Body Type**: This option assigns a color based on the different types of bodies within a part, such as beam and link types within a Line Body.
- **By Cross Section** (line bodies only): The application assigns a color to a body based on its specified cross section.
- **Condensed Parts Colors** (Rigid Dynamics analysis only): the application assigns colors per condensed part.

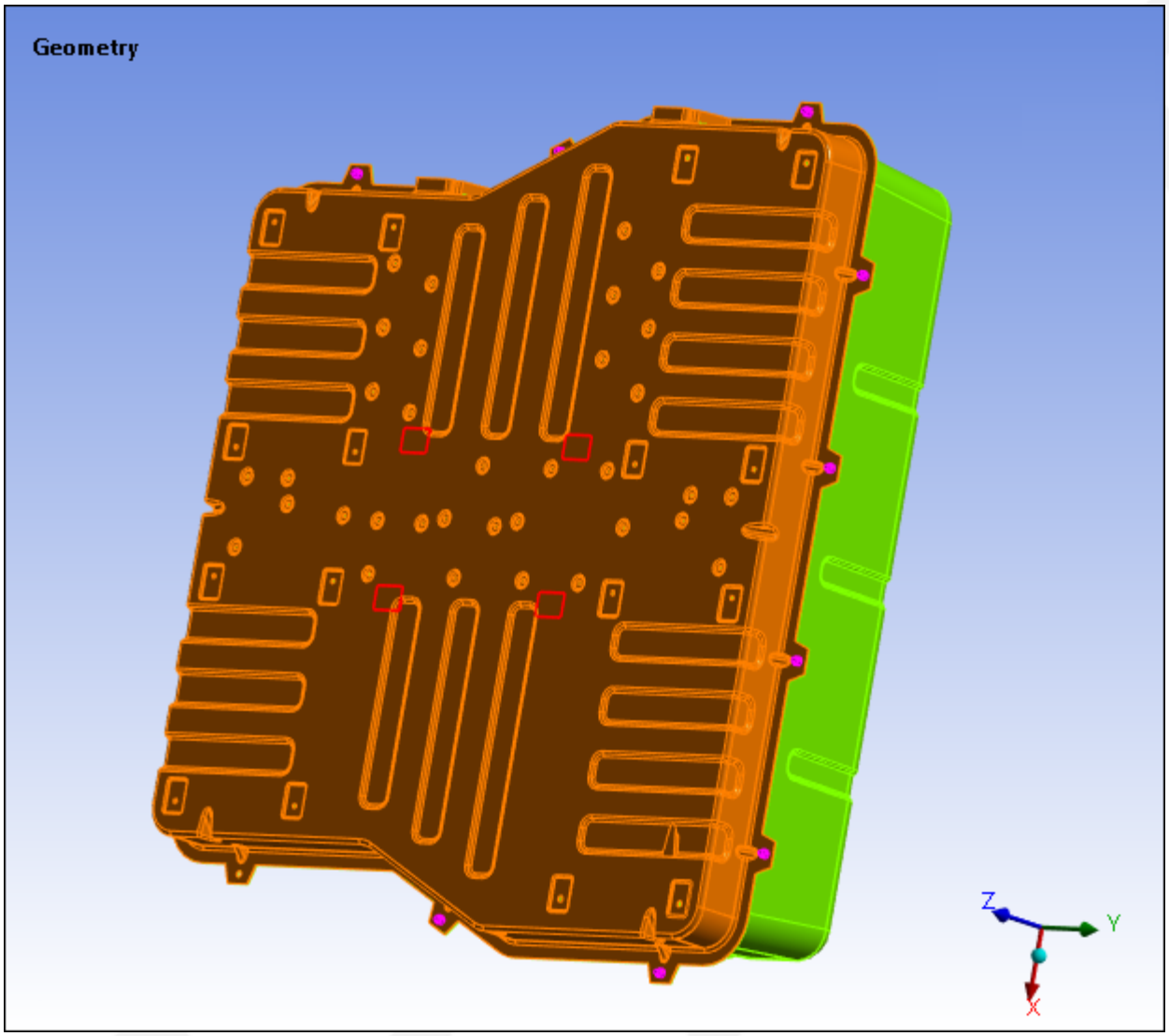
Note:

A maximum of 15 distinct materials can be shown in the legend. If a model has more than 15 materials, coloring by material will not have any effect unless enough parts are hidden or suppressed.

You can reset the colors back to the default color scheme by right-clicking on the Geometry object in the tree and selecting **Reset Body Colors**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example 3: Color by Parts



Solid Bodies

You can process and solve solid models, including individual parts and assemblies. An arbitrary level of complexity is supported, given sufficient computer time and resources.

Surface Bodies

You can import surface bodies from an array of sources (see Geometry Preferences). Surface bodies are often generated by applying mid-surface extraction to a pre-existing solid. The operation abstracts away the thickness from the solid and converts it into a separate modeling input of the generated surface.

Surface body (see Body object reference section) models may be arranged into parts. Within a part there may be one or more surface bodies; these may even share the part with line bodies.

Parts that feature surface bodies may be connected with the help of spot welds and contacts.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The following topics are addressed in this section.

Preparing Assemblies of Surface Bodies
Using Thickness Mode
Importing Surface Body Models
Importing Surface Body Thickness
Understanding Surface Body Shell Offsets
Specifying Surface Body Thickness
Specifying Surface Body Layered Sections
Specifying Surface Body Reinforcements
Specifying Faces With Multiple Thicknesses and Layers

Preparing Assemblies of Surface Bodies

While preparing an assembly of surface bodies for solution you may find the need to understand and modify the connectivity of the bodies involved. Mechanical offers tools to help you accomplish these tasks. For example, you may:

- Confirm whether two surface bodies are topologically connected. This may be especially useful for surface bodies obtained from a mid-surface operation on solids and created artificial gaps in their proximity.
- Confirm the connectivity of individual elements in the mesh of the surface bodies.
- Mend missing connections between surface bodies by joining their meshes with shared nodes.

To confirm the connectivity of surface bodies it is useful to review the connectivity of their edges using a number of features in both Mechanical and DesignModeler. Edges can be classified depending on the number of faces they topologically connect. For example, the boundary edge of a surface body connects to a single face and is classified as a "single edge", whereas an interior edge connecting two faces of the surface body will be classified as a "double edge". Single and double edges can be distinguished visually using the options of the **Edge** group (p. 84) on the **Display** tab. As an alternative, you can Create a Named Selection Object (p. 1057) that groups all edges of a given topological connectivity by using the **Face Connections** (p. 1061) criterion.

Edge group (p. 84) options can also be used to review the connectivity of not only the geometry, but also the mesh elements. The same principles applied to the connectivity of a surface body edge apply to element edges.

Mechanical provides Mesh Connections to mend surface body assemblies at locations that are disjointed. With this feature, the meshes of surface bodies that may reside in different parts can be connected by joining their underlying elements via shared nodes. The Mesh Connection does not alter the geometry although the effect can be conveniently previewed and toggled using the option in the **Edge** group (p. 84).

Using Thickness Mode

You can determine the source that controls the thickness of a surface body using the **Thickness Mode** indication combined with the **Thickness** field, both located in the Details view of a surface **Body** object. Upon attaching a surface body, the **Thickness Mode** reads either **Auto** or **Manual**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- In **Auto** Mode the value of thickness for a given surface body is controlled by the CAD source. Future CAD updates will synchronize its thickness value with the value in the CAD system.
- In **Manual** mode the thickness for the surface body is controlled by the Mechanical application, so future updates from the CAD system will leave this value undisturbed.
- A **Thickness Mode** will be **Automatic** until the **Thickness** is changed to some non-zero value. Once in **Manual** mode, it can be made **Automatic** once again by changing the **Thickness** value back to zero. A subsequent CAD update will conveniently synchronize the thickness with the value in the CAD system.

Thicknesses for all surface bodies are represented in a dedicated column on the **Worksheet** that is displayed when you highlight the **Geometry** object.

Importing Surface Body Models

To import a surface body model (called a sheet body in NX), open the model in the CAD system and import the geometry as usual. If your model mixes solid bodies and surface bodies, you should select which type of entity you want to import via the **Geometry** preferences in the Workbench **Properties** of the **Geometry** cell in the **Project Schematic**. Once in the Mechanical application, you can adjust the **Geometry** preferences in the Details view (p. 119), where they take effect upon updating.

Note:

If you want to retain a preference selection in the Workbench **Properties**, you must first save before exiting the Ansys Workbench.

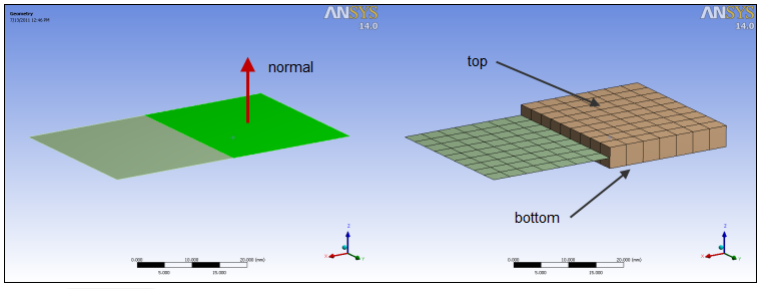
Importing Surface Body Thickness

When thickness is defined on the entire surface body

Surface body thickness will be imported from CAD (including DesignModeler) if, and only if, the existing surface body thickness value in the Mechanical application is set to 0 (zero). This is true on initial attach and if you set the surface body thickness value to zero prior to an update. This allows you the flexibility of updating surface body thickness values from CAD or not.

Understanding Surface Body Shell Offsets

Surface bodies have a normal direction, identified by a green coloring when the surface body face is selected. Shell elements have a "top" surface (farthest in the positive normal direction) and a "bottom" (farthest in the negative normal direction).

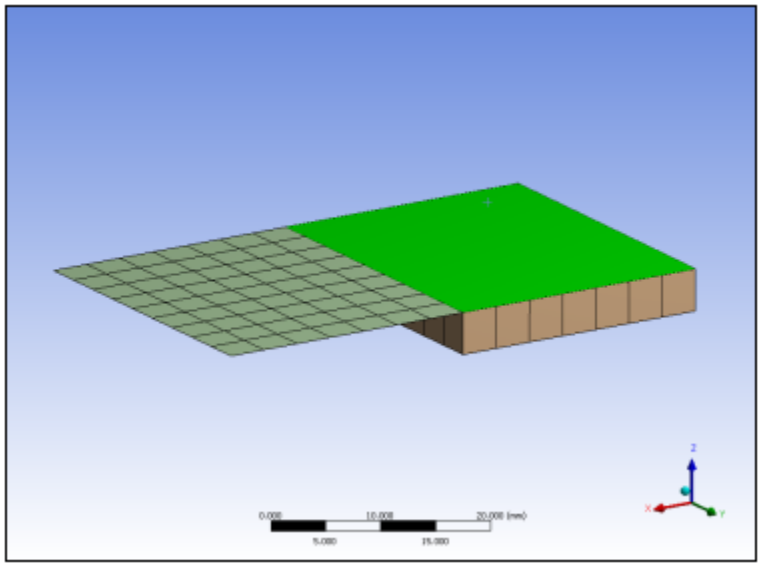


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

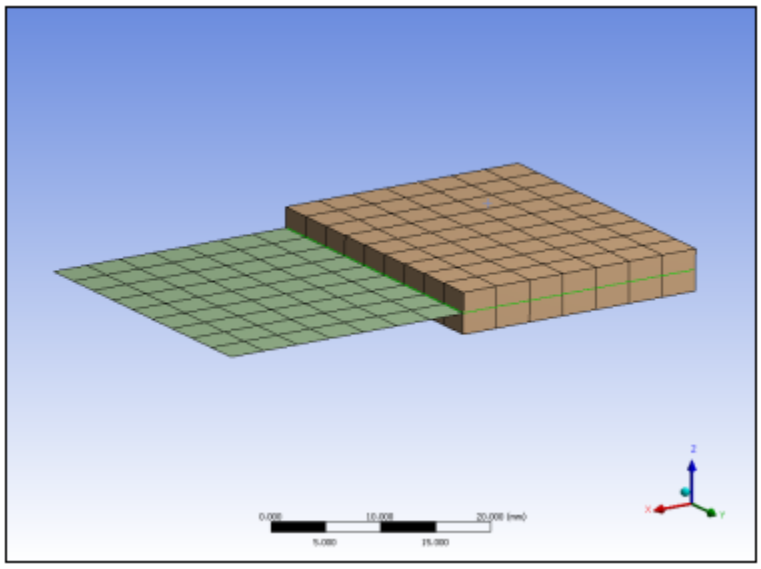
Published: 2023-10-23

By default, the shell section midsurface is aligned with the surface body, but you can use the **Offset Type** drop-down menu located in the Details view of a **Surface Body** object or an object scoped to a surface body to offset the shell section midsurface from the surface body:

- **Top** - the top of the shell section is aligned with the surface body.

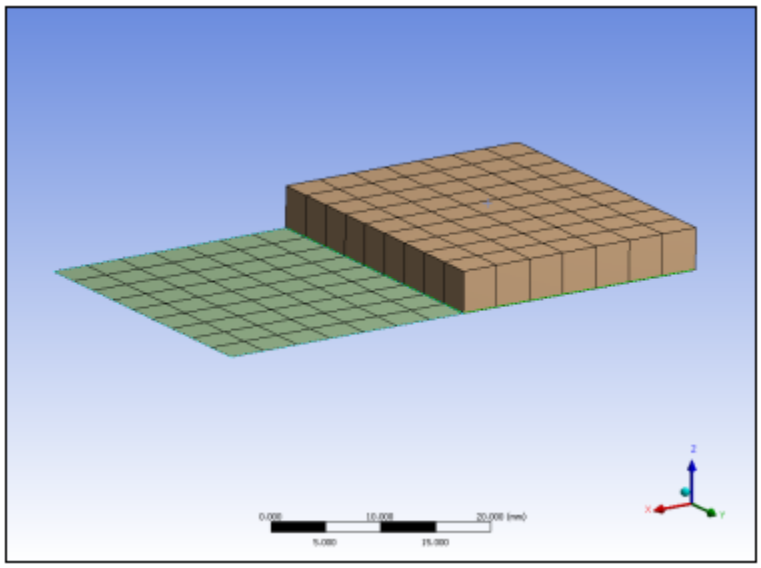


- **Middle (Membrane)** (default) - the middle of the shell section is aligned with the surface body.

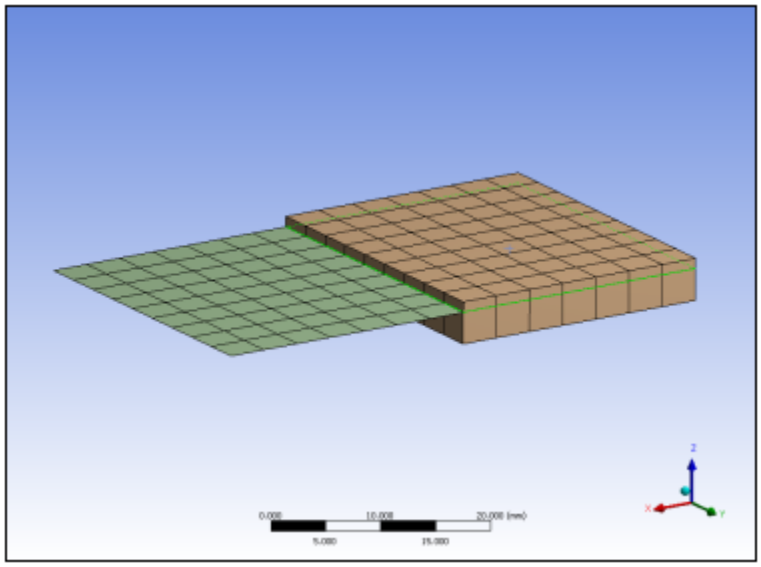


- **Bottom** - the bottom of the shell section is aligned with the surface body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- **User Defined** - the user defines the amount of offset (**Membrane Offset**), measured in the positive normal direction from the middle of the shell section to the surface body (may be positive or negative value).



Specifying Surface Body Thickness

The thickness of surface bodies can be defined in several ways:

1. A uniform thickness over the entire body which can be defined inside Mechanical or imported from a CAD system (p. 862). Thicknesses imported from CAD can be overridden by the Thickness Mode (p. 861)
2. A constant or spatially varying thickness applied to a selection of surfaces or bodies.
3. Thickness values imported from an upstream system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Layer information can be specified using a Layered Section (p. 868) object or imported through an Imported Plies object.

Note:

See Specifying Faces With Multiple Thicknesses and Layers (p. 871) for information on how Mechanical resolves conflicts when multiple thickness specifications are applied to the same geometry.

Application

To specify the thickness of an entire surface body:

Highlight the Surface Body object and, in the Details, enter a value in the **Thickness** property. A value greater than 0 must be present in this field.

To specify the thickness of selected faces on a surface body:

1. Highlight the **Geometry** object and insert a Thickness object from the **Geometry** Context tab or right-click and select **Insert** > **Thickness**.

Note:

The **Thickness** object overwrites any element that is scoped to the selected surfaces that has thickness greater than 0 defined in the Details of the Surface Body object (See above).

2. Scope the Thickness to a desired geometric or mesh entity. The application supports Body, Face, and Element scoping.
3. Define the thickness as a constant (default), with a table, or as a function:
   a. To define the thickness as a constant, enter the value in the **Thickness** field in the Details view.
   b. To define the thickness with a table:
      i. Click the **Thickness** property in the Details, then select **Tabular** from the flyout menu.
      ii. Set the **Independent Variable** in the Details to **X**, **Y**, or **Z**.
      iii. Select a **Coordinate System**. The **Global Coordinate System** (Cartesian) is the default.
      iv. Enter data in the **Tabular Data** window. The **Graph** window displays the variation of the thickness.
   c. To define the thickness with a function:
      i. Click the **Thickness** field in the Details, then click **Function** from the flyout menu.
      ii. Enter the function in the **Thickness** field. (Example: 45+10*x/591)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

iii. Adjust properties in the **Graph Controls** category as needed:

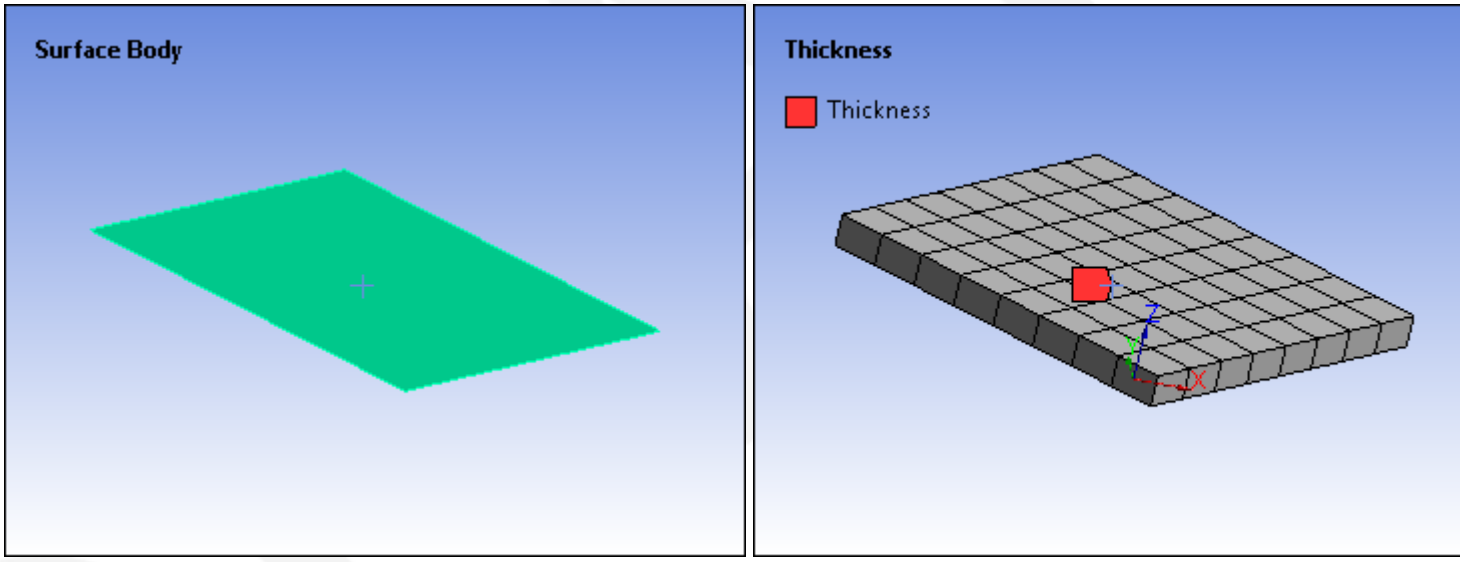
- **Number of Segments** - The function is graphed with a default value of 200 line segments. You can change this value to better visualize the function.
- **Range Minimum** - The minimum range of the graph.
- **Range Maximum** - The maximum range of the graph.

4. Set the desired shell offset (p. 862).

Review the requirements and limitations for this feature in the notes (p. 866) topic below.

Thickness Display Options

In the **Geometry** window, the application draws the thickness value you specify in the **Thickness** *property* when you 1) select the **Show Mesh** option and the **Thick Shells and Beams** option, 2) when you have the **Mesh** object selected, or 3) when you have a result selected. An example of this display is shown below. As illustrated, there is a Surface Body with no thickness and a **Thickness** object selected with a **Thickness** property value of 20mm displayed using the **Show Mesh** and **Thick Shells and Beams** options.



Important:

This display feature is not supported when you 1) scope the **Thickness** object to Elements and 2) specify the **Thickness** property using the **Tabular** or **Function** options.

Note:

- Surface body thicknesses must be greater than zero. Failures will be detected by the solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- When importing surfaces bodies from DesignModeler, the associated thickness is automatically included with the import. See Importing Surface Body Thickness (p. 862) for details.
- For the following, the nominal thickness of the body is used as opposed to the thickness controlled by the **Thickness** objects being added (**Thickness**, **Layered Thickness**, **Imported Layered Thickness**):
  - **Geometry**: the read-only physical properties, **Volume**, **Mass**, **Centroid**, and **Moment of Inertia** provided under the Details view **Properties** category of the Geometry object, Part object, and the Body object. The properties based on any variable thickness are correctly calculated in the solver and can be verified through miscellaneous record results for Mechanical APDL based solutions.
  - **Meshing**: during the meshing process, auto-detection based on surface body thickness, automatic pinch controls, surface body thickness used as mesh merging tolerance.
  - **Solution**: Heuristics used in beam properties for spot welds as well as moments applied to vertices.
- **Thickness** objects are not supported for rigid bodies.
- Variable thickness is displayed only for mesh and result displays. Location probes, **Path** scoped results, and **Surface** scoped results do not display nor account for variable thickness. They assume constant thickness.
- If multiple **Thickness** objects are applied to the same face, only those properties related to the last defined object will be sent to the solver, regardless of whether the object was defined in DesignModeler or in Mechanical. See Specifying Faces With Multiple Thicknesses and Layers (p. 871) for details.

You can import thicknesses from an upstream system. Basic setup steps are given below. You can find more information on mapping data in the Mechanical application in the appendix (Appendix B: Data Transfer Mapping and Validation (p. 2389)).

Note:

Thickness import is supported for 3D shell bodies or planar 2D bodies using Plane Stress. The Mechanical APDL Solver for 3D shell bodies will use the nodal thicknesses directly via the **SECFUNCTION** command. For the Explicit Solver or MAPDL solver for 2D bodies, the element's nodal thicknesses are converted to an average element thickness.

To import thicknesses from an upstream system:

1. In the project schematic, create a link between the **Solution** cell of a system and the **Model** cell of an upstream system.
2. Attach geometry to the analysis system, and then double-click **Model** cell to open Mechanical. An **Imported Thickness** folder is added under the **Geometry** folder and an imported thickness is added to the **Imported Thickness** folder, by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Select the appropriate options in the Details.
4. Select Imported Thickness and select **Import Thickness** from the context menu.

Specifying Surface Body Layered Sections

Layers applied to a surface body can be prescribed in several ways:

- A defined Layered Section object can be scoped to a selection of surfaces on the geometry.
- An Imported Plies object can provide layer information for the elements within a surface body.

Note:

Layered Section objects can only be used in the following analysis types:

- Explicit Dynamics
- Harmonic Response
- Eigenvalue Buckling
- Modal
- Random Vibration
- Response Spectrum
- Static Structural
- Transient Structural

The following sections describe the use of the Layered Section object.

Defining and Applying a Layered Section
Viewing Individual Layers
Layered Section Properties
Notes on Layered Section Behavior

Defining and Applying a Layered Section

1. Select the **Geometry** object in the tree and insert a **Layered Section** object from the **Geometry** Context tab or choose **Insert > Layered Section** (right-click and choose from context menu).
2. Select the **Scoping Method** that you will use:
   - Geometry Selection: Click in the **Geometry** field that appears, to enable you to pick surface bodies or individual faces from the model and select **Apply**.
   - Named Selection: Click on the **Named Selection** drop down that appears and select one of the available named selections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. Choose a **Coordinate System**. You may choose any user-defined Cartesian or Cylindrical coordinate system. The Body Coordinate System option specifies that the coordinate system selected for each body will be used. There is no default.

4. Set the desired **Offset Type** (p. 862). Offset Type is not supported in Explicit Dynamics analyses.

5. Click on the arrow to the right of Worksheet in the **Layers** field then select **Worksheet** to enter the layer information for this Layered Section. The Layered Section worksheet can also be activated by the **Worksheet** option.

   The worksheet displays a header row, and two inactive rows labeled +Z and -Z to indicate the order in which the materials are layered. Layer one will always be the layer at the bottom of the stack (closest to -Z). When you insert a layer, all of the layers above it will renumber.

   To add the first layer, right-click anywhere in the Layered Section Worksheet and select **Add Layer**. Once the layer is added:

   - Click in the **Material** column of the row and select the material for that layer from the drop-down list.
   - Click in the **Thickness** column and define the thickness of that layer. Individual layers may have zero thickness, but the total layered-section thickness must be nonzero.
   - Click in the **Angle** column and define the angle of the material properties. The angle is measured in the element X-Y plane with respect to the element X axis. This value can be entered as degrees or radians, depending on how units are specified.

   To add another layer, do one of the following:

   - With no layers selected, you can right-click the header row, +Z row, or -Z row to display a context menu. Select Add Layer to Top to add a layer row at the top (+Z) of the worksheet. Select Add Layer to Bottom to add a layer row to the bottom of the worksheet (-Z).
   - With one or more layers selected, you can right-click any selected layer to display a context menu. Select Insert Layer Above (which inserts a layer row above the selected row in the +Z direction) or Insert Layer Below (which inserts a layer row below the selected row in the -Z direction).

   To delete a layer, select one or more rows, right-click any selected row, and select **Delete Layer**.

6. Select the **Nonlinear Effects** (p. 345) and **Thermal Strain Effects** (p. 346) settings in the Material category of the Details view. The reference temperature specified for the body on which a layered section is defined is used as the reference temperature for the layers.

   Nonlinear Effects and Thermal Strain Effects are not supported in Explicit Dynamics analyses.

Viewing Individual Layers

In the Graphics Properties section of the Details panel, the **Layer To Display** field allows the visualization of the thickness/offset/layer sequence of the layers composing a Layered Section object. To view a particular layer, click the field and enter the layer number. You can use the up and down buttons or enter a layer number directly. If you enter a number larger than the maximum number

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

of layers in that layered section, the value will be set to the maximum number of layers in that layered section. If layer zero is selected, all the layers will be drawn (without the delineation between layers) as a compact entity, shown the same as when the Mesh node is selected in the tree. All other geometry not scoped to the current Layered Section object is shown with thickness zero.

Individual layers will be visible only when Show Mesh is enabled (if the model has been meshed previously), and only on Layered Section objects. If Show Mesh is not enabled, just the geometry and the scoping will be shown on the model.

When a layer is selected to display, the layer with its defined thickness, offset, and sequence will be displayed in the graphics window. Due to the limitations described for the Show Mesh option, it is recommended that the user switch back and forth if needed to Wireframe/Shaded Exterior View mode to properly see annotations.

Note:

When viewing Imported Plies, the thickness that you see is not relative to the geometry like it is with a **Layered Section** object.

Layered Section Properties

The following Properties are displayed in Details panel for Layered Sections:

- **Total Thickness** - Total thickness of the section, including all of the layers defined for the section. Used when displaying the mesh.
- **Total Mass** - Total mass of all of the layers in the section. The density of the material for each layer is calculated at a reference temperature of 22° C.

Notes on Layered Section Behavior

Note:

- If multiple thickness objects (including **Layered Section** objects) are applied to the same face, only those properties related to the last defined object will be sent to the solver, regardless of whether the object was defined in DesignModeler or in Mechanical. See Specifying Faces With Multiple Thicknesses and Layers (p. 871) for details.
- If adjacent elements within the same part have different thickness values, the elements will appear to be ramped.
- Layered Sections cannot be scoped to rigid bodies.
- Layered Sections do not affect the following items:
  - Assembly properties: volume, mass, centroid, and moments of inertia. This is for display in the Details view only. The correct properties based on any variable thickness are correctly calculated in the solver and can be verified through miscellaneous record results for Mechanical APDL based solutions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Meshing: auto-detection based on surface body thickness, automatic pinch controls, surface body thickness used as mesh merging tolerance.
  - Solution: Heuristics used in beam properties for spot welds.
- A Thermal Condition applied to a Layered Section is only valid if applied to both shell faces (**Shell Face** is set to **Both**, not to **Top** or **Bottom**).
- Layered Sections are not valid with cyclic symmetry.
- The following material properties are supported by Layered Sections in an Explicit Dynamics analysis:
  - Isotropic Elasticity, Orthotropic Elasticity
  - Johnson Cook Strength, Zerilli Armstrong Strength, Steinberg Guinan Strength, Cowper Symonds Strength
  - Orthotropic Stress Limits, Orthotropic Strain Limits, Tsai-Wu Constants
  - Plastic Strain, Principal Stress, Stochastic Failure,
- For orthotropic materials in Explicit Dynamics, the Z material direction is always defined in the shell normal direction. The X material direction in the plane of each element is determined by the x-axis of the coordinate system associated with the Layered Section. If the x-axis of this coordinate system does not lie in the element plane, then the x-axis is projected onto the shell in the coordinate system z-axis direction. If the z-axis is normal to the element plane, then the projection is done in the coordinate system y-axis. For cylindrical systems, it is the y-axis that is projected onto the element plane to find the Y material direction.

Specifying Surface Body Reinforcements

The **Model Type** property for the Surface Body object enables you to specify the body as a reinforcement. The options for the property include **Shell** (default) and **Reinforcement**. The **Shell** setting treats the body as a surface body - there is no reinforcement treatment.

See the Reinforcement Specification Using Mesh-Independent Method (p. 775) section for the steps to specify a Surface Body as a reinforcement.

Specifying Faces With Multiple Thicknesses and Layers

Thickness and **Layered Section** (p. 868) objects may be scoped to more than one face of a surface body. As a result, a face may have more than one thickness definition.

The order of precedence used to determine the thickness that will be used in the analysis is as follows:

1. Imported Plies objects
2. Imported Thickness objects

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Layered Section objects
4. Thickness objects
5. Thickness as a property of a body/part

For multiple objects of the same type, the object lower in the tree (more recently created) will be used in the analysis.

This thickness may not be the desired thickness to be used in the analysis. In a large model, you may want to fix this problem prior to solving the model.

You can search for faces with multiple thicknesses by selecting **Search Faces with Multiple Thicknesses** from the context menu of any of the following: the **Geometry** folder, a **Body** object (individual or group of objects), a **Thickness** object or a **Layered Section** object. For each face found with multiple thicknesses, a warning message similar to the one shown below will be displayed in the message box.

This face has more than one thickness defined. You may graphically select the face via RMB on this warning in the Messages window.

To find the face and its corresponding thickness objects for a particular message, highlight that message in the message pane, right-click the message and choose **Go To Face With Multiple Thicknesses** from the context menu. The face associated with this message is highlighted in the **Geometry** window and the corresponding thickness objects are highlighted in the tree.

If there is no face with multiple definitions, the following information will be displayed in the message box.

```
No faces with multiple thicknesses have been found.
```

A related Go To (p. 110) option is also available. If you highlight one or more faces with thickness definition of a surface body, then right-click in the **Geometry** window and choose **Go To** > **Thicknesses for Selected Faces**, the corresponding thickness objects will be highlighted in the tree.

Note:

You cannot search for Imported Plies that overlap with other thickness objects. However a warning will be generated during the solution if this situation might exist.

Line Bodies

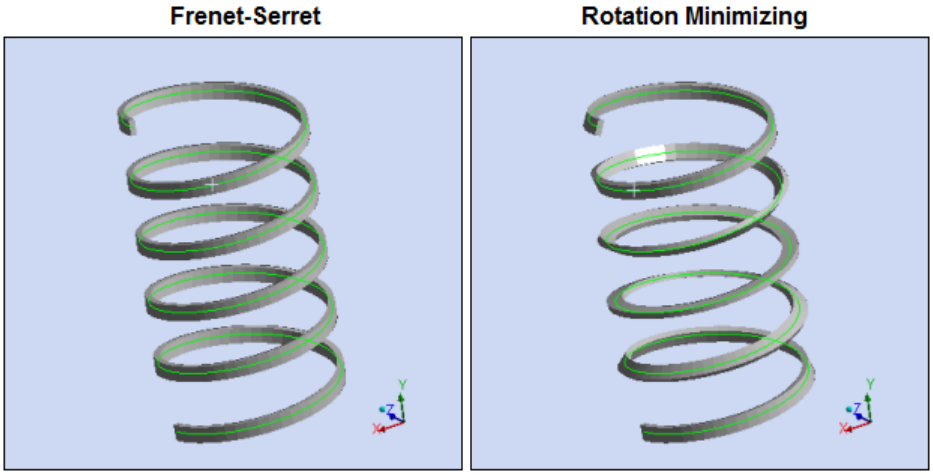
A line body consists entirely of edges and does not have a surface area or volume. Although multiple CAD sources can provide line bodies to Ansys Workbench, only DesignModeler and Ansys SpaceClaim Direct Modeler provide the additional cross section data needed to use line bodies in an analysis. For those CAD sources that cannot provide the cross section data, you need to import them into DesignModeler or Ansys SpaceClaim Direct Modeler, define the cross sections, and then send the geometry to the Mechanical application in Ansys Workbench. Cross sectional data is imported into Mechanical and provided by Cross Section Objects.

In addition to specifying the cross section type and offset, DesignModeler and SpaceClaim also allow you to align cross sections to ensure they have the proper orientation. For more information on Frame

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Alignment and how to override the default alignment algorithm, see Cross Section Alignment. Illustrated below are examples of the alignment algorithm options from DesignModeler.



Go to a section topic:

- Overview (p. 873)
- Application (p. 875)
- Thickness Display (p. 876)
- Cross Sections (p. 877)

Note:

See the Line Body Results (p. 1938) section for additional information about the display of results scoped to line bodies.

Overview

Once imported, a line body is represented by a **Line Body** object in the tree, where the **Details** pane includes the associated cross section information of the line body that was defined in SpaceClaim Direct Modeler, DesignModeler, or supported CAD system. Depending on your application, you can further define the line body using the **Model Type** property. Options include **Beam**, **Pipe**, **Link/Truss**, **Cable**, **Thermal Fluid** (thermal analysis only), **Reinforcement**, or the read-only option **Axisymmetric Shell**. Review the following guidelines:

- **Beam** (BEAM188/BEAM189): This option is usually a suitable option when analyzing thin to moderately thick beam structures. A variety of cross-sections can be associated with beams.

Note:

When a beam body is meshed with BEAM188, KEYOPT(3) = 2 (the quadratic shape function) is written to the input file. This differs from the usual default setting of BEAM188 in Mechanical APDL (KEYOPT(3) = 0, linear shape function). Note that this element is a lower order element and is only used when the Mesh object property **Element Order** is specified

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

as **Linear**, either by you as the user or by the application. See the *Mechanical APDL Element Reference* for BEAM188 for more information.

- **Pipe** (PIPE288/PIPE289/ELBOW290): This option is suitable for analyzing initially circular cross-sections and thin to moderately thick pipe walls. You can apply special loads on pipes such as Pipe Pressure (p. 1532) and Pipe Temperature (p. 1534). Curved pipe zones or high deformation zones in pipes can be further modeled using the Pipe Idealization (p. 1704) object.
- **Link/Truss** (LINK180): This option is a suitable option when analyzing uniaxial tension-compression scenarios. For stress and strain output, the Mechanical APDL solver only produces the AXL component (such as SX, EPELX, EPPLX, ...). This element type is not supported by the Beam Tool or the Beam Probe.

  Note:

  Meshing does not distinguish between the different types of 1D elements in the meshing process. Hence, beam orientation nodes are created for beam elements and Link elements even though Link elements do not need any orientation nodes. The beam orientation nodes created for the Link elements are not getting connected to the Link elements but are simply ignored during the solution process.

- **Cable** (LINK180/CABLE280): This option enables you to simulate tensile load transfer along the length of the line body. For a linear order mesh, this option uses the LINK180 element. When the **Element Order** property of the Mesh object is set to **Program Controlled** or for a quadratic order mesh, this option uses the CABLE280 element. For stress and strain output, the Mechanical APDL solver only produces the AXL component (for example, SX, EPELX, EPPLX,...). The element types do not support the **Beam Tool** or the **Beam Probe**.

  Note:

  Linear element order creates LINK180 elements with **Tension** and **Compression** setting as default TENSKEY for **SECCONTROL**.

- **Thermal Fluid**: Thermal fluid flow models heat distribution between fluid and solid bodies during steady-state and transient thermal analyses. This selection activates the Mechanical APDL element FLUID116 with **TEMP** (Temperature) as the degree of freedom. Activating Fluid Flow (via **Convection** (p. 1602)) is equivalent to a reduced-order model for a Computational Fluid Dynamic (CFD) analysis, for a one-dimensional fluid flow. This provides an accurate solution that does not require significant computation time. Heat flow is generated by the conduction within the fluid and the mass transport of the fluid (p. 1622).

  Note:

  If your analysis includes the material property **Specific Heat**, defined in Engineering Data, the **Thermal Fluid** setting only supports Specific Heat defined at Constant Pressure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Reinforcement** (p. 775) (REINF264): This option models a line body as a reinforcement within a structure. It is supported for 3D Static Structural and Steady-State Thermal analyses only and requires that your imported model already include the line bodies you wish to specify as reinforcements as well as an appropriate material to assign to the reinforcement bodies.
- **Axisymmetric Shell** (SHELL208/SHELL209): This read-only option is **only** displayed when you create a Line Body from a Line Coating (p. 1038) using the Pull feature.

Note:

For the **Link/Truss** and **Cable** options, the application meshes the line body with only one element if the body is a line with a single edge. You can override this behavior by specifying a mesh **Sizing** control that has the **Behavior** property set to **Hard**.

Application

To define your line body, highlight the **Line Body** object and set the following in the Details view:

1. **Stiffness Behavior** (p. 343): As applicable, use this property to modify the stiffness behavior of the line body. Options include **Flexible** (default) and **Stiff Beam**.
2. **Coordinate System** (p. 343): As needed, use this property to change the assigned default coordinate system.
3. **Reference Temperature** (p. 344): As needed, use this property to change the default reference temperature that is taken from the environment.
4. **Cross Section**: Select a cross section from the drop-down list of available Cross Section Objects. This property is not visible when the **Model Type** property (below) is set to **Thermal Fluid**.
5. **Offset Mode**: Set to **Refresh on Update** (default) to enable the values in the Details view to update when the CAD system updates or to **Manual**, to enable the Details view values to override the CAD system updates.
6. **Offset Type**: Set to **Centroid**, **Shear Center**, **Origin**, or **User Defined**, where **Offset X** and **Offset Y** are available.
7. **Model Type**: Options include **Beam** (default), **Thermal Fluid** (thermal analysis only), **Pipe**, **Link/Truss**, **Cable**, or **Reinforcement**. Note the following specifics for certain options.

   Thermal Fluid

   When you select the **Thermal Fluid** option, you need to evaluate the following additional properties:

   - **Fluid Cross Area**: Defines the fluid cross-section area of the Line Body. You can modify this value, however; by default, the application uses the value provided by your geometry application. And, updating your source Geometry resets this property to the default value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Discretization**: Options include **Upwind/Linear**, **Central/Linear**, and **Upwind/Exponential**. Refer to the Discretization scheme described in the *Mechanical APDL Element Reference* for FLUID116.

Link/Truss

When you select the **Link/Truss** option, the **Behavior** property also displays. Options include **Tension and Compression** (default), **Tension Only**, and **Compression Only**.

Reinforcement

This option enables you to specify line bodies as reinforcing fibers (p. 775) inside of a structure.

Axisymmetric Shell (Read-Only)

Only visible for a Line Body created from a Surface Coating (p. 1038). This is available using the mesh-based Pull feature. Review display characteristics in the Line Body Results (p. 1938) section.

8. **Cross Section (For Solver)**: This property displays when your Line Body includes a user-defined cross-section. It enables you to send user-defined cross-sections to the MAPDL solver as either a **Pre-Integrated** (default) cross-section or as a **Mesh** section. These options are specified in the Geometry (p. 225) preference category of the **Options** dialog.

   Selecting **Pre-Integrated** sends the integrated cross-section inertia properties, calculated in DesignModeler or Ansys SpaceClaim Direct Modeler, to the solver using the **SECTYPE**,,**BEAM**,**ASEC** command. The **Mesh** option sends the mesh section data, generated in DesignModeler or Ansys SpaceClaim Direct Modeler, to the solver using the **SECTYPE**,,**BEAM**,**MESH** command. The **Mesh** option enables you to post process results on the entire mesh section.

   Note:

   You may see slight result differences between the **Pre-integrated** and **Mesh** settings as a result of how the application performs the calculations. There are certain limitations associated with the cross-section input data when using the **Pre-integrated** setting.

9. Specify **Material** category properties as needed.

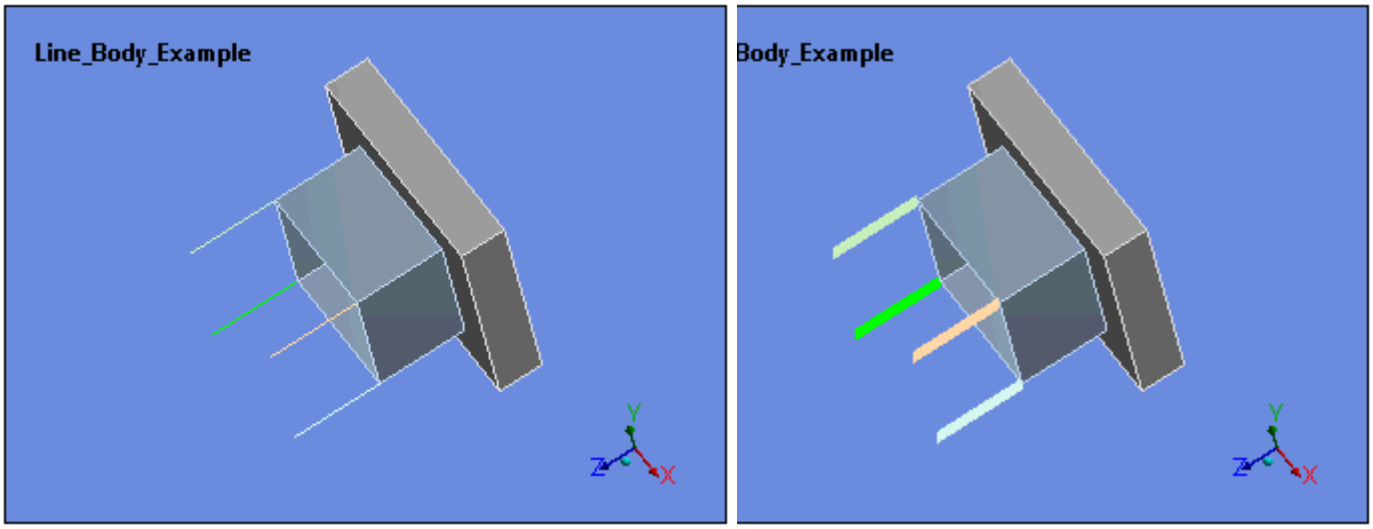
See the Body object reference section for a complete listing of all **Line Body** properties.

Thickness Display

As illustrated below, you can increase the thickness of a line body's graphical display in the **Geometry** window by changing the default setting of the **Line Body Thickness** property in the **Graphics** (p. 228) preference of the application **Options**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- Beams can also be used as connections within a model. See Beam Connections (p. 1404) for further information on this application.
- Pipes are only realized in structural analyses. All line bodies defined in other analysis types are always realized as beams. This extends to linked analyses as well. For example, in a thermal-structural linked analysis where line bodies are defined as pipes, the thermal component of the analysis will only realize the line bodies as beams.

Cross Sections

By default, line bodies are displayed as lines in the **Geometry** window, with no graphical indication of a cross section. If cross sections are defined in a line body, you can select the **Cross Section** option from the **Style** (p. 80) group of the **Display** tab to display the line bodies as solids (3D), enabling you to visually inspect the cross sections. This visualization can be useful in determining the correct orientation of line bodies. For circular and circular tube cross sections, the number of divisions used for rendering the line bodies as solids has an adjustable range from 6 to 360 with a default of 16. You can adjust this using the **Number of Circular Cross Section Divisions** option in the **Graphics** (p. 228) category of the **Options** dialog.

The **Cross Section** display option has the following characteristics:

- By default, the feature is not enabled. However, when activated, it will persist as a session preference.
- Only geometry displays are applicable. The feature is not available for mesh displays.
- When the feature is enabled, both normal lines and solid representations are drawn.
- The solid representation of the geometry cannot be selected nor meshed, and has no effect on quantitative results.
- The feature supports section planes and works with all line body cross sections (primitive and user defined).
- User Integrated cross sections (direct entry of the properties) have no display.
- The feature is not available for use with viewports (p. 284).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- This feature is not supported on the Linux platform.
- When you import geometry from an External Model system, geometric edges may be collapsed and not display the actual orientations that are specified in the external model file.

Tip:

Use the **Show Mesh** (p. 97) option to visualize the proper orientations as defined in the file. These are the orientations that the application sends to the solver.

Simulation without Geometry

The application does not require you to import a geometry for your analysis. You can open an analysis system without importing a geometry and then specify all of your desired environmental conditions. Once complete, you save your project for use with any desired model. See the Create a Simulation Template (p. 346) section for additional information.

2D Analyses

Mechanical enables you to run two-dimensional (2D) simulations for structural and thermal analyses. Two-dimensional simulations can save processing time and conserve machine resources for models and environments that involve negligible effects from a third dimension. You specify your analysis as two-dimensional on the Workbench project page (**Analysis Type** property for the **Geometry** cell set to **2D**). However, as desired, you can change the setting of the Analysis Type property to switch your analysis to 3D at any time.

Important:

- Switching your analysis from 2D to 3D (or vise versa) could invalidate certain specifications you have defined in your analysis. Specifically, if you have defined parameters, they may become invalid and although it would display in the Parameter Manager, it would not be available in Mechanical.
- When you are performing a 3D simulation on shell bodies (only), you can specify a shell body as 2D using the **Dimension** property of the Body object.
- For a line body in a 2D thermal analysis, you can set the **Model Type** property to **Thermal Fluid**.

Go to a section topic:

- Specify 2D Analysis in Workbench (p. 879)
- Define the 2D Behavior of Geometry (p. 879)
- Review Requirements and Limitations (p. 880)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specify 2D Analysis in Workbench

For a 2D analysis, Mechanical supports surface (p. 860) and line (p. 872) bodies that are planar bodies on the X-Y plane. To create a 2D analysis:

1. Add an analysis to the Workbench Project page
2. Open your model in the appropriate Ansys application, such as Discovery, or another supported CAD system, and specify that the model's orientation is in the x-y plane.
3. On the project page, select the **Geometry** cell of the analysis system and set the **Analysis Type** property to **2D** (Advanced Geometry Options).
4. Open the model in Mechanical.

Define the 2D Behavior of Geometry

Specify the **2D Behavior** property of the Geometry object. Options include:

- **Plane Stress** (default): Assumes zero stress and non-zero strain in the z direction. Use this option for structures where the z dimension is smaller than the x and y dimensions. Example uses are flat plates subjected to in-plane loading, or thin disks under pressure or centrifugal loading. A **Thickness** field is also available if you want to enter the thickness of the model.
- **Axisymmetric**: Assumes that a 3D model and its loading can be generated by revolving a 2D section 360^{o} about the y-axis. The axis of symmetry must coincide with the global y-axis. The geometry must lie on the positive x-axis of the x-y plane. The y direction is axial, the x direction is radial, and the z direction is in the circumferential (hoop) direction. The hoop displacement is zero. Hoop strains and stresses are usually very significant. Example uses are pressure vessels, straight pipes, and shafts. You may wish to review the Axisymmetric Loads and Reactions section, of the *Mechanical APDL Basic Analysis Guide*, for a description about how to apply constraints in order to prevent unwanted rigid-body motions.

  Note:

  Certain CAD applications automatically increase the bounding box size beyond the exact limits of the geometry and can cause the geometry to appear in the negative X plane. This causes Mechanical to generate an error and prohibit a solution. In this scenario, you can change the error setting to a warning in order to perform a solution. You use the **Geometry** (p. 225) preference **2D Axisymmetric Check** in the **Options** dialog to change this setting.

- **Plane Strain**: Assumes zero strain in the z direction. Use this option for structures where the z dimension is much larger than the x and y dimensions. The stress in the z direction is non-zero. Example

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

uses are long, constant, cross-sectional structures such as structural line bodies. **Plane Strain** behavior cannot be used in a thermal analysis (steady-state (p. 657) or a transient (p. 680)).

Note:

Since thickness is infinite in plane strain calculations, different results (displacements/stresses) will be calculated for extensive loads (that is, forces/heats) if the solution is performed in different unit systems (MKS vs. NMM). Intensive loads (pressure, heat flux) will not give different results. In either case, equilibrium is maintained and thus reactions will not change. This is an expected consequence of applying extensive loads in a plane strain analysis. In such a condition, if you change the Mechanical application unit system after a solve, you should clear the result (p. 1981) and solve again.

- **Generalized Plane Strain**: Assumes a finite deformation domain length in the z direction, as opposed to the infinite value assumed for the standard **Plane Strain** option. **Generalized Plane Strain** provides more practical results for deformation problems where a z direction dimension exists, but is not considerable. See the next section, Using Generalized Plane Strain (p. 881), for more information.

  The **Generalized Plane Strain** option requires you to specify the following properties:

  – **Fiber Length**: Sets the length of the extrusion.

  – **End Plane Rotation About X**: Sets the rotation of the extrusion end plane about the x-axis.

  – **End Plane Rotation About Y**: Sets the rotation of the extrusion end plane about the y-axis.

- **By Body**: Enables you to set the **Plane Stress** (with **Thickness** option), **Plane Strain**, or **Axisymmetric** options for individual bodies that appear under **Geometry** in the tree. If you choose **By Body**, then click an individual body, these 2D options are displayed for the individual body.

Review Requirements and Limitations

Requirements for Specifying Boundary Conditions

For a 2D analysis, you use the same procedure for applying loads and supports (p. 1497) as you would use in a 3D analysis. However, the loads and results are in the x-y plane and there is no z component. All loads and supports, except the following:

- Line Pressure
- Simply Supported
- Fixed Rotation

Important:

When your 2D analysis includes any of the following boundary conditions, the results may not be accurate in certain cases where more than one element type has already been created on the scoped surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Hydrostatic Pressure
- Force (Applied By property set to Surface Effect)
- Pressure (Applied By property set to Surface Effect)
- Bearing Loads
- Elastic Support
- Heat flux
- Radiation
- Convection

Please review the output file carefully for related messages before looking at the results. See the SURF151 and SURF153 sections in the Mechanical APDL Element Reference for more information.

General Limitations

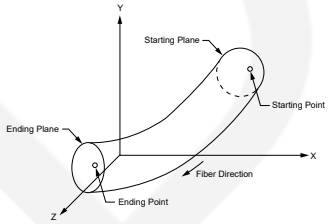
Note the following limitations:

- A Pressure load can only be applied to an edge.
- A Bearing Load and a Cylindrical Support can only be applied to a circular edge.
- For analyses involving axisymmetric behavior, Rotational Velocity loads can only be applied about the y-axis.
- For loads applied to a circular edge, the direction flipping in the z axis will be ignored.
- Only Plain Strain and Axisymmetric are supported for Explicit Dynamics analyses.
- Mechanical does not support Cyclic results for a 2D Analysis.

Using Generalized Plane Strain

This feature assumes a finite deformation domain length in the z direction, as opposed to the infinite value assumed for standard plane strain. It provides a more efficient way to simulate certain 3D deformations using 2D options.

The deformation domain or structure is formed by extruding a plane area along a curve with a constant curvature, as shown below.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The extruding begins at the starting (or reference) plane and stops at the ending plane. The curve direction along the extrusion path is called the fiber direction. The starting and ending planes must be perpendicular to this fiber direction at the beginning and ending intersections. If the boundary conditions and loads in the fiber direction do not change over the course of the curve, and if the starting plane and ending plane remain perpendicular to the fiber direction during deformation, then the amount of deformation of all cross sections will be identical throughout the curve, and will not vary at any curve position in the fiber direction. Therefore, any deformation can be represented by the deformation on the starting plane, and the 3D deformation can be simulated by solving the deformation problem on the starting plane. The **Plane Strain** and **Axisymmetric** options are particular cases of the **Generalized Plane Strain** option.

All inputs and outputs are in the global Cartesian coordinate system. The starting plane must be the x-y plane, and must be meshed. The applied nodal force on the starting plane is the total force along the fiber length. The geometry in the fiber direction is specified by the rotation about the x-axis and y-axis of the ending plane, and the fiber length passing through a user-specified point on the starting plane called the starting or reference point. The starting point creates an ending point on the ending plane through the extrusion process. The boundary conditions and loads in the fiber direction are specified by applying displacements or forces at the ending point.

The fiber length change is positive when the fiber length increases. The sign of the rotation angle or angle change is determined by how the fiber length changes when the coordinates of the ending point change. If the fiber length decreases when the x coordinate of the ending point increases, the rotation angle about y is positive. If the fiber length increases when the y coordinate of the ending point increases, the rotation angle about x is positive.

For Eigenvalue Buckling (p. 487) and Modal (p. 515) analyses, the **Generalized Plane Strain** option usually reports fewer Eigenvalues and Eigenvectors than you would obtain in a 3D analysis. Because it reports only homogeneous deformation in the fiber direction, generalized plane strain employs only three DOFs to account for these deformations. The same 3D analysis would incorporate many more DOFs in the fiber direction.

Because the mass matrix terms relating to DOFs in the fiber direction are approximated for Modal and Transient analyses, you cannot use the lumped mass matrix for these types of simulations, and the solution may be slightly different from regular 3D simulations when any of the three designated DOFs is not restrained.

Overall steps to using Generalized Plane Strain

1. Attach a 2D model in the Mechanical application.
2. Click on **Geometry** in the tree.
3. In the Details view, set **2D Behavior** to **Generalized Plane Strain**.
4. Define extrusion geometry by providing input values for **Fiber Length**, **End Plane Rotation About X**, and **End Plane Rotation About Y**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. Add a **Generalized Plane Strain** load (p. 1582) under the analysis type object in the tree.

   Note:

   The **Generalized Plane Strain** load is applied to all bodies. There can be only one **Generalized Plane Strain** load per analysis type so this load will not be available in any of the load drop-down menu lists if it has already been applied.

6. In the Details view, input the x and y coordinates of the reference point, and set the boundary conditions along the fiber direction and rotation about the x and y-axis.
7. Add any other loads or boundary conditions that are applicable to a 2D model.
8. Solve. Reactions (p. 2107) are reported in the Details view of the **Generalized Plane Strain** load.
9. Review results.

Construction Geometry

The **Construction Geometry** object is a **Model**-level object in the Outline hierarchy. The **Construction Geometry** drop-down menu in the **Prepare** group of the **Model Context** tab enables you to insert a desired **Construction Geometry** type, including:

Path

Surface

Solid

STL

Construction Line

Note:

The **Path** and **Surface** construction geometry features enable you to create a specific geometry scoping (curve or plane) and then use this scoping to specifically define a result item.

Path

A **Path** is categorized as a form of construction geometry and is represented as a spatial curve to which you can scope path results (p. 1940). The results are evaluated at discrete points along this curve.

You defined a **Path** using the following methods:

By Start and End Point

These points can be specified directly or can be calculated from the entry and exit point (intersections) of the positive X-axis of a coordinate system through a mesh. The path may be a straight line segment or a curve depending on the type of coordinate system (Cartesian or Cylindrical).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You can control the discretization by specifying the number of sampling points, and these will be evenly distributed along the path up to a limit of 200.

Note:

Paths defined in this manner will only be mapped onto solid or surface bodies. If you wish to apply a path to a line body you must define the path by an edge (as described below).

Edge

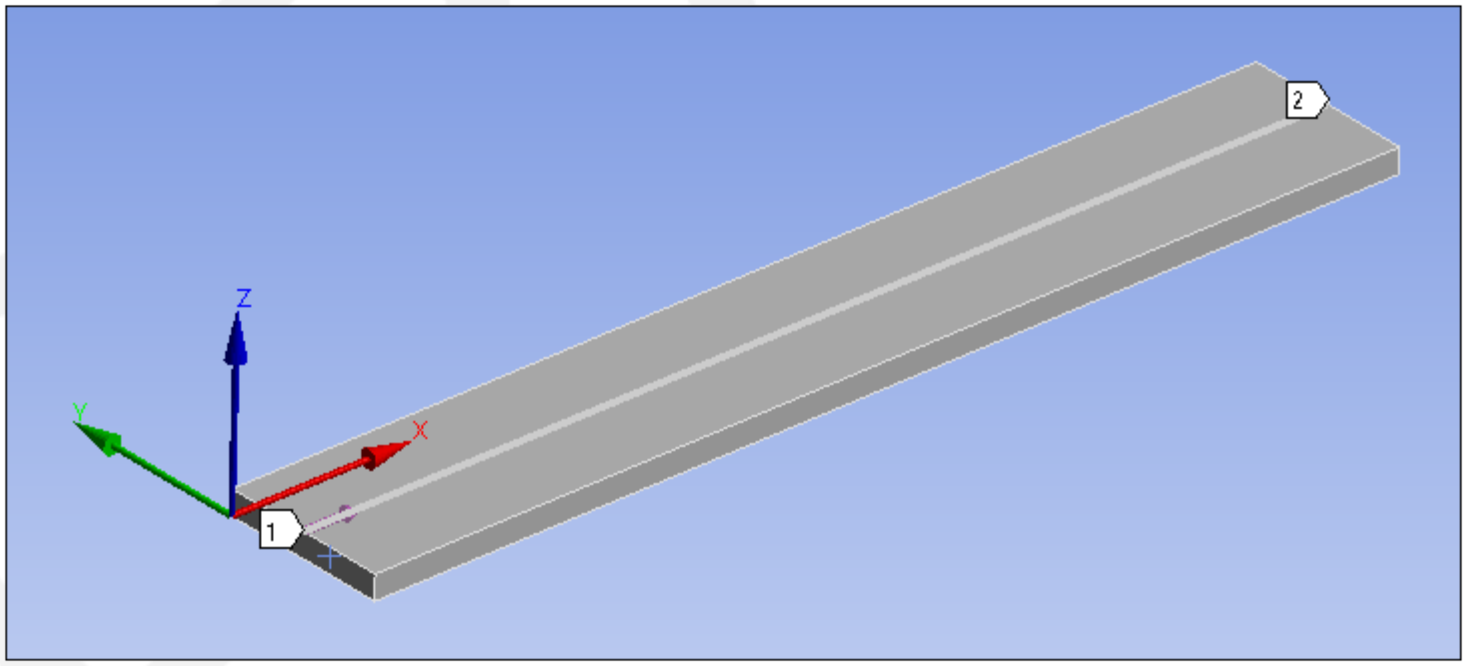
The discretization will include all nodes in the mesh underlying the edge. Multiple edges may be used but they must be continuous.

Result Graphical Display

For each result scoped to a **Path**, the **Graph Controls** category provides an option to display the result in the **Graph** on **X-axis**, as a function of **Time** or with **S**, the length of the path. Note that Path results have the following restrictions:

- They are calculated on solids and surfaces but not on lines.
- They can be collected into charts as long as all of the other objects selected for the chart have the same **X-axis** (**Time** or **S**).
- You can define a path in the geometry by specifying two points, an edge, or an axis. Before you define a path, you must first add the **Path** object from the **Construction Geometry** context toolbar.

You can then define the path using any of the methods presented below.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Application

There are a number of methods to define a **Path**, including:

- Defining a Path using Two Points (p. 885)
- Defining a Path using an Edge (p. 886)
- Defining a Path from Results Scoped to Edges (p. 886)
- Defining a Path using X-axis Intersection (p. 886)
- Defining a Path from Probe Labels (p. 887)

Additional options for this feature include:

- Snap to Mesh Nodes (p. 888)
- Exporting Path Data (p. 890)

Defining a Path using Two Points

Using this method you define the path by specifying two points in any of the following ways:

Define path using the Hit Point Coordinate option:

1. In the Details, select **Two Points** in the **Path Type** list.
2. Under **Start**, choose **Click to Change** in the **Location** row.
3. Select the **Hit Point Coordinate** option on the Graphics Toolbar (p. 97).

   As you move the cursor across the model, the coordinates display and update as you reposition the cursor.
4. Click at the desired start location for the path. A small cross hair appears at this location. You can click again to change the cross hair location.
5. Click **Apply**. A "1" symbol displays at the start location. Also, the coordinates of the point display in the Details view. You can change the location by repositioning the cursor, clicking at the new location, and then clicking **Click to Change** and **Apply**, or by editing the coordinates in the Details view.
6. Repeat steps 2 through 5 to define the end point of the path under **End** in the Details view. A "2" symbol displays at the end location.
7. Enter the **Number of Sampling Points**.

Define path using coordinates:

1. In the Details view, select **Two Points** in the **Path Type** list.
2. Under **Start**, enter the X, Y, and Z coordinates for the starting point of the path.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Under **End**, enter the X, Y, and Z coordinates for the ending point of the path.
4. Enter the **Number of Sampling Points**.

Define Path using vertices, edges, faces, or nodes:

1. In the Details view, select **Two Points** in the **Path Type** list.
2. Select one or more vertices or nodes, a single edge, or a face where you want to start the path, and then click **Apply** under **Start**, **Location**. An average location is calculated for multiple vertex or node selections.
3. Select the vertices, nodes, face, or the edge where you want to end the path, and then click **Apply** under **End**, **Location**.
4. Enter the **Number of Sampling Points**.

Note:

The start and end points need not both be specified using the same procedure of the three presented above. For example, if you specify the start point using the **Coordinate** toolbar button, you can specify the end point by entering coordinates or by using a vertex, edge, or face. Any combination of the three procedures can be used to specify the points.

Defining a Path using an Edge

This method helps you define a path by selecting an edge. To define a path:

1. In the Details view, select **Edge** in the **Path Type** list.
2. Specify an edge geometry using **Geometry Selection** or by specifying an edge-based **Named Selection**.

Defining a Path from Results Scoped to Edges

In order to help better quantify the variation of a result along a set of edges, path results are available. For a result that is scoped to an edge or multiple contiguous edges, you can convert the scoping to the equivalent **Path**, by:

1. Selecting the result object that is scoped to an edge or contiguous edges.
2. Display the context menu by right-clicking the mouse, and the select **Convert To Path Result**.

A **Path** object is automatically created under the Construction Geometry folder and the **Path Type** property is set to **Edge**.

Defining a Path using X Axis Intersection

For this option, the application automatically creates the path on the first body the line intersects from the coordinate system origin, as illustrated below. A line is extended along the positive X axis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

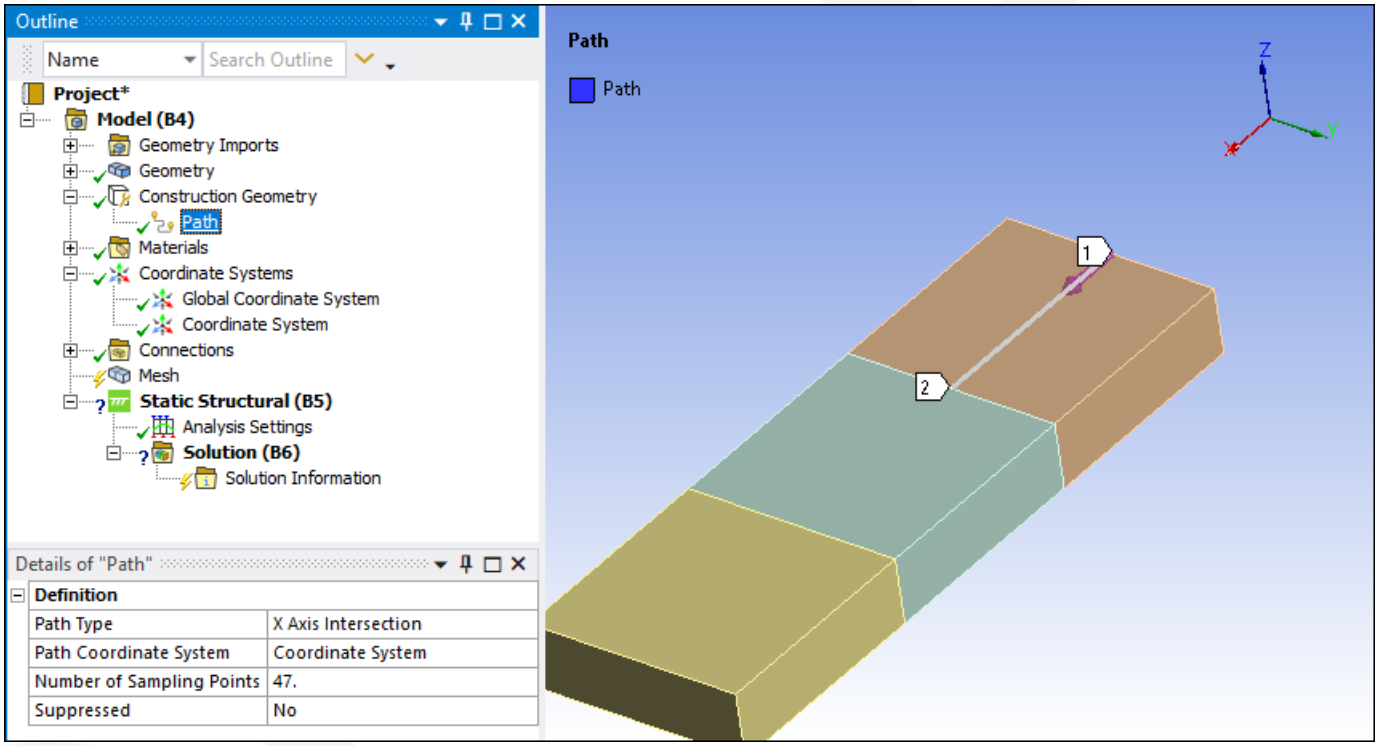
Published: 2023-10-23

of the selected coordinate system, and the path is created between the first and last points where this line intersects the first body. To create this Path:

Important:

When you specify a **Path** as the scoping for a result (p. 1940), the application re-calculates the path using the mesh instead of geometry. As a result, the locations of path points may vary slightly.

1. Set the **Path Type** property to **X Axis Intersection**.
2. Select the desired coordinate system that defines the x-axis using the **Path Coordinate System** property.
3. Specify the **Number of Sampling Points**. The default is value is **47**.



Defining a Path from Probe Labels

While reviewing results, you can define a path automatically from two probe labels. To define the path:

1. Create two probe annotations (p. 207) by choosing the **Probe** button from the Result Context Tab (p. 62).
2. Choose the **Label or Imported Objects** button from the Graphics Toolbar (p. 97) and select the two probe annotations. (Hold the **Ctrl** key to select both probe annotations.)
3. Right-click in the **Geometry** window and choose **Create Path From Probe Labels** from the context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. A path is automatically created between the probe annotations. A corresponding **Path** object is displayed in the tree with a **Path Type** of **Two Points**.

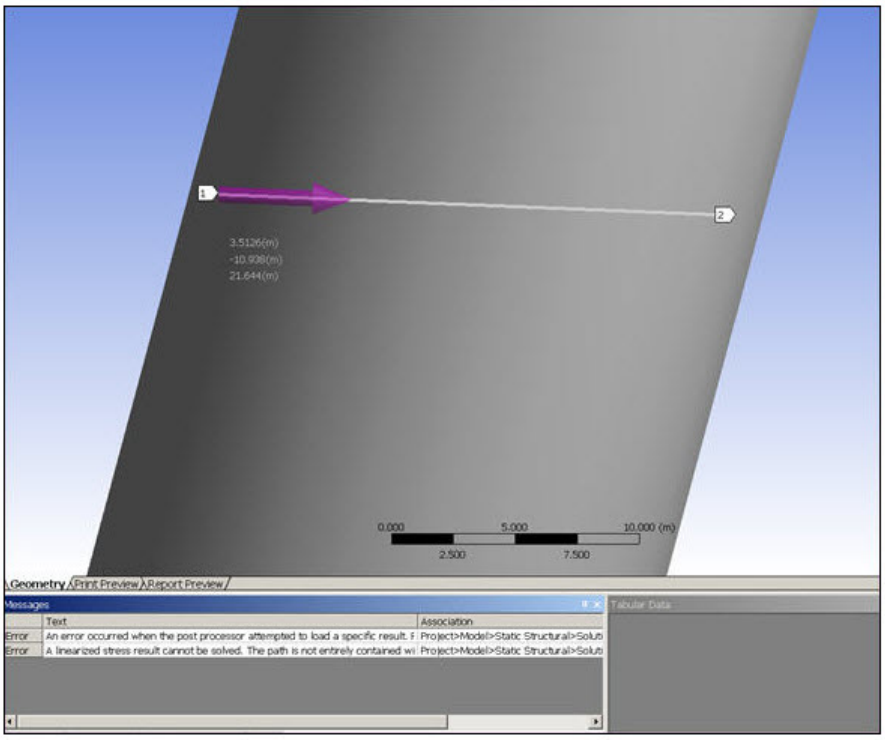
Snap to Mesh Nodes

When solving linearized stresses (p. 2041), the path you define by two points must be contained within the finite element mesh to avoid an error. Because the two points can be derived from the tessellation of the geometric model, the points may be contained within the geometry but may not be contained within the mesh. This is especially true for curved geometry faces. After defining the two points using the **Coordinate** toolbar button method (see above), you can ensure that the path is contained within the mesh by using the **Snap to mesh nodes** feature. To use the feature, set **Show Mesh** to **Yes** in the Details view of the **Construction Geometry** object in order to see the location of the nodes in the mesh. Then, right-click the **Path** object and select **Snap to mesh nodes** from the context menu. This action alters the path, as necessary, such that both the start point and end point of the path snap to the closest node in the mesh.

The **Snap to mesh nodes** feature avoids the error and allows the solve to continue provided the path you define does not traverse through any discontinuities in the model, such as a hole. For these cases, even though the **Snap to mesh nodes** feature alters the path endpoints to coincide with the nearest nodes in the mesh, the linearized stress result still fails because the path is defined through the discontinuity.

The following pictures illustrate this feature.

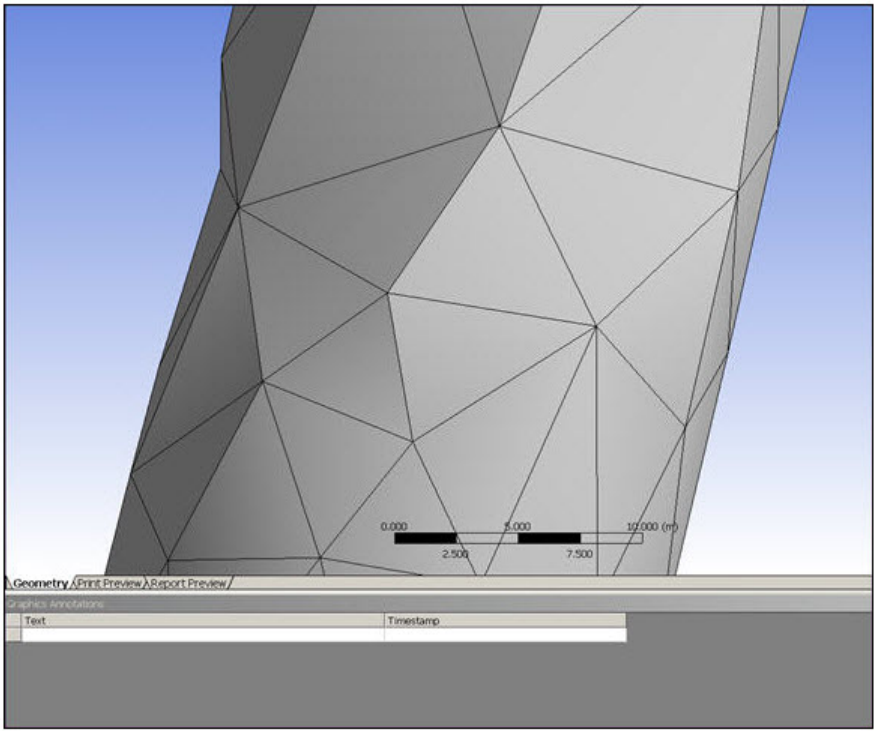
Attempt to solve for linearized stress. Path defined within geometric model:



*Corresponding mesh used for geometric model, obtained by setting **Show Mesh** to **Yes**:*

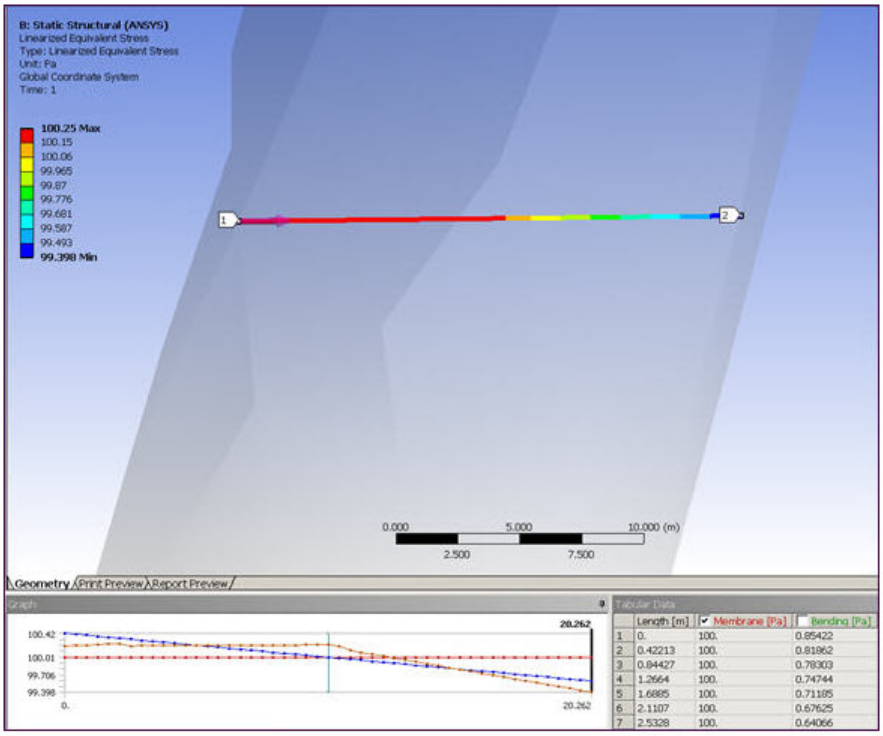
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



*Path contained within mesh after choosing **Snap to mesh nodes**. Solution completes:*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

If the model is re-meshed after choosing **Snap to mesh nodes**, the feature is not automatically applied to the newly meshed model. You must choose **Snap to mesh nodes** again to alter the path start and end points to the new mesh.

Exporting Path Data

You can export coordinate data for a defined path by clicking the right mouse button on a **Path** object and choosing **Export** from the context menu.

Surface

A surface is categorized as a form of construction geometry and is represented as a section plane to which you can scope surface results (p. 1959) or reaction probes (p. 2107).

To define a surface:

1. Highlight the **Model** object, open the **Construction Geometry** drop-down menu on the **Model** Context Tab (p. 48), and select **Surface**.
2. Define a coordinate system whose X-Y plane will be used as a cutting plane, as follows:
   a. Create a local coordinate system (p. 1192).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

b. Define the origin (p. 1192) of the local coordinate system.

Note:

With respect to the facets of the surface:

- For a Cartesian coordinate system, the surface is the intersection of the model with the X-Y plane of the coordinate system.
- For a cylindrical coordinate system, the surface is the intersection of the model with the cylinder whose axis is the Z axis of the coordinate system. In this case, you must specify the radius in the Details view of the **Surface** object.

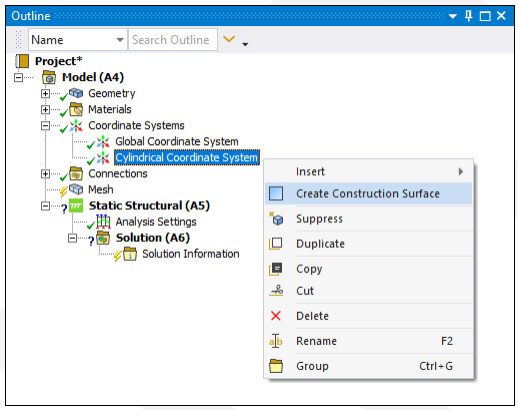
Tip:

For an existing coordinate system, you can define a **Surface** Construction Geometry object by selecting a desired Coordinate System object, right-clicking, and selecting **Create Construction Surface**. This feature allows you to define the coordinate system first. See the procedure below.

Create Construction Geometry Surface from Coordinate System

As illustrated below, you can create a **Surface Construction Geometry** from any existing coordinate system using the right-click feature **Create Construction Surface**.

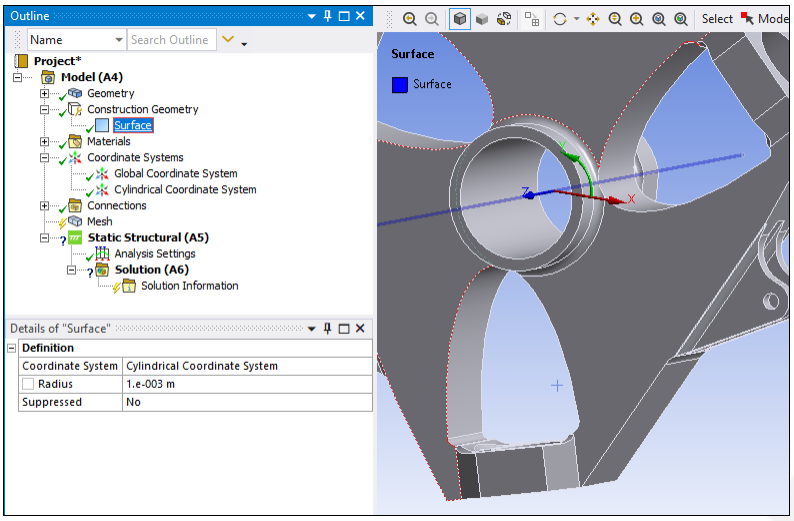
Right-click Menu for



Details for Surface Object

The **Details** display the defined coordinate system, Radius, and enables you to suppress the object if desired.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Solid

The **Solid** option of the Construction Geometry feature enables you to create and add a solid part to the model you have imported into Mechanical.

To define a **Solid** Construction Geometry:

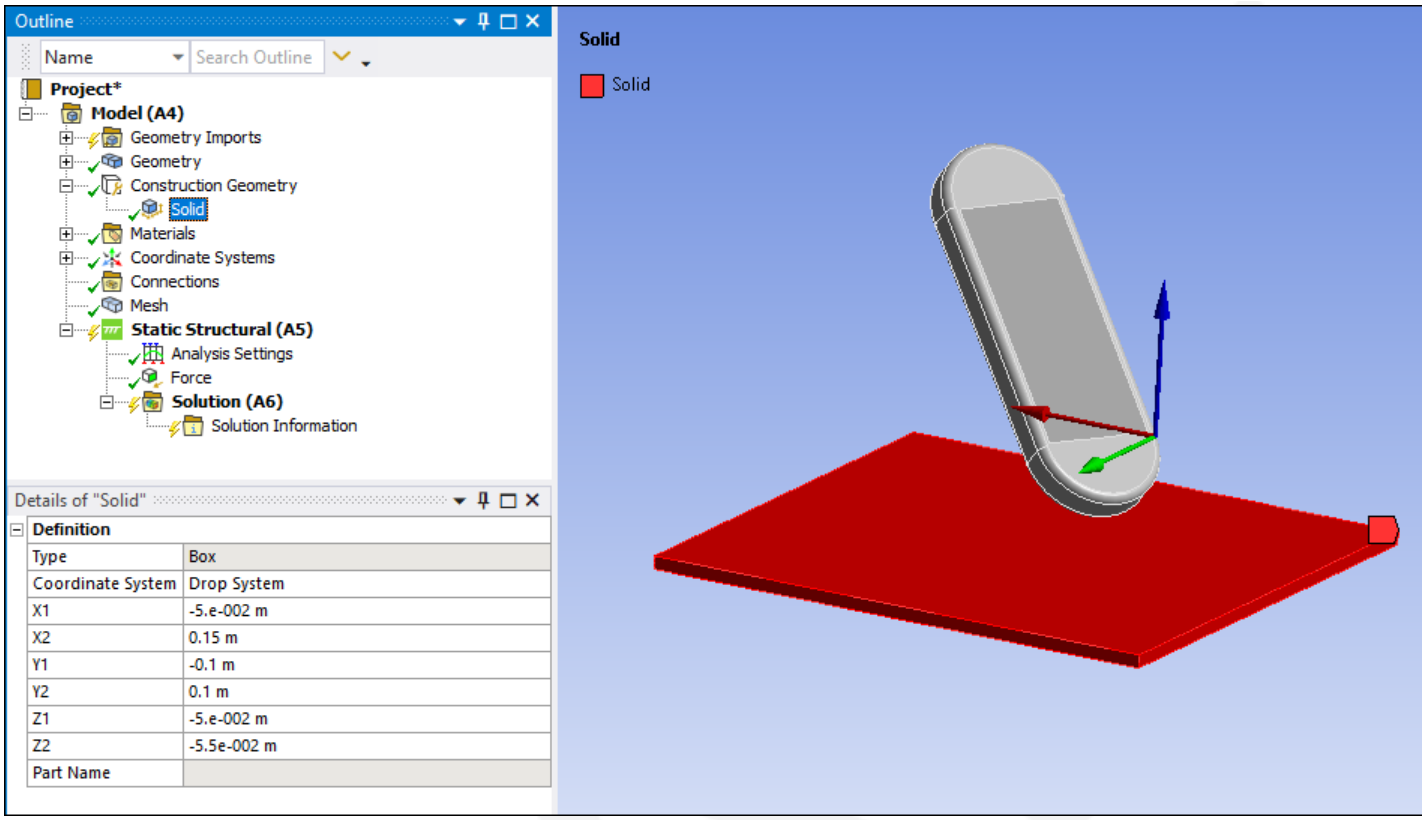
1. Highlight the **Model** object, open the **Construction Geometry** drop-down menu on the **Model** Context Tab (p. 48), and select **Solid**.

2. As needed, specify a coordinate system. The default is the **Global Coordinate System**.

3. Specify the dimensions of the solid part using the **X1** through **Z2** properties. These properties define the start point and the end point for each dimension. The application displays a frame of your geometry as you enter values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

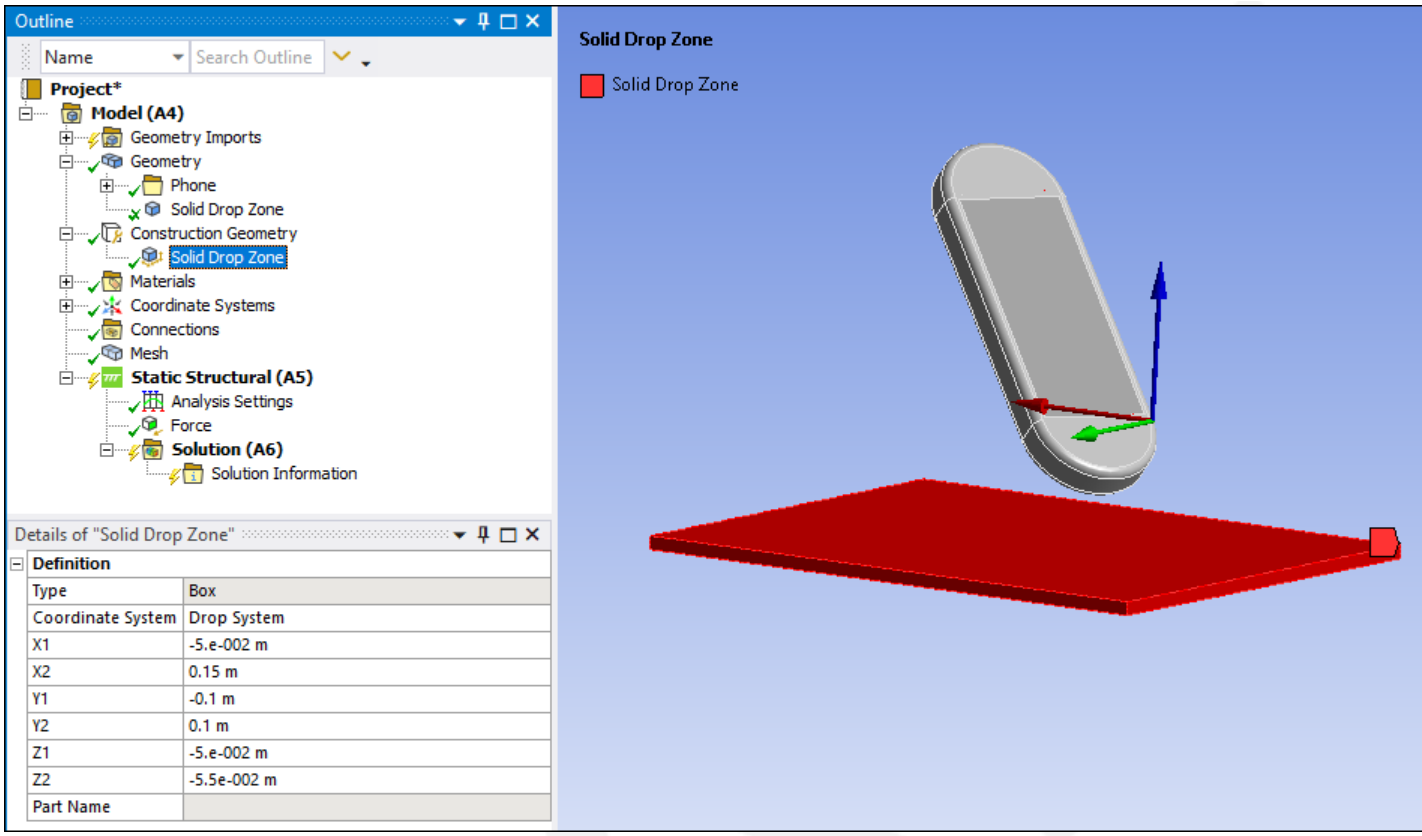


Important:

- The difference between the X, Y, and Z values must be greater than zero and within a tolerance of $1e^{-10}$. All comparisons are performed in the associated CAD units.
- Automatic contact detection is not performed when a solid part is generated. In order for the solid part to be included in automatic contact generation, you need to either select **Create Automatic Connections** option via the **Connections** folder or update the geometry from the CAD source by selecting the **Update Geometry from Source** option via the **Geometry** folder.
- Making a vertex or node selection on your model displays the Location of the vertex/node in the Status Bar (p. 135). This information can be helpful when constructing your solid part.

4. Right-click the object and select **Add to Geometry**. Once created, a new Part is placed in the **Geometry** folder. This part behaves as and may be used like any part of the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

Review the following requirements:

- You must update the Geometry object (**Update Geometry from Source** option) in order to transfer changes made on the Solid object to the Part object.
- In order to delete a **Solid** object, you must first right-click the object and select **Remove Geometry**. This action removes the part from the **Geometry** folder. You can then delete the **Solid** object as needed.
- Selective Update (p. 340) is not supported for solids created in Mechanical.

STL

The **STL** option of the **Construction Geometry** feature enables you to import and view an STL (Stereolithography) file in Mechanical.

To import an **STL** file:

1. Highlight the **Model** object, open the **Construction Geometry** drop-down menu on the **Model** Context Tab (p. 48), and select **STL**.
2. At this point, define the following properties as needed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Length Units**: Meters is the default setting. Changing this property rescales the STL geometry according to the new unit. Changing from Meters to Centimeters resizes the model accordingly.
- **Show Always**: Options include **No** (default) and **Yes**. If you set this property to **Yes**, the application displays the STL geometry in the **Geometry** window regardless of what object is selected in the tree.
- **Color**: Change the color of the STL geometry. This enables you to easily identify your STL geometry.

3. Right-click the **STL** object and select **Import STL File**. This option imports and displays the new model in the **Construction Geometry** folder.

Note:

When generated, the position of the imported STL geometry in the **Geometry** window is determined by the coordinates from which the original STL file is generated. To ensure that you can see the geometry, use the **F7** key (**Zoom to Fit**) to properly display all geometries.

The **Statistics** property displays the number of triangles included in the STL geometry.

Also see the STL object reference page of the Help for more information.

Construction Line

The **Construction Line** feature is a unique geometry sketching tool you use to create line segments on or around your model that you eventually convert into line bodies. This section describes the methods and options used to sketch and produce these line bodies.

- Defining a Construction Line (p. 895)
- Generating Line Bodies (p. 897)
- Understanding the Tools (p. 897)

Defining a Construction Line

Select the **Construction Line** option from the **Construction Geometry** drop-down menu on the **Model** Context tab (p. 48). Once you have inserted the object, several additional options are available to insert additional objects. See the **Construction Line** object reference section for the other methods.

Note:

Using the Image Plane from File option of the **Insert** (p. 42) group on the **Home** tab, you can import an image into Mechanical and place it on or around your model based the on XY-plane of the selected coordinate system. You then use this image to very accurately sketch line segments. To properly trace over an imported image, you need to position your

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

model using the same orientation that was used to capture the image, otherwise, your new sketch may be in an incorrect position compared to the model.

Once inserted, the object has the following **Details** pane categories and properties:

Category	Property/Description
Sketch Selection	**Method**: Options for this property include: • **Interactive** (default): This option enables you to sketch line s using the **Planar** (p. 898) or **Freeform** (p. 898) options. • **Coordinate System**: This option enables you to sketch line s hand using the **Planar** (p. 898) or **Freeform** (p. 898) options.
Active Sketch Details	**Sketch ID**: This read-only property shows an application define Line ACT APIs. **Association Type**: This read-only property shows the entity typ **System**. **Association ID**: This read-only property shows the ID of the ske property, either **Geometric** or **Coordinate System**. If set to **Geo** the ACT API Reference Guide). If set to **Coordinate System**, the **Note:** These ID values are compatible with the Mechanical ACT A
Image Plane Properties	The properties of this category enable you to import and overl or around your model. **Image File**: Import an image that overlays on the XY-plane of t can very accurately sketch one or more line segments. Note tha your model using the same orientation that was used to captur position compared to the model. **Coordinate System**: Specify where to place your image based **Show Coordinate System**: Show the selected Coordinate Syste **Width**: Specify the physical width of your image. For the best re physical height matches the aspect ratio of your image. **Height**: Specify the physical height of your image. For the best physical height matches the aspect ratio of your image. **Translucency**: Increasing the value makes your image more and line. **Horizontal Flip**: Reverse your image horizontally.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property/Description
	Vertical Flip: Reverse your image vertically.

For an individual **Construction Line** object, you can create as many line bodies as desired. You can switch back and forth between each of the above methods for each line body, or group of line bodies, you create. The **Sketch Selection** property simply defines the method to select the active plane for planar sketching. The option specified for this property has no effect on the final geometry that will be added to the model. See the Understanding the Tools (p. 897) topic below for descriptions of the various tools and options available for this feature.

Generating Line Bodies

Once you have completed sketching your line bodies, use the **Construction Line** object's context (right-click) menu option **Add to Geometry** to generate line bodies. Use the **Update Geometry** option to incorporate any desired additions/changes.

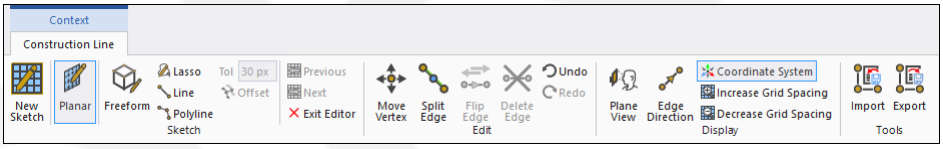
Once you have created a Line Body (p. 872), you need to specify a cross section and assign a material. Review the Cross Section and material assignment (p. 345) topics as well and the Body object reference section for descriptions of all of the associated geometry properties.

Understanding the Tools

This topic describes the features and tools of the **Construction Line** Context menu that are organized into the following groups:

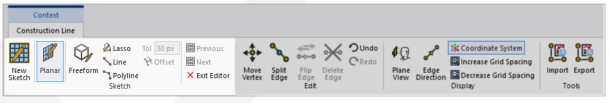
- Sketch (p. 897)
- Edit (p. 899)
- Display (p. 900)
- Tools (p. 900)

This is the default display of the Context tab for the **Construction Line** object.



Sketch

The **Sketch** options group (highlighted below) contains draw tools that enable you to create line segments.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

New Sketch

This option is active by default when you insert the object. Select a desired plane on your model (based on a face selection) or you can set the **Sketch Selection** property to **Coordinate System** and select a user-defined coordinate system (p. 1192).

Available when the **Sketch Selection Method** property is set to **Interactive**, the **New Sketch** option specifies a new sketch plane on a face of your model that you select. Once specified, the **Planar** editor becomes active. Select a desired sketch option and create line segments.

Planar

The **Planar** editor enables you to sketch 2D line segments on a given specified plane. Your line segments are bound to the specified geometry face or a Coordinate System, that is, your sketch can only be performed within this plane.

To begin sketching line segments, select the **Lasso** (p. 898), **Line** (p. 898) or **Polyline** (p. 898) tool. You can also use the **Offset** tool.

Freeform

The **Freeform** editor enables you to sketch 3D line segments between geometric entities (including face centroids) or the vertices of existing line segments. To begin sketching line segments, select the **Line** (p. 898) or **Polyline** (p. 898) tool.

Lasso

The **Lasso** tool, only available for the 2D (Planar) editing mode, enables you to click and drag the mouse cursor to trace a line segment (beginning to end). This tool has an accompanying Tolerance (**Tol**) field that defines the number of pixels in between each vertex the tool automatically creates. Use the **[Esc]** key to complete a sketch.

Line

The **Line** tool is available for both editing modes and enables you to make individual line segments. By default, when you create a line segment using this tool, point selections "snap" into place:

- Using the **Planar** editor. This snapping takes precedence for sketch vertex points and to cursor placement. Holding the **[Shift]** key enables you to snap to a point on the plane grid.
- On sketch vertices, edges, and face centroids using the **Freeform** editor. However, using **Freeform**, you can freely place points on faces without any snapping.

Polyline

The **Polyline** tool is available for each editing mode and enables you to make a series of line segments that the application will use to generate a single line. The snapping capability is the same as it is for the **Line** tool. Use the **[Esc]** key to complete a sketch.

Offset

The **Offset** option only becomes active when you 1) select one or more edges, and 2) select the **Planar** option. It enables you to create an offset of an edge(s), within the 2D sketching plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

When you activate the tool, drag and drop the edge(s) to a desired offset location. You can define multiple offsets. Note that you can also click a location on the plan where you would like the offset located.

Previous/Next

The **Previous** and **Next** options cycle through the sketch planes you have created for the current Construction Line object.

Exit Editor

The **Exit Editor** option deactivates all **Construction Line** Context tab options.

Tip:

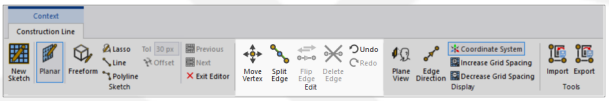
Note the following selection features:

- Holding the **[Shift]** key enables grid point "snapping." The application highlights a point on the grid to which you can connect your selection. This is helpful when placing and/or moving a vertex.
- **Box Selection**: You can perform box selections by holding the **[Shift]** key and left mouse button and dragging right or dragging left.
  - Dragging Left: Select only segments that have both vertices in the box.
  - Dragging Right: Select segments with at least one vertex in the box.

  For **Planar** mode, box-selection selects segments with both vertices lying in the sketching plane only.

Edit

The **Edit** group (highlighted below) contains tools for modifying or moving existing lines and edges.



Move Vertex

When you select this option, you can highlight an existing vertex and move it via drag and drop. You can select multiple vertices using the **[Ctrl]** key. When you hold the **[Shift]** key and move a vertex, the application will snap it to the grid.

Split Edge

The **Split Edge** option creates a new vertex on an existing line segment. You can use the **[S]** key to perform this action.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Flip Edge

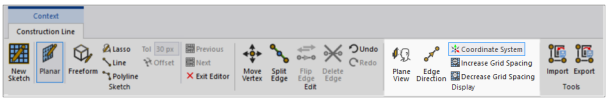
Active when a line is segment selected, this option enables you to invert the Edge Direction of the line segment. You can also use the **[F]** key to perform this action. Edge direction is the direction of the vector of the first point of your line segment to the second point. This direction can be displayed using the **Edge Direction** option.

Delete Edge

Active when a line segment is selected, this option deletes the selected line segment. You can also use the **[Delete]** key.

Display

The options of the **Display** group are highlighted and described below.



Plane View

Reorient the view of the active sketch plane to the XY-plane.

Edge Direction

Display line segment edge direction. Edge direction is the direction of the vector of the first point of your segment to the second point.

Coordinate System

Toggles the display of the triad for the active sketch's Coordinate System on and off.

Increase/Decrease Grid Spacing

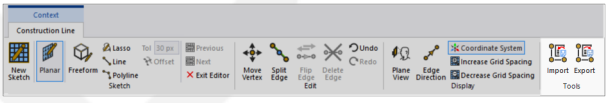
Increase or decrease the spacing of the grid display (by power of 10 based on the CAD unit).

Tip:

You can use the key combinations **[Ctrl]**+**[+]** and **[Ctrl]**+**[-]** to increase and decrease the grid.

Tools

The **Import** options group (highlighted below) contains tools to import or export currently or previously defined line bodies.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Import

Import previously exported line bodies in XML file format.

Export

Export the currently defined line bodies in XML file format.

Point Mass

Overview

You can idealize the inertial effects from a body using a **Point Mass**. Applications include applying a force with an acceleration or any other inertial load; or adding inertial mass to a structure, which affects modal and harmonic solutions.

The location of the **Point Mass** can be anywhere in space and can also be defined in a local coordinate system if one exists. The default location is at the centroid of the geometry. The **Point Mass** will automatically be rotated into the selected coordinate system if that coordinate system differs from the global coordinate system. You can also input moment of inertia values for each direction.

The Rigid Body Dynamics and Explicit Dynamics Solvers consider moments of inertia in the selected coordinates system, rather than in the global coordinate system.

Application

To define a **Point Mass**:

1. Select the Geometry object (or a child object).
2. You can then add a **Point Mass** object by:
   - Selecting the **Point Mass** option from the **Mass** group on the **Geometry** Context tab (p. 56).

     or...

   - Right-clicking the mouse button and selecting **Insert** > **Point Mass**.

     or...

   - Selecting the desired geometry in the graphics window, right-clicking the mouse, and then selecting **Insert** > **Point Mass** from the context menu.
3. Specify the **Scoping Method** property as either **Geometry Selection**, **Named Selection**, or **Remote Point**. Based on the selection made in this step, select a:
   - Geometry (faces, edges, or vertices) and click **Apply** in the Details view for the **Geometry** property.

     or...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Single node and click **Apply** in the Details view for the **Geometry** property. In order to select an individual node, you need to first generate a mesh on the model, and then select the **Node** filter on the Graphics Toolbar (p. 97).

  or...

- User-defined node-based named selection from the drop-down list of the **Named Selection** property.

  or...

- User-defined remote point from the drop-down list of the **Remote Point** property.

  or...

4. Specify the **Point Mass** as a **Remote Attachment** (default) or a **Direct Attachment** using the **Applied By** property. The **Remote Attachment** option uses either a user-defined or a system-generated **Remote Point** as a scoping mechanism. **Remote Attachment** is the required **Applied By** property setting if the geometry scoping is to a single face or multiple faces, a single edge or multiple edges, or multiple vertices. The **Direct Attachment** option allows you to scope directly to a single vertex (Geometry) or a node (using an individually selected node or a node-based Named Selection) of the model.

5. Enter a **Mass** value.

6. Modify Point Mass object Details properties as needed.

When defined as a **Remote Attachment**, the **Point Mass** is considered a remote boundary condition (p. 1735). It can make use of remote points that are either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and remote points by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Support Limitations

A **Point Mass** cannot:

- span multiple bodies if the **Stiffness Behavior** of the bodies is declared as **Rigid** (see Rigid Bodies (p. 852) section for additional information).
- be applied to a vertex scoped to an end release (p. 1409).

Explicit Dynamics solutions do not allow scoping via a Direct Attachment.

Distributed Mass

You can use this feature to distribute additional mass across the faces or edges of the flexible parts in your model. Using this feature, you can idealize the inertial effects from the bodies/entities that are evenly spread across the surfaces of your model, for example, mass contribution from paint, external equipment, a large number of small objects spaced evenly across the surfaces, etc.

To define a **Distributed Mass**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

1. Select the Geometry object (or a child object).
2. You can then add a **Distributed Mass** object by:
   - Selecting the **Distributed Mass** option from the **Mass** group on the **Geometry** Context tab (p. 56).

     or...

   - Right-clicking the mouse button and selecting **Insert**>**Distributed Mass**.

     or...

   - Selecting the desired geometry in the Geometry window, right-clicking the mouse, and then selecting **Insert**>**Distributed Mass**.
3. Specify the **Scoping Method** property as either **Geometry Selection** or **Named Selection**. Based on the selection made in this step, select a:
   - Geometry (faces and edges *only*) and click **Apply** in the Details view for the **Geometry** property.

     or...

   - Face-based or edge-based user-defined named selection from the drop-down list of the **Named Selection** property.
4. Specify the **Mass Type** as either **Total Mass** or **Mass per Unit Area**. Based on this selection, enter a value for **Total Mass** or **Mass per Unit Area**.

See the Distributed Mass object reference page for additional information about the properties of this feature.

Limitations

Note the following with regards to a **Distributed Mass**. A Distributed Mass can only be:

- Applied in 3D analyses.
- Scoped to faces or edges of bodies whose **Stiffness Behavior** is set as **Flexible**.
- Used in structural analyses (that is, Static Structural Analysis (p. 652), Transient Structural Analysis (p. 665), Transient Structural Analysis Using Linked Modal Analysis System (p. 674), and Linear Dynamic Analysis (p. 486) types).

Surface Coating

This feature enables you to apply a surface coating (shell layer) of a specified material and thickness over one or more faces of your model. This feature is beneficial when you want to include the ability to accurately evaluate surface stresses or to overlay your structure with thin parts, such as modelling Thermal Barrier Coatings or sheet metal over support structures. To accurately model this type of ap-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

plication, Mechanical enables you to specify the thickness, stiffness behavior, coordinate system, and material.

Note:

Surface Coating (p. 2008) is also a **Scoping Method** option for results.

Important:

During a Cyclic Symmetry analysis, the application does not expand Surface Coating results.

Requirements

Note the following requirements for the **Surface Coating** feature:

- It can only be applied to the surfaces (faces) of solid bodies (shells/sheets are not supported.).
- Supported by 3D analyses only.
- Is only supported for structural analysis types that are using the Mechanical APDL Solver. Analyses include Static Structural Analysis (p. 652), Transient Structural Analysis (p. 665), Transient Structural Analysis Using Linked Modal Analysis System (p. 674), and Linear Dynamic Analysis (p. 486) types.
- It uses SHELL181 and SHELL281.
- The mesh of the bodies onto which you apply a Surface Coating should contain same **Element Order** (**Linear** or **Quadratic**).

Application

To create a **Surface Coating**:

1. You can either:
   a. Select the Geometry object and click **Surface Coating** option from the **Modify** group on the **Geometry** Context tab (p. 56).

      Or...

   b. Right-click on the Geometry object or in the **Geometry** window and select **Insert** > **Surface Coating**.

      Or...

   c. Select the desired faces available in the **Geometry** window, right-click, and select **Insert** > **Surface Coating**. This option automatically specifies the Geometry property in the Details view and therefore negates the next step.
2. Specify the **Scoping Method** property as either **Geometry Selection** or **Named Selection**. Based on the selection made in this step, select a:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Geometry (faces *only*) and click **Apply** in the Details view for the **Geometry** property.

  or...

- Face-based user-defined named selection from the drop-down list of the **Named Selection** property.

3. Specify the **Stiffness Behavior** as either **Stress Evaluation Only** (default), **Membrane Only**, or **Membrane and Bending**. If you set this property to **Stress Evaluation Only**, no **Thickness** entry is necessary.

   Note:

   By default, Mechanical automatically sets the Shell Offset to Bottom for all **Stiffness Behavior** settings.

4. Select a material for the surface coating from the fly-out menu of the **Material** property.
5. Specify a **Thickness** for the surface body.
6. As needed, change the **Coordinate System** setting. The **Default Coordinate System** option is based on the geometry scoping specified above.

See the Surface Coating object reference page for additional information about the properties of this feature.

Thermal Point Mass

Overview

For **Transient Thermal** analyses, you can idealize the thermal capacitance of a body using a thermal point mass. Thermal Capacitance replaces the need to calculate the body's internal thermal gradient. The **Thermal Point Mass** is commonly used as a medium to store or draw heat from surrounding objects. Applications include the heat dissipation of refrigerators, cooling electronic devices, and heat sinks of computer motherboards.

This section examines the following feature applications and requirements:

- Apply Thermal Point Mass Object (p. 905)
- Behavior Property Specifications (p. 907)
- Support Limitations (p. 908)

Application

To define a **Thermal Point Mass** in your Transient Thermal analysis:

1. Select the Geometry object (or a child object).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. You can then add a **Thermal Point Mass** object by:

   - Select the Geometry object and click **Thermal Point** option from the **Mass** group on the **Geometry** Context tab (p. 56).

     or...

   - Right-clicking the mouse and selecting **Insert** > **Thermal Point Mass**.

     or...

   - Select the desired geometry in the graphics window, right-click the mouse, and then select **Insert** > **Thermal Point Mass**.

3. Specify the **Scoping Method** property as either **Geometry Selection**, **Named Selection**, or **Remote Point**. Based on the selection made in this step, select a:

   - face, edge, or vertex of a solid or surface model or on an edge or vertex of a surface model and click **Apply** in the Details view for the **Geometry** property.

     or...

   - single node and click **Apply** in the Details view for the **Geometry** property. In order to select an individual node, you need to first generate a mesh on the model, and then select the **Node** filter on the Graphics Toolbar (p. 97).

     or...

   - user-defined node-based named selection from the drop-down list of the **Named Selection** property.

     or...

   - user-defined remote point from the drop-down list of the **Remote Point** property.

4. Specify the **Thermal Point Mass** as a **Remote Attachment** (default) or a **Direct Attachment** using the **Applied By** property. The **Remote Attachment** option uses either a user-defined or a system-generated **Remote Point** as a scoping mechanism. **Remote Attachment** is the required **Applied By** property setting if the geometry scoping is to a single face or multiple faces, a single edge or multiple edges, or multiple vertices. The **Direct Attachment** option allows you to scope directly to a single vertex (Geometry) or a node (using an individually selected node or a node-based Named Selection) of the model.

   Note:

   The location of the **Thermal Point Mass** can be anywhere in space. The default location is at the centroid of the geometry.

5. Modify coordinate system properties as needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

6. Enter a **Thermal Capacitance** value. **Thermal Capacitance** refers to ability of the material to store heat. The higher the thermal capacitance, the more heat can be stored for each degree rise in temperature of the **Thermal Point Mass**.

7. When the **Thermal Point Mass** is defined as a **Remote Attachment**, the **Behavior** property displays: define as **Isothermal**, **Coupled**, or **Heat-Flux Distributed**. See the *Behavior Property Specifications* topic below for additional information about how to make the appropriate selection.

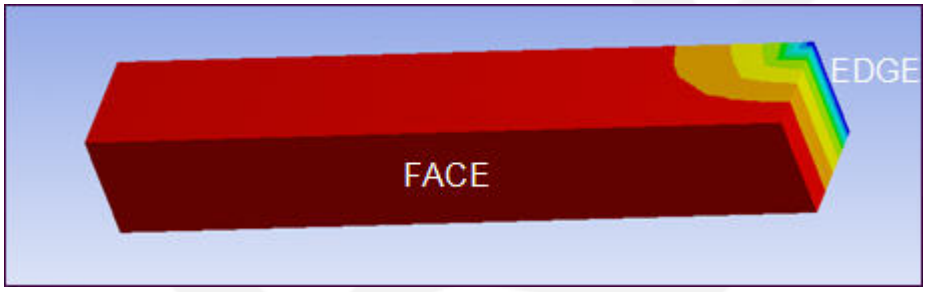
8. Modify additional Thermal Point Mass object Details view properties as needed.

When defined as a **Remote Attachment**, the **Thermal Point Mass** is considered a remote boundary condition (p. 1735). It can make use of remote points that are either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and remote points by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

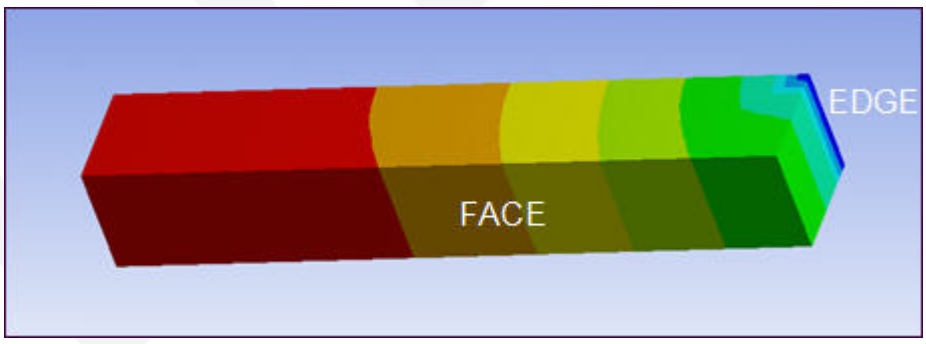
Behavior Property Specifications

The **Thermal Point Mass** includes three **Behavior** options in the Details View that control its interaction with the bodies in the geometry selection: **Isothermal**, **Coupled**, and **Heat-Flux Distributed**:

- For the **Isothermal** behavior, temperatures throughout the geometry selections and the **Thermal Point Mass** are constrained to be the same. The following is an example of a **Thermal Point Mass** using **Isothermal** behavior applied to the FACE while a temperature boundary condition is located at the EDGE. While there is a temperature distribution from the boundary condition (EDGE) up to the surface (FACE), the temperature on the FACE in the pinball region, itself takes a single value that matches that of the **Thermal Point Mass**.



- For **Heat-Flux Distributed** behavior, however, the temperature of the geometry selection and the point mass are not constrained to be the same. The temperature of the **Thermal Point Mass** becomes a weighted average of those on the geometry selection. For comparison, the previous example has been modified to use the **Heat-Flux Distributed** behavior. The FACE, no longer constrained to be isothermal to the point mass, displays a gradient.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For **Coupled** behavior, the geometry has the same DOF solution on its underlying nodes as the remote point location. This formulation is similar to the Mechanical APDL constraint defined by the **CP** command.

Support Limitations

A **Thermal Point Mass** cannot be applied to a vertex scoped to an end release (p. 1409).

Models from External Meshes and Model Assemblies

The following sections examine the features in Ansys Workbench and Ansys Mechanical that enable you to generate and assemble geometries from external component systems:

Importing Mesh-Based Geometry

Assembling External Models and Mechanical Models

Importing Mesh-Based Geometry

The Workbench External Model system enables you to either import your mesh or to synthesize a geometry from the mesh file. When generating a geometry, supported geometric entities include solids, shells, as well as beam element-based bodies (that have an assigned cross-sectional definition). The resulting geometry is the culmination of the use of the implicit (angle-based tolerance) and explicit (based on node-based components in the file) methods that work in combination to synthesize geometry and create surfaces that enclose the mesh volume.

This feature supports data import of lines, shells, or solids or a mix of lines, shells, and solids. See the External Model Supported Element Types (p. 969) section for a list of the available element types for the Mechanical APDL common database (.cdb), NASTRAN, and ABAQUS file formats.

This feature supports all Mechanical analysis types. For the specific instructions to import a finite element mesh file using this tool, see the Creating and Configuring an External Model System section of the Workbench Help.

The following sections describe how to work with the finite element data that you import from an upstream **External Model** system.

Supported External File Types

Import Workflow and Interface Options

Supported Finite Element Data Types

Automatic Material Assignment

Reference Node Naming

Importing Mesh-Based Databases in Batch

External Model Supported Element Types

Supported External Model Commands

Accessing Imported Mesh-Based Databases through ACT

Examples of a Synthesized Mesh using Tolerance Angles

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Supported External File Types

You can import the following finite element mesh files directly into Mechanical using the Workbench External Model system:

- Mechanical APDL common database (.cdb)[1]
- Workbench mesh data file (.acmo)
- ABAQUS Input (.inp)[2]
- NASTRAN Bulk Data (.bdf, .dat, .nas)
- Fluent Input (.msh, .cas)
- ICEM CFD Input (.uns)
- LS-DYNA Input (.k and .key)[3]

[1] The Mechanical APDL application defines contact conditions as elements and writes this data as such to the Mechanical APDL .cdb file. Conversely, the Mechanical application does not consider contact conditions as a part of the geometry (defined through elements). Mechanical defines its own entity. Therefore, when you import Mechanical common data base (.cdb) files into Mechanical, the application:

- Translates MAPDL contact elements to the equivalent Mechanical Connections entry.
- Does not process components (as Named Selections) that include contact elements. Furthermore, if you have a .cdb files that contains only contact element-based components, the application does not transfer these components.

As a result, these process requirements can lead to differences in the number of elements displayed by each application as well as the absence of Named Selections specified in one application but not available in the other.

[2] Note the following Workbench support limitations when importing **ABAQUS Input files**:

- Parts and Assemblies are not supported. That is:
  - → Only the data from the very first **Instance* command is read by External Model. All other data from any additional **Instance* command is ignored.
  - → The element and node sets, as well as the materials, that are associated with the first **Instance* command are processed. Any data that follows the first **End Part* or **End Instance* commands is ignored.
- Workbench does not support keywords that generate additional items, such as keyword **NGEN*. The commands **NSET*, **ELSET*, and **NODAL THICKNESS* are the only commands that support the Generation parameter.
- Only the first load step of the ABAQUS file is read by External Model.

[3] Note the following Workbench support limitations when importing **LS-DYNA Input files**:

- Workbench will generally only import mesh information. For a list of supported keywords, go to the List of LS-DYNA supported keywords (p. 972)
- Workbench does not support the *INCLUDE feature of LS-DYNA, and will only read the file you supply as an input.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

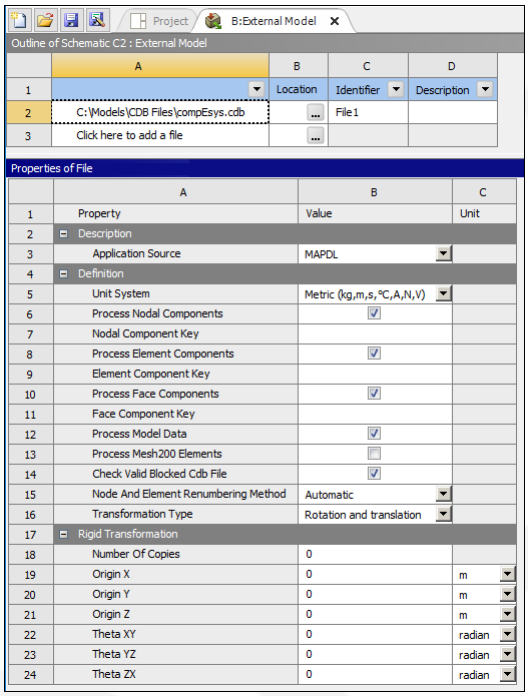
Import Workflow and Interface Options

Go to a section topic:

- External Model Properties in Workbench (p. 910)
- Model Cell Properties in Workbench (p. 910)
- Mechanical Interface Options (p. 913)
- Requirements and Limitations (p. 917)

External Model Properties in Workbench

The **External Model** tab is illustrated below. From this tab, you to modify various properties prior to importing your finite element data into Mechanical. Refer to the Creating and Configuring an External Model System topic in the **External Model** section for a description of the properties available on the External Model tab.

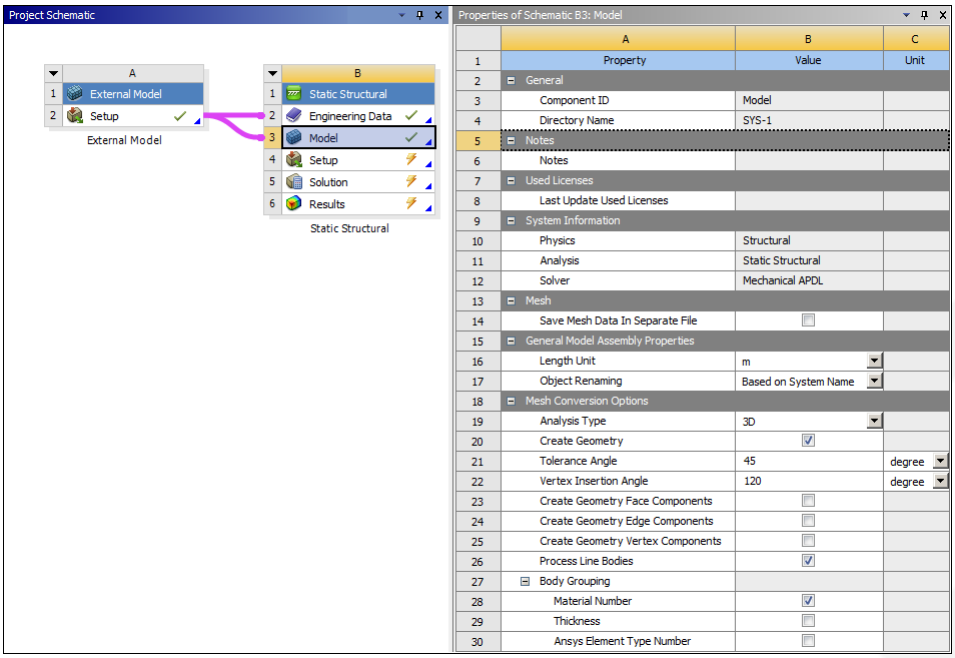


Model Cell Properties in Workbench

The properties of the **Model** cell on the Workbench **Project** page are illustrated below. You use the **Mesh Conversion Options** group of properties to specify import processes used on the finite element data. Properly defining these properties is important for you to accurately generate the desired geometries in Mechanical.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Mesh Conversion Options include:

- **Analysis Type**: Defines the mesh file as **3D** (default) or **2D**. When working with 2D analysis types, make sure that all of your model's surface normals point in the same direction using the **Rigid Transformation** properties available through the **External Model** feature.

  When you set this property to the **2D** option, in Mechanical, the application validates that the geometry is two-dimensional (2D) by checking the value of the **Length Z** property, using CAD units, in the **Bounding Box** category of the Geometry object. The tolerance used for this check is based on the setting of the **2D Tolerance** property (**Properties** category of the **Geometry** object). If you change the value of the **2D Tolerance** property, you must refresh the analysis on the Workbench project page in order for the application to re-validate the geometry.

- **Create Geometry**: This option is active by default. When selected, the application automatically creates your geometry in Mechanical based on solids, shells, and beam elements (that have an assigned cross-sectional definition) present in the mesh. When imported, a **Geometry** object is placed in the **Outline**. Deselecting this option instructs the application to import the mesh only (elements and nodes). When imported, an **Element Groups** object is placed in the **Outline**. **Element Groups** object and its children behave as a **Geometry** object and its children. That is, the Solid, Surface, and Line elements behave as bodies.

  Not creating a geometry dramatically decreases your import time as well as significantly reducing the amount of memory used during the process.

- **Tolerance Angle**: This value determines if adjacent elements are of the same face during the geometry creation process. The geometry creation process identifies groups of element facets on the exterior of the mesh. These generated facets create geometric faces in Mechanical. Then skin detection algorithm scans the exterior element facets and groups them based on a tolerance angle. For example, two adjacent element facets are grouped into the same face if the angle between their normals is less than or equal to the given tolerance angle. Therefore, an angle tolerance of 180^{o} creates only a single face for the whole body while a tolerance of 1^{o} creates

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

an amount of geometric faces which approaches the number of element faces if any curvature is present.

Calculations to synthesize geometries using tolerance angles use the implicit method. Processing nodal components on the same topology will override this method. See the illustrations below for examples of this behavior.

The default **Tolerance Angle** is **45** degrees. This is the recommended setting.

- **Vertex Insertion Angle**: The Vertex Insertion Angle is the minimum angle to insert a vertex between two free edges of mesh. The default value is **120** degrees. During the generation of the geometry, if two segments of an edge abruptly make an angle greater than the Vertex Insertion Angle, then the edge is split and a vertex is inserted.
- **Create Geometry Face/Edge/Vertex Components**: These options become active when your mesh file contains node-based named selections. When active, the application automatically creates geometric named selections (face/edge/vertex). For a vertex or an edge, the node-based named selection can include only include one node or one edge. For faces, nodes from multiple faces can be included in the node-based named selection. This option will not affect the geometry creation algorithm itself, which is based purely on the mesh adjacency angles as detailed above.

  Note:

  The application automatically transfers geometry-based Named Selections for faces.

- **Component Key**: If you select one or more of the **Create Geometry Face/Edge/Vertex Components** option, this property enables you to generate geometric components specific to the key name that you enter. The application evaluates keys from the beginning of the string value. For example, a given mesh file has the following components: Fixed_Support1, Fixed_Support2, and Force1. If you enter "Fixed_Support", Mechanical automatically produces Named Selections for Fixed_Support1 and Fixed_Support2, but not Force1.
- **Process Line Bodies**: This property enables you to import finite element line bodies (see Line Bodies (p. 872)) that have a properly defined cross-section. For these finite element line bodies, the following element types are supported (p. 969).
  - For CDB files: see the **Beam Shape Category** in the External Model Supported Element Types (p. 969) section of the Help.
  - For NASTRAN files: the CBAR and CBEAM element types are supported and require a cross-section property defined via PBAR, PBARL, PBEAM, or PBEAML.
  - For ABAQUS files: multiple options are available. Primary properties include: *BEAM SECTION, *FRAME SECTION, and *BEAM GENERAL SECTION.
- **Body Grouping**: This property tells Mechanical how to group the elements (of the same type) from the data file. Elements are always grouped by shape and different shaped elements are placed in separate groups. This option controls the number of element groups/bodies created in Mechanical. You use the following options to specify how you want to group elements. You can select one or more (or all) options. Property options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Material Number** (default): Group elements based on the material number assigned to the elements. The application creates a group for **each** Material Number.
- **Thickness**: Group all elements that have the same thickness number.
- **Ansys Element Type Number**: For Mechanical APDL common database (.cdb) files only, this option enables you to group elements by their local Ansys element type that was assigned from the element library. Note that the element type number is the first argument to the **ET** command in the (.cdb) file.

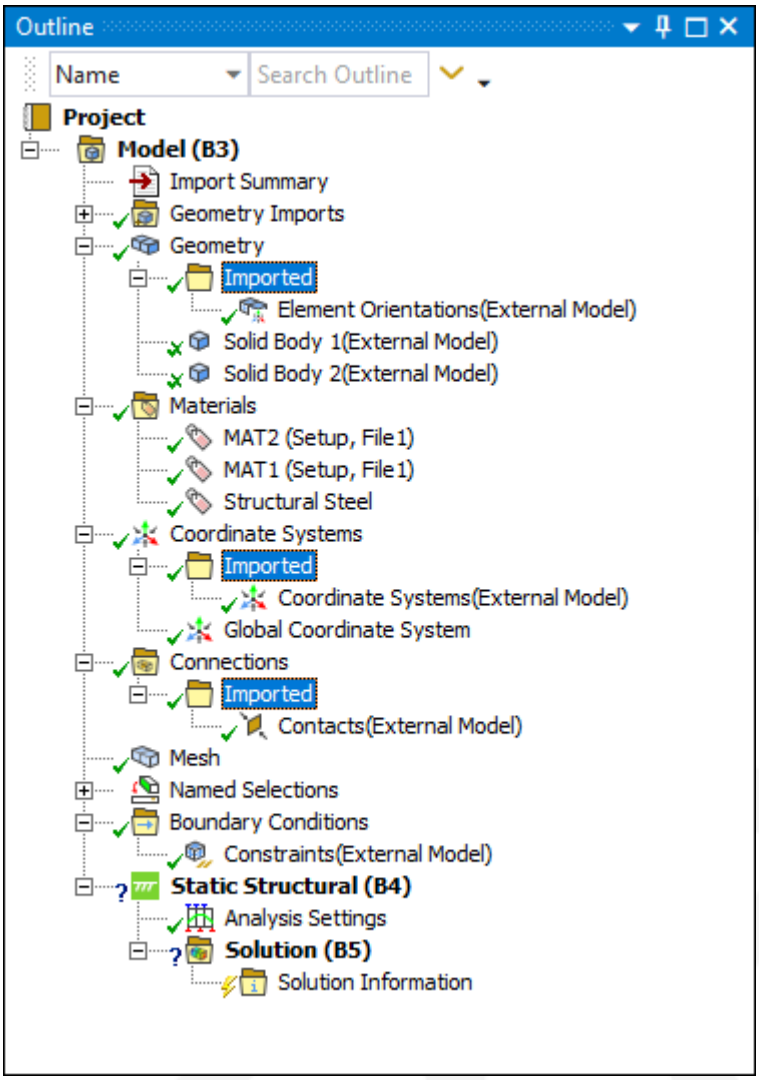
Important:

- For LS-DYNA files, Mechanical creates only one part. Elements are grouped into bodies first by element shape then by Part ID as defined in the LS-DYNA file and then (if chosen), by **Material Number** or **Thickness**. Therefore each element in a body will have same shape, LS-DYNA Part ID and (if chosen), and same Material Number and Thickness. In addition, each body listed in the Outline has its part name specified using the *PART keyword in the input file.
- For ICEM CFD Files, the application uses data internal to ICEM CFD files to group elements into bodies and to create geometric faces. Therefore, the **Tolerance Angle** is ignored.

Mechanical Interface Options

When you import an **External Model** source file that includes any of the finite element data listed below (p. 918), Mechanical inserts an "Imported" folder (excluding Named Selections) beneath the corresponding **Outline** object. As illustrated in the example **Outline** below, this **Imported** folder object is a simple group folder that contains child objects for the associated data type.

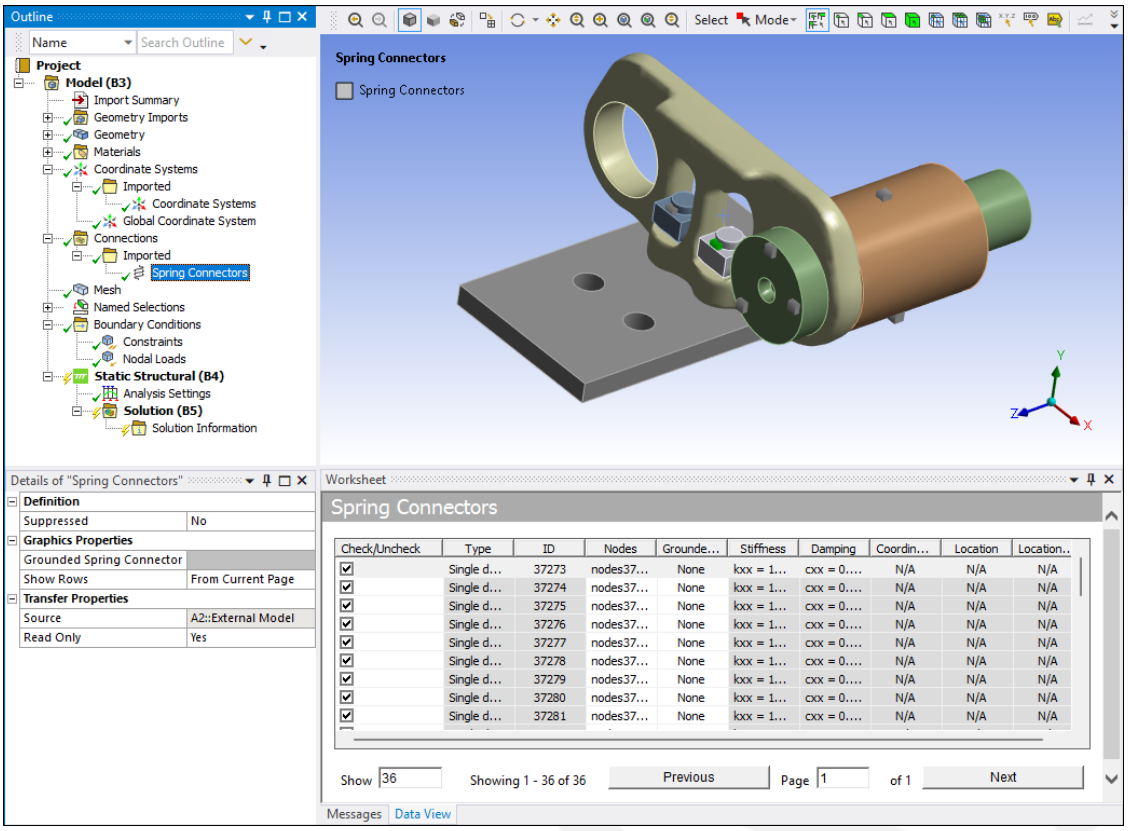
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Selecting a child object of the **Imported** folder displays the mesh/geometry in the **Geometry** window, with the imported data type highlighted on the mesh/geometry, as well as the **Worksheet**. An example of a selected imported spring (p. 956) is illustrated below. The **Worksheet** displays a summary of all imported data including column headings for the specific data associated with each data entry (connection Type, node ID, etc.). Each row of the **Worksheet** represents a different set of data. You can sort the table data by clicking on a column heading. The check-box for each **Worksheet** row enables you to deactivate (suppress) the row. And, based on the number of data items imported, the **Worksheet** displays 500 (default) data items per worksheet page. Use the display and/or navigational options at the bottom of the window to display more data items per page as well as to step through the available pages of data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



A graphical representation of the data is also shown in the **Geometry** window. You can choose which data is displayed in the **Geometry** window using the **Show Rows** property under **Graphics Properties** category in the **Details** view. The **Show Rows** property includes the following options:

- **From Current Page** (default): Only display the data from the current page in the **Worksheet**.
- **From All Pages**: Display all of the data for the object independent of the rows visible in **Worksheet**.
- **None**: Do not display any data in the **Geometry** window.

In addition, for the following objects, you have the ability to select the color used for the displayed data in the **Geometry** window through a color selection property in the **Details** view:

- Imported Boundary Conditions (p. 919)
- Imported Constraint Equations or Coupling (p. 928)
- Imported Flexible Remote Connectors (p. 942)
- Imported Point Masses (p. 947)
- Import Rigid Bodies (p. 950)
- Imported Rigid Remote Connectors (p. 951)
- Imported Spring Connectors (p. 956)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Worksheet

Furthermore, when you select a **Worksheet** row, the graphical representation of the data in the **Geometry** window becomes highlighted, such as the green spring shown in the above image. You can also select the graphical representation directly in the **Geometry** window by activating the **Imported Data Highlight** option on the Graphics Toolbar (p. 97). Normal mouse and key combinations for selecting, deselecting, modifying, or clearing selections act just as they do with geometry selections, etc. This selection feature is available for the following types of imported data:

- Imported Constraint Equations or Coupling (p. 928)
- Imported Contacts (p. 930)
- Imported Flexible Remote Connectors (p. 942)
- Imported Point Masses (p. 947)
- Imported Rigid Remote Connectors (p. 951)
- Imported Spring Connectors (p. 956)

In addition, for the above data types, when you select the graphical representation (Point Mass, Spring, etc.) in the **Geometry** window and right-click, the context menu provides the following applicable options:

- **Go To** > **Corresponding Bodies in Tree**: This option navigates to attached bodies in the tree.
- **Filter Worksheet Based on Selection**: This option only displays the selected objects in the worksheet. You can revert to viewing a specified number of rows in the Worksheet by editing the column headings via the right-click **Show** option in the Worksheet.

When the finite element data is displayed in the **Worksheet**, there are common options you can employ when you right-click on a **Worksheet** row, including:

- **Promote**: When you promote a **Worksheet** entry, the data, in whichever form (Coordinate System, Element Orientation, etc.), remains linked to the external system. You can modify the data within Mechanical, however; your changes are not transfer to the external system. In addition, any changes made in the external system will overwrite any modifications you make in Mechanical. Generally, you promote a Worksheet as a scoping in the form of a Named Selection. However, for Imported Bolt Pretensions, you can promote a Bolt Pretension, to a desired environment, in the form of a load.
- **Copy as new**: When you copy a Worksheet entry, a new independent object, based on the type of data, is inserted into the Tree outline. The application assigns a default name to the object based on the data type (Coordinate System, Element Orientation, etc.).
- **Show**: This option enables you to select which columns you wish to display on the Worksheet.
- **Check/Uncheck**: This option enables you to deactivate (suppress) a worksheet row.
- **Go To Promoted**: This option becomes available after you have promoted a Worksheet entry. This option takes to you the corresponding Tree outline object. This is typically related

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to a scoping that you have created through promotion. However, for Imported Bolt Pretensions, it can correspond to a Bolt Pretension promoted in the form of a load.

- **Edit Items**: This option becomes available when your finite element data type has editable Worksheet content. It enables you to make changes to the associated data items, such as a node ID. You can select individual or multiple rows as well as the data type tree object. When you select multiple rows or the tree object, the application applies all changes to all of the selected rows or to all of the Worksheet content, respectively.

Note:

Currently, Mechanical supports the Promote and Copy options for the following imported data:

- **Bolt Pretensions**
- **Contacts**
- **Coordinate Systems**
- **Element Orientations**
- **Flexible Remote Connectors**
- **Point Masses** (when scoped to a node of the geometry only)
- **Rigid Remote Connectors**

Requirements and Limitations

Note the following requirements and limitations for importing mesh-based geometries:

- The application does not copy input files specified in **External Model** into the project folders of downstream systems. The application references these files by absolute path only. Be sure you don't move or rename these files.
- Geometry construction is for 3D lines, solids, and shells and 2D planar bodies only. Mechanical ignores any other element types contained in the mesh file.
- Mechanical only processes node-based components when attempting to create geometry-based Named Selections for the faces. The application ignores element components.
- You cannot change the meshes. That is, you cannot change, clear, or re-mesh once the file has been imported into Mechanical.
- Mesh controls (Refinement, etc.) are not supported.
- Adaptive Mesh Refinement is not supported.
- Geometry is not associative. As a result, if you update the environment, for example, by adding another mesh file, any scoping that you have performed on an object will be lost. To avoid losses

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

to your analysis environment, make sure that you have properly defined the imported Named Selections or criterion-based Named Selections.

- The application specifies the **Stiffness Behavior** property as either **Flexible** or **Rigid**. This is a read-only property.
- The **Scale Factor Value** property on the **Geometry** object is not supported.

Supported Finite Element Data Types

Mechanical enables you to import the following finite element data types from **External Model**:

Imported Bolt Pretensions and Premeshed Bolt Pretensions

Imported Boundary Conditions

Imported Composite Plies

Imported Constraint Equations or Coupling

Imported Contacts

Imported Coordinate Systems

Imported Cross Sections

Imported Element Orientations

Imported Flexible Remote Connectors

Imported Named Selections

Imported Nodal Orientations

Imported Point Mass

Imported Rigid Bodies

Imported Rigid Remote Connectors

Imported Shell Thicknesses

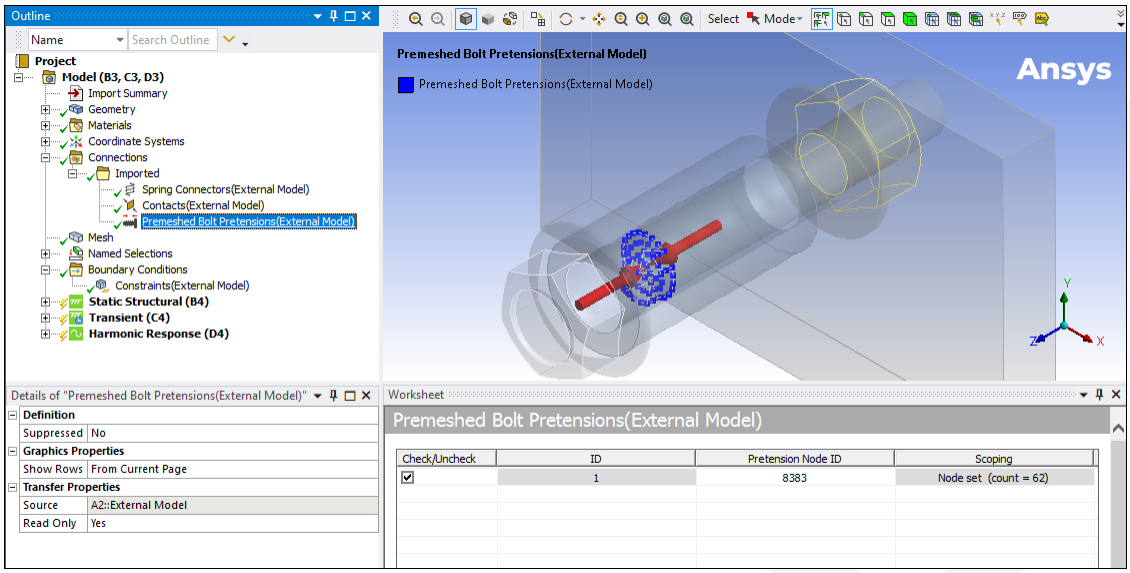
Imported Spring Connectors

Imported Initial Stresses

Imported Bolt Pretensions and Premeshed Bolt Pretensions

When your upstream External Model source file includes bolt pretension loads, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder. It contains the child object: **Bolt Pretensions** (or **Premeshed Bolt Pretensions**). The data associated with the imported bolt pretension is provided through the **Worksheet**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

For **Premeshed Bolt Pretensions**, the nodes at the pretension section interface are coupled using **CP** commands to provide continuity for all other analyses types where Bolt Pretension is not applicable.

Supported Analysis Types

Imported Bolt Pretensions and Premeshed Bolt Pretensions is supported by the following analysis types:

- Coupled Field Analyses
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data as contact:

- **CDB**: PRETS179 elements are processed as Pre-meshed Bolt Pretensions.
- **NASTRAN**: Not supported.
- **ABAQUS**: The **PRE-TENSION** and **PRE-TENSION SECTION** commands are processed as Bolt Pretensions. **NODE** (as integer or component name), **SURFACE** (as component name), and **ELEMENT** (as integer or component name) are the supported parameters.

See the Imported: Bolt Pretensions object reference section for additional information.

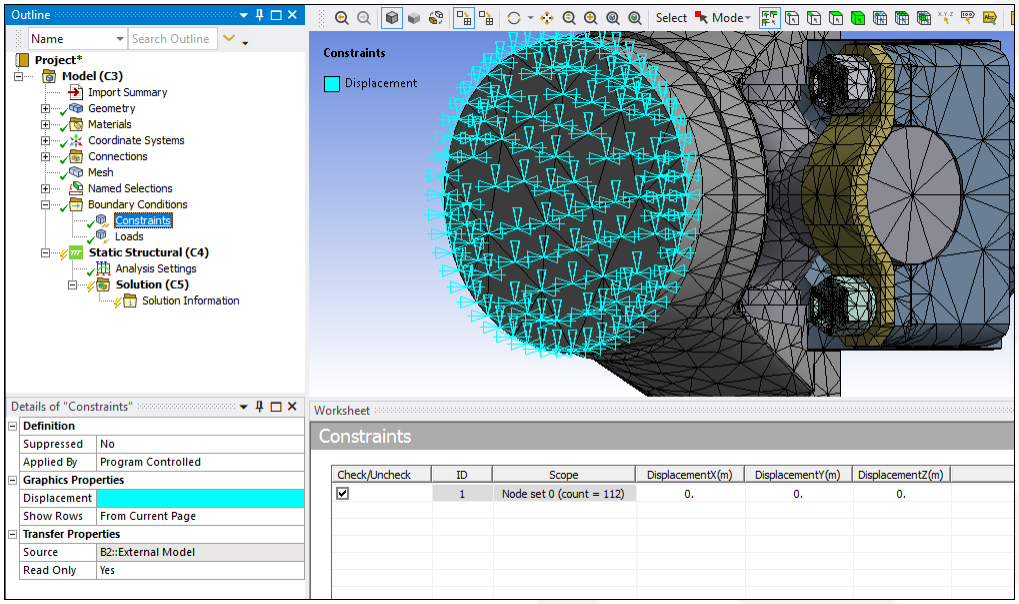
Imported Boundary Conditions

When your upstream External Model source file includes constraint and/or loading conditions, a **Boundary Conditions** folder appears beneath the **Model** parent folder when you open the file

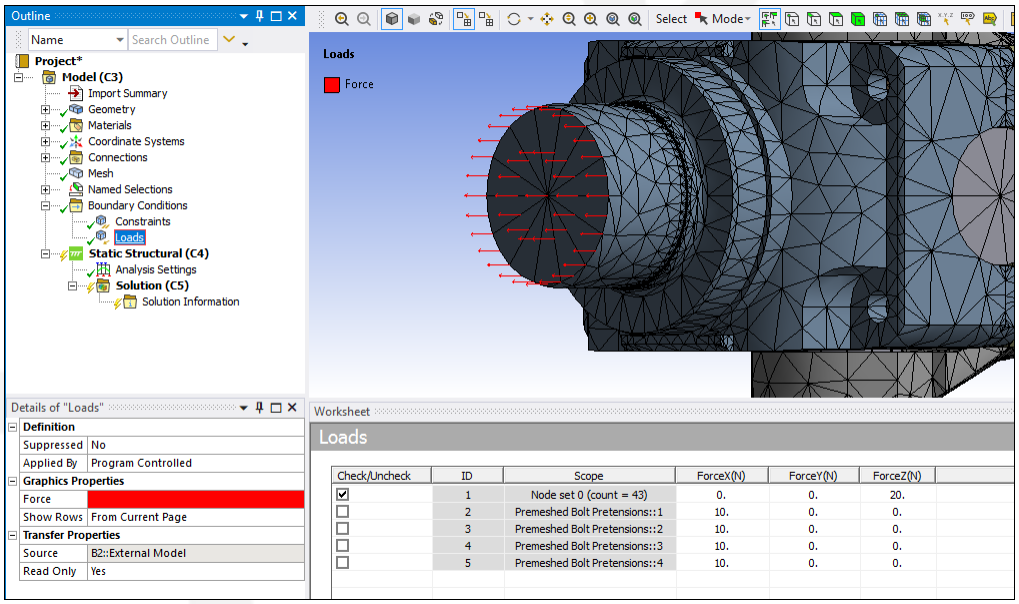
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

in Mechanical. It can contain the child objects: **Constraints**, **Nodal Loads**, **Surface Loads**, **Body Loads**, and **Acceleration**. When required, the data associated with the imported boundary conditions is provided through the **Worksheet**. **Worksheet** content, including all default settings, is based on the data in the external file.

Constraints Example



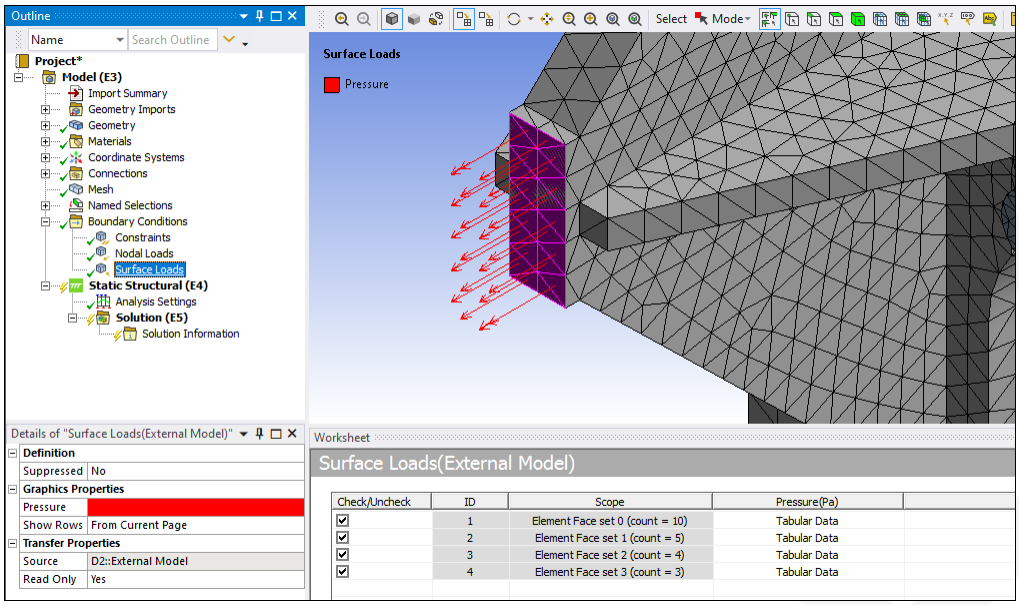
Nodal Loads Example



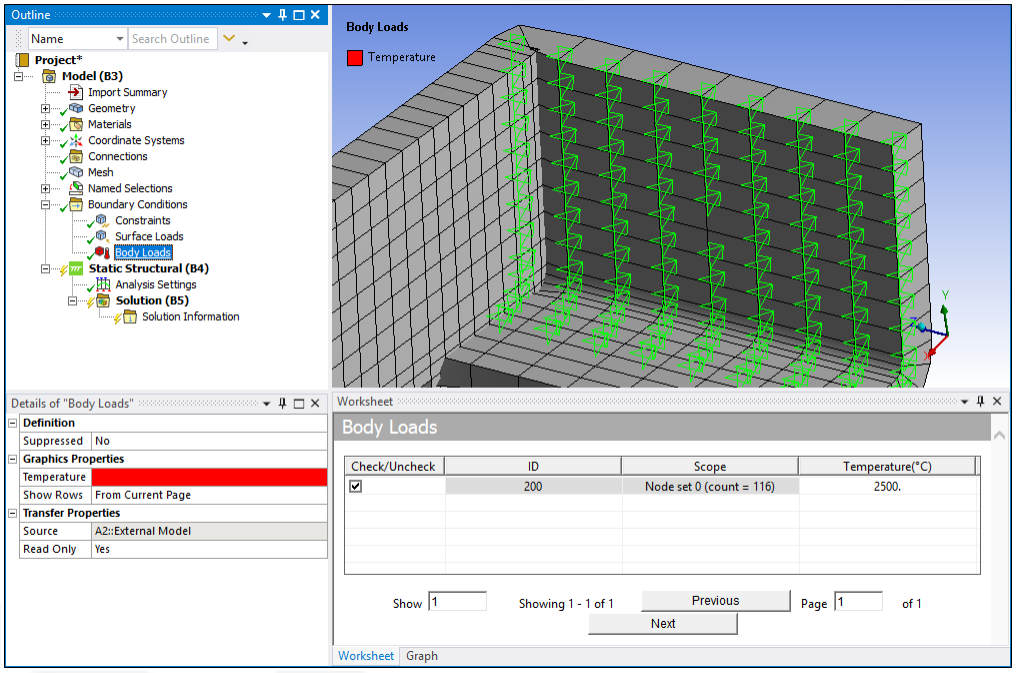
Surface Loads Example

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



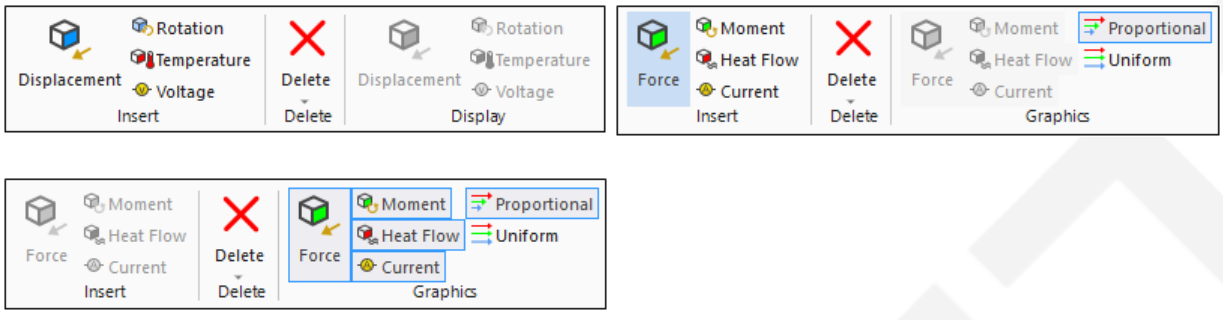
Body Load Example



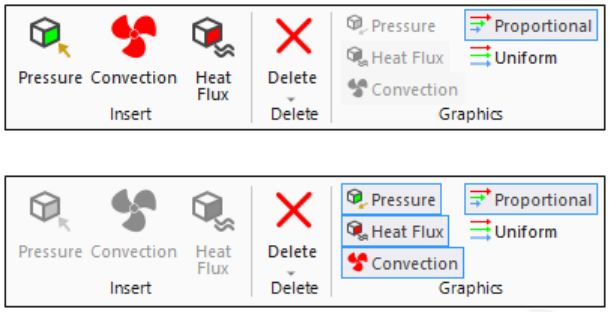
In addition to the degrees of freedom (DOFs) imported from the source file, you have the ability to expand the applicable DOFs for an imported Constraints/Nodal Loads/Surface Loads object. For example, an imported file used to define a thermal analysis that you also wish to use for a structural analysis. The group options illustrated below display when you select the **Constraints**, **Nodal Loads**, or the **Surface Loads** objects. These options enable you to add (or remove) additional DOFs to the existing Constraints/Nodal Loads/Surface Loads. Once added, the DOFs show up in the worksheet, where you can then specify desired values. No ribbon options are available for Body Loads as there is only one DOF.

Constraints Options **Nodal Loads Options**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Surface Loads Options



Graphics Options for Constraints, Nodal, Surface, and Body Loads

When you have one of the objects selected, a corresponding **Graphics** group, as illustrated above, is also displayed. These tab groups enable you to turn the display of any of the options, Displacement, Rotation, etc., on or off. In addition, the graphics options for Nodal Loads and Surface Loads provides two buttons that change the display of vectors from a Proportional display to a Uniform display.

Load Step Support

The imported Constraint/Nodal Load/Surface Load objects support step dependent loading. Selecting a row in the worksheet activates the **Tabular Data** window that displays the constraint/load values for each step present in the analysis. You can edit the data in tabular view to apply values varying across steps. In addition, user has the ability to activate/deactivate the constraint/load components for each step through the context (right-click) menu on the **Tabular Data** window. See the Activation/Deactivation of Loads (p. 1423) topic for additional information.

Supported Analysis Types

Imported Boundary Conditions (**Constraints**, **Nodal Loads**, **Surface Loads**, and **Body Loads**) support the following analysis types:

- Coupled Field Analyses
- Static Structural
- Transient Structural (except when linked to Modal Solution with Mode-Superposition Solution Method.)
- Eigenvalue Buckling
- Harmonic Response

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Modal
- Steady-State Thermal
- Transient Thermal
- Electric
- Thermal-Electric

Supported Source File Commands

The application imports the following source file data/commands as loading conditions:

- **CDB**: The supported Mechanical APDL commands include:
  - **BF**/**BFBLOCK** and **BFE**/**BFEBLOCK** commands, with the degree of freedom **TEMP**, imported as body loads. Note that the **BFE**/**BFEBLOCK** commands support the elements SHELL181 and SHELL281 only.
  - **D** commands, imported as constraints.
  - **F** commands, imported as loads.
  - **SFE**/**SFEBLOCK** commands, imported as surface loads, applied using the elements SURF154, SOLID185, SOLID186, SOLID187, SHELL181, SOLID285, and SHELL281. Only normal pressures are supported. Tractions pressures are not supported. Thermal convection and heat flux loads, for solids and surfaces, are applied using the elements SURF152, SHELL131, SHELL132, SOLID70, SOLID87, SOLID90, SOLID278, and SOLID279.
  - **ACEL** command to simulate gravity.

  Important:

  Mechanical does not import **ACEL**, **D**, **F**, **SFE**/**SFEBLOCK**, **BF**/**BFBLOCK**, and **BFE**/**BFEBLOCK** commands specified using tables.

- **NASTRAN**: The supported NASTRAN commands include:
  - **SPC** command is imported as a constraint for a NASTRAN structural analysis and a thermal constrain for a NASTRAN thermal analysis.
  - **SPC1**, **SPCD**, and **SPCADD** commands that are imported as constraints.
  - **LOAD** and **SUBCASE** commands imported as load cases.
  - **FORCE**, **FORCE1**, **FORCE2**, **MOMENT**, **MOMENT1**, and **MOMENT2** commands that are imported as loads.
  - **TEMP** command, is imported as either a temperature Load (Body) for NASTRAN structural analyses or as temperature Constraint for NASTRAN thermal analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - **TEMPD** command, is imported as a temperature Load (Body) for NASTRAN structural analyses.

  - **PLOAD**, **PLOAD2**, and **PLOAD4** commands are imported as Surface Loads. Only normal pressures are supported. Tractions pressures are not supported.

  - **GRAV** command defines gravity vectors to determine gravity loading on the model.

  In the absence of **SPC** or **LOAD** and **SUBCASE**, all of the above commands are imported. If the input file includes the **LOAD**, **SPC**, or **SUBCASE** commands, then load steps are determined and only those commands which are selected in the Case Control Section are imported.

- **ABAQUS**: The supported ABAQUS commands include:

  - *BOUNDARY command. The supported include TYPE with value as DISPLACEMENT, FIXED, or OP. The data lines using both Direct format and Type format are supported. For the type format, all are supported except NOWARP, NOOVAL, and NODEFORM.

  - *CLOAD command imported as loads. The OP parameter is supported. And, all of the data lines are supported in this command category.

  - *STEP and *END STEP commands are processed as load steps in the analysis. The parameters for both the commands are ignored.

  - *DSLOAD and *DLOAD commands are supported for surface and gravity loads. For surface loads, the OP parameter is the only optional parameter supported. And, all of the data lines whose distributed load type label is P are supported in this command category. For gravity loads, use the GRAV option.

  - DSFLUX and *DFLUX commands are supported for Heat Flux. The OP parameter is the only optional parameter supported. And, all of the data lines whose distributed load type label is S are supported in this command category.

  - *SFILM and *FILM commands are supported for Convection. The OP parameter is the only optional parameter supported. And, all of the data lines whose distributed load type label is F are supported in this command category.

- **LS-DYNA**: The supported LS-DYNA commands are *BOUNDARY_SPC_NODE and *BOUNDARY_SPC_SET.

Also see the Imported: Boundary Conditions object reference section for additional information.

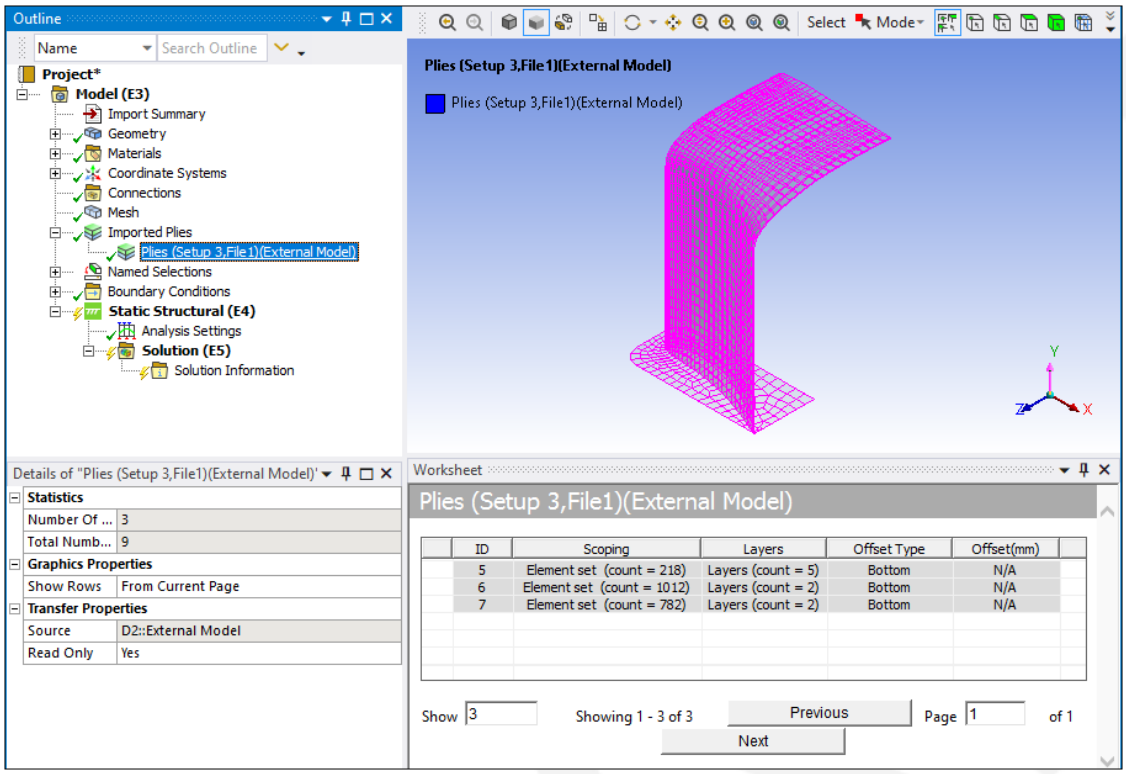
Imported Composite Plies

When your upstream External Model source file includes composite data, an **Imported Plies** folder appears beneath the **Model** parent folder when you open the file in Mechanical.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

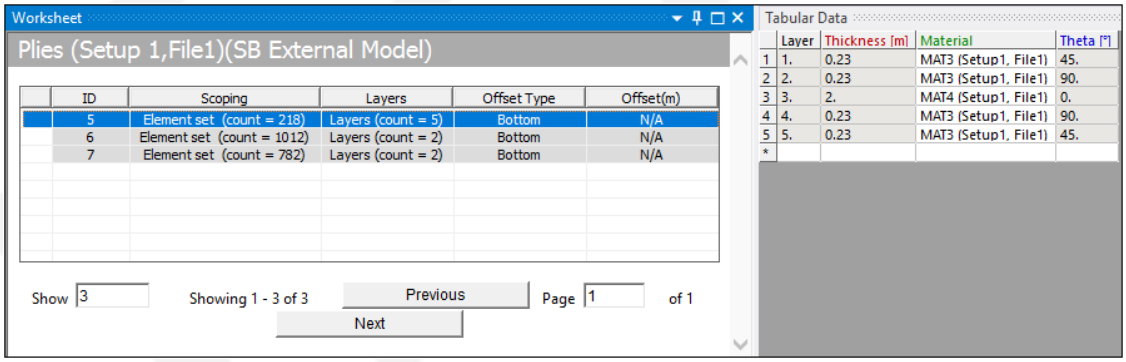
Published: 2023-10-23

Imported Plies Object



Plies Object

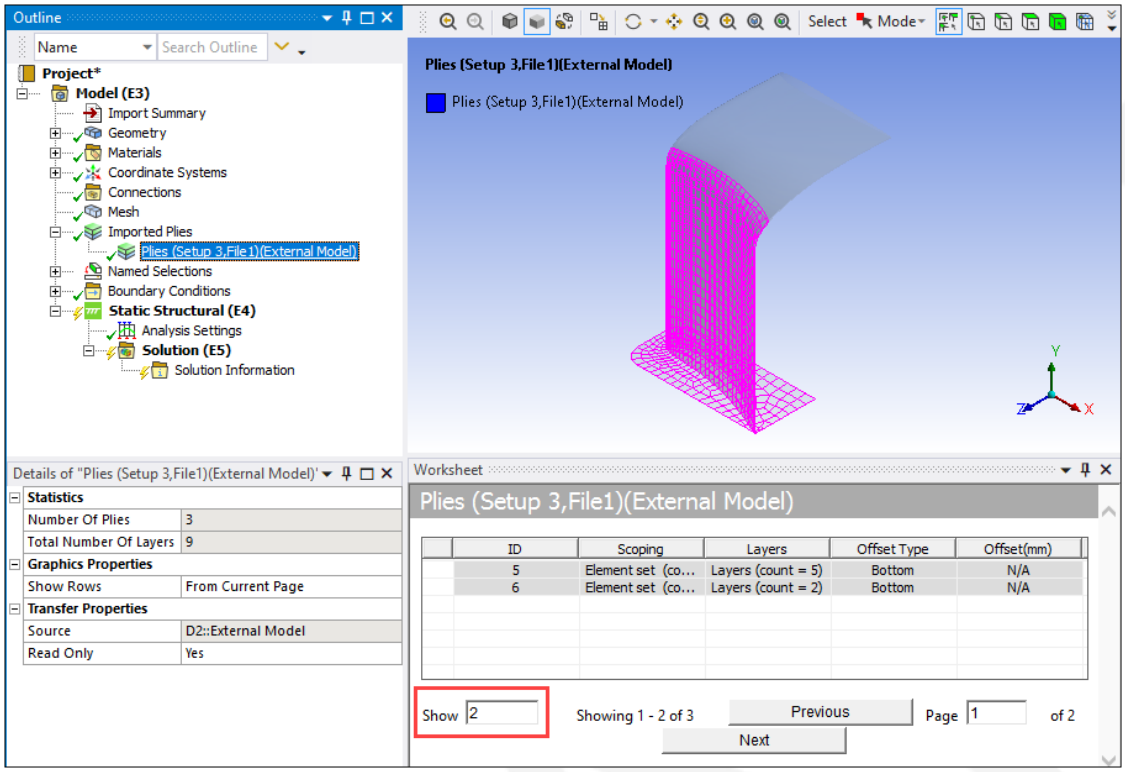
The **Imported Plies** object includes the child object **Plies**. This object displays a parenthetical that displays the upstream Component ID and file Identifier (Setup 1, File 1) as well as the system name (External Model). When selected, the **Worksheet** displays the source file data, and all ply data is highlighted in the Geometry window.



Display Feature

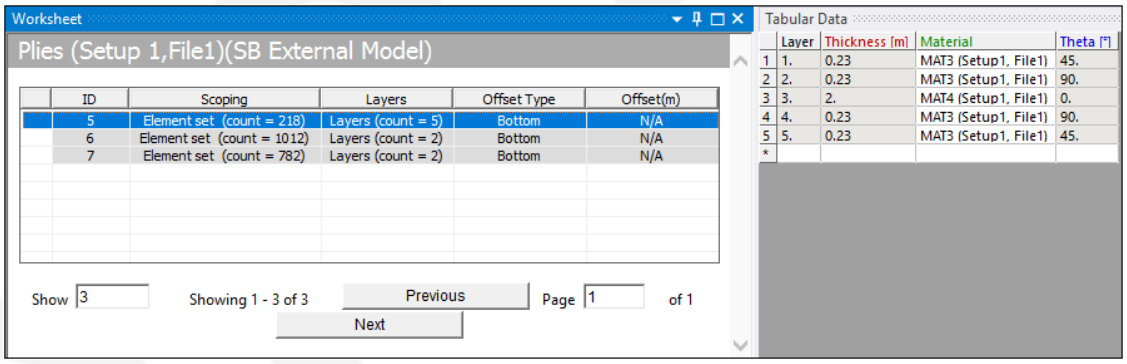
To visually check the sections, you can change the entry of the **Show** field, based on the available rows, to change which sections are highlighted in the Geometry window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Worksheet and Tabular Data

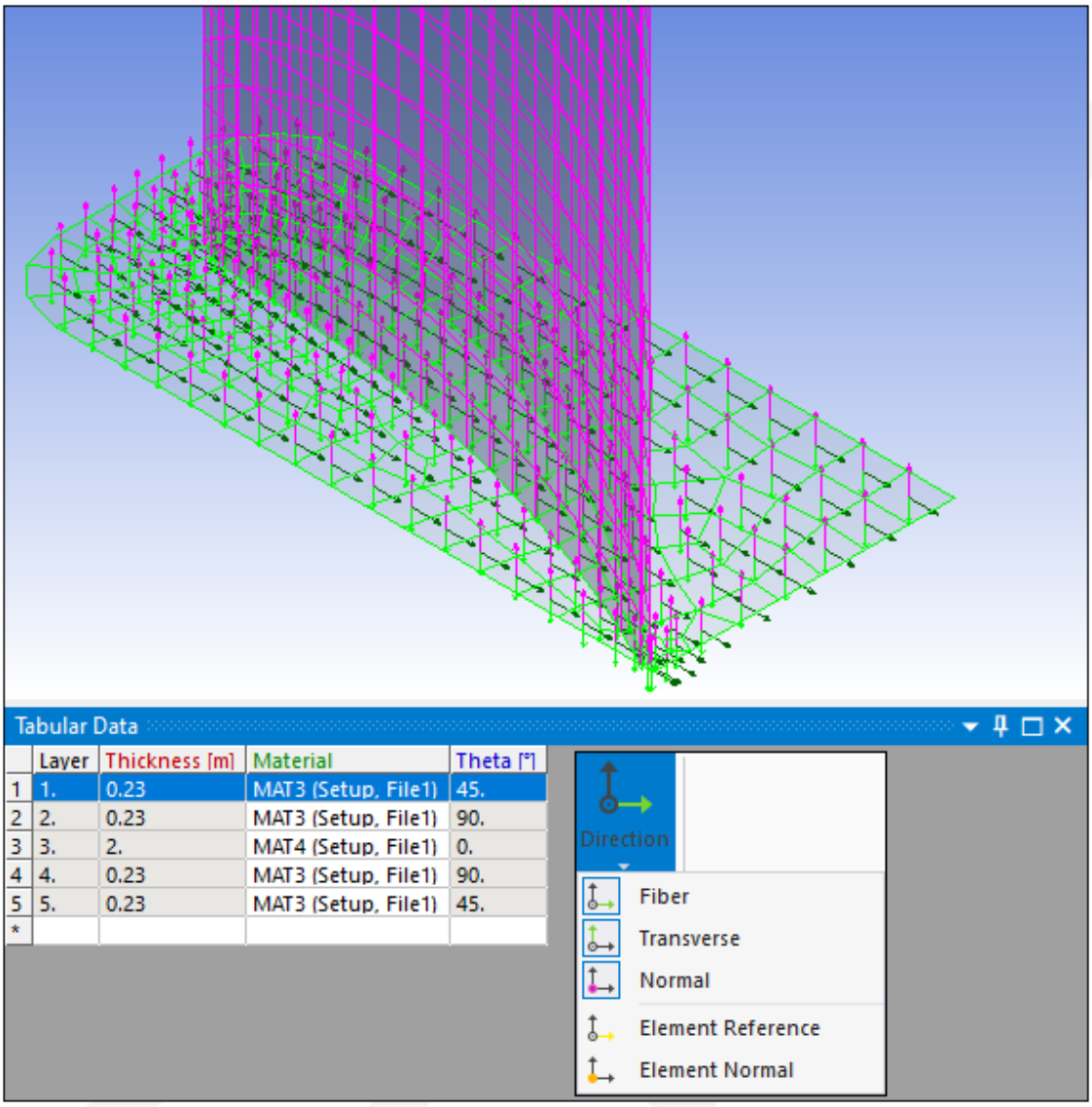
When you select a row in the **Worksheet**, 1) the elements for the selected row are highlighted in green in the Geometry, and 2) the **Tabular Data** window displays the associated layer data. You can modify the **Material** for the ply as needed in this window.



When you select a row in **Tabular Data**, you can use the ply-based display options of the **Direction** drop-down menu of the **Ply** group (p. 50) on the **Model** tab to display ply and element directions for imported ply structures.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Supported Analysis Types

The composite plies import type supports the following analysis types:

- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands for composites:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **CDB**: **SECTYPE**, **SECDATA**, **SECBLOCK**, and **SECOFFSET**.

  Note:

  Sections defined with the SECBLOCK command that have a single layer, where theta=0.0, are only imported as **Imported Plies** if the section is defined with one of the following element types: SOLID185, SOLID186, SOLSH190, SOLID278, and SOLID279.

- **NASTRAN**: PCOMP, PCOMPS, PCOMPG.
- **ABAQUS**: *SHELL SECTION. Note the following processing requirements:
  - The *SHELL SECTION command uses the ORIENTATION parameter to define ply directions. External Model only supports the use of ORIENTATION parameter on the command line. That is, the application does not support the use of the ORIENTATION parameter on data lines (for each layer). Here is an example of a supported *SHELL SECTION composite definition:

Command	*SHELL SECTION, ORIENTATION=Orient1, elset=elset1, composite, layup=Composite_Layup
Data Lines	0.125, 3, Glass-Epoxy, 45, L1_IS_1 0.500, 3, Glass-Epoxy, 0, L1_IS_2 0.125, 3, Glass-Epoxy, -45, L1_IS_3

  - If an ORIENTATION parameter is specified, the SYSTEM parameter must be set to RECTANGULAR for the associated ORIENTATION (SYSTEM= RECTANGULAR).
  - External Model only supports the STACK DIRECTION parameter value equal to 3 for the *SHELL SECTION commands. If this parameter has any other value specified in the input file, the associated *SHELL SECTION command is not imported/processed.

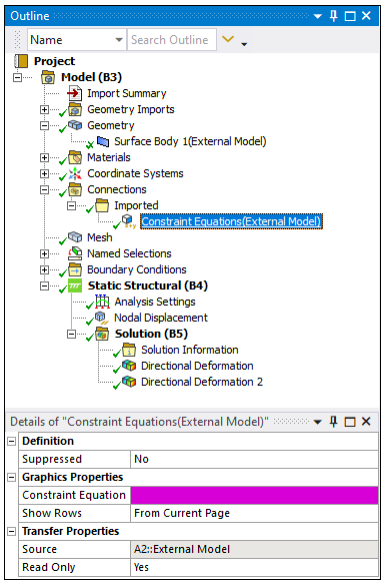
For additional information, see the Imported: Composite Plies object reference section as well as the **SECBLOCK for Shells** topic in the SECBLOCK Command section of the *Mechanical APDL Programmer's Reference*

Imported Constraint Equations or Coupling

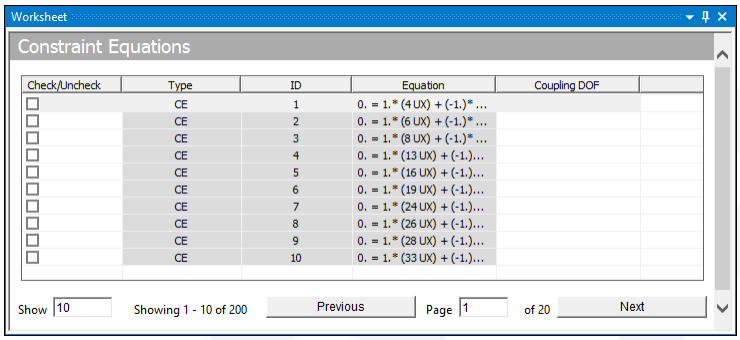
When your upstream External Model source file includes constraint equations or coupling, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder. It contains the child object: **Constraint Equations**. The data associated with the imported equations is provided through the Worksheet. You can Check/Uncheck (p. 916) Worksheet entries using the selection box in the table or through the right-click context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

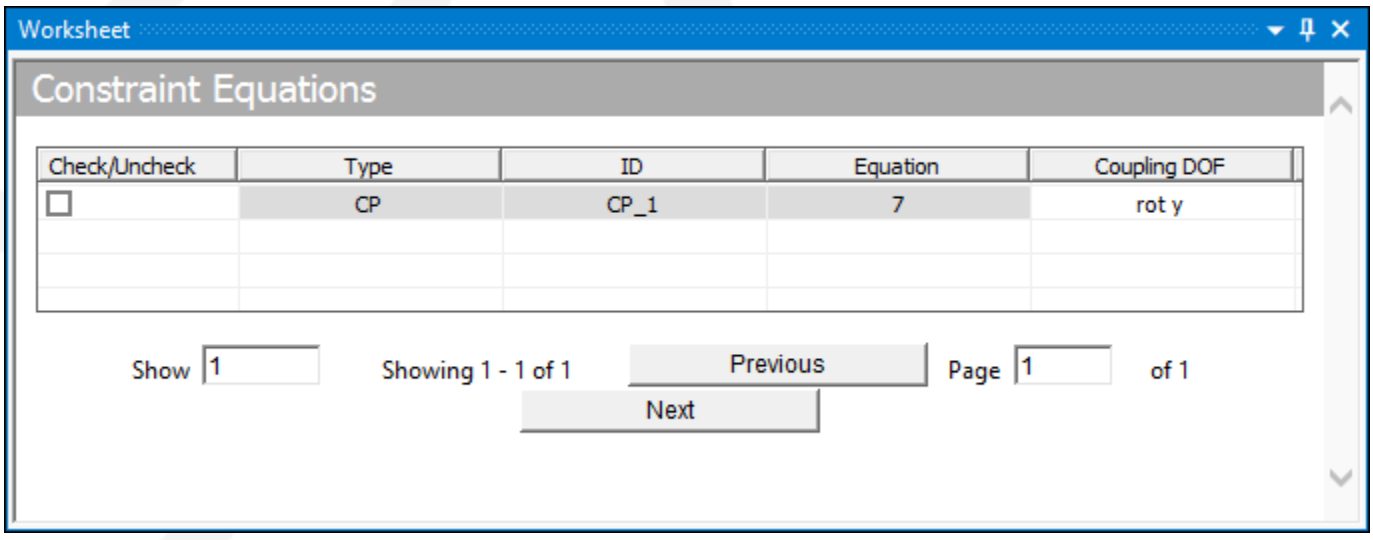
Published: 2023-10-23



Imported Constraint Worksheet Data



Imported Coupling Worksheet Data



Supported Analysis Types

Constraint Equations and Coupling import types are supported by the following analysis types:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural
- Steady-State Thermal (Coupling only)
- Transient Thermal (Coupling only)

Supported Source File Commands

The application imports the following source file data/commands as constraint equations or coupling:

- **CDB**: The **CE** commands are processed as constraint equations and the **CP** commands are processed as couplings.
- **NASTRAN**: MPC commands are processed as constraint equations.
- **ABAQUS**: *EQUATION commands are processed as constraint equations and *MPC TIE commands as couplings.

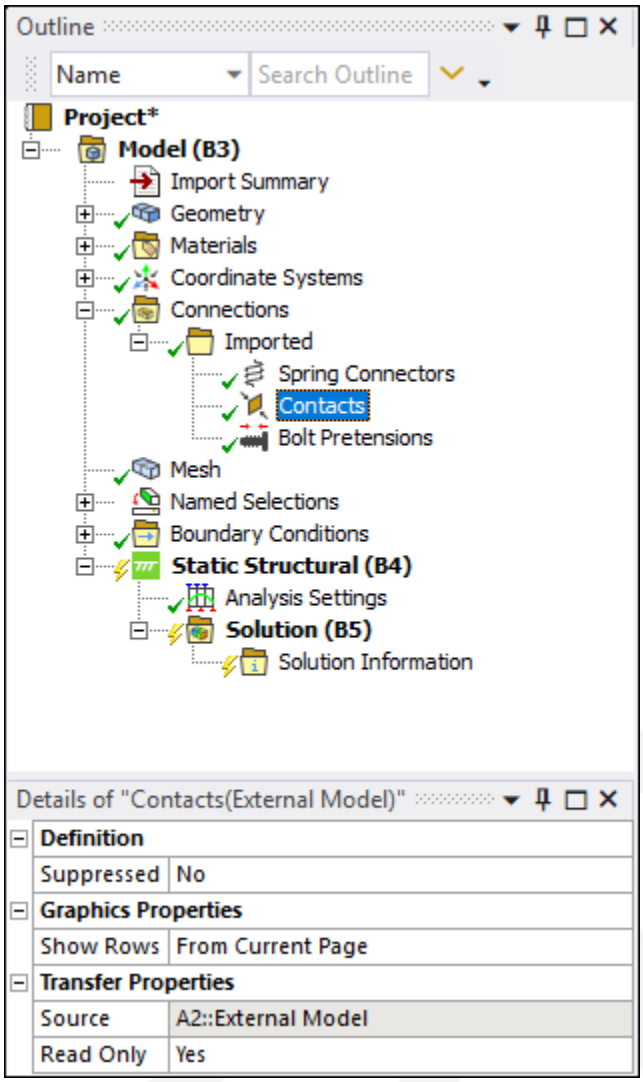
See the Imported: Constraint Equation object reference section for additional information.

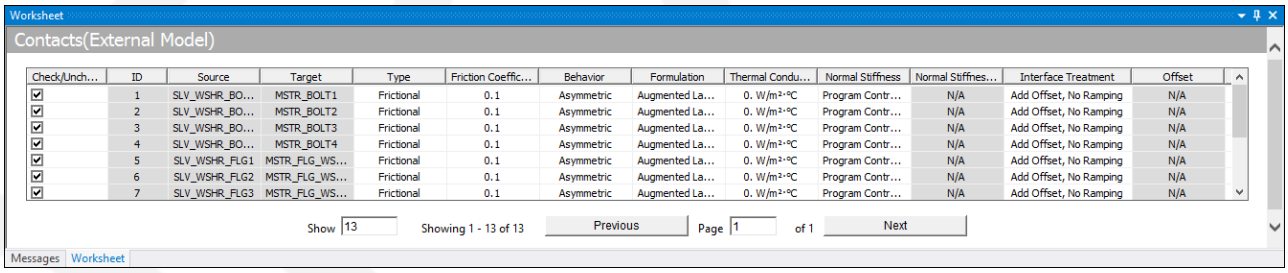
Imported Contacts

When your upstream External Model source file includes surface to surface contact, from either solid or shell bodies, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder. It contains the child object: **Contacts**. The data associated with the imported contact is provided through the **Worksheet**. You can Check/Uncheck (p. 916) Worksheet entries using the selection box in the table or through the right-click context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Supported Analysis Types

Imported contact is supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Transient Structural
- Steady State Thermal
- Transient Thermal

Note:

The application imports thermal conductivity data for thermal analyses.

Supported Source File Commands

The application imports the following source file data as contact:

CDB

- Surface contacts in the form of CONTA173 (source) and CONTA174 (source), CONTA175 (source), and TARGE170 (target) elements.
- Target elements will usually define the element shape using the element shape flag (column 8) in the EBLOCK section. If not defined, the number of nodes for that target element will be used. The following target element shapes are supported:
  - Linear triangular shape – 6
  - Linear quadrilateral shape – 7
  - Quadratic triangular shape – 8
  - Quadratic quadrilateral shape - 9
- The following Element Type Options (KEYOPT) are supported:
  - KEYOPT(2) - Contact Algorithm
  - KEYOPT(3) – Units of normal contact stiffness

    Values 0 and 1 only
  - KEYOPY(8) – Symmetric Contact Behavior
  - KEYOPT(9) – Effect of Initial Penetration or Gap

    Values 0, 1, 2, 4, and 5 supported
    All other values default to 0
  - KEYOPT(12) – Behavior of Contact Surface
  - KEYOPT(18) – Sliding Behavior
- The following Real Constants are supported:
  - REAL(3) - Normal Penalty Stiffness Factor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - REAL(10) – Contact Surface Stiffness
  - REAL(14) – Thermal Contact Conductance
- Material Property MU – Coefficient of Friction

NASTRAN

Not supported.

ABAQUS

Mechanical supports the following ABAQUS contact keywords:

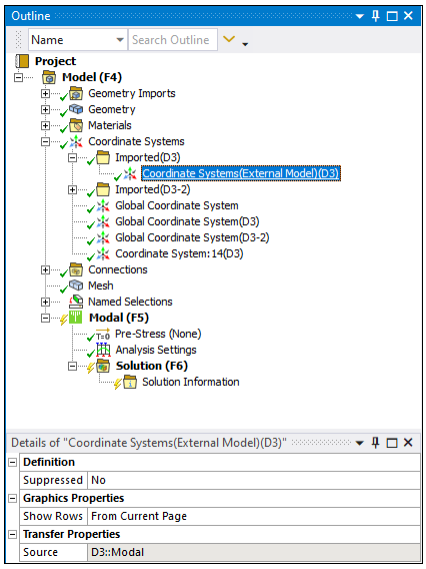
- *CONTACT PAIR. Supported parameters:
  - TYPE
  - SMALL SLIDING
  - HCRIT
  - INTERACTION
  - TIED
- *CONTACT INTERFERENCE (TYPE = CONTACT PAIR). SHRINK is the only option supported.
- *CLEARANCE. Parameters MASTER, SLAVE, and VALUE are supported.
- *SURFACE BEHAVIOR. Supported parameters:
  - NO SEPARATION
  - AUGMENTED LAGRANGE
- *FRICTION. Supported parameters:
  - ROUGH
  - LAGRANGE
- *CHANGE FRICTION: This keyword is supported for the first *STEP keyword specified in the input file only. Only the INTERACTION parameter is supported on this keyword.
- *GAP CONDUCTANCE: The application supports gap conductance as a table of gap pressure or gap clearance. All the tabular values are averaged and used as a single thermal conductance value in the contact pair.
- *TIE: Parameters NAME and POSITION TOLERANCE are supported.

See the Imported: Contact object reference section for additional information.

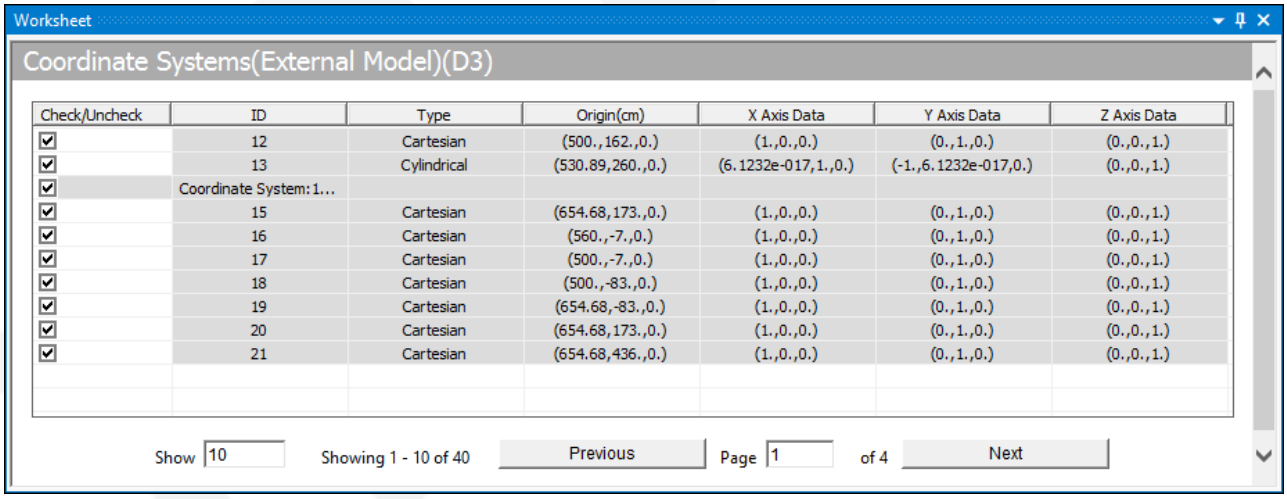
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Imported Coordinate Systems

When your upstream External Model source file includes coordinate systems, an **Imported** folder appears beneath the Coordinate System parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child object: **Imported Coordinate Systems (External Model)**.

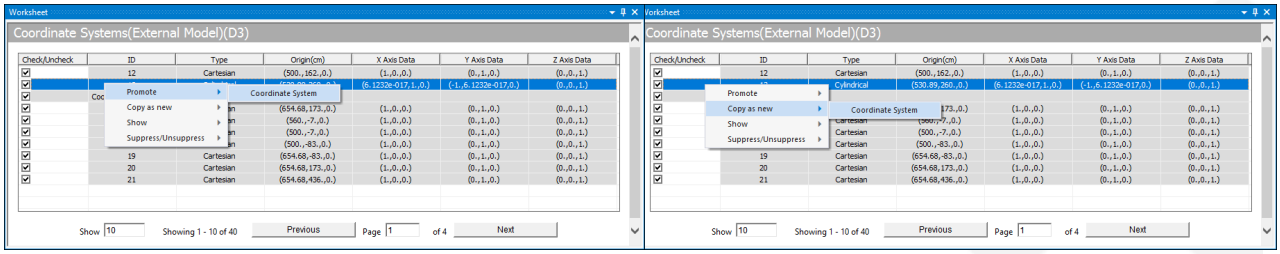


Selecting this singular object displays the Worksheet. The Worksheet displays all imported coordinate systems. Each coordinate system includes an ID from the source file, the Type of coordinate system (Cartesian or Cylindrical), and the location and orientation of the coordinate system.



As shown in the following illustrations, on the individual coordinate systems, you can **Promote** or **Copy as New** (p. 916).

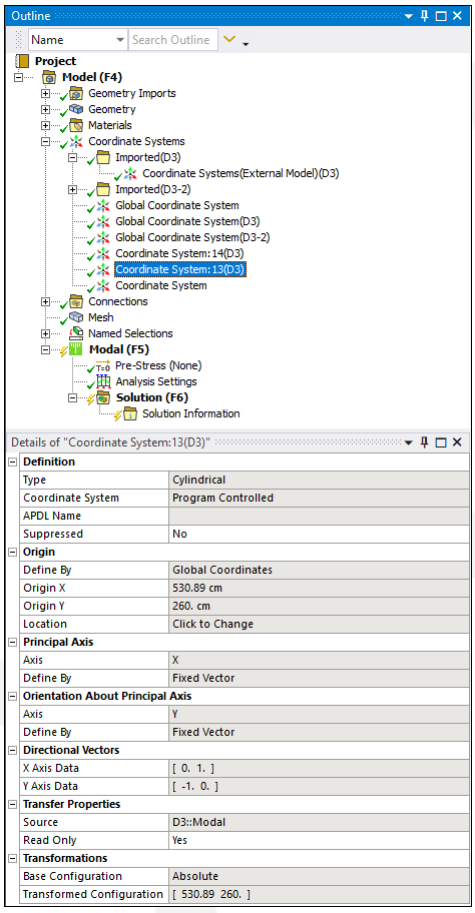
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



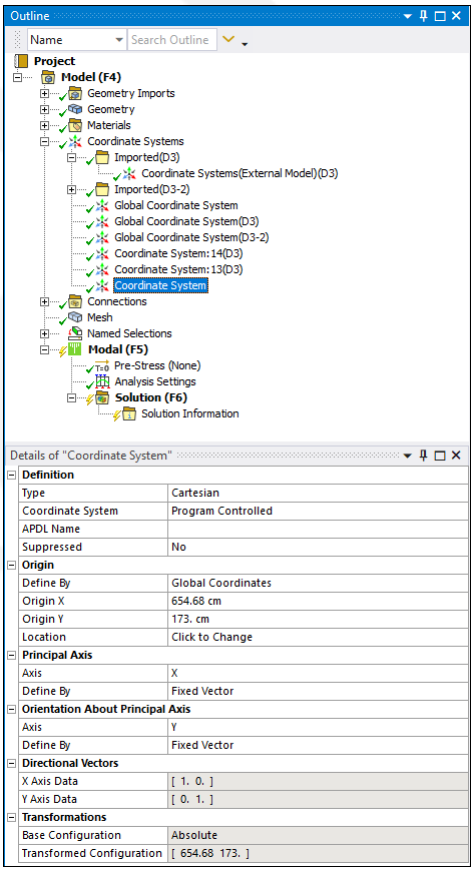
As illustrated below, the application creates new coordinate objects through promotion or copy. It renames promoted coordinates using the ID and the source file name and maintains a link (Transfer Properties) to the source system. Any source system updates or changes affect the promoted coordinate. You can make changes to promoted coordinates by setting the **Read Only** property of the Transfer Properties to **No** (default = **Yes**).

Copied coordinates are simply a new independent coordinate and named as such. These promote and copy features support multiple Worksheet selections.

Promoted Coordinate System



Copied Coordinate System



Supported Analysis Types

Imported Coordinate Systems are supported by the following analysis types:

- Coupled Field Analyses

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands as coordinate systems:

- **CDB**: **LOCAL** command.
- **NASTRAN**: Coordinate systems are defined by the following commands:
  - cord1c
  - cord1r
  - cord2c
  - cord2r
- **ABAQUS**: *ORIENTATION command.

  This command must be defined by coordinates or nodes. Any other definition is not supported. The types of coordinate system supported are Cartesian and cylindrical.

See the Imported: Coordinate System object reference section for additional information.

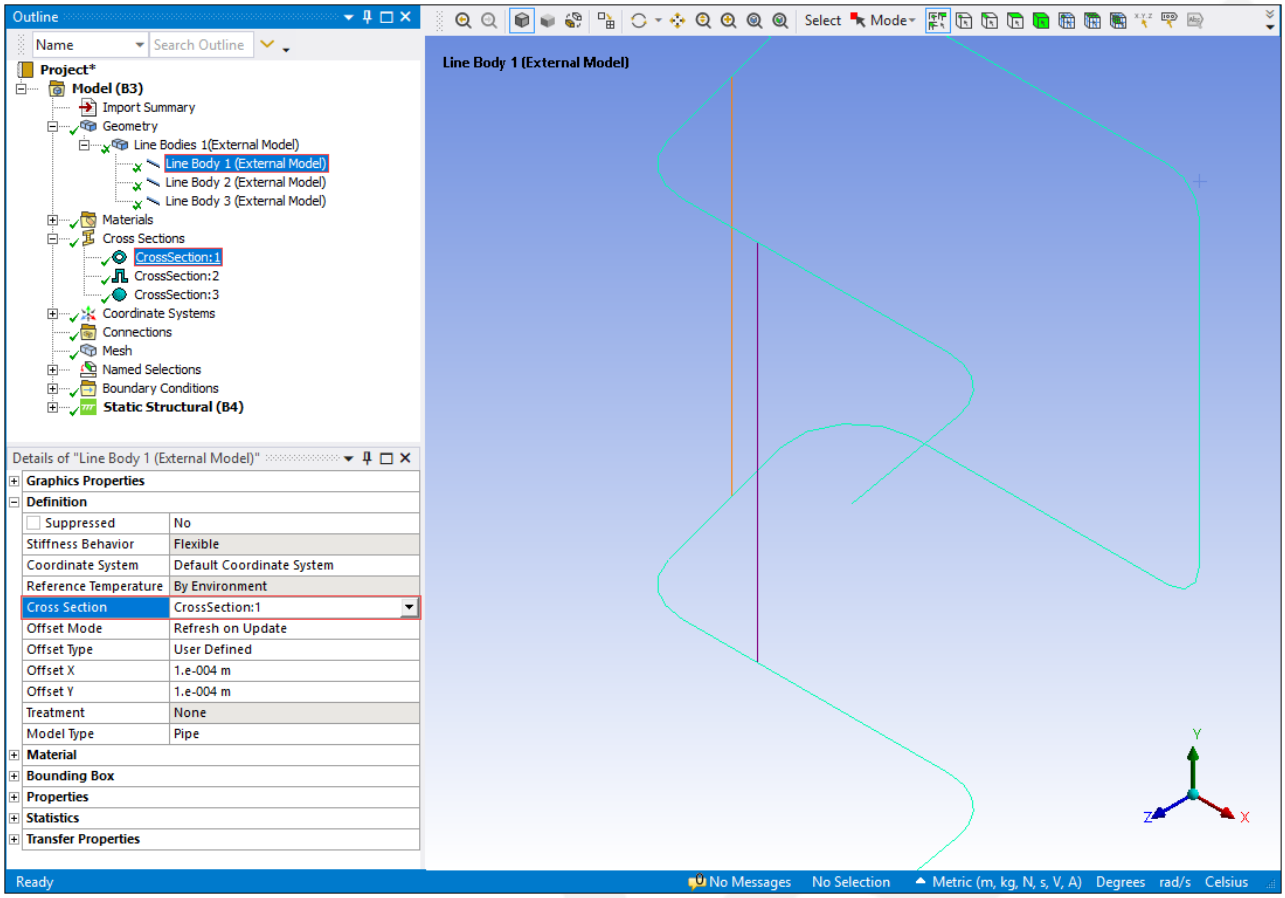
Imported Cross Sections

When your upstream External Model source file includes Line Body (p. 872) cross section data, cross section objects are automatically created/inserted under the Cross Sections folder. These imported cross sections act as any other user-defined cross section.

Highlighted below is an example of an imported Line Body. It automatically includes a property reference to the accompanying imported cross section (**CrossSection1**).

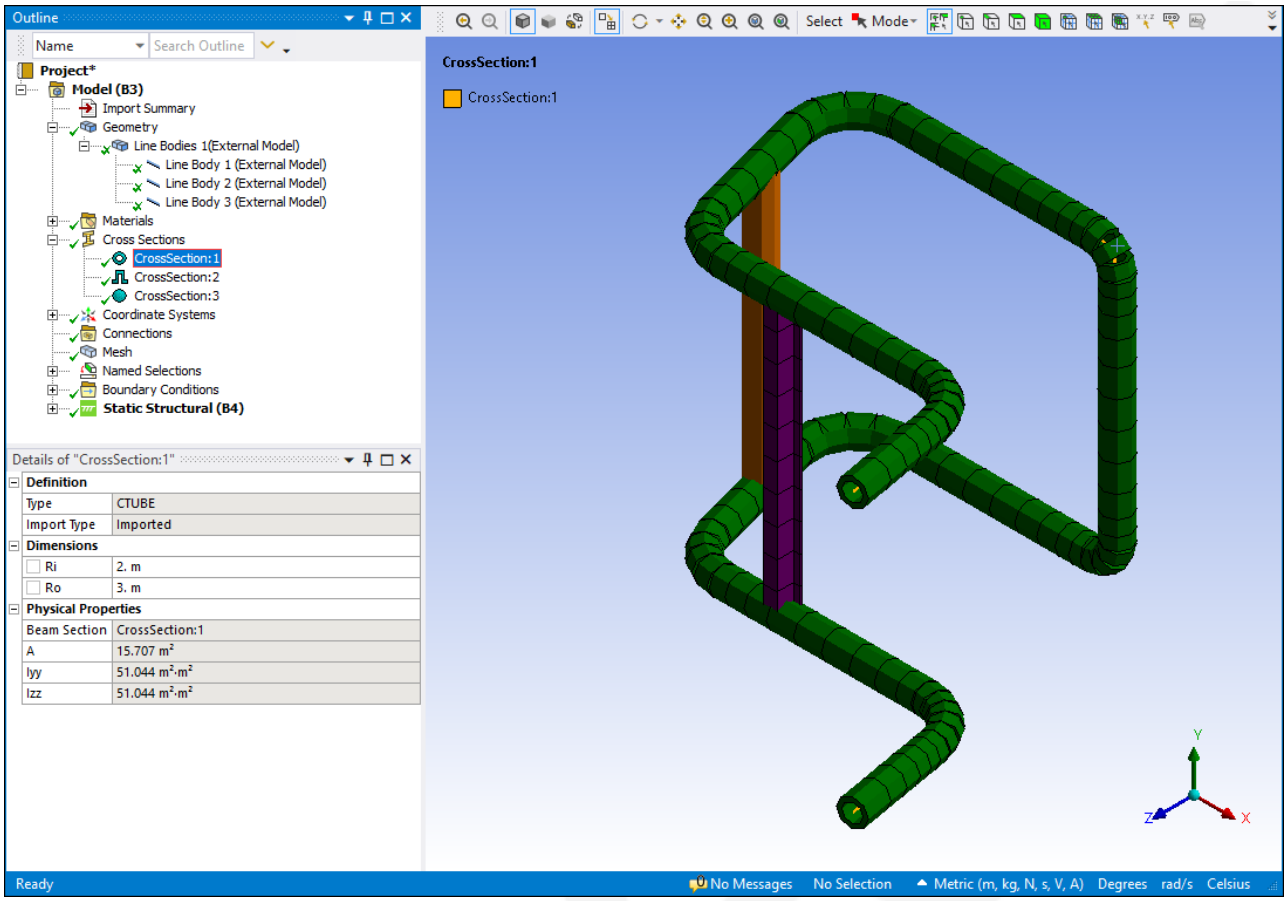
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Here is **CrossSection1** highlighted. The mesh display is turned on for this image.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Supported Source File Commands

The application imports the following source file data/commands as line body/beam cross sections:

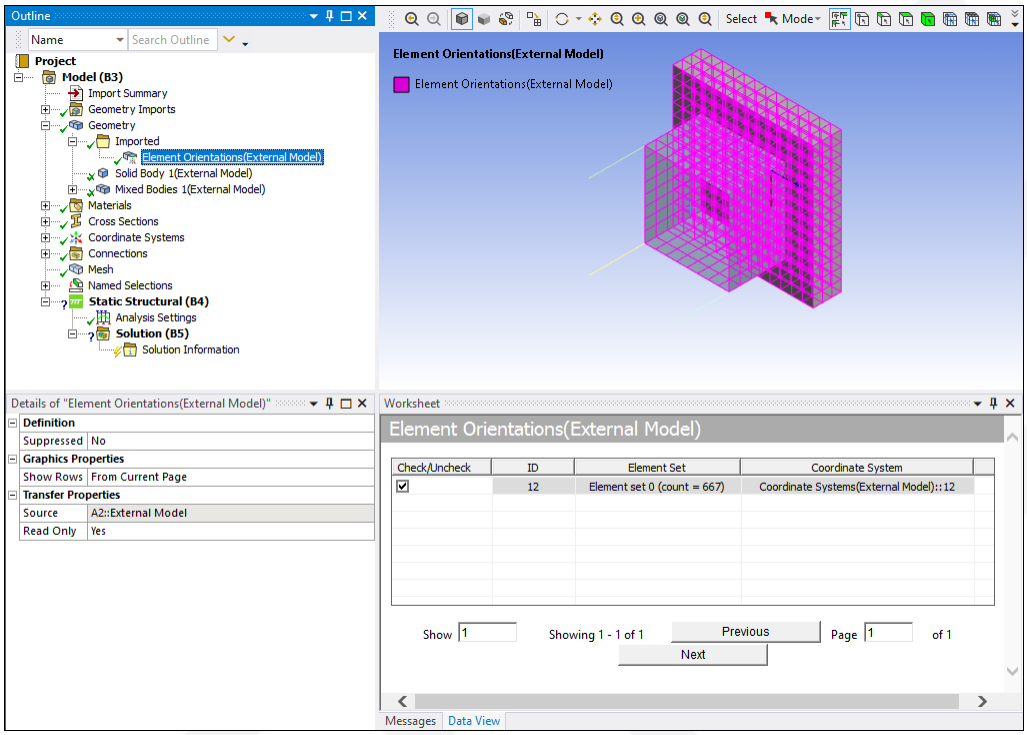
- **CDB**: The SECTION/REAL attribute defined with the element in the EBLOCK section.
  - BEAM, LINK and PIPE Types are supported.
  - The MESH Subtype is not supported.
- **NASTRAN**: Property ID designated with the PBAR, PBARL, PBEAM, PBEAML, PROD and CONROD bulk data entry.
  - CHAN2, H, CROSS, T1, HEXA, and HAT1 section types are not supported in Mechanical. The equivalent ASEC properties are calculated and used for these sections.
- **ABAQUS**: On the each "*BEAM SECTION","*BEAM GENERAL SECTION" and "FRAME SECTION" keyword:
  - ARBITRARY, HEX, TRAPEZOID, and ELBOW Sections are not supported and are not used.
  - Tapered beams are not supported in Mechanical. If a tapered beam is specified in the CDB or ABAQUS file the section data from the first node is used.

Also see the Cross Section Objects section for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

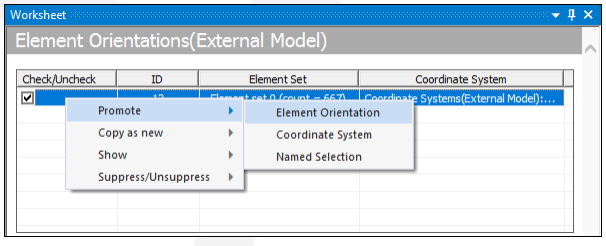
Imported Element Orientations

When your upstream External Model source file includes element orientation data, an **Imported** folder appears beneath the **Geometry** folder when you open the file in Mechanical. As with the imported coordinate systems, this object is a simple group folder (p. 208). It contains the child object: **Element Orientations** (p. 1008). Selecting this singular object displays the Worksheet. The Worksheet displays all of the imported element orientations, including a source-system ID, the Element Set, and the coordinate system of the element orientation.



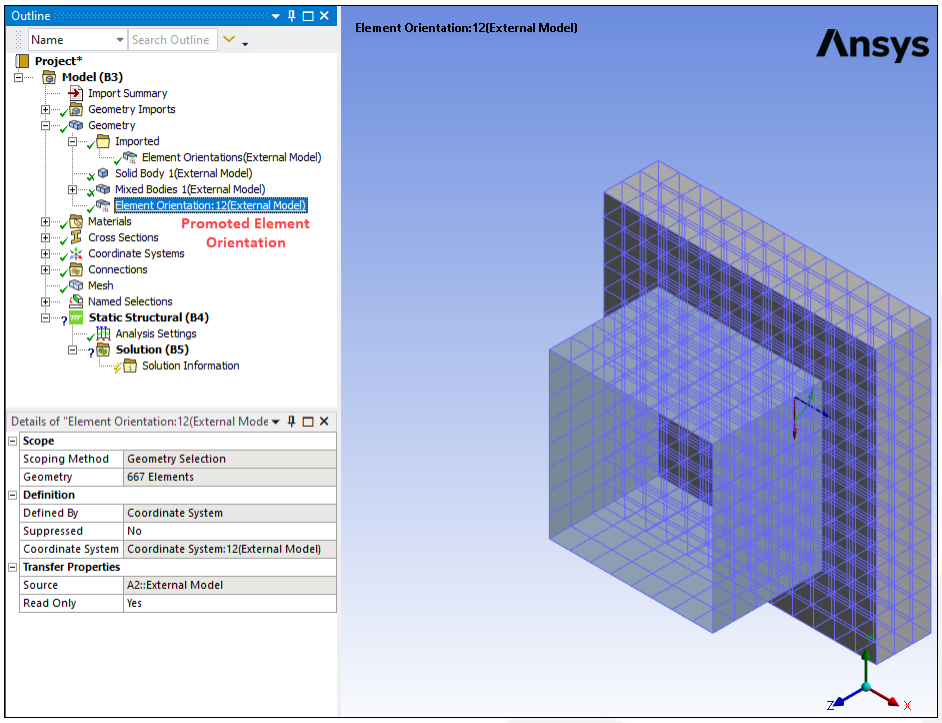
Promoting Element Orientations

As illustrated below, the application gives you the option on the Worksheet to promote or copy your Element Orientations. In addition, you can promote or copy (p. 916) the scoped Element Orientation set as a Named Selection and the coordinate system as a native coordinate system. Promoted objects maintain a link to the source file. Any source system updates or changes affect the promoted object.

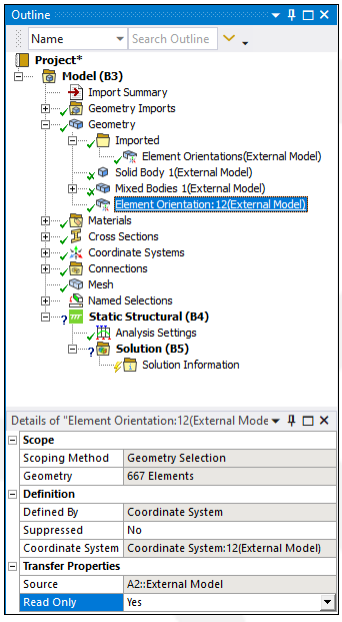


As illustrated here, the promoted Worksheet entry becomes read-only and no operations are supported for the row. The newly promoted object is highlighted below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The Details view of the promoted object is illustrated below. You can make changes to promoted objects by setting the **Read Only** property of the **Transfer Properties** category to **No** (default = **Yes**).

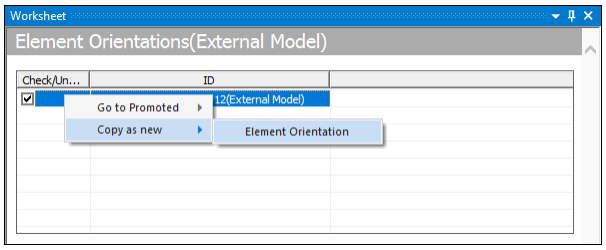


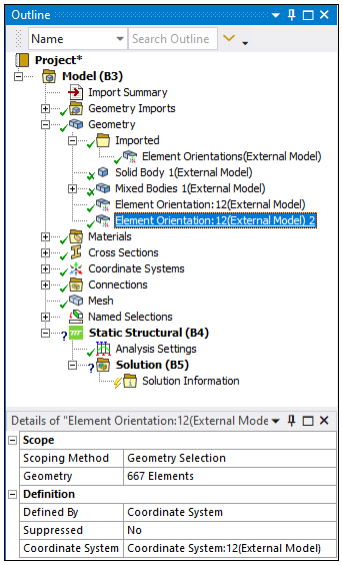
Copying Element Orientations

Copied Worksheet entries become a new independent object and are named based on your selection: Element Orientation, Coordinate System, or Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Supported Analysis Types

Imported Element Orientations are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands as element orientations:

- **CDB**: All the elements defined by the **ESYS** command are processed.
- **NASTRAN**: For the Material Coordinate System field of the supported NASTRAN elements, Mechanical uses the default coordinate system if the field is blank or the imported coordinate system if the field contains an integer value. If the field contains a real value (angle), the application generates a coordinate system for each element based on the centroid of the elements and rotated by the given angle.
- **ABAQUS**: It is processed via the *ORIENTATION command.

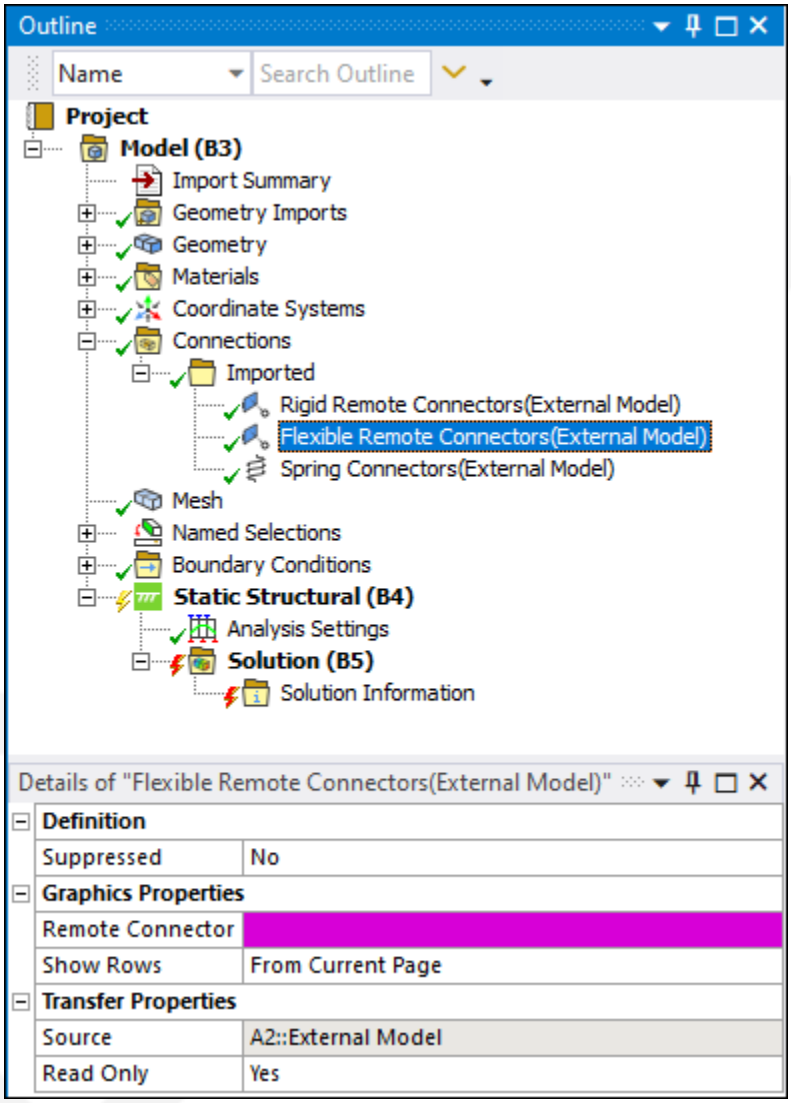
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

See the Imported: Element Orientation object reference page as well as the associated object reference pages for additional details about the properties and use of Element Orientation, Coordinate System, or Named Selections objects.

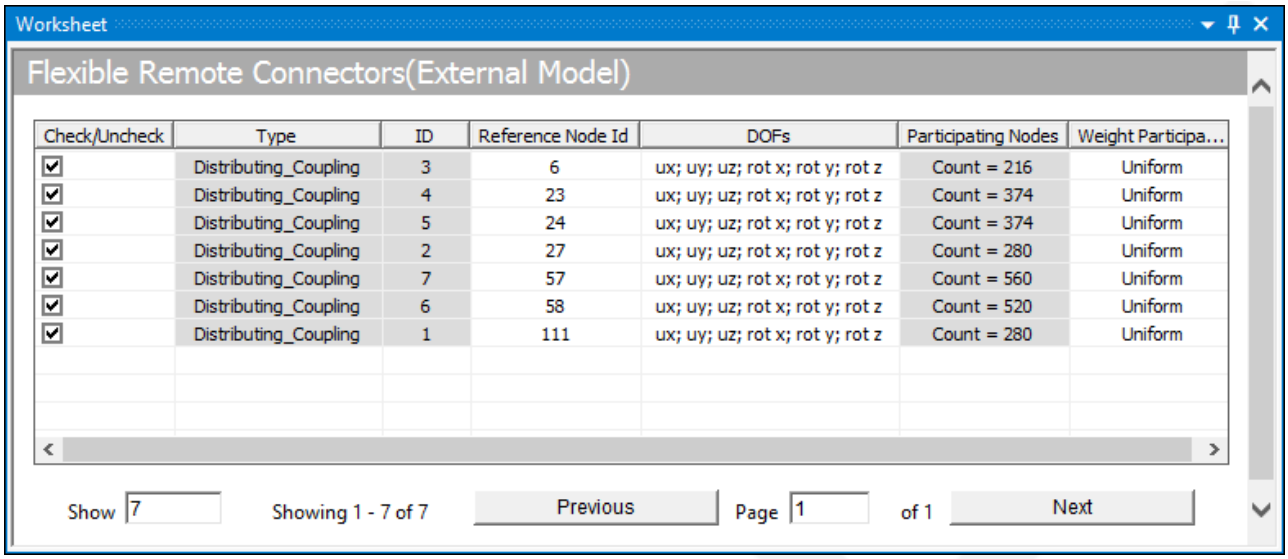
Imported Flexible Remote Connectors

When your upstream External Model source file includes flexible remote connections, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child objects: **Flexible Connectors**. The data associated with the imported connections is provided through the Worksheet. Worksheet content, including default settings, is based on the data in the external file. If a connector is scoped to a node that is **not** attached to the geometry, you can promote and/or copy (p. 916) the connector, otherwise, no promote and/or copy options are available.

Worksheet content is generally read-only, however, you can Check/Uncheck (p. 916) Worksheet entries using the selection box in the table, through the right-click context menu, or you can select the cell at the top of the column to check/uncheck all table rows.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Supported Analysis Types

Imported Flexible Remote Connectors are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal

Supported Source File Commands

The application imports the following source file data/commands as flexible remote connections:

- **CDB**: Contact/Target pairs with TARGE169/TARGE170 elements with pilot node (**TSHAP**, *PILO*) and Contact elements with the following KEYOPT settings:

KEYOPT (2) = 2	MPC-based approach.
KEYOPT (12) = 5	Bonded (always).
KEYOPT (4) = 1	Force-distributed constraint for CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177.

- **NASTRAN**: RBE3. For this command:
  - The reference node the reference node's degrees of freedom (DOFs) are considered.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– For a single dependent DOFs/Weight Set, the connection is considered Uniform.

– For multiple dependent DOFs/Weight Sets, the connection is considered user-defined.

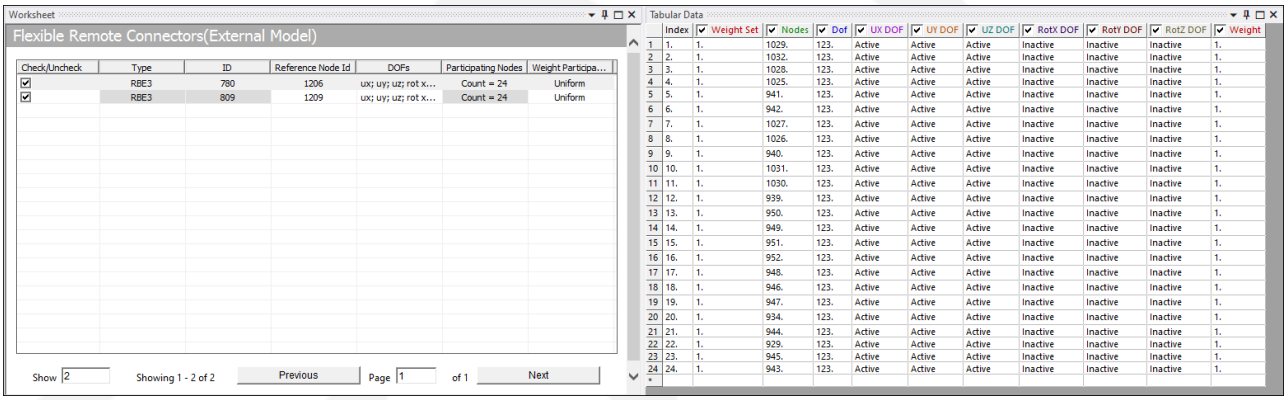
– If the reference DOFs are 123456 (UX, UY, UZ, ROTX, ROTY, ROTZ) and all the dependent DOFs are 123, a combination DOFSEL/RBE3 command will be sent to the solver for these DOFs. The dependent weights are also considered.

– For multiple slave DOF/Weight Sets with all the dependent DOFs that are not 123 and the reference DOFs are 123456, the connection is considered user-defined, but the dependent DOFs will be ignored. The dependent weights are also considered.

– For all other conditions the connection will be sent to the solver as a deformable RBE3 style bonded contact.

– If a non-structural analysis is being solved, the connection will be seen in the solver as a deformable RBE3 style bonded contact.

– When you select a connection (row) in the **Worksheet**, the **Tabular Data** window displays and includes the degree of freedom for each participating node.



– The dependent data table is not shown if there are no dependent nodes for a row or if any dependent nodes are not selected or active (a body is suppressed).

Note:

If you are resuming an earlier version of Mechanical and there are no dependent DOFs present, you must specify the DOFs for each set.

- **ABAQUS**: *DISTRIBUTING COUPLING and *COUPLING, *DISTRIBUTING.

  Processing is limited to required parameters only, not the optional parameters, such as orientation or mass.

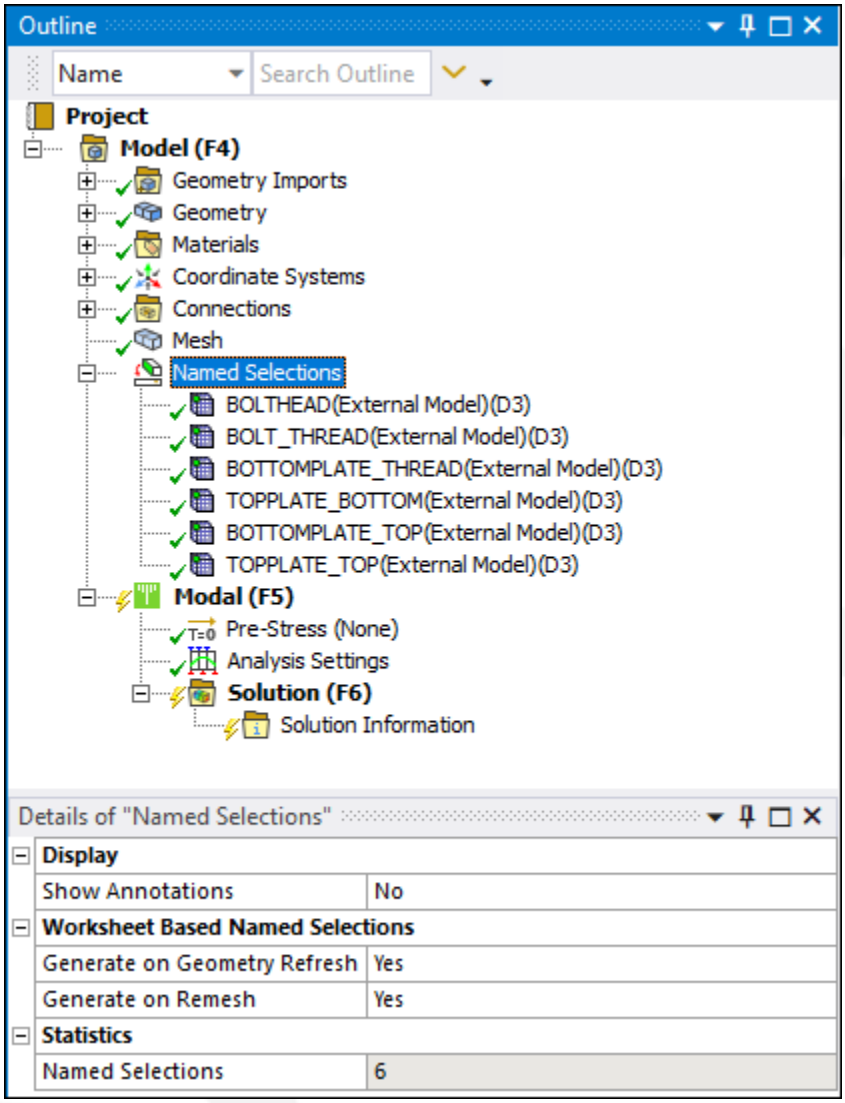
See the Imported: Flexible Remote Connectors object reference section for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Imported Named Selections

When your upstream External Model source file includes Named Selections, they are inserted under the Named Selections folder. These imported Named Selections act as any other user-defined Named Selection.

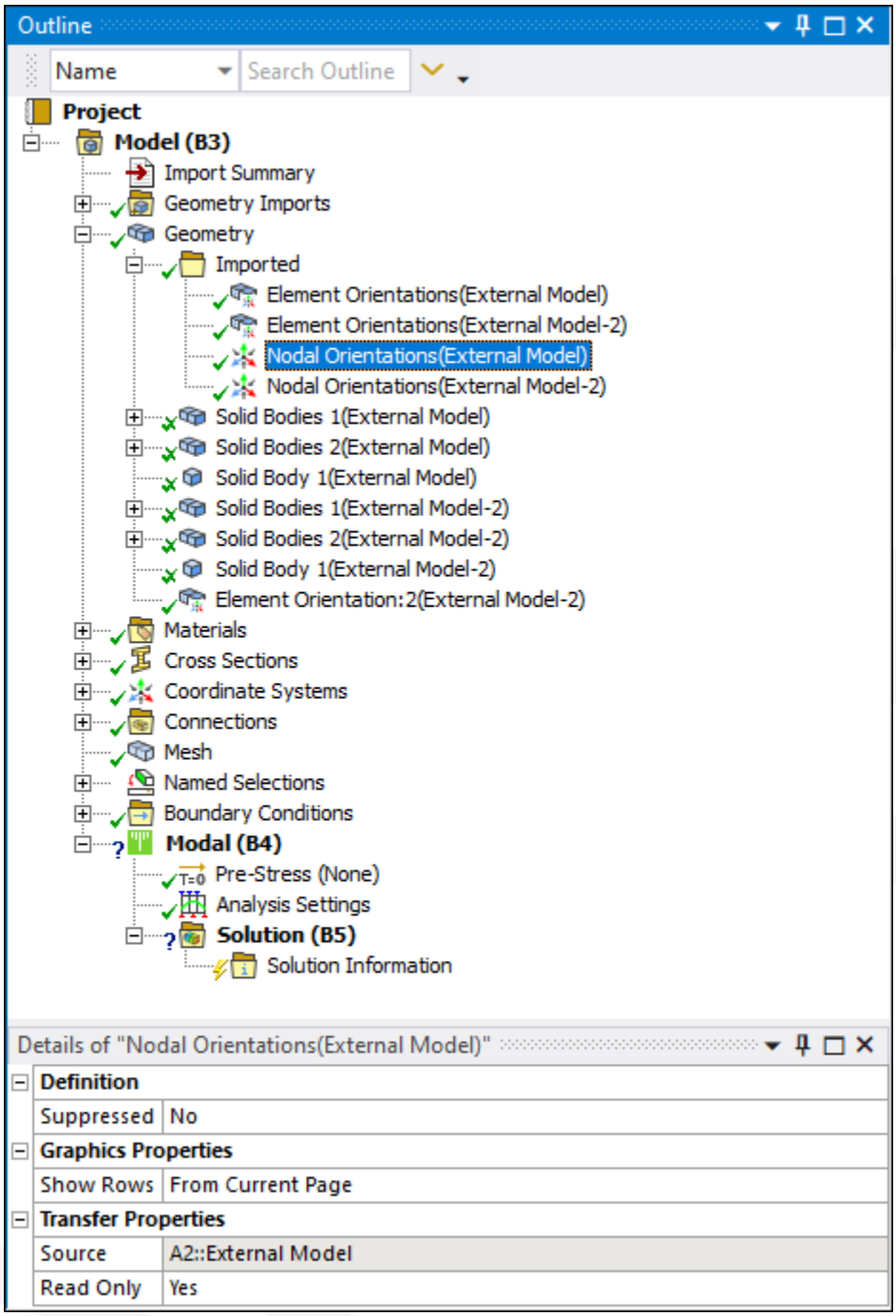


Review the Named Selections object reference page as needed for additional details about the properties and use of Named Selection objects.

Imported Nodal Orientations

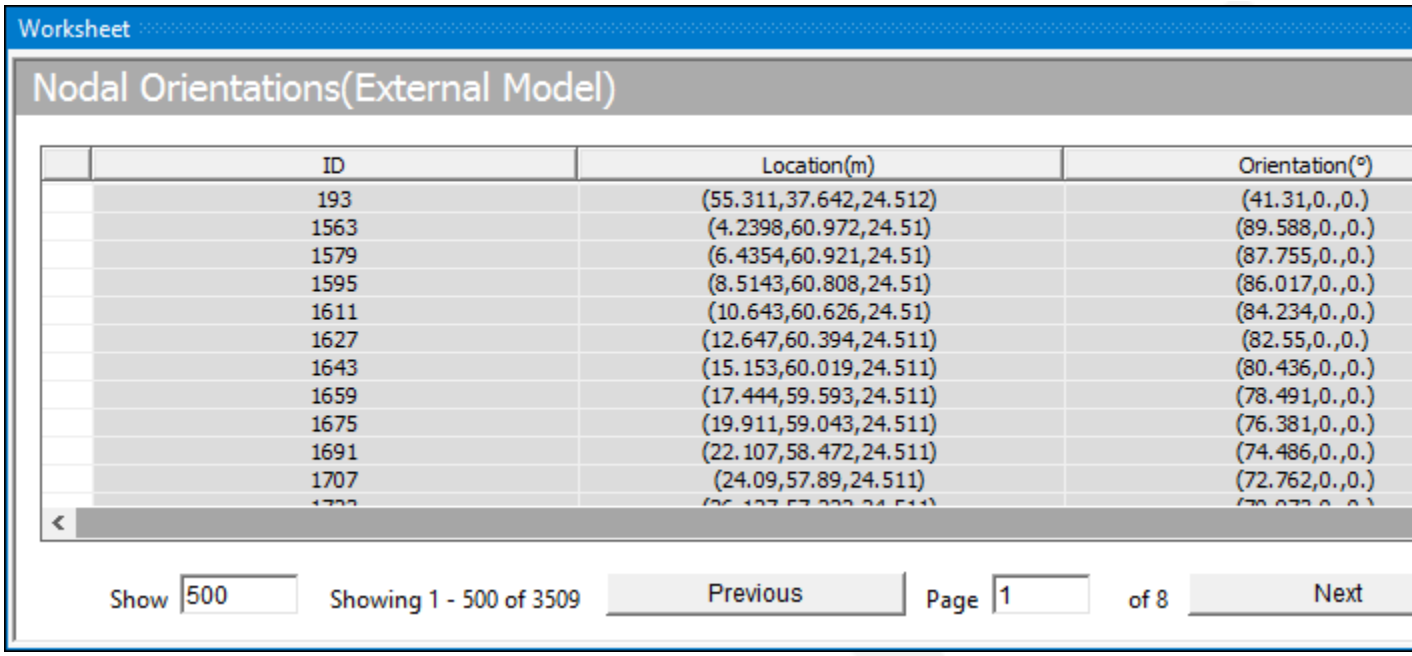
When your upstream External Model source file includes Nodal Orientations. An **Imported** folder appears beneath the **Geometry** folder when you open the file in Mechanical. This object is a simple group folder. It contains the child object: **Nodal Orientations**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23


Worksheet

Nodal Orientations(External Model)

	ID	Location(m)	Orientation(°)
	193	(55.311,37.642,24.512)	(41.31,0.,0.)
	1563	(4.2398,60.972,24.51)	(89.588,0.,0.)
	1579	(6.4354,60.921,24.51)	(87.755,0.,0.)
	1595	(8.5143,60.808,24.51)	(86.017,0.,0.)
	1611	(10.643,60.626,24.51)	(84.234,0.,0.)
	1627	(12.647,60.394,24.511)	(82.55,0.,0.)
	1643	(15.153,60.019,24.511)	(80.436,0.,0.)
	1659	(17.444,59.593,24.511)	(78.491,0.,0.)
	1675	(19.911,59.043,24.511)	(76.381,0.,0.)
	1691	(22.107,58.472,24.511)	(74.486,0.,0.)
	1707	(24.09,57.89,24.511)	(72.762,0.,0.)

Show 500 Showing 1 - 500 of 3509 Previous Page 1 of 8 Next

Supported Analysis Types

Imported Nodal Orientations are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands as nodal orientations:

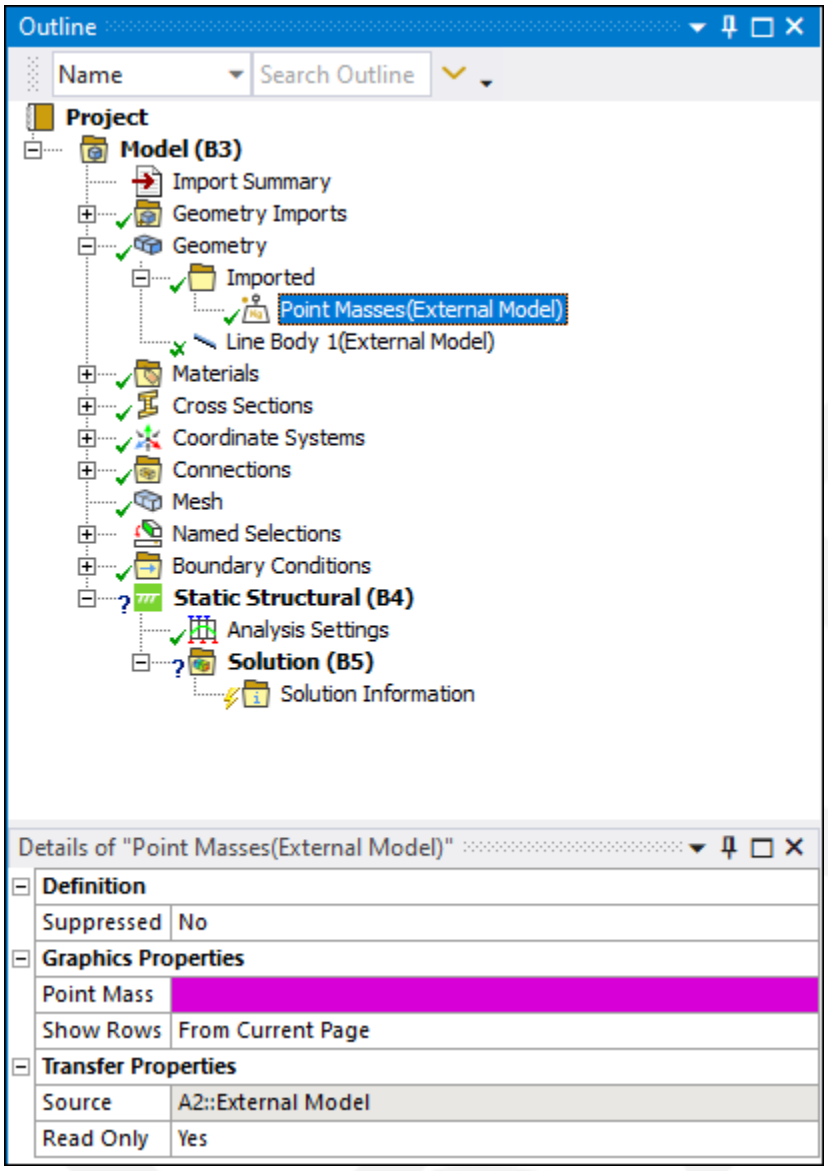
- **CDB**: All orientations from the `.cdb` file that are included in NBLOCK are processed.
- **NASTRAN**: Orientations are processed via the commands of the Imported Coordinate Systems (p. 934).
- **ABAQUS**: Orientations are processed via the *TRANSFORM command.

Also see the Imported: Nodal Orientation object reference section for additional information.

Imported Point Mass

When your upstream External Model source file includes Point Masses, an **Imported** folder appears beneath the **Geometry** folder when you open the file in Mechanical. This object is a simple group folder. It contains the child object: **Point Masses**. If the Point Masses are scoped to a node attached to the geometry, you can promote and/or copy (p. 916) them, otherwise, no promote and/or copy options are available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Worksheet

Point Masses(External Model)

Check/Uncheck	Type	ID	Reference No...	Location(m)	Mass(kg)	Mass Moment of Inertia X(kg·m²
☑	MASS	101	101	(0.,0.,0.6)	300.	0.
☑	MASS	102	102	(0.6,0.,0.)	300.	0.
☑	MASS	111	111	(0.,3.,0.6)	300.	0.
☑	MASS	112	112	(0.6,3.,0.)	300.	0.
☑	ROTARYI	202	102	(0.6,0.,0.)	0.	1.e-002
☑	ROTARYI	212	112	(0.6,3.,0.)	0.	1.e-002

Show 6 | Showing 1 - 6 of 6 | Previous | Page 1 of 1 | Next

Important:

If your Imported Point Mass is connected to a body through an imported connection, i.e. Remote Connection, Constraint Equation, or Spring, Mechanical does not include the Imported Point Mass in any Rotational Velocity or Rotational Acceleration scoping.

Supported Analysis Types

Imported Point Masses are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands as point masses:

- **CDB**: Element MASS21 is processed.
  - If 2D, MASS and IZZ are supported.
  - If 3D, MASSX, IXX, IYY and IZZ are supported.
  - Rotary Inertia only if KEYOPT(3) = 0 or 3.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **NASTRAN**: The CONM2 command is processed. We process the complete command comprising of Node IDs, Coordinate System IDs, Mass Value, offset distances, and mass moments of inertia. Any defined orientation is processed via the nodes (p. 945).

  Note:

  If there are any non-zero offsets, you must set them to zero in order to proceed with the solution.

- **ABAQUS**: Only the *ELEMENT, TYPE=MASS and *ELEMENT, TYPE=ROTARYI Point Mass elements are supported. The Mass Value is the only value obtained from the MASS command. All other *MASS command values are ignored. Any defined orientation is processed via the nodes (p. 945).

  Rotary Inertial and Orientation from *Rotary Inertia command are processed and all other data from *Rotary Inertia command are ignored.

Review the Point Mass object reference page as needed for additional details about the properties and use of Point Mass objects.

Imported Rigid Bodies

When the source file contains rigid body (p. 852) definitions, Mechanical imports them as bodies with the **Stiffness Behavior** property of the Body object automatically set to **Rigid** (read-only). Currently this feature is only supported for ABAQUS files.

Supported Analysis Types

Imported Rigid Bodies are supported by the following analysis types:

- Harmonic Response
- Modal
- Static Structural
- Transient Structural
- Response Spectrum
- Random Vibration

Supported Source File Commands

The application imports the following source file data/commands as a rigid body:

CDB

Imported Rigid Bodies are not currently supported by CDB files.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

ABAQUS

The ABAQUS command *RIGID BODY command is processed to import rigid bodies. The command supports the following arguments:

*RIGID BODY, ELSET=<elset_name>, REF NODE=<ref_node>

Where:

"<elset_name>" is the name of an element set (*ELSET). Supported elements are usual 2D and 3D continuum elements and rigid elements R3D3, R3D4.

"<ref_node>" is either a node number or a name of a nodal NSET that contains only one node number.

Note:

- Once imported, the reference node (REF NODE) is available as a normal node and Mechanical automatically generates a node-based Name Selection for each REF NODE. You can use these system generated Name Selections to scope loads and boundary conditions.
- The application supports contact between flexible and rigid bodies only when defined in the ABAQUS file. This means that manual contact definition (after import) is not supported as this requires to remesh your Model and will invalidate all your objects scoped on mesh entities.
- Mechanical does not support multi body parts that have a mix of flexible and rigid bodies. In this instance, the application automatically sets the **Stiffness Behavior** to **Flexible** and issues a warning.

NASTRAN

Imported Rigid Bodies are not currently supported by NASTRAN files.

LS-DYNA

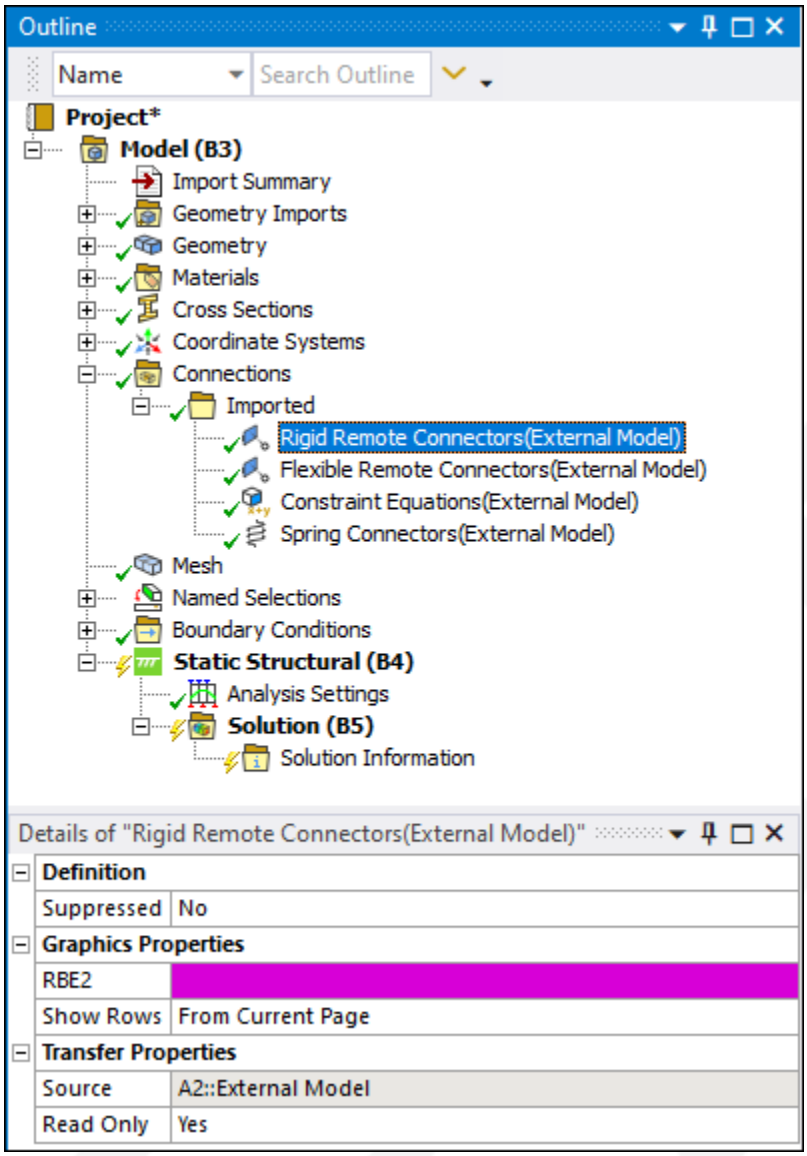
Imported Rigid Bodies are not currently supported by LS-DYNA files.

Imported Rigid Remote Connectors

When your upstream External Model source file includes rigid remote connections, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child objects: **Rigid Connectors**. The data associated with the imported connections is provided through the Worksheet. Worksheet content, including default settings, is based on the data in the external file. If a connector is scoped to a node that is **not** attached to the geometry, you can promote and/or copy (p. 916) the connector, otherwise, no promote and/or copy options are available.

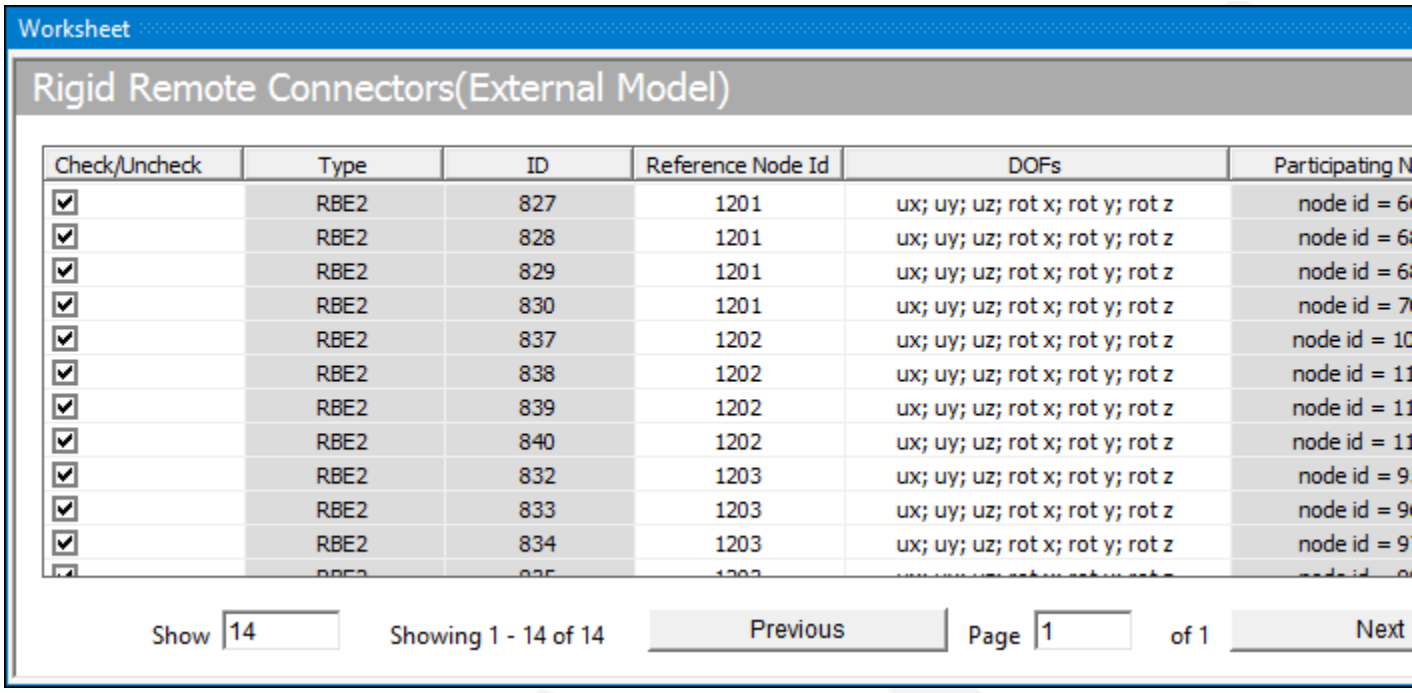
Worksheet content is generally read-only, however, you can Check/Uncheck (p. 916) Worksheet entries using the selection box in the table, through the right-click context menu, or you can select the cell at the top of the column to check/uncheck all table rows.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23


Worksheet

Rigid Remote Connectors(External Model)

Check/Uncheck	Type	ID	Reference Node Id	DOFs	Participating N
☑	RBE2	827	1201	ux; uy; uz; rot x; rot y; rot z	node id = 6
☑	RBE2	828	1201	ux; uy; uz; rot x; rot y; rot z	node id = 6
☑	RBE2	829	1201	ux; uy; uz; rot x; rot y; rot z	node id = 6
☑	RBE2	830	1201	ux; uy; uz; rot x; rot y; rot z	node id = 7
☑	RBE2	837	1202	ux; uy; uz; rot x; rot y; rot z	node id = 10
☑	RBE2	838	1202	ux; uy; uz; rot x; rot y; rot z	node id = 11
☑	RBE2	839	1202	ux; uy; uz; rot x; rot y; rot z	node id = 11
☑	RBE2	840	1202	ux; uy; uz; rot x; rot y; rot z	node id = 11
☑	RBE2	832	1203	ux; uy; uz; rot x; rot y; rot z	node id = 9
☑	RBE2	833	1203	ux; uy; uz; rot x; rot y; rot z	node id = 9
☑	RBE2	834	1203	ux; uy; uz; rot x; rot y; rot z	node id = 9

Show 14 Showing 1 - 14 of 14 Previous Page 1 of 1 Next

Supported Analysis Types

Imported Rigid Remote Connectors are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal

Supported Source File Commands

The application imports the following source file data/commands as rigid remote connections:

- **CDB**: Contact/Target pairs with:
  - TARGE169/TARGE170 elements with pilot node (**TSHAP**, *PILO*) and Contact elements with the following KEYOPT settings:

KEYOPT (2) = 2	MPC-based approach.
KEYOPT (12) = 5	Bonded (always).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

KEYOPT(4) = 2	Rigid surface constraint for CONTA171, CONTA172, CONTA173, and CONTA174.
KEYOPT(4) = 0	Rigid surface constraint for CONTA175, CONTA176, and CONTA177

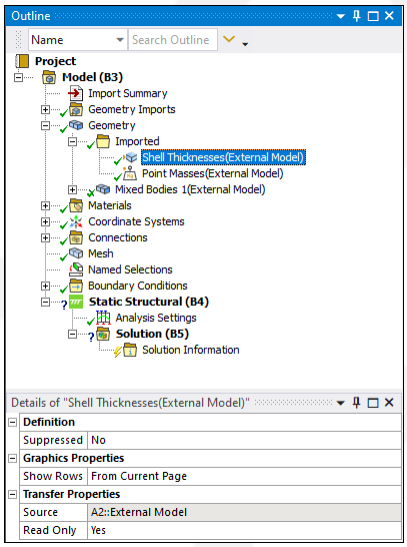
  - TARGE170 line segment (**TSHAP**, *LINE*) and no Contact elements.

- **NASTRAN**: RBE2. For this command, Mechanical processes the master node and the dependent nodes and their degrees of freedom (DOFs). All other data is ignored.
- **ABAQUS**:
  - *COUPLING and *KINEMATIC commands are processed but the orientation field is ignored.
  - *KINEMATIC COUPLING command is processed but the orientation field is ignored.
  - *MPC, TYPE = BEAM.
- **LS-DYNA**: *CONSTRAINED_NODAL_RIGID_BODY is processed, but the INERTIA option of this keyword is not supported.

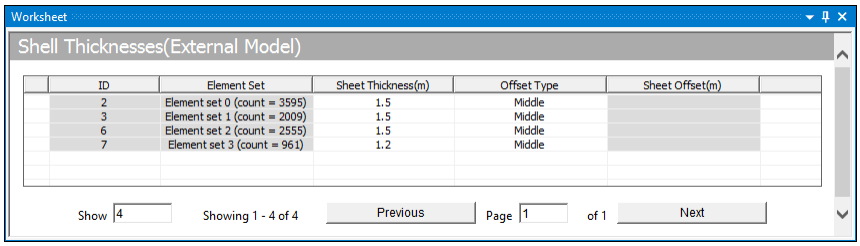
See the Imported: Rigid Remote Connectors object reference section for additional information.

Imported Shell Thicknesses

When your downstream External Model source file includes shell thicknesses, an **Imported** folder appears beneath the **Geometry** parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child objects: **Shell Thickness**.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



For Shell Thicknesses, you can modify the following Worksheet table options:

- **Sheet Thickness**: This field enables you to change the value of the Shell Thickness.
- **Offset Type**: This field provides a drop-down list of options, that include **Top**, **Middle**, **Bottom**, and **User Defined**.
- **Sheet Offset**: You use this property when the **Offset Type** is **User Defined**.

Supported Analysis Types

Imported Shell Thicknesses are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural
- Steady-State Thermal (Coupling only)
- Transient Thermal (Coupling only)

Supported Source File Commands

The application imports the following source file data/commands as shell thicknesses:

- **CDB**: The SECTION/REAL attribute defined with the element in the EBLOCK section.
- **NASTRAN**: Property ID designated with the PSHELL bulk data entry.
- **ABAQUS**: On the each "*MEMBRANE SECTION", "*SHELL GENERAL THICKNESS" or "*SHELL SECTION" keyword. If the NODAL THICKNESS parameter is used with one of these keywords, the application automatically sets element thickness to the thickness of the lowest node number. You cannot change the thickness values.
- **LSDYNA**: *SECTION_SHELL keyword

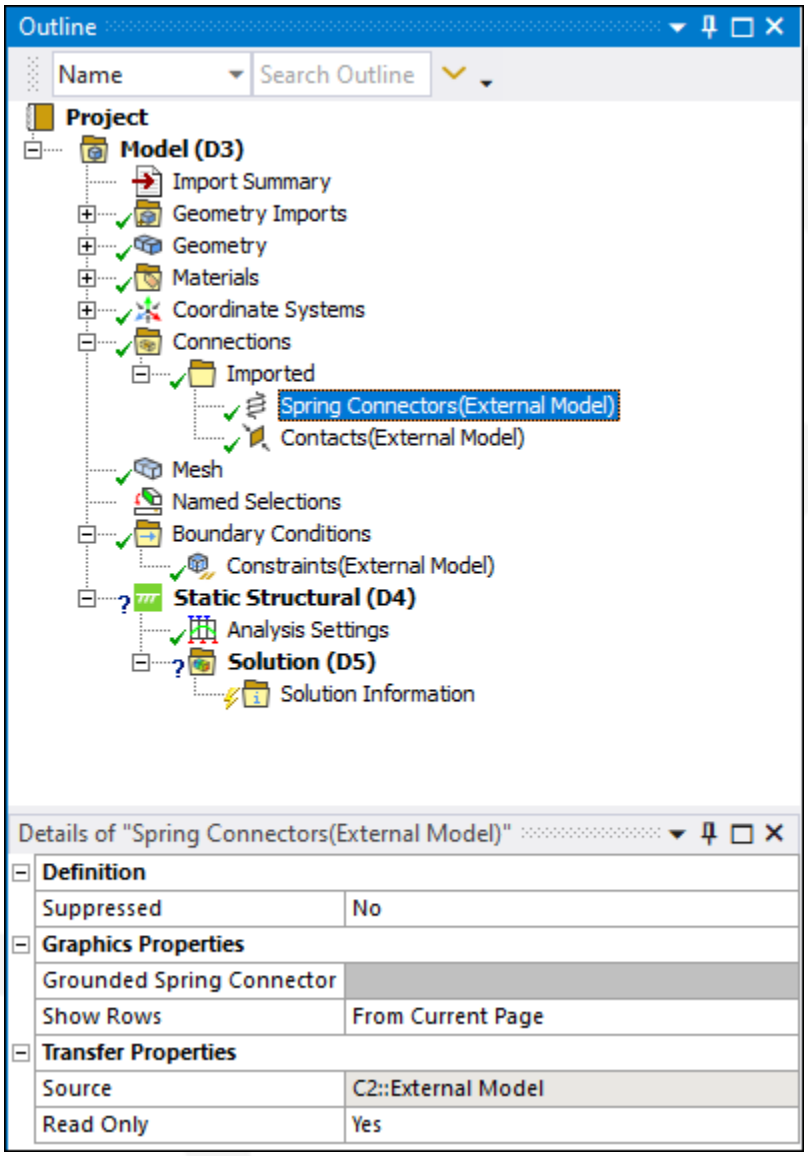
See the Imported: Shell Thicknesses object reference section for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

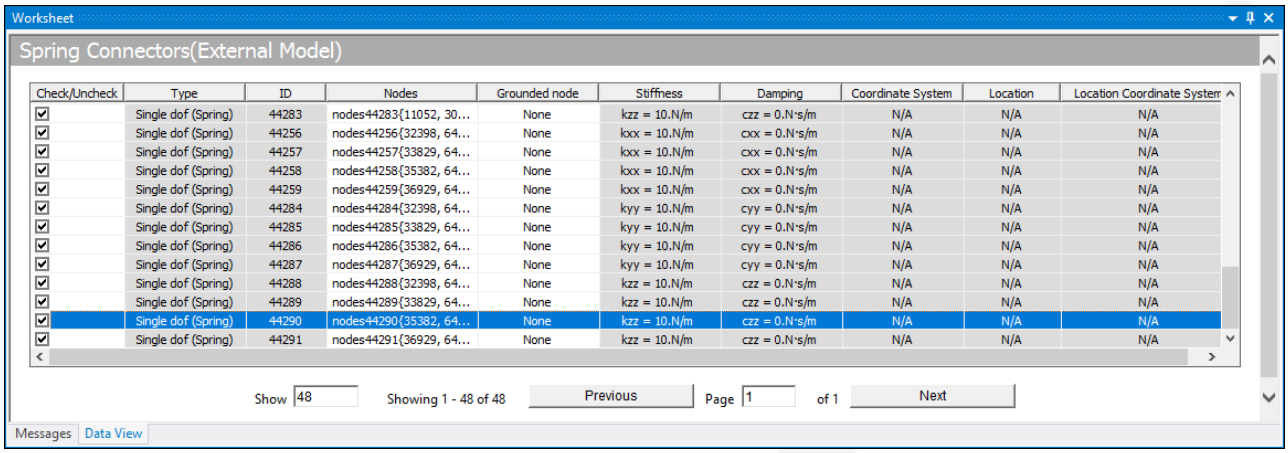
Imported Spring Connectors

When your upstream External Model source file includes spring connections and bushings, an **Imported** folder appears beneath the **Connections** parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child objects: **Spring Connectors**. The data associated with the imported springs and bushing is provided through the Worksheet. Worksheet content, including default settings, is based on the data in the external file.

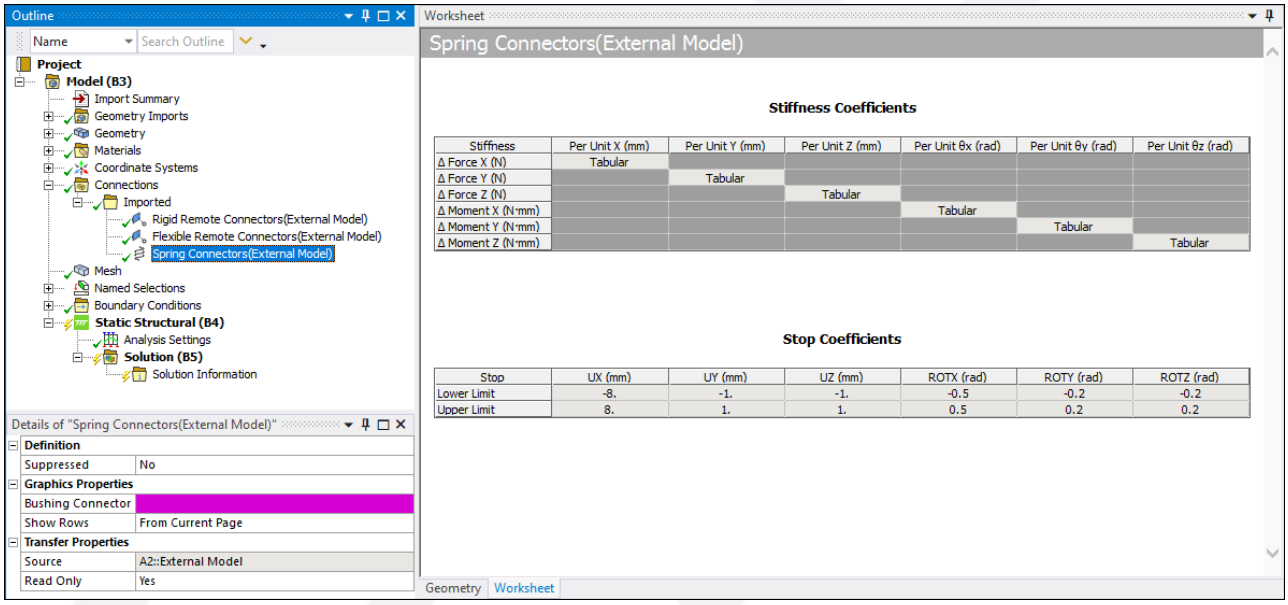
Spring Example



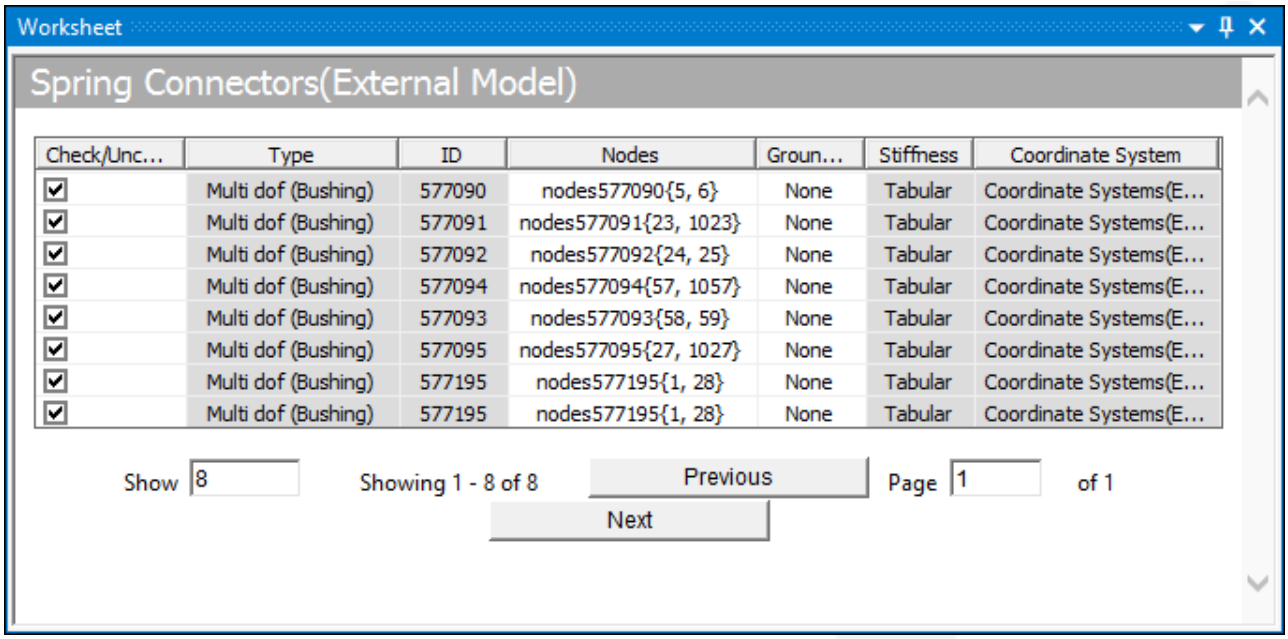
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



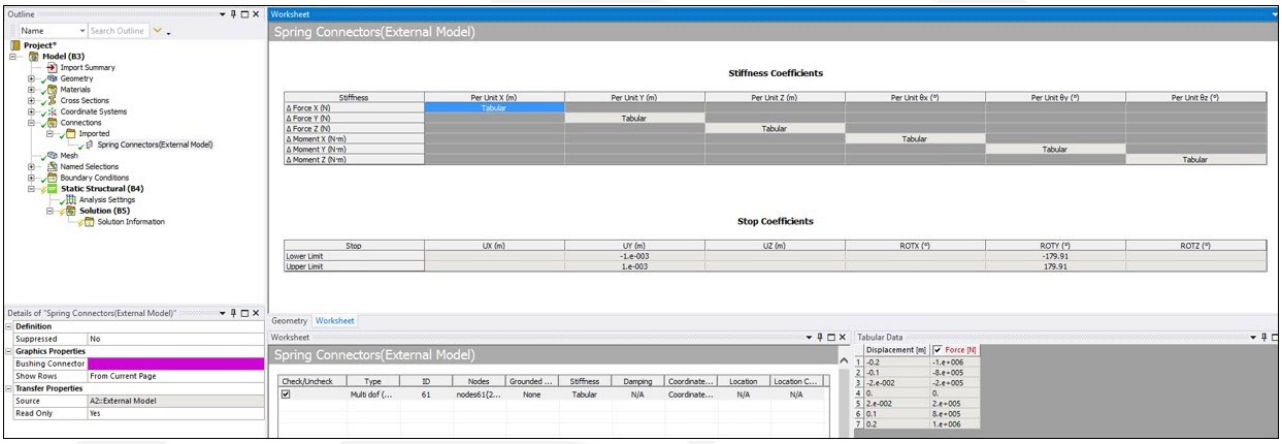
Bushing Example



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Nonlinear Bushing Example



Supported Analysis Types

Imported Spring Connectors are supported by the following analysis types:

- Coupled Field Analyses
- Harmonic Response
- Modal
- Static Structural
- Transient Structural

Supported Source File Commands

The application imports the following source file data/commands as springs:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

CDB

For the CDB file type imported into Mechanical through **External Model**, the application only processes:

- COMBIN14 linear spring elements with the following restrictions:
  - For the supported constants of this element type, Mechanical only processes Real Constants Stiffness (K) and structural Damping Coefficient (CV1). If your input data includes any real constant in addition to K or CV1 (such as CV2), these additional values are not processed.
  - Mechanical does not process material commands.
  - Supported KEYOPT,2 numbers include structural degrees of freedom only.
- COMBIN39 nonlinear spring elements with the following restrictions:
  - Supported KEYOPT,3 numbers include translational degrees of freedom only.
  - Mechanical does not process material commands.

NASTRAN

Mechanical supports the processing the CELAS1, CELAS2, CBUSH, and PBUSH commands only from a NASTRAN file with the following requirements:

- Mechanical only processes stiffness(K) and structural damping (GE). The stress coefficient is not processed.
- If GB (grid point B) is blank, the application assumes that the next node is grounded and creates a new node with the same location as the previous node.
- For CELAS1 and CELAS2 commands the first DOF will be taken as the operating DOF for the spring.
- For the PBUSH command, Mechanical supports the stiffness coefficients K1, K2, K3, K4, K5, K6 and structural damping coefficients (GE1, GE2, ... GE6).

ABAQUS

Spring Connections

For spring data imported from ABAQUS, Mechanical processes the element types:

- SPRING1 (spring element between a node and ground acting in a fixed direction).
- SPRING2 (spring element between two nodes acting in a fixed direction). For this element type, the application will import nonlinear force-displacement spring data that contains the origin, if the NONLINEAR parameter is used with the *SPRING keyword.
- SPRINGA (spring element between two nodes with its line of action being the line joining the two nodes).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In addition, the application supports the following *SPRING options, however, all other fields are ignored:

- Orientation via Imported Nodal Orientation (p. 945).
- Degree of freedom, stiffness and structural damping coefficients.
- Linear and nonlinear behavior.

Important:

For SPRING1 elements, an additional node is created in the same spot as the given node and that node is grounded when sent to the solver.

Bushing Connections

Mechanical supports the following ABAQUS bushing keywords:

- *ELEMENT: This keyword requires the TYPE parameter CONN3D2 as well as the Data Lines that include the Element IDs and two node IDs (two nodes must be present). You can also use the optional parameter ELSET. Note that the ELSET Name must match name the *CONNECTOR SECTION keyword. Grounded bushings are not supported.
- *ELSET: This keyword requires the ELSET parameter whose name must be included on either the *ELEMENT or *ELSET keywords as well as the name of the *CONNECTOR SECTION keyword.
- *ORIENTATION: This keyword requires the NAME Parameter and this parameter must match the name given on the second line of the *CONNECTOR SECTION keyword.
- *CONNECTOR SECTION: This keyword requires the ELSET and BEHAVIOR parameters whose names must be included on either the *ELEMENT or *ELSET keywords or the *CONNECTOR BEHAVIOR keyword. In addition, the first Data Line must define BUSHING as the type of connector and the second line must define the name of the coordinate system and must match a name on the *ORIENTATION keyword. Only the first Orientation name is used.
- *CONNECTOR BEHAVIOR: This keyword requires the NAME Parameter. This Parameter must also match name used with the BEHAVIOR parameter on the *CONNECTOR SECTION keyword. In addition, the *CONNECTOR BEHAVIOR keyword must be immediately followed by the keywords *CONNECTOR ELASTICITY and *CONNECTOR STOP (as described below).
- *CONNECTOR ELASTICITY: This keyword may be repeated up to six times for the values of a DOF. It requires the COMPONENT Parameter and the COMPONENT Parameter must represent a DOF direction. Directions values include: 1 = UX, 2 = UY, 3 = UZ, 4 = ROTX, 5 = ROTY, 6 = ROTZ

  The NONLINEAR Parameter is optional unless your input includes a table of Stiffness vs Constraint, in which case it is required.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The solution does not allow both nonlinear and linear data. Linear data is ignored during the solution. To ensure proper nonlinear effects, set the **Large Deflection** property to **Yes**.

Corresponding Data Lines include, if Nonlinear, Stiffness and Constraint. And, the line is repeated for each Stiffness, Constraint pair in a table. If the Data Line is Linear, it includes Stiffness.

- *CONNECTOR STOP: This keyword requires the COMPONENT Parameter and a data line that includes an Upper limit and a Lower limit value.

Also see the Imported: Spring Connectors object reference section.

Imported Initial Stresses

When your downstream External Model source file includes initial stresses and initial equivalent plastic strains, an **Imported** folder appears beneath the **Geometry** parent folder when you open the file in Mechanical. This object is a simple group folder (p. 208). It contains the child objects: **Initial Stresses**. The External Model can only reference LS-DYNA (. k) files.

For Initial Stresses, none of the fields can be modified.

Supported Analysis Types

Imported Initial Stresses are supported by the following analysis types:

- Coupled Field Analyses
- Static Structural
- Transient Structural
- LS-DYNA

Supported Source File Commands

The application imports the following **LSDYNA** source file data/commands as initial stresses:

*INITIAL_STRESS_SOLID

*INITIAL_STRESS_SHELL

See Explicit-to-Implicit Sequential Solutions for additional information.

Automatic Material Assignment

If more than one body in your upstream **External Model** system is assigned the same material, Mechanical automatically creates a Material Assignment object for the associated bodies. This action makes sure that each body has the same unique material identifier.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Reference Node Naming

For the imported FE data types listed below, Mechanical enables you to assign custom names to reference nodes in the Worksheet. This enables you to access the nodes in the solver file (through command snippets (p. 2303)). Your name can contain only standard English letters and numbers, spaces, underscores, hyphens, and/or parenthesis characters.

- Imported Bolt Pretensions and Premeshed Bolt Pretensions (p. 918)
- Imported Flexible Remote Connectors (p. 942)
- Imported Point Masses (p. 947)
- Imported Rigid Remote Connectors (p. 951)
- Imported Spring Connectors (p. 956)

Mechanical automatically makes the following changes to your custom name in order to prepare the new reference name for the Mechanical APDL application:

- A name exceeding 32 characters is truncated.
- A name that begins with a number, space, hyphen, or parentheses character, is renamed to include “C_” before the character.
- Spaces, hyphens, or parentheses between characters in a name are replaced with underscores.

For example, a Reference Node in the Mechanical application called "1 Edge" appears as component "C_1_Edge" in the Mechanical APDL application input file.

Importing Mesh-Based Databases in Batch

The Mechanical installation includes an executable file (`Ans.StandaloneExternalModelConsole.exe`) that enables you to import solver input files. This input file describes a model through various data, including, but not limited to, mesh data, materials, loads and boundary conditions, as well as analysis settings. The executable is able to parse an input file and produce two types of output files:

1. A solver file that is based on a specific solver, such as the Mechanical APDL solver or the LS-DYNA solver. This means that the output solver file can be processed by the specified solver. Furthermore, you can optionally choose to save this output file as a Mechanical database (`*.mechdb`) file, which includes a Mechanical system as well as the target system.
2. A Mechanical database (`*.mechdb`) file. You can then use this database in the Mechanical application, where you can add a target analysis system. When opened in the application, `*.mechdb` files produce an analysis system in Mechanical that generates objects included with the Model object, not including an analysis system. Although it is the simplest workflow, this method requires you to manually complete setting up the analysis.

Based on your platform, the executable file is located in the following directory:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Windows

"<installation_directory>\aisol\bin\winx64\Ans.StandaloneExternalModelConsole.exe"

Linux

"<installation_directory>/aisol/.workbench -cmd <installation_directory>/aisol/bin/linx64/Ans.StandaloneExternalModelConsole.exe"

Go to a section topic:

- Requirements and Limitations (p. 963)
- Required Command Arguments (p. 963)
- Command Entry (p. 963)
- Supported Arguments (p. 965)
- TOML File (p. 966)

Requirements and Limitations

This feature supports:

- The input of the following solver files:
  - Mechanical APDL common database (`.cdb`)
  - ABAQUS Input (`.inp`)
  - LS-DYNA Input (`.k` and `.key`)
- Single input files only. This feature does not support model assembly.

Important:

For ABAQUS and LS-DYNA input files that contain the *INCLUDE keyword, the application also processes the dependent files.

Required Arguments

You must use one of the following arguments:

- **`--input_solver_file`** (Import a solver file using default settings).
- **`--input_settings`** (Import a solver file using settings specified in TOML file format).

Command Entry

Use the following to execute the command and specify the required and optional arguments.

1. Open the Command Prompt window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. At the command prompt, specify the desired directory location. For example:

   "C:\ANSYS Inc\v241\aisol\bin\winx64\Ans.StandaloneExternalModelConsole.exe"

3. Based on your desired output, specify one of the following required argument combinations:

   - **--input_solver_file**: Import a solver file and create a Mechanical database (up to **Model** object level).

     For example: **Ans.StandaloneExternalModelConsole.exe --input_solver_file <file_path to solver input file> --save_mech_db**

   - **--input_settings**: Create solver file based on target analysis as well as creating an optional Mechanical database with the analysis system.

     For example: **Ans.StandaloneExternalModelConsole.exe --input_settings <file_path to TOML file> --save_mech_db**

4. As desired, specify additional arguments as described below.

Tip:

You can drag and drop files into the Command Prompt window to automatically generate the file path entry.

Command Entry Example 1

"C:\Program Files\ANSYS Inc\v241\aisol\bin\winx64\Ans.StandaloneExternalModelConsole.exe" --input_solver_file \\Parts\Mesh\Mech_CDB\Esys.cdb --save_mech_db --output_dir_path c:

This command entry imports a cdb solver file and generates a Mechanical database.

Where:

This entry initiates the execution of the file.

"C:\Program Files\ANSYS Inc\v241\aisol\bin\winx64\Ans.StandaloneExternalModelConsole.exe"

This argument specifies that you want to import a cdb file during the batch process.

--input_solver_file \\Parts\Mesh\Mech_CDB\Esys.cdb

This argument specifies that you want to generate and save a Mechanical database to a file during the batch process.

--save_mech_db

This argument specifies where you want the executable to save the generated file.

--output_dir_path c:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Command Entry Example 2

```
"C:\Program Files\ANSYS Inc\v241\aisol\bin\winx64\Ans.StandaloneExternalModelConsole.exe" --input_settings Example2.toml --write_solver_file --output_dir_path c:\
```

This command entry uses the `Example2.toml` file to 1) import a solver file (as defined in the **`--input settings argument`**) and 2) output a solver file (as defined by the **`SolverType`** option).

The output solver file is written to **`c:\`** as prescribed by the **`--output_dir_path`**.

The `Example2.toml` file is a text file with the following content:

```
[MeshFileOptions]
FilePath = 'C:\Example2.k'
FileUnitSystemName = "MKS_STANDARD"

[ConversionOptions]
TopologyCreationMode = "MeshOnly"

[EnvironmentOptions]
AnalysisType = "Static"
PhysicsType = "Structural"
SolverType = "ANSYS"
```

These entries specify the following:

`FilePath`: The path to and location of the input file.

`FileUnitSystemName`: Units system.

`TopologyCreationMode`: Specify whether to import the mesh, geometry, or both.

`AnalysisType`: Analysis type.

`PhysicsType`: Physics type.

`SolverType`: Specifies the solver output file type.

Supported Arguments

Supported arguments include:

Argument	Description
`--help`	Produce a help message.
`--input_solver_file <file_path>`	Import a solver input file and output a Mechanical `*.mechdb` file. Mesh data, materials, loads and boundary conditions will be created in the Mechanical database for objects included at the **Model** object level. Analysis specific settings will not be processed because a target solver is not known.
`--input_settings <file_path>`	Point to a TOML file, which in turn points to a solver input file and additional settings. The TOML file needs to be complete as describes below. The **[MeshFileOptions]** need to point to an input solver file and the **[EnvironmentOptions]** need to point to the desired target solver system.
`--logging`	Create External Model and Mechanical log files in the temporary directory.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Argument	Description
--output_dir_path <directory_path>	Create output files in the provided directory. If this argument is missing, the directory behind the Temp environment variable is used.
--save_mech_db	Save the current session as a Mechanical database.
--write_solver_file	Write a solver file. This argument requires the **--input_settings** argument that includes defined analysis options.

TOML File

To use the **--input_settings <file_path>** argument, you must create a TOML file - Tom's Obvious, Minimal Language TOML (`.toml`). This file is referenced by the input argument. The settings in the TOML file are grouped in categories (as a TML table) and each category contains a list of options as pairs: name = value.

This file has mandatory and optional entries.

Example File

The following example shows the settings for the categories **MeshFileOptions**, **ConversionOptions**and **EnvironmentOptions**:

```
# Example TOML File
[MeshFileOptions]

FilePath =
FileUnitSystemName = 'BIN_CONSISTENT'
NodeComponentPrefixes = 'ROT'
ElementComponentPrefixes = 'ROT'
ElementFaceComponentPrefixes = 'ROT'

[ConversionOptions]
FaceAngle = 45.0
EdgeAngle = 120.0
TopologyCreationMode = 'GeometryAndMesh'
CreateGeometryFaceComponents = true
CreateGeometryEdgeComponents = true
CreateGeometryVertexComponents = true

[EnvironmentOptions]
AnalysisType = 'Static'
PhysicsType = 'Structural'
SolverType = 'ANSYS'
```

Categories and Options

The supported TOML categories and their options are described below.

[MeshFileOptions]

Category	Option
FilePath: [string][mandatory]	Specify the full path to the solver file to be imported. Use single quotes to enclose the value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Option
FileUnitSystemName: [string][optional]	Unit system name to use when importing. Supported values include: "**SI**" "**MKS_STANDARD**" "**CGS_STANDARD**" "**NMM_STANDARD**" "**NMMTON_STANDARD**" "**BFT_STANDARD**" "**BIN_STANDARD**" "**UMKS_STANDARD**" "**MKS_CONSISTENT**" "**CGS_CONSISTENT**" "**NMM_STANDARD**" "**BFT_CONSISTENT**" "**BIN_CONSISTENT**" "**UMKS_CONSISTENT**" "**KGMMMS_CONSISTENT**" "**MKS_CONSISTENT**" "**CGS_CONSISTENT**" "**NMM_STANDARD**" "**BFT_CONSISTENT**" "**BIN_CONSISTENT**" "**UMKS_CONSISTENT**" "**KGMMMS_CONSISTENT**"
SupportingFiles: [array of strings][optional]	List of included files to use as included file paths. These paths appear in the commands *INCLUDE for ABAQUS, *INCLUDE/*INCLUDE_TRANSFORM for LS-Dyna, INCLUDE for Nastran. If specified, only the specified files will be processed. If not specified, all the included files will be processed.
ProcessMesh200FromCdb: [boolean][optional]	For `.cdb` files process, MESH200 element types. Supported values include **true** and **false**.
NodeComponentPrefixes: [string][optional]	Only import nodal components whose names start with the provided prefixes. The prefixes are to be separated by ';' in the provided string.
ElementComponentPrefixes: [string][optional]	Only import elemental components whose names start with the provided prefixes. The prefixes are to be separated by ';' in the provided string.
ElementFaceComponentPrefixes: [string][optional]	Only import element-face components whose names start with the provided prefixes. The prefixes are to be separated by ';' in the provided string.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[ConversionOptions] (Optional Entries)

Option	Description
FaceAngle: [double][optional]	Tolerance angle for faces.
EdgeAngle: [double][optional]	Tolerance angle for edges.
ProcessLineBodies: [boolean][optional]	Import line elements as line bodies. Supported values include **true** and **false**.
BodyGrouping: [array of strings][optional]	Body grouping to use for import. Supported values include: "**Material**" "**Thickness**" "**AnsysElementNumber**"
TopologyCreationMode: [string][optional]	Specifies whether geometry and/or mesh should be generated by the import. Supported values: "**GeometryAndMesh**" "**MeshOnly**" "**GeometryOnly**"
KeepOnlyXYPlanarElements: [boolean][optional]	Only import 2d elements in the X-Y plane. Supported values include **true** and **false**.
TwoDTolerance: [double][optional]	Tolerance value to use for the **KeepOnlyXYPlanarElements** option.
CreateGeometryFaceComponents: [boolean][optional]	Attempt to create geometric face named selections from components. Supported values include **true** and **false**.
CreateGeometryEdgeComponents: [boolean][optional]	Attempt to create geometric edge named selections from components. Supported values include **true** and **false**.
CreateGeometryVertexComponents: [boolean][optional]	Attempt to create geometric face vertex selections from components. Supported values include **true** and **false**.

[MechanicalOptions]

The only option for this is **AutoContactDetectionOnAttach** [boolean][optional]. This option runs Mechanical auto-contact detection when importing the solver file. Supported values include **true** and **false**.

[EnvironmentOptions] (Mandatory Entries only when **--write_solver_file** is used.)

Option	Description
AnalysisType: [string][mandatory]	Specify the analysis type. Supported values include: "**Static**"

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	Description
	"**SteadyState**" "**Harmonic**" "**Transient**" "**RandomVibration**" "**ResponseSpectrum**" "**Buckling**" "**Modal**" "**ShapeOptimization**" "**TopologyOptimization**" "**SubstructureGeneration**"
PhysicsType: [string][mandatory]	Specify the physics type. Supported values include: "**Structural**" "**Thermal**" "**Electromagnetics**" "**Electric**" "**ThermalElectromagnetics**" "**Multiphysics**"
SolverFileName: [string][optional]	Specify the solver file name produced when using with --write_solver_file. If missing, the defaulted value is `ds.dat`.
SolverType: [string][mandatory]	Specify the solver. The only supported value is "**ANSYS**"

External Model Supported Element Types

Mechanical APDL Elements

The following element types are supported when `.cdb` files are processed via the **External Model** system.

Shape Category	Supported Mechanical APDL Element Type
2D and 3D Point	MASS21
2D Linear Quadrilateral	PLANE13[a], PLANE25[a], FLUID29[a], PLANE55[a], PLANE75[a], PLANE182[a], INTER192, INTER202, CPT212[a], PLANE292[a], MESH200[b]
3D Linear Quadrilateral	FLUID136, SHELL281, SHELL131[a], SHELL157[a], SHELL181[a], MESH200[b]
2D Quadratic Triangle	PLANE35, MESH200[b]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Shape Category	Supported Mechanical APDL Element Type
2D Quadratic Quadrilateral	PLANE77 [a], PLANE78[a], PLANE83[a], PLANE121[a], PLANE183[a], INTER193, INTER203, CPT213[1], PLANE223[a], PLANE230[a], PLANE233[a], PLANE293[a], MESH200[b].
3D Quadratic Quadrilateral	FLUID139, FLUID136, SHELL132[a], SHELL281, MESH200[b]
Beam	BEAM4, BEAM188[c], BEAM189[b], and PIPE288/PIPE289, MESH200[b].
Linear Hexahedral	SOLID5 [a], FLUID30[a], PLANE223, SOLID65[a], SOLID70[a], SOLID96[a], SOLID185[a], SOLID278[a], SOLSH190[a], INTER195, CPT215[a], , MESH200[b]
Link	LINK33, LINK34, LINK180, FLUID116[d]
Quadratic Tetrahedral	SOLID87, SOLID98, SOLID123, SOLID187, CPT217, SOLID227, SOLID232, SOLID237, SOLID285, SOLID291, FLUID221, MESH200[b]
Quadratic Hexahedral	SOLID90 [a], SOLID122[a], SOLID186[a], INTER194, INTER204, CPT216[a], SOLID226[a], SOLID231[a], SOLID236[a], SOLID279[a], FLUID220, , MESH200[b]

[a] This element supports multiple shapes. This list displays the elements in their most basic and fundamental form.

[b] Section data and Real Constants are not imported for this element type.

[c] Pre-integrated composite beam sections are not supported by these elements types.

[d] The application does not use KEYOPTs for FLUID116. Therefore, you should verify the Fluid Descretization setting. In addition, this element uses the Flow cross-sectional area only.

NASTRAN Elements

The following element types are supported when NASTRAN Bulk Data (.bdf, .dat, .nas) files are processed via the **External Model** system.

Shape Category	Supported NASTRAN Element Type
2D Linear Quadrilateral	CQUAD
3D Linear Quadrilateral	CQUAD4, CQUADR
3D Linear Triangle	CTRIAR , CTRIA3
2D Quadratic Triangle	CTRIAX, CTRIAX6
3D Quadratic Triangle	CTRIA6
2D Quadratic Quadrilateral	CQUADX
3D Quadratic Quadrilateral	CQUAD8
Beam	CBAR, CBEAM
Link	CROD[a], CONROD[a]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Shape Category	Supported NASTRAN Element Type
Quadratic Tetrahedral	CTETRA[b]
Quadratic Hexahedral	CHEXA[b]
Quadratic Prism	CPENTA[b]

[a] The torsional constant specified with rod element types is not used if you change the **Model Type** property is changed from **Link** to **Beam**. Instead, a circular cross section is assumed and the calculated torsional constant of the cross section is used.

[b] These elements have the option of being linear.

ABAQUS Elements

The following element types are supported when ABAQUS Input (.inp) files are processed via the **External Model** system.

Shape Category	Supported ABAQUS Element Type
2D Linear Quadrilateral	CPE4, CPE4H, CPE4I, CPE4IH, CPE4R, CPE4RH, CPS6, CPS6M, CPS6MH, CPS6MT, CPS6E, CPEG4, CPEG4H, CPEG4I, CPEG4IH, CPEG4R, CPEG4RH, CPEG4HT, CPEG4RT, CPEG4RHT, DC2D4, DCC2D4, DCC2D4D, DC2D4E, AC2D4, AC2D4R, CAX4, CAX4H, CAX4I, CAXEIH, CAX4R, CAX4RH, DCAX4, DCCAX4, DCCAX4D, DCAX4E, CAXET, CAX4HT, CAX4RT, CAX4P, CAX4PH, CAX4RP, CAX4RPH, ACAX4, ACAX4R, CAX4E, CGAX4, CGAX4H, CGAX4R, CGAX4RH, CGAXET, CGAX4HT, CGAX4RT, CGAXERHT, CAXA4N, CAXA4HN, CAXA4RN, CAXA4RHN
2D Linear Triangle	CPE3, CPE3H, CPE3S, CPE3T, CPE3E, CPS3, CPS3T, CPS3E, CPEG3, CPEG3H, CPEG3HT, CPEG3T, C2D3, DC2D3, DC2D3E, AC2D3, WARP2D3, CAX3, CAX3H, CGAX3, CGAX3H, DCAX3, DCAX3E, CAX3T, CGAX3T, CGA3HT, ACAX3, CAX3E
3D Linear Quadrilateral	ASI4, S4, S4R, S4RS, S4RSW, S4R5, M3D4, MSD4R, MCL6, R3D4, F3D4
3D Linear Triangle	S3, S3R, S3RS, DS3, STRI3, F3D3, M3D3, SFM3D3
2D Quadratic Triangle	CPE6, CPE6H, CPE6M, CPE6MH, CPE6MT, CPE6MHT, CPE6MHT, CPE6MP, CPE6MPG, CPE6E, CPS6, CPS6M, CPS6MT, CPS6E, CPEG6, CPEG6H, CPEG6M, CPEG6MH, CPEG6MT, CPEG6MHT, DC2D6, DC2D6E, AC2D6, CAX6, CAX6H, CAX6M, CAX6MH, DCAX6, DCAX6E, CAX6MT, CAX6MHT, CAX6MP, CAX6MPH, ACAX6, CAX63, CGAX8, CGAX8H, CGAX8M, CGAX8MH, CGAX8MT, CGAX8MHT
3D Quadratic Triangle	STRI65, DS6, M3D6
2D Quadratic Quadrilateral	CPE8, CPE8H, CPE8R, CPE8RH, CPE8T, CPE8HT, CPE8RT, CPE8RHT, CPE8P, CPE8PH, CPE8RP. CPE8RPH, CPE8E, CPE8RE, CPS8, CPS8R, CPS8T, CPS8HT, CPE8RT, CPE8RHTM CPE8P. CPE8PH, CPE8RP, CPE8RPH, CPE8E, CPE8RE, DC2D8, DC2D8E, AC2D8, CAX8, CAX8H, CAX8R, CAX8RH, CGAX8, CGAX8H, CGAX8R CGAX8RH, DCAX8, CAX8T, CAX8HT, CAX8RT, CAX8RHT, CGAX8T, CGAX8HT, CGAX8RT, CGAX8RHT, CAX8P,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Shape Category	Supported ABAQUS Element Type
	CAX8PH, CAX8RP, CAX8PH, ACAX8, CAX8E, CAX8RE, CAXA8N, CAXA8HN, CAXA8RN, CAXA8RHN, CAXA8PN, CAXA8RPN
3D Quadratic Quadrilateral	S8R, S8R5, S8RT, S9R5, ASI8, M3D8, M3D8R, M3D9, M3D9R, MCL9
Beam/Pipe	B21, B21H, PIPE21, PIPE21H, BEAM23, BEAM23H, FRAME2, B31, PIPE31, FRAME3, B31, B22, PIPE22, B32, PIPE32
Linear Tetrahedral	C3D4, C3D4H, C3D4T, C3D4E, DC3D4, DC3D4E, AC3D4
Link	T3D2, T3D3 (midside nodes are ignored)
Quadratic Tetrahedral	C3D10, C3D10H, C3D10M, C3D10MH, C3D10HT, C3D10MHT, C3D10MP, C3D10MPH, C3D10E, DC3D10, DC3D10E, AC3D10
Linear Hexahedral	C3D8, C3D8H, C3D8I, C3D8IH, C3D8R, C3D8RH, C3D8T, C3D8HT, C3D8RT, C3D8RHT, C3D8RPH, C3D8E, DC3D8, DC3D8E, DCC3D8, DCC3D8D, AC3D8, AC3D8R, SC8R
Quadratic Hexahedral	C3D20, C3D20H, C3D20R, C3D20RH, C3D20T, C3D20HT, C3D20RT, C3D20RHT, C3D20P, C3D20PH, C3D20RP, C3D20RPH, C3D20E, C3D20RE, DC3D20, DC3D20E, AC3D20, C3D27, C3D27R, C3D27RH
Linear Prism	C3D6, C3D6H, C3D6T, C3D6E, DC3D6, DC3D6E, AC3D6, SC6R
Quadratic Prism	C3D15, C3D15H, C3D15V, C3D15VH, C3D15E, DC3D15, DC3D15, DC3D15E

LS-DYNA Elements

The following element types are supported when LS-DYNA Input (.k) files are processed via the **External Model** system.

Shape Category	Supported LS-DYNA Element Type
2D Linear Quadrilateral	*ELEMENT_SHELL
2D Linear Triangle	*ELEMENT_SHELL
3D Linear Quadrilateral	*ELEMENT_SHELL , *ELEMENT_SHELL_THICKNESS , *ELEMENT_SHELL_THICKNESS_OFFSET
3D Linear Triangle	*ELEMENT_SHELL , *ELEMENT_SHELL_THICKNESS , *ELEMENT_SHELL_THICKNESS_OFFSET
Linear Tetrahedral	*ELEMENT_SOLID
Quadratic Tetrahedral	*ELEMENT_SOLID
Linear Hexahedral	*ELEMENT_SOLID
Linear Prism	*ELEMENT_SOLID

List of LS-DYNA Supported Keywords

*NODE

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

*ELEMENT_SOLID

*ELEMENT_SHELL

*ELEMENT_SHELL_THICKNESS

*ELEMENT_SHELL_THICKNESS_OFFSET

*PART

*SECTION_SOLID

*SECTION_SHELL

*SECTION_SOLID

*SET_NODE

*SET_NODE_LIST

*CONSTRAINED_NODAL_RIGID_BODY

*BOUNDARY_SPC_NODE

*BOUNDARY_SPC_SET

*INITIAL_STRESS_SOLID

*INITIAL_STRESS_SHELL

Supported External Model Commands

The following sections contain lists of Interfaces provided by the supported commands repository.

External Model CDB Commands

External Model NASTRAN Commands

External Model ABAQUS Commands

External Model FE Commands Repository

External Model CDB Commands

This is the list of Interfaces provided by CDB Commands Repository. All the processed commands are represented by their specialized types, like I**CE**Command, I**CP**Command etc. The unprocessed commands are represented by the IGenericCommand.

```
IGenericCommand : public ICommand
{
    int ArgumentCount();
    string GetArgument(int index);
};

ICETerm
{
    uint Node();
    string Dof();
    double Coefficient();
};
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
ICECommand : public ICommand
{
    uint Nce();
    double Constant();
    int  TermCount();
    ICETerm GetTerm(int index);
};

ICECMODCommand : public ICommand
{
    uint Nce();
    double Constant();
};

ICPCommand : public ICommand
{
    uint Ncp();
    string Dof();
    int NodeCount();
    int GetNode(int index);
};

INCommand : public ICommand
{
    uint Node();
    string Type();
    int ValCount();
    int GetVal(int index);
};

INode
{
    uint Id();
    double GetLocation(int index);
    double GetRotation(int index);
};

INBlockCommand : public ICommand
{
    int NodeCount();
    INode GetNode();
};

IKeyopt
{
    uint Knum();
    int Value();
};

IETCommand : public ICommand
{
    uint Id();
    uint Ename();
    int KeyoptCount();
    IKeyopt GetKeyopt(int index);
};

IENCommand : public ICommand
{
    string Type();
    int ValueCount();
    int GetValue(int index);
};

IElement
{
    uintId();
    uint Mat();
    uint Type();
    uint Real();
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
    uint Section();
    uint Csys();
    int NodeCount();
    int GetNode(int index);
};

IEBlockCommand : public ICommand
{
    int ElementCount();
    IElement GetElement(int index);
};

ICMBlockCommand : public ICommand
{
    string Cmname();
    string Type();
    int IdCount();
    uint GetId(int index);
};

ICMGRPCommand : public ICommand
{
    string Aname();
    int CnameCount();
    string GetCname(int index);
};

ICMEDITCommand : public ICommand
{
    string Aname();
    string Oper();
    int CnameCount();
    string GetCname(int index);
};

ILocalCommand : public ICommand
{
    string Type();
    uint Ncsy();
    uint Cstyp();
    int ValueCount();
    double GetValue(int index);
};

IPreadCommand : public ICommand
{
    string Id();
    int ValueCount();
    double GetValue(int index);
};

IRCommand : public ICommand
{
    uint Nset();
    uint Stloc();
    int ValueCount();
    double GetValue();
};

IReal
{
    uint Id();
    int ValueCount();
    double GetValue(int index);
};

IRLBlockCommand : public ICommand
{
    int RealCount();
    IReal GetReal(int index);
};
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
ISecdataCommand : public ICommand
{
    int ValueCount();
    double GetValue(int index);
};

ISectypeCommand : public ICommand
{
    uint Secid();
    string Type();
    string Subtype();
    string Secname();
    int Refinekey();
};

ISecoffsetCommand : public ICommand
{
    string Location();
    int ValueCount();
    int GetValue(int index);
};

ILayer
{
    double Thick();
    uint Mat();
    double Theta();
    uint Numpt();
};

IShellSecBlockCommand : public ICommand
{
    string Type();
    int LayerCount();
    ILayer GetLayer(int index);
};

ICell
{
    int NodeCount();
    uint GetNode(int index);
    uint MatId();
};

IBeamSecBlockCommand : public ICommand
{
    string Type();
    int CellCount();
    ICell GetCell(int index);
    int NodeCount();
    INode GetNode(int index);
};

IMPDataCommand : public ICommand
{
    uint Mat();
    string Lab();
    int TempCount();
    double GetTemp(int index);
    int ValueCount();
    double GetValue(int index);
};

IMPCommand : public ICommand
{
    uint Mat();
    string Lab();
    double C0();
    double C1();
    double C2();
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
    double C3();
    double C4();
};

IBFCommand : public ICommand
{
    int Node();
    string Lab();
    int ValCount();
    object GetVal(int index); // object is double or string
};

IBFECommand : public ICommand
{
    int Elem();
    string Lab();
    int Stloc();
    int ValCount();
    object GetVal(int index); // object is double or string
};

INodalBodyForce
{
    int Node();
    int ValCount();
    object GetVal(int index);
};

IBFBlockCommand : public ICommand
{
    string Lab();
    int NodalBodyForceCount();
    INodalBodyForce GetNodalBodyForce(int index);
};

IElementBodyForce
{
    int Elem();
    int Count();
    int GetStloc(int index);
    object GetVal(int index); // object is double or string
};

IBFEBlockCommand : public ICommand
{
    string Lab();
    int ElementBodyForceCount();
    IElementBodyForce GetElementBodyForce(int index);
};
IElementSurfaceLoadValues

{
    int ValCount();
    object GetVal(int index); // object is double or string
}

IElementSurfaceLoad
{
    int Elem();
    int Count();
    int GetLKey(int index);
    int GetKey(int index);
    IElementSurfaceLoadValues GetElementSurfaceLoadValues(int index);
};

ISFEBlockCommand : public ICommand
{
    string Lab();
    int ElementSurfaceLoadCount();
    IElementSurfaceLoad GetElementSurfaceLoad(int index);
};
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
ITBDataValues
{
    double Temp();
    int ValueCount();
    double Value(int index);
};

ITBDataCommand : public ICommand
{
    string Lab();
    int Mat();
    int Ntemp();
    int Npts();
    string Tbopt();
    int DataValueCount();
    const ITBDataValues* DataValues(int index);
};

ITBPTValues
{
    int ValueCount();
    double Value(int index);
};

ITBPTData
{
    double Temp();
    int TBPTCount();
    ITBPTValues TBPTValues(int index);
};

ITBPTCommand : public ICommand
{
    string Lab();
    int Mat();
    int Ntemp();
    int Npts();
    string Tbopt();
    int TBPTDataCount();
    ITBPTData TBPTData(int index);
};

IElementType
{
    uint Id();
    uint Ename();
    int KeyoptCount();
    IKeyopt GetKeyopt(int index);
};

IETBlockCommand : public ICommand
{
    int ElementTypeCount();
    IElementType GetElementType(int index);
};

ISeccontrolCommand : public ICommand
{
    int ValueCount();
    double GetValue(int index);
};
```

External Model NASTRAN Commands

This is the list of Interfaces provided by the NASTRAN Commands Repository. All Bulk Data entries are represented by IGenericCommand, except ADAPT, OUTPUT and OUTRCV which are represented

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

by the IOptionsControlCommand. And, the case control section commands are represented by ICaseControlCommand.

```
IGenericCommand : public ICommand
{
    int ArgumentCount();
    string GetArgument(int index);
};

ICaseControlCommand : public ICommand
{
    string Text();
};

INastranOption
{
    string Name();
    string Value();
};

INastranOptionLine
{
    int OptionCount();
    INastranOption GetOption(int index);
};

IOptionsControlCommand : public ICommand
{
    int ArgumentCount();
    string GetArgument(int index);
    int OptionLineCount();
    INastranOptionLine GetOptionLine(int index);
}
```

External Model ABAQUS Commands

This is the list of Interfaces provided by ABAQUS Commands Repository.

```
IAbaqusKeywordArgument
{
    string Key();
    string Value();
};

IAbaqusKeywordArgumentColl
{
    int Count();
    IAbaqusKeywordArgument Item(uint index);
};

IAbaqusKeywordDataLine
{
    int Count();
    string Item(uint index);
};

IAbaqusKeywordDataLineColl
{
    int Count();
    IAbaqusKeywordDataLine Item(uint index);
};

IAbaqusKeyword : public ICommand
{
    IAbaqusKeywordArgumentColl Arguments();
    IAbaqusKeywordDataLineColl DataLines();
};
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Model FE Commands Repository

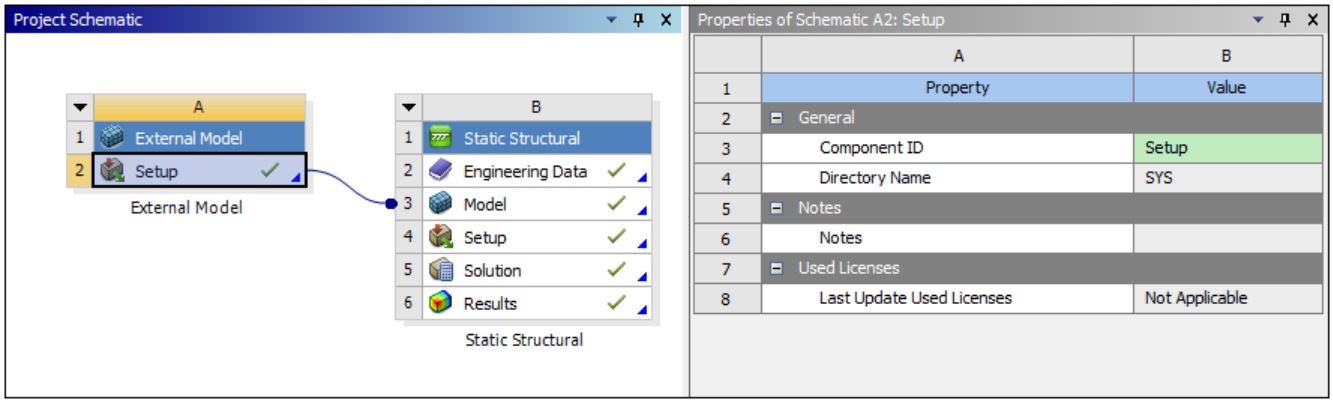
```
enum Ansys.ACT.Automation.Mechanical.FECommandsType
{
 Processed,
 UnProcessed,
 All = Processed | UnProcessed
};

ICommand
{
 string Name ();
 uint Index ();
};

ICommandColl
{
 uint Count ();
 ICommand Item (uint lNum);
};

ICommandRepository
{
 ICommandColl GetCommandsByName (string name);
 ICommand GetCommandByIndex (int index);
 int GetCommandNamesCount (FECommandsType eCommandsType);
 string GetCommandName (FECommandsType eCommandsType, int index);
};
```

Accessing Imported Mesh-Based Databases through ACT

The Mechanical application enables you to import supported finite element data types from External Model and represents them natively through various imported objects in the tree. In addition, imported databases almost always contain additional commands that are not natively supported by the application. Mechanical exposes these commands (along with the commands natively processed in Mechanical) through the ACT extensions and console window. The following APIs enable you to access the underlying commands repository for a file specified in **External Model**.

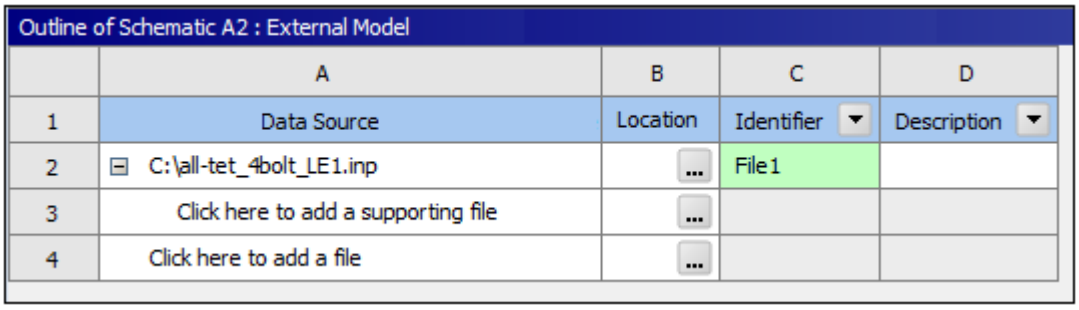
```
commands = ExtAPI.DataModel.Project.Model.GetFECommandsRepository("Setup", "File1");
```

Users familiar with ACT APIs for Ansys Mechanical will notice that the command **`GetFECommandsRepository`** on the **Model** object provides access to the commands repository. The two arguments ("Setup" and "File1") specify the source External Model **Component Id** and "File Identifier" in the External Model Component.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The return object commands is an instance of the interface **ICommandRepository** (p. 980).

Once user has access to the commands database, you can access:

- Processed Commands: Commands processed and natively represented in Mechanical.
- Unprocessed Commands: Everything other than Processed Commands.

User can access the unprocessed commands names using the combination of following commands:

```
unprocessedCmdNamesCount = commands.GetCommandNamesCount(Ansys.ACT.Automation.Mechanical.FE.CommandsType.UnPro
```

And...

```
unprocessedCmdName = commands.GetCommandName(Ansys.ACT.Automation.Mechanical.FE.CommandsType.UnProcessed, inde
```

As you might notice, the command **GetCommandNamesCount** on **ICommandRepository** (p. 980) provides the count of the command names of type specified as first argument:

```
Ansys.ACT.Automation.Mechanical.FE.CommandsType.UnProcessed
```

Or...

```
Ansys.ACT.Automation.Mechanical.FE.CommandsType.Processed
```

And the command **GetCommandName** provides the command name given the type and index (index is 1 based, therefore, if the **unprocessedCmdNamesCount = n**, the valid index value are **(1,n)**).

Given a command name, user can access all instances of a command with a given name as:

```
namedCommands = commands.GetCommandsByName(commandName);
```

The return value **namedCommands** is an instance of the interface **ICommandColl**. A command can be accessed from the **namedCommands** as:

```
namedCommand = namedCommands.Item(index);
```

The index argument is **1** based, therefore valid index value are **(1,namedCommands.Count())**)

The return value **namedCommand** is a derived instance of the interface **ICommand**, i.e. the **namedCommand** instance exposes the methods available on the **ICommand** interface:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
Name(): Command name

Index(): 1 based global index of the command in command repository
```

And any additional methods exposed the specific instance of the command. For example, if the **namedCommand** is an instance of **ICECommand** (a type of processed command from CDB file), then the following additional methods are available:

```
Nce(): Constraint Equation ID

Constant(): Constant Term

TermCount(): Number of Terms

GetTerm(int index): Term at Index
```

Similarly, if the **namedCommand** is an instance of **IGenericCommand** (unprocessed command from CDB file), then the following additional methods are available:

```
ArgumentCount(): Number of arguments

GetArgument(int index): Argument at index
```

Refer to the External Model CDB Commands (p. 973), External Model NASTRAN Commands (p. 978), and the External Model ABAQUS Commands (p. 979) sections for all the available processed and unprocessed commands from CDB, NASTRAN, and ABAQUS databases.

In addition, you can traverse the command repository by going to the next and previous commands in the database using the following APIs:

```
prevCommand = commands.GetCommandByIndex(currentCommand.Index()-1);
```

```
nextCommand = commands.GetCommandByIndex(currentCommand.Index()+1);
```

Given the information provided in the above section, let's take a look at some examples.

CBD File Examples

The following examples describe how to query the command repository.

```
commands = ExtAPI.DataModel.Project.Model.GetFECommandsRepository("Setup", "File1")
```

Example: List All Command Names

This example enables you to generate a listing all command names in the command repository.

```
commandType = Ansys.ACT.Automation.Mechanical.FE.CommandsType.All
commandCount = commands.GetCommandNamesCount(commandType)
commandsNames = []
for iCommand in range(0, commandCount):
 commandsNames.append(commands.GetCommandName(commandType, iCommand+1))
commandsNames
```

Example: List All Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

This example enables you to generate a listing all nodes in the command repository.

```
NBLOCK,6,SOLID,     430,     10
(3i9,6e21.13e3)
      421        0        0 6.3203350000000E+004-4.2985880000000E+004 6.3500000000000E+004
      422        0        0 6.1696900000000E+004-4.5619000000000E+004 6.3500000000000E+004
      423        0        0 5.9365700000000E+004-4.7562160000000E+004 6.3500000000000E+004
      424        0        0 5.6517810000000E+004-4.8605830000000E+004 6.3500000000000E+004
      425        0        0 5.3482670000000E+004-4.8603130000000E+004 6.3500000000000E+004
      426        0        0 5.0634300000000E+004-4.7562160000000E+004 6.3500000000000E+004
      427        0        0 4.8295410000000E+004-4.5625390000000E+004 6.3500000000000E+004
      428        0        0 4.6795660000000E+004-4.2986070000000E+004 6.3500000000000E+004
      429        0        0 4.6250000000000E+004-4.0000000000000E+004 6.3500000000000E+004
      430        0        0 4.6795660000000E+004-3.7013930000000E+004 6.3500000000000E+004
N,R5.3,LOC,       -1,

/*Commands*/
nodeIds = []
nblockCommands = commands.GetCommandsByName("NBLOCK")
nblockCount = nblockCommands.Count
for iNBlock in range(0, nblockCount):
 nblock = nblockCommands[iNBlock+1]
 nodeCount = nblock.NodeCount()
 for iNode in range(0, nodeCount):
  node = nblock.GetNode(iNode+1)
  nodeIds.append(node.Id())
nodeIds

/*Output*/
[421, 422, 423, 424, 425, 426, 427, 428, 429, 430]
```

Example: Extract Analysis Settings

This example enables you to extract all Analysis Setting in the command repository.

```
ANTYPE, 2
MODOPT,LANB,     6,  0.00000000    ,  0.00000000    ,     0, 0,  0.00000000    , 0

/*Commands*/
antypeCommands = commands.GetCommandsByName("ANTYPE")
if(antypeCommands.Count > 0):
    antypeCommand = antypeCommands[1]
    if(antypeCommand.ArgumentCount() > 0):
        antype = antypeCommand.GetArgument(1)
        print antype
        if(antype == "2"):
            modoptCommands = commands.GetCommandsByName("MODOPT")
            if(modoptCommands.Count > 0):
                modoptCommand = modoptCommands[1]
                for iArg in range(0, modoptCommand.ArgumentCount()):
                    print modoptCommand.GetArgument(iArg+1)

/*Output*/
2
LANB
6
0.00000000
0.00000000
0
0
0.00000000
0
```

ABAQUS File Example

This example enables you to generate a listing all nodes in the command repository.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
/*Input*/
*NODE, NSET=nset_csys2
       421, 6.320335E+01,-4.298588E+01, 6.350000E+01
       422, 6.169690E+01,-4.561900E+01, 6.350000E+01
       423, 5.936570E+01,-4.756216E+01, 6.350000E+01
       424, 5.651781E+01,-4.860583E+01, 6.350000E+01
       425, 5.348267E+01,-4.860313E+01, 6.350000E+01
       426, 5.063430E+01,-4.756216E+01, 6.350000E+01
       427, 4.829541E+01,-4.562539E+01, 6.350000E+01
       428, 4.679566E+01,-4.298607E+01, 6.350000E+01
       429, 4.625000E+01,-4.000000E+01, 6.350000E+01
       430, 4.679566E+01,-3.701393E+01, 6.350000E+01

/*Commands*/
nodeIds = []
nodeKeywords = commands.GetCommandsByName("NODE")
nodeKeywordCount = nodeKeywords.Count
for iNodeKeyword in range(0, nodeKeywordCount):
 nodeKeyword = nodeKeywords[iNodeKeyword+1]
 arguments = nodeKeyword.Arguments()
 for iArg in range(0, arguments.Count()):
  arg = arguments.Item(iArg+1)
  print arg.Key()+"="+arg.Value()
 datalines = nodeKeyword.DataLines()
 for iLine in range(0, datalines.Count()):
  dataline = datalines.Item(iLine+1)
  nodeIds.append(dataline.Item(1))
nodeIds

/*Output*/
NSET=nset_csys2
[421, 422, 423, 424, 425, 426, 427, 428, 429, 430]
```

NASTRAN File Examples

Example: Print Case Control Section

This example enables you to print the case control section in the command repository.

```
/*Input*/
CEND
TITLE = MSC.Nastran job created on 01-May-09 at 15:46:33
ECHO = NONE
MPC = 101107
SUBCASE 1
   TITLE=REDESIGNED EAI PAN WITH PRODUCTION LOADS
   SUBTITLE=VIBE_FX
   SPC = 2
   LOAD = 101108
   DISPLACEMENT(PLOT,SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   MPCFORCES(SORT1,REAL)=ALL
   STRESS(PLOT,SORT1,REAL,VONMISES,BILIN)=ALL
   FORCE(PLOT,SORT1,REAL,BILIN)=ALL
SUBCOM 2
   TITLE=REDESIGNED EAI PAN WITH PRODUCTION LOADS
   SUBTITLE=COMBINED VIBE LOADS (PRODUCTION)
   LABEL=COMBINE VIBE PRODUCTION WITH AXIAL LINK
   SUBSEQ = 1.,
            1.,
            1.,
            1.,
            1.,
            1.
   DISPLACEMENT(PLOT,SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   MPCFORCES(SORT1,REAL)=ALL
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
   STRESS(PLOT,SORT1,REAL,VONMISES,BILIN)=ALL
   FORCE(PLOT,SORT1,REAL,BILIN)=ALL
BEGIN BULK

/*Commands*/
commands = ExtAPI.DataModel.Project.Model.GetFECommandsRepository("Setup", "File1")
caseControlCommands = commands.GetCommandsByName("CASE")
caseCount = caseControlCommands.Count
offset = ""
for iCaseCmd in range(0, caseCount):
 caseCmd = caseControlCommands[iCaseCmd+1]
 text = caseCmd.Text()
 if (text.startswith("SUBCASE") or text.startswith("SUBCOM")):
  print text
  offset = "\t"
 else:
  print offset + text

/*Output*/
TITLE = MSC.Nastran job created on 01-May-09 at 15:46:33
ECHO = NONE
MPC = 101107
SUBCASE 1
TITLE = REDESIGNED EAI PAN WITH PRODUCTION LOADS
SUBTITLE = VIBE_FX
SPC = 2
LOAD = 101108
DISPLACEMENT(PLOT ,SORT1,REAL)=ALL
SPCFORCES(SORT1 ,REAL)=ALL
MPCFORCES(SORT1 ,REAL)=ALL
STRESS(PLOT ,SORT1,REAL,VONMISES,BILIN)=ALL
FORCE(PLOT ,SORT1,REAL,BILIN)=ALL
SUBCOM 2
TITLE = REDESIGNED EAI PAN WITH PRODUCTION LOADS
SUBTITLE = COMBINED VIBE LOADS (PRODUCTION)
LABEL = COMBINE VIBE PRODUCTION WITH AXIAL LINK
SUBSEQ = 1.,1.,1.,1.,1.
DISPLACEMENT(PLOT ,SORT1,REAL)=ALL
SPCFORCES(SORT1 ,REAL)=ALL
MPCFORCES(SORT1 ,REAL)=ALL
STRESS(PLOT ,SORT1,REAL,VONMISES,BILIN)=ALL
FORCE(PLOT ,SORT1,REAL,BILIN)=ALL
```

Example: List All Bushings

This example enables you to generate a listing all CBUSH elements in the command repository.

```
/*Input*/
BEGIN BULK
CBUSH    5344    1       15301   15307                           3
        .5
CBUSH    5345    1       15302   15312                           3
        .5
CBUSH    5346    1       15303   15311                           3
        .5
CBUSH    5347    1       15304   15310                           3
        .5
CBUSH    5348    1       15305   15309                           3
        .5
CBUSH    5349    1       15306   15308                           3
        .5

/*Commands*/
commands = ExtAPI.DataModel.Project.Model.GetFECommandsRepository("Setup", "File1")
cbushCommands = commands.GetCommandsByName("CBUSH")
cbushCount = cbushCommands.Count
for iCBushCmd in range(0, cbushCount):
 cbushCmd = cbushCommands[iCBushCmd+1]
 args = []
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

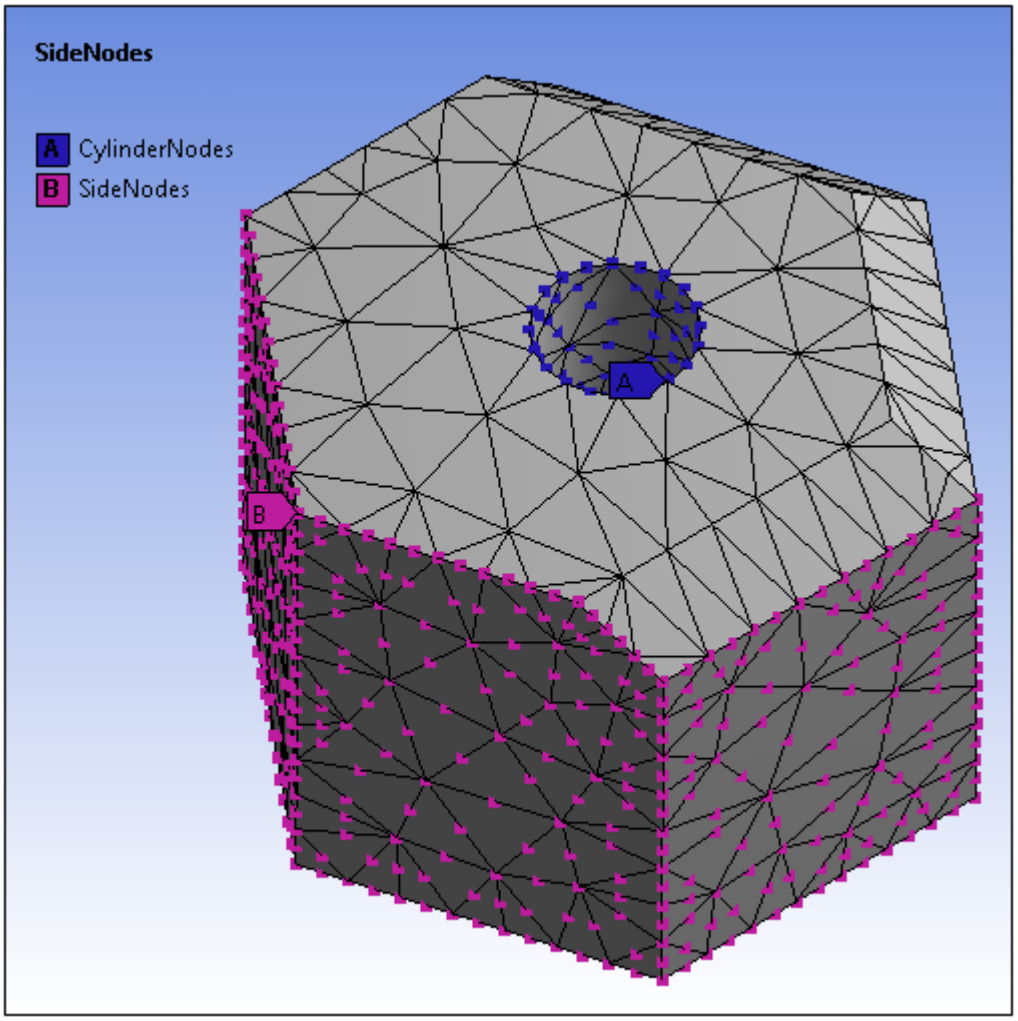
```
 args.append(cbushCmd.Name())
 argCount = cbushCmd.ArgumentCount()
 for iArg in range(0, argCount):
  args.append(cbushCmd.GetArgument(iArg+1))
 print args

/*Output*/
['CBUSH', 5344, 1, 15301, 15307, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
['CBUSH', 5345, 1, 15302, 15312, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
['CBUSH', 5346, 1, 15303, 15311, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
['CBUSH', 5347, 1, 15304, 15310, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
['CBUSH', 5348, 1, 15305, 15309, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
['CBUSH', 5349, 1, 15306, 15308, 0.0, 0.0, 0.0, 3, 0.5, 0, 0.0, 0.0, 0.0]
```

Examples of a Synthesized Mesh using Tolerance Angles

Examples of a geometry that results from a synthesis for a given mesh with different **Tolerance Angle** settings and **Component Key** specifications are illustrated below.

Meshed Model

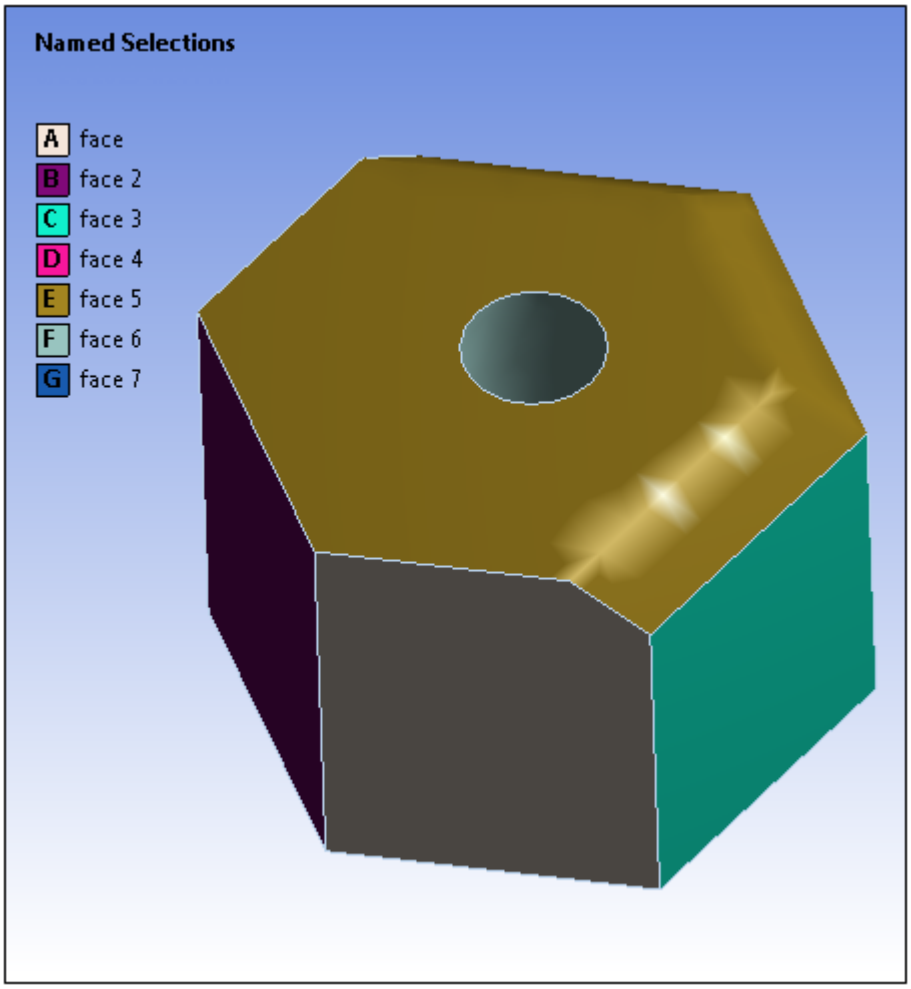
This illustration is a graphical representation from Mechanical of the node data provided by a `.cdb` file. Two nodal components have been processed: **CylinderNodes** and **SideNodes**.



45° Tolerance Angle

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

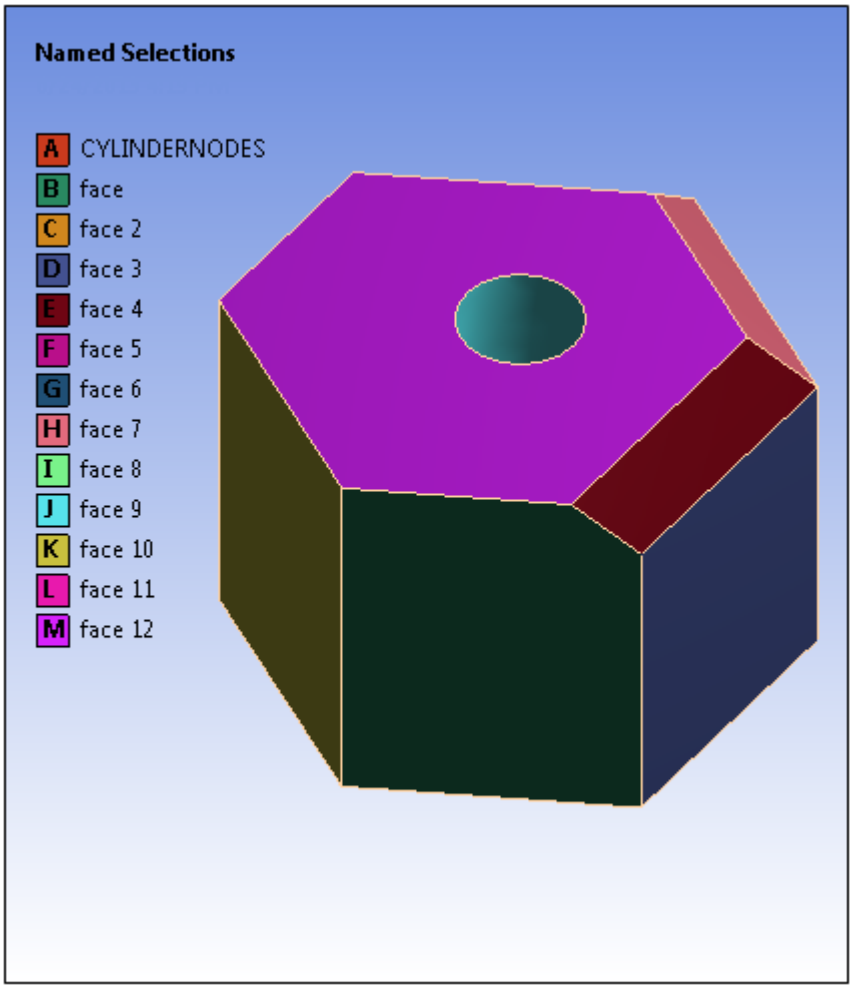
This illustration represents a synthesized geometry that includes faces created using a 45° Tolerance Angle.



25° Tolerance Angle

In this illustration, the **Tolerance Angle** was specified as 25°. This resulted in an accurate synthesis of the geometry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

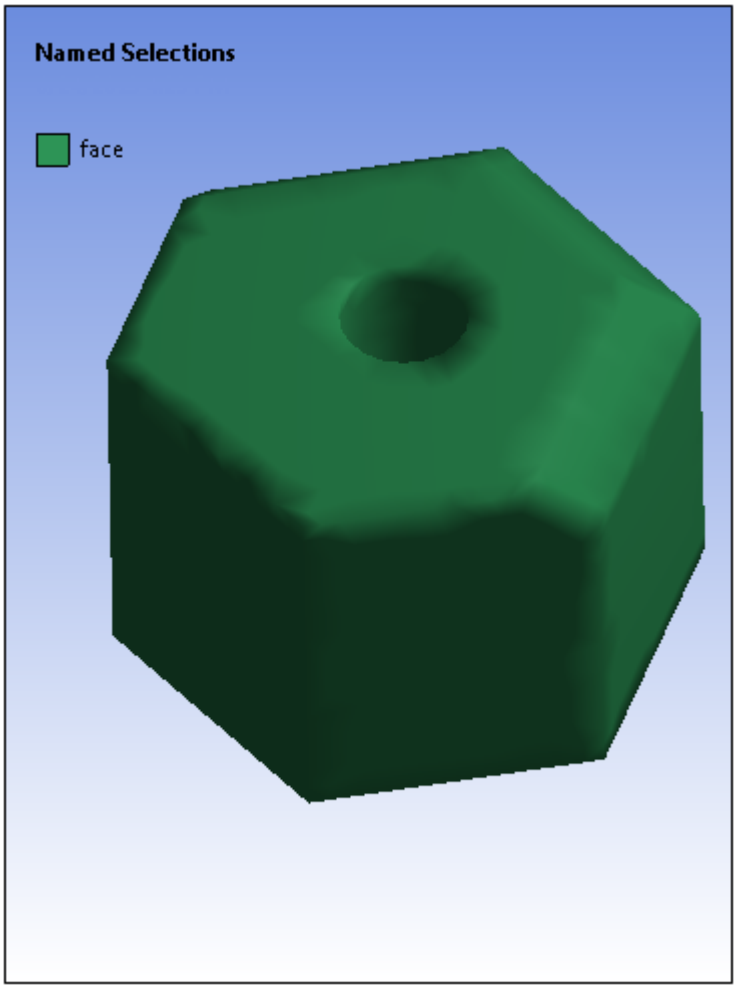


180° Tolerance Angle

This example illustrates how only one face is generated for the geometry when no tolerance angle (180°) is specified and no node-based components are processed. This type of result can also occur when a node-based component contains all of the nodes for a given body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Assembling External Models and Mechanical Models

You can import multiple meshed-based geometries into the **Model** cell of a Mechanical analysis system. The sources of these geometries can be:

- **External Model** component systems
- **Mechanical Model** component systems
- Mechanical analysis systems

The source systems define the geometry and meshes for the downstream Mechanical system. In addition, upstream Mechanical Models, Mechanical analysis systems, and External Model systems support the capability to transfer the following data/objects:

Mechanical Model/Mechanical Analysis System	*External Model*
• Beam Connections (p. 1404) • Contacts (p. 1222) and Spot Welds (p. 1407)	• Constraint Equation (p. 2269) • Coordinate Systems • Element Orientation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

• Commands Object (p. 2303) (that are children of objects that are transferred) • Coordinate Systems (p. 1191) • Element Orientation (p. 1008) (supported only when defined using a Coordinate System) • Engineering Data • Grouping (p. 208) Objects • Joints (p. 1292) • Named Selections (p. 1057) • Point Mass (p. 901) • Remote Points (p. 1089) • Springs (p. 1395)	• Flexible Remote Connectors • Named Selections (p. 1057) • Nodal Orientation • Point Mass • Rigid Remote Connectors • Shell Thicknesses • Spring Connectors

Note:

Model-to-Model linking does not support upstream **End Release** objects.

Refer to the sections below for additional information about the requirements, capabilities, and treatment of assembled models.

Assembly Examples
Model Assembly Specification
Model Alignment
Object Renaming
Associativity of Properties
Contact Detection
Mesh Modification
Using Legacy Databases
Limitations and Restrictions for Model Assembly

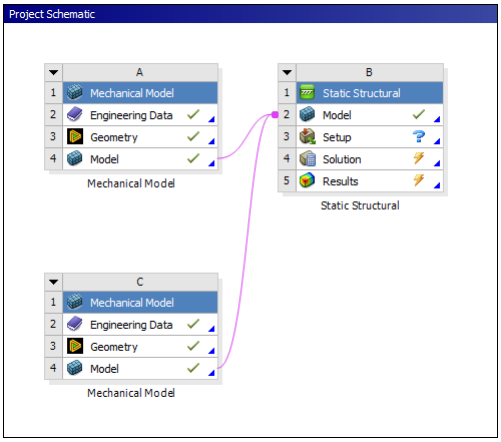
Assembly Examples

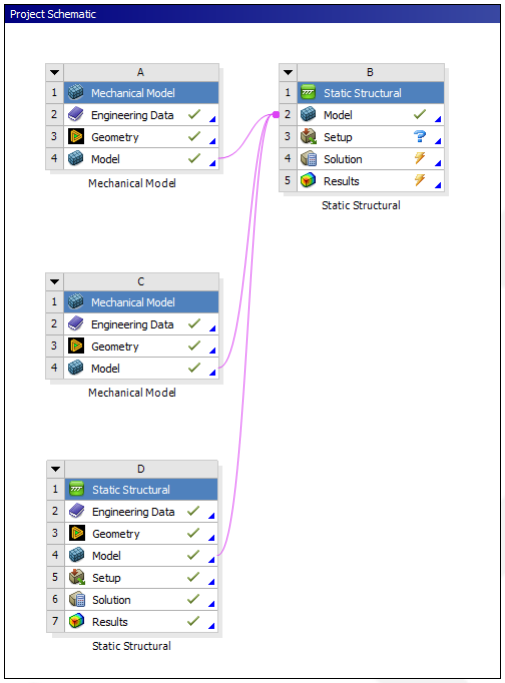
Examples of the Model Assembly feature are illustrated below. Model cells are linked (Model-to-Model linking). You must first mesh all of the upstream systems in order to open the models in Mechanical.

Assembling Mechanical Model Systems	**Assembling Mechanical Model Systems and Analysis Systems**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

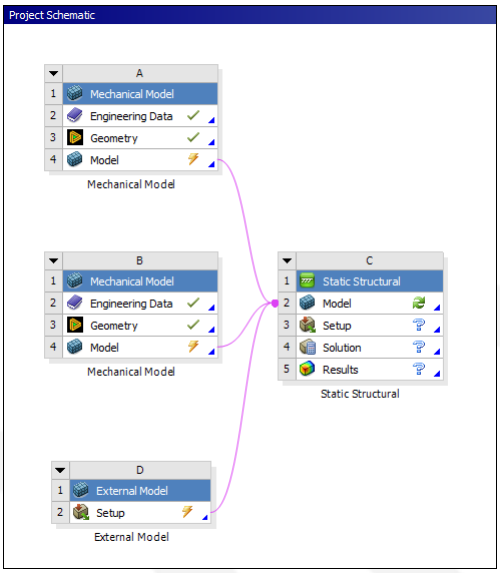
Published: 2023-10-23





This example incorporates the **External Model** feature.

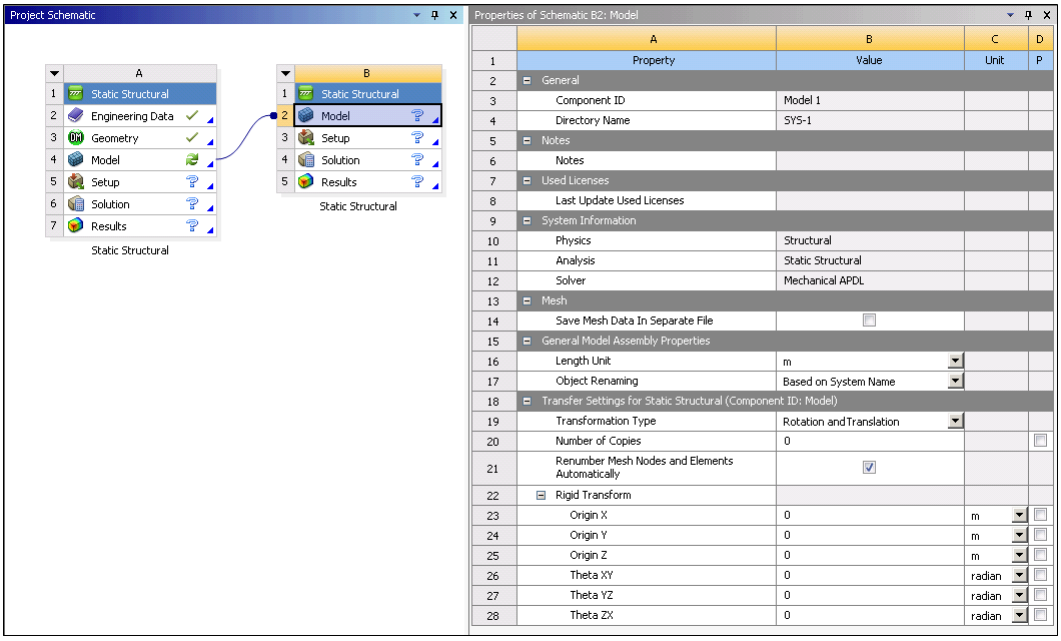
Assembling Mechanical Model Systems and External Model Systems



Model Assembly Specification

Similar to defining Mesh-to-Mesh Connections, Model-to-Model linking provides certain **Project Schematic** properties on the downstream **Model** cell prior to import. These properties can be grouped into two categories as described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



General Model Assembly Properties

The following properties are applicable to all the objects imported into the downstream system.

Property	Description
Length Unit	Units in which the geometry is assembled in Mechanical.
Object Renaming	Downstream system allows you to rename imported objects to easily identify the source of the object and also to differentiate objects from different sources with same name. The drop-down list of automatic naming options includes: • **Off**: no automatic naming of imported objects occurs. • **Based on Cell ID**: automatic naming uses the upstream system's cell ID. • **Based on System Name** (default): automatic naming uses the upstream system's system name. • **User Specified Tag**: when selected, a **User Specified Tag** entry field displays in the Transfer Settings property category (describe below). The default value in this field is the System Name of the upstream model. You can change this name. **Note:** – This option is only applicable for connections from **Mechanical Model** systems. For connections from an **External Model** or **ACP** Component Systems,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	Description
	this option behaves as the **Off** setting and no automatic naming occurs for objects from the **External Model** or **ACP** systems. – Changing the **Object Renaming** option immediately renames the corresponding imported objects in the assembled system. This behavior is different from **Group Objects By Source** option which groups/ungroups the objects based on source on the next refresh operation.
Group Objects By Source	Select the checkbox to automatically group the various source objects created/imported during the assembly process. The assembly process can generate a large number of objects. Grouping them enables you to more easily manipulate the objects in the tree structure. This feature works independently or it can group existing groups of objects that you have defined in an upstream system(s). See the Grouping Tree Objects (p. 208) section of the Help for additional information. **Note:** • This option is only available when you have more than one connection to the model or when you have multiple copies specified for the source **Mechanical Model**. • Changing this option groups/ungroups the imported objects on the next refresh operation. This behavior is different from **Object Renaming** option that immediately renames the corresponding imported objects in the assembled system.

Transfer Settings for "System Name"

Similar to importing mesh-based files using the **External Model** component system, these properties are applicable to each source (upstream) model imported into the downstream system.

Note:

The application does not apply Rigid Transform values if they expand to the "identity transformation." The identity transformation is when the values for Origin X, Origin Y,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Origin Z are all zero (within a tolerance) and Theta XY, Theta YZ, Theta ZX are all multiples of 360 degrees (within a tolerance). The application uses an absolute and relative tolerance equal to the 2D Tolerance specified in the Geometry (p. 225) category of the Options (p. 217) preferences. You can change the specified default value (1e-5) using the preference dialog.

Property	Description
Transformation Type	You use this property to apply a transformation to your upstream model. The transformation options for this property include **Rotation and Translation** (default) and **Mirroring**. The **Rotation and Translation** option enables you to translate the origin of the upstream model along the X, Y, or Z axis or to rotate the model about its origin in the XY, YZ, or ZX plane. Using the **Mirroring** option enables you to import your upstream model as well as a mirror copy of the model about a specified plane. When you create a mirrored copy of your model, the corresponding coordinate system is mirrored by first mirroring the X and Y axes and then the Z axis is computed to make sure the coordinate system adheres to the right hand rule. **Note:** For the **Rotation and Translation** option, transformations are applied in the following order: 1. Rotation about the Y Axis 2. Rotation about the X Axis 3. Rotation about the Z Axis 4. Translations
User Specified Tag	This property allows you to append a label on to the name of the imported objects. The default value is the System Name from the upstream model. This property is only visible when the **Object Renaming** property is set to **User Specified Tag**.
Number of Copies	Specifying a number of copies greater than zero creates copies of the source mesh. For example, if you import a Mechanical Model with a single part and set **Number Of Copies** to 2, you will get three parts in the downstream Mechanical system.
Renumber Mesh Nodes and Elements Automatically	This property is active by default and automatically renumbers the nodes and elements of your assembly. Deselecting this option maintains the existing node and element numbering. Any numbering duplication and the application will display an error condition. You can modify node and element numbers with the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	Description
	Mesh Numbering (p. 1035) feature once you import your assembly into Mechanical.
Rigid Transform (**Transformation Type** = **Rotation and Translation**)	**Transform the Original**: This property is only available when **Number Of Copies** is set to 1 or greater. Select the checkbox if you want to apply the specified transformation to the source mesh. **Origin X, Y, and Z**: Translate the origin of the model along the X, Y, or Z axis. If you specify any copies, the translation will be applied relative to the previous copy (or source mesh in the case of the first copy). **Theta XY, Theta YZ, and Theta ZX**: Rotate the model about its origin in the XY, YZ, or ZX plane. If you specify any copies, the rotation will be applied relative to the previous copy (or source mesh in the case of the first copy).
Mirror Transform (**Transformation Type** = **Mirroring**)	**Plane Point X/Y/Z** These properties enable you to define the coordinates of a point on the mirroring plane. **Plane Normal X/Y/Z**: These properties enable you to define a vector that is normal (orthogonal) to the mirroring plane.

Model Alignment

Introduction

Source models may need to be transformed (translated and rotated in space) to position and align them. The transformations can be pre-calculated or you may need to calculate the transformations based on the fit of your different sub-models. For example, you may need to align the hole from one model to the bolt from another model.

There are two methods available to specify the rigid transformations for a source model:

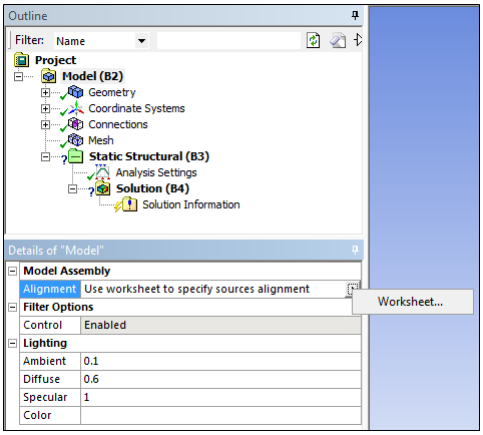
1. Specifying **Rigid Transform** properties in **Transfer Settings** (p. 991) for each **Source Model**. These settings are useful if the required translation and rotation values are already available, or can easily be calculated for a source.

2. Using the **Worksheet** on the **Model** node in the assembled system that enables you to properly position and align your source models using coordinate systems that you import from sources systems or that you create in Mechanical for the assembled system. The details of alignment using **Worksheet** are discussed below.

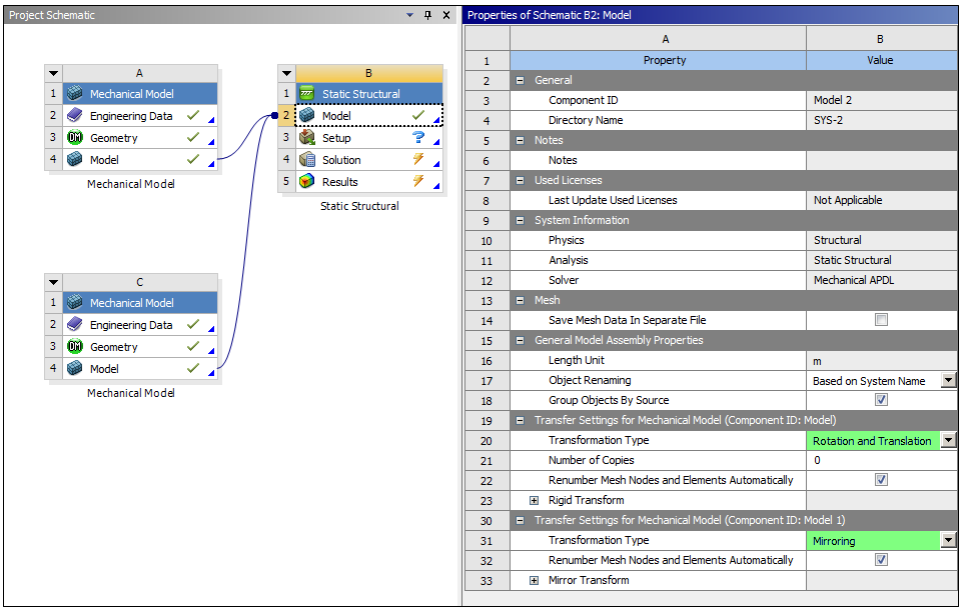
Specifying Rigid Transforms Using the Worksheet

You access the Model Assembly **Worksheet** by selecting the **Model** object in Mechanical and the selecting the fly-out menu of the **Alignment** property, as shown below, or by selecting the Worksheet button on the Home Tab (p. 40).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

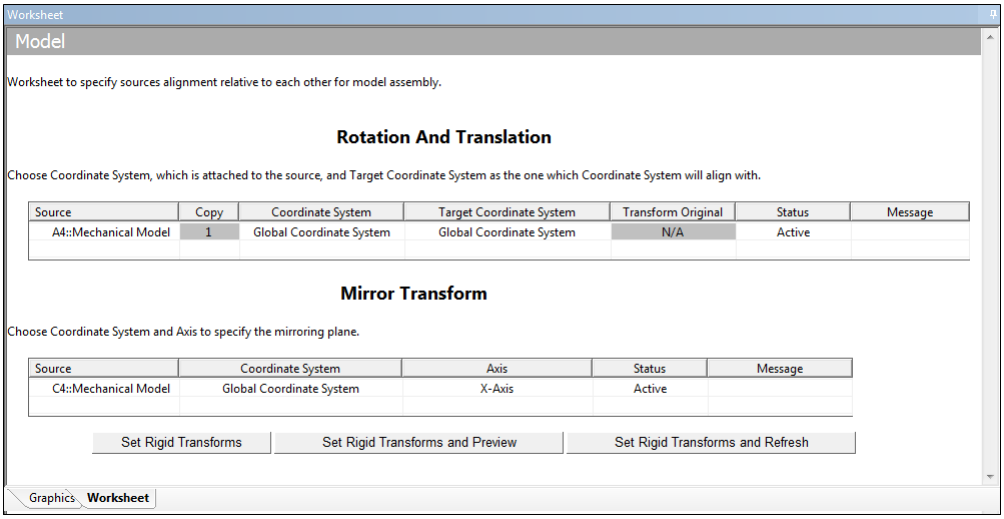


The **Transfer Settings** (p. 991) selections you make on the Workbench **Project** page determine what is displayed in the Mechanical **Worksheet**. In the example below, **Rotation and Translation** and **Mirror Transform** have both been specified. The **Worksheet** sections populate based on your **Transformation Type** entries.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

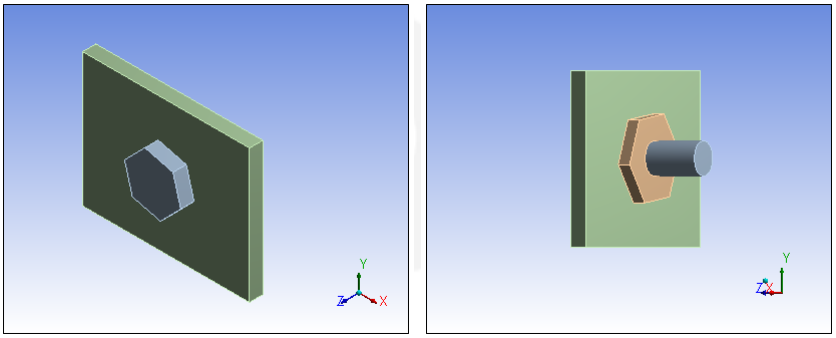
Published: 2023-10-23



The **Worksheet** contains a row for every Mechanical source model. For **Rotation And Translation**, you need to specify a source and a target coordinate system, for each source model. For alignment, the transforms are calculated such that the source coordinate system after transformation is aligned with the target coordinate system. For **Mirror Transform**, you need to specify a Coordinate System and an Axis to define the mirroring plane.

Example

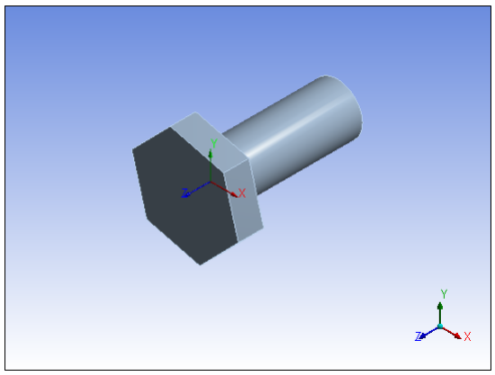
The following example demonstrates the assembly of three components (a bolt, plate, and nut) using the **Rotation and Translation** transformation. The unassembled geometry is shown below.



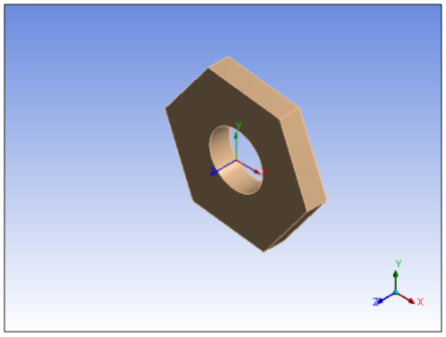
In this example, the plate needs to be aligned between the bolt head and the nut. To specify alignment, the following coordinate systems are created:

- **Coordinate System (Bolt)**: Centered at intersection of bolt axis and base of the bolt head.

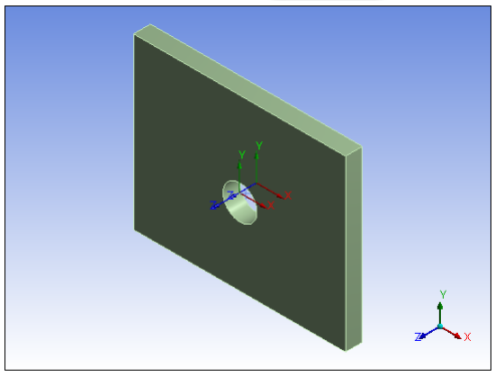
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- **Coordinate System (Nut)**: Centered at intersection of nut axis and top face of the nut.



- **Coordinate System (Plate - Bolt)** and **Coordinate System (Plate - Nut)**: Centered at intersection of hole axis and top and bottom faces of the plate, respectively.

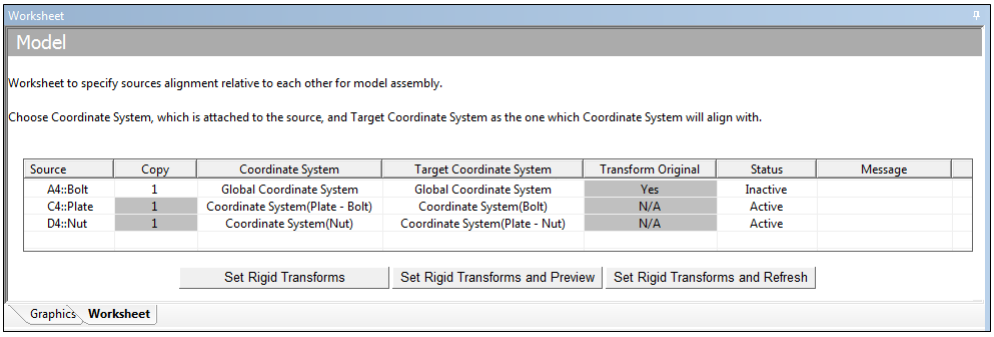


You use the features of the Worksheet to specify alignment. In the following example, the:

- bolt is in its desired location so no transforms are specified for the bolt.
- plate needs to be aligned with the bolt head. The **Coordinate System (Plate - Bolt)** is specified as source coordinate system and **Coordinate System (Bolt)** is specified as target coordinate system.
- top face of the nut needs to be aligned with bottom face of the plate so **Coordinate System (Nut)** is specified as source coordinate system and **Coordinate System (Plate-Nut)** is specified as target coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

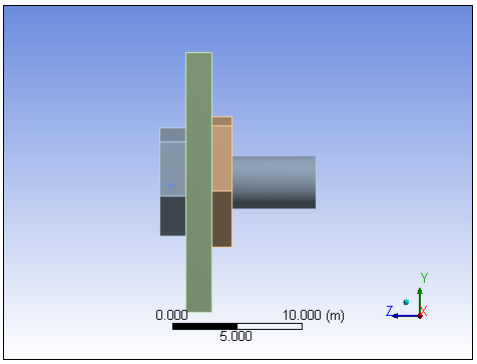
Published: 2023-10-23



Once the transforms are specified using coordinate systems, you have three options to update the rigid transform values in workbench, which include:

- **Set Rigid Transforms**: Calculates rigid transform value and sets them in Transfer Settings in Workbench
- **Set Rigid Transforms and Preview**: Calculates rigid transform value, set them in Transfer Settings in Workbench and Preview Geometry.
- **Set Rigid Transforms and Refresh**: Calculates rigid transform value, set them in Transfer Settings in Workbench and Refresh Model.

The assembled system is shown below.



The worksheet contains the following additional columns:

- **Copy**: Enables you to specify a copy of the model that needs to be aligned. Calculates the transforms such that the specified copy is aligned with the target coordinate system.
- **Transform Original**: Enables you to control **Transform Original** property when the number of copies is greater than zero. If the **Copy** specified is **1**, then the field is read-only and set to **Yes**.
- **Status**: Enables you to control how the transformations specified (using **Coordinate System** and **Target Coordinate System**) in a particular row are applied during **Rigid Transform** calculation. Options include:
  - **Active**: The application performs a complete calculation to align **Coordinate System** with **Target Coordinate System**.
  - **Inactive**: The transformations for the Source are not calculated. The model does not move as a result of calculating **Rigid Transforms**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - **Locked**: The application locks the source relative to the target, i.e. the source moves by the same amount the target does. This option is useful when, for example, you need to move a complete (and aligned) subassembly due to some additional requirement. In this case, for all sources of the subassembly, set the **Status** to **Locked** and then specify the transformation (using **Coordinate System** and **Target Coordinate System**) only for the root node (first source in the chain).

- **Message**: Displays any validation messages if any of the entry is invalid.

Note:

For legacy databases (created in Mechanical version 15.1 or before), the Worksheet for alignment is only available once you have reassembled your model in the current version.

Object Renaming

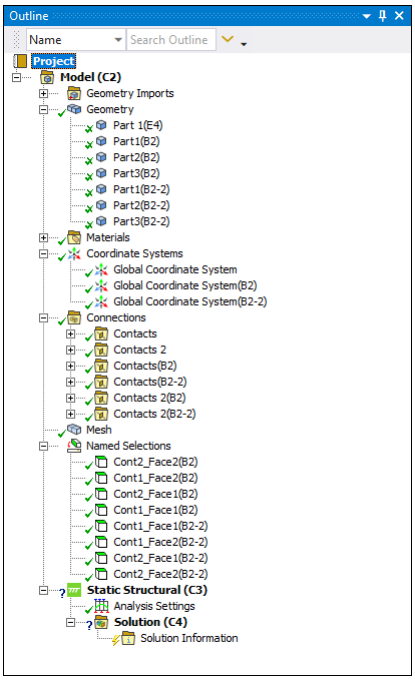
Once imported into Mechanical, automatic renaming occurs for the following objects imported from upstream systems.

- Beams (p. 1404)
- Command objects (p. 2303)
- Contacts (p. 1222)
- Coordinate Systems (p. 1191)
- Cross Sections
- Geometry (p. 847) Bodies (Solid, Surface, and Line)
- Grouped tree objects (p. 208)
- Joints (p. 1292)
- Named Selections (p. 1057)
- Point Mass (p. 901)
- Remote Points (p. 1089)
- Springs (p. 1395)

Based on the **Object Renaming** property specified in the **General Model Assembly** properties group, the application appends the **System Name**, **Cell Id**, or the **User Specified Tag** to the object name (in parenthesis). The example shown below is using the system name. In addition, if you create copies of a mesh, the application incrementally appends the name (**-2**, **-3**, etc.).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note that there may be limitations on the naming conventions used on for Named Selections objects. See the Limitations and Restrictions for Model Assembly (p. 1005) topic in the *Mechanical Model Systems and Analysis Systems* section for additional information.

Associativity of Properties

During model assembly, the application maintains the properties and scoping of the objects imported from upstream systems (geometry bodies, coordinate systems, contact conditions, Joints, Named Selections, and Command objects). These specifications are, by default, read-only in the downstream analysis system. A Details view category, **Transfer Properties**, as illustrated below for examples of a Named Selection and a Coordinate System, provides the **Source** name of the upstream system and provides an option to change the object's properties from **Read Only**. Refreshing your system returns these properties to a read-only state.

The properties of imported objects in a downstream system are by default, **Read-Only**, and inherit their values from the upstream system, except for the **Suppressed** property. The **Suppressed** property inherits its value from the upstream-system during the initial import only. The application maintains this setting as long as you do not change the property in the downstream system. Unlike all of the other properties that update when you modify the upstream system and refresh or reimport, if you change the setting of the **Suppressed** property, the application preserves your change and any subsequent upstream system updates and/or refresh or reimport does not affect this setting update. Based on this unique characteristic of the **Suppressed** property, the application does not treat the property as **Read-Only**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Model A Named Selection(Mechanical Model A)"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Imported
Total Selection	1 Body
Suppressed	0
Used by Mesh Worksheet	No
Transfer Properties	
Source	A4::Mechanical Model A
Read Only	Yes

Details of "CoordinateSystem:2(External Model)"	
Definition	
Type	Cylindrical
Coordinate System	Program Controlled
Suppressed	No
Origin	
Define By	Global Coordinates
Origin X	-3.5355 m
Origin Y	-3.5355 m
Origin Z	0. m
Location	Click to Change
Principal Axis	
Axis	X
Define By	Fixed Vector
Orientation About Principal Axis	
Axis	Y
Define By	Fixed Vector
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[-3.5355 -3.5355 0.]
Transfer Properties	
Source	A2::External Model
Read Only	Yes

Contact Detection

During the import process when creating an assembly, the application not only imports and automatically renames contact conditions, it automatically identifies and creates any new contact conditions created as a result of the assembly process. During the import process, two types of contact conditions are detected:

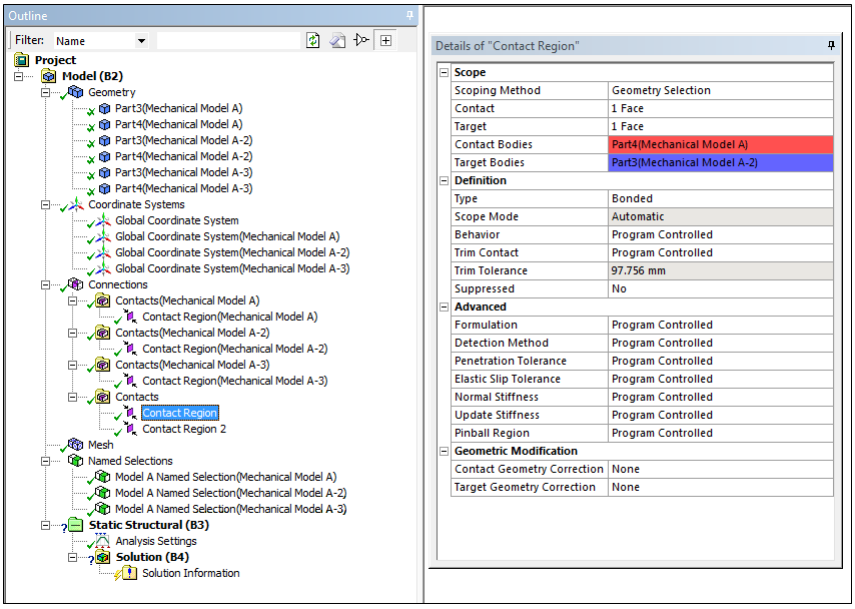
1. Contacts between sub-assemblies imported from different sources. This type of contact detection happens for all sources, either imported from **Mechanical Model** or mesh-based geometry systems like **External Model** or ACP.

2. Contacts within an imported sub-assembly. This type of contact detection happens only for mesh-based geometry systems like ACP or Deformed Mesh (except **External Model**).

The example below illustrates the generation and renaming of contact conditions for two copies of a **Mechanical Model**. In addition, the highlighted contact condition is for the new contact conditions between the assembled geometries.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



In addition to contact detection during the import process, the Mechanical application allows you to detect contact between sub-assemblies or within a sub-assembly after importing (assembling) the models.

To automatically generate connections between sub-assemblies:

1. Insert a Connection Group object under the Connections object using the Connections (p. 50) Context tab or by right-clicking on the **Connections** object and selecting **Insert**>**Connection Group**.
2. In the Details view for the new **Connection Group**, select the **Assemblies** option for the **Search Across** property.
3. As needed, specify any other **Auto Detection** properties.
4. Right-click the **Connection Group** object and select **Create Automatic Connections**.

To automatically generate connections within a sub-assembly:

1. Specify **Source Assembly** for the **Scoping Method** property
2. Select the desired assembly from the **Source Assembly** property's drop-down list.
3. As needed, specify any other **Auto Detection** properties.

> **Note:**
>
> The application only applies updates to the **Source Assembly** for this **Connection Group**.

4. Right-click the **Connection Group** object and select **Create Automatic Connections**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To automatically generate connections between the files of an External Model source:

1. Set the **Scoping Method** property to **Source Assembly**.
2. Select the desired source file from the **Source Assembly** property's drop-down list.
3. As needed, specify any other **Auto Detection** properties.

> **Note:**
>
> The application only applies updates to the **Source Assembly** for this **Connection Group**.

4. Set the **Search Across** property to **Files**.
5. Right-click the **Connection Group** object and select **Create Automatic Connections**.

Mesh Modification

When you assemble files from upstream sources (Mechanical Model, External Data, and/or analysis systems), the **Mesh** object presents an additional category called **Model Assembly** that includes a property called **Read Only**. The options for this property are **Yes** (default) and **No**. It should be a rare requirement, but you can change the setting of this property (to **No**) if all of the upstream systems are either Mechanical Model sources or analysis system sources. When set to **No**, all of the Mesh object's default properties display and although they are not in sync with the upstream systems, you can edit the assembly's mesh and regenerate the mesh.

Setting the **Read Only** property to **Yes** following changes to the model that cause the mesh to become out-of-date, you can clear and resume your original mesh using the **Generate Mesh** option.

An associated process for the resume operation on **Generate Mesh** is that during the attachment process the application verifies that the mesh is available on all unsuppressed bodies and that the **Stiffness Behavior** settings for all of the assembled bodies match the stiffness behavior from the source file. If this verification is unsuccessful, then the attachment process will not complete and the application generates error messages that list the bodies in error.

In addition, when the **Read Only** property is set to **Yes**, the use of the following features is not supported:

- Cyclic Symmetry (p. 1134)
- Mesh Controls (Sizing, etc.)
- Mesh Connections: when at least one upstream system is **ACP** or **External Model**.
- Node Merge: when at least one upstream ACP system has an Interface Layer defined.
- Mesh Numbering (p. 1035): when at least one upstream system is ACP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Fracture Meshing (p. 415): when at least one upstream system is ACP or External Model.

Note:

It is recommended that you define all mesh controls and settings in your upstream systems.

Mesh settings on upstream systems take priority over any downstream mesh settings. That is, any changes to an upstream system will overwrite your mesh setting changes on your downstream system once updated. As a result, you could see differences between the assembled mesh and the settings of the downstream meshed model. Therefore, to have your downstream mesh to be updated per the mesh setting changes, you need to re-mesh your downstream model once it has been refreshed.

The application suppresses Mesh controls if they are present in the assembled system during the Mesh transfer process. You may unsuppress the mesh controls and remesh if the assembled system allows for it (assembled system does not have a connection from **External Model** or **ACP**).

Using Legacy Databases

You may encounter compatibility issues when importing model assemblies that you saved in a previous version of the application. To address this, you can first update the legacy system to your current version using the following steps:

From Workbench

To update your systems to the current version: right-click on the **Model** cell of the upstream system and select **Update**. The application automatically saves the project and as a result, updates the systems to the current version of the software.

From Mechanical

In the event you open a legacy system in Mechanical and are prompted with a warning message regarding the version, you can:

1. Make a minor change that does not affect the solution states (hide a body and unhide it again, change the name of an object, etc.).
2. Save the Mechanical database or close the application. The systems are updates to the newer version.

Limitations and Restrictions for Model Assembly

Note the following requirements for model assembly:

- Parts are made up of one or more bodies. As a result, when working with model systems, the application treats meshed parts and meshed bodies differently with regards to whether the mesh is transferred to the downstream system. Bodies meshed in an upstream system always transfer the mesh to the downstream system. However, parts (single-body or multi-body) meshed and suppressed later in an upstream system; do not have their mesh transferred to the downstream

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system during assembly. Consequently, when the downstream system supports unsuppression (**Mesh** object **Read Only** property set to **No**), any unsuppressed parts require you to generate a new mesh (unlike an unsuppressed body). This is a different behavior compared to transferring a single model. The application transfers the mesh of a suppressed part for a single model and enables you to unsuppress parts without generating a new mesh.

- The application does not support **Mesh Edit** features, such as Mesh Connections, Node Merge, etc. in the assembled system if the **Read-Only** property on the **Mesh** object in the assembled system is set to **Yes** and if at least one of the upstream systems includes mesh edits.

- **Don't lose scoping**: Geometry from **External Model** (.cdb) files is partially associative. When you have geometry from multiple **External Model** system assembled, and you refresh upstream model data into the downstream system, any geometry scoping that you have performed on an object in the downstream analysis will be lost for the modified External Model system only. That is, only **External Model** systems that you change lose scoping. For example, if you have two **External Model** systems assembled, System 1 and System 2, and you have objects scoped to geometry in the assembled system. If you modify System 1 and then refresh the upstream system, geometry scoping on objects is lost only for System 1. System 2 experiences no scoping losses. A more robust way to maintain scoping is to properly define imported Named Selections or criterion-based Named Selections.

  In addition, if you are using the Inventor application, Autodesk Inventor, any changes you make in the application will cause the loss of scoping in the downstream Mechanical system following an update.

- The Geometry object property **Scale Factor Value**, allows you to modify the size of imported geometries in the upstream systems. The scale factor value of newly imported geometries is **`1.0`**. You can modify the value and that modified value is expected to be preserved on updated models.

  Be aware that when you assemble models and change the associated unit of measure, you are limited by a scale factor limit of **`1e-3`** to **`1e3`**. This scale factor limit is the limit for any combination of models. Factor values are totaled and anything outside of this range is ignored. As a result, due to these tolerances, scaled models, especially larger and/or combined models, sometimes have problems importing geometry/mesh.

- You need to perform material assignment in the upstream systems. The Material category property, **Assignment**, in the downstream system is read-only.

- For Named Selections, Mechanical sends only 32 characters to the Mechanical APDL application input file (p. 1087).

  The use of **Based on System Name** or **User Specified Tag** options for the **Object Renaming** property can easily generate names which are same for the first 32 characters and therefore cause Named Selections created later in the process to overwrite earlier ones. The application issues a warning if this scenario is detected during the solution process.

  Note that this limitation only affects the use of Named Selections in a **Commands** (p. 2303) object.

  For all other objects scoped to named selections, the length of named selection has no bearing on the scoping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- When you assemble rigid body systems or systems that include rigid bodies, Mechanical may require you to remesh the rigid bodies if the application detects new contact conditions on the rigid bodies. If this is the case, the application sets the **Read Only** property on the **Mesh** object to **No** and also cleans any imported mesh-based Named Selections. These objects need to be re-scoped. The **Mesh** object property, **Read Only** is set to **No** when importing rigid bodies or rigid body systems from **Mechanical Model** and/or analysis systems. Note that it is not possible to remesh (**Read Only** cannot be set to **No**) when assembling upstream sources from **External Model** or **ACP**, so contact conditions with rigid bodies are not allowed in such cases if remeshing is required.

- You cannot create or define Virtual Topology in the assembled system. As needed, define the required virtual topology features in the upstream systems. Furthermore, if you do define virtual topology, any change to the topology can cause scoping losses in the downstream system. You can correct scoping losses using the geometry Scoping Wizard (p. 147).

- The application does not currently support the ability to transfer parameters from upstream systems to the assembled system. As a result, upstream parameters will not be present in the assembled system.

- Model systems do not support the following features. If present, updates to the project fail for the system transferring data to a downstream system. You need to suppress or delete these features before transferring data.

  – Crack Objects

  – Interface layers Imported from ACP

  – Substructuring

  – Cyclic Symmetry

  – Symmetry Region

You may wish to refer to the Mechanical Model section of the Workbench Help for additional information about this component system.

Additional Limitations and Restrictions for External Model Component System

When an **External Model** component system is incorporated into model-to-model assembly, certain additional limitations and restrictions arise:

- Any suppressed bodies from other upstream systems can be unsuppressed in the downstream environment provided they were meshed prior to being suppressed in the upstream system. However, suppressed parts from other upstream systems can never be unsuppressed in the downstream environment when using the **External Model** component system. These restrictions also apply when using the options **Unsuppress All Bodies** and/or **Invert Suppressed Body Set**.

- Geometry from **External Model** is partially associative. When you have geometry from multiple **External Model** system assembled, and you refresh upstream model data into the downstream system, any geometry scoping that you have performed on an object in the downstream analysis will be lost for the modified **External Model** system only. That is, only **External Model** systems that you change lose scoping. For example, if you have two **External**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Model systems assembled, System 1 and System 2, and you have objects scoped to geometry in the assembled system. If you modify System 1 and then refresh the upstream system, geometry scoping on objects is lost only for System 1. System 2 experiences no scoping losses. A more robust way to maintain scoping is to properly define imported Named Selections or criterion-based Named Selections.

- Limitations similar to geometry associativity also apply to properties on bodies imported from **External Model** systems. Any changes made to properties on bodies imported from an **External Model** system in the assembled system are lost if the source **External Model** system is modified. Properties on bodies from other (unmodified) External Model(s) are maintained.
- Cylindrical supports (p. 1688) cannot be applied to mesh-based files imported using the **External Model** Component. You can instead apply equivalent loading using a **Displacement** (p. 1666) load applied in appropriate cylindrical coordinate system.
- During the assembly process, if named selections from different files (sources) have the same name, the application merges these named selections.

Element Orientation

The Element Orientation feature is used define the orientations and alignments of elements/bodies for a given model. It enables you to define the coordinate system of one or more manually selected elements or each element of a specified body. As described below, there are two application methods for the Element Orientation feature: either the Surface and Edge Guide (p. 1009) option that is the default, or the Coordinate System (p. 1011) option. An example of each option accompanies the application steps. In addition, review the procedure used to generate orientations (p. 1013) when using the Surface and Edge Guide option as well as the display options (p. 1013) that enable you to change the how you wish to view the orientation vectors in the graphics window.

Also see the Element Orientation object reference page for additional information about this feature.

Note:

Element Orientations can also be imported through External Data (p. 736) or the **External Model** system. See the Imported Element Orientation (External Data) and Imported: Element Orientations (External Model) object reference pages for more information.

Requirements

Note the following requirements for this feature:

- It is supported for 3D solids and shells.
- If multiple edges are selected in the **Edge Guide** category, the edges must be connected.
- The feature is not supported by the Rigid Body or Explicit Dynamics analysis systems.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The LS-DYNA analysis system supports element orientation.

Caution:

When solving your analysis, you may receive the following message if you are using the Element Orientation feature and your Ply/Layer data has overlapping elements.

"Some Element Orientations will be ignored in the solution process due to overlapping elements from Imported Plies."

Note:

- For shell elements, this feature uses the projection of the local system on the shell surface. As a result, the element orientations displayed in the graphics window will be the same as those used by the solver.
- If you want to import Element Orientations from an upstream Mechanical system to a downstream Mechanical system, the application only supports Element Orientations defined by a coordinate system.

For specific technical information about this feature, refer to the **ESYS** command in the *Mechanical APDL Command Reference* as well as the Understanding the Element Coordinate System in the *Mechanical APDL Element Reference*.

Application

Surface and Edge Guide Option (default option)

To define your element orientation using the **Surface and Edge Guide** option:

1. Select the **Geometry** object in the tree.
2. Select the **Element Orientation** option on the **Geometry** Context tab (p. 56) or right-click and select **Insert**>**Element Orientation** or right-click in the Geometry window and select **Insert**>**Element Orientation**. The new object becomes the active object.

   Note:

   You can insert multiple **Element Orientation** objects into the tree. The last object inserted supersedes previous objects that have the same body scoping.

3. In the **Scope** category of the Details view:

   a. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

b. Select the body you wish to apply orientations to and, click in the **Geometry** property's field, and then click the **Apply** button. You can also specify an element or elements (p. 276) as well as a desired geometry- or element-based (p. 276) **Named Selection**.

Note:

The body you select may have an existing coordinate system scoping; however, once defined by an **Element Orientation** system, the feature overwrites any existing coordinate systems.

---

4. In the **Definition** category of the Details view specify the **Defined By** property as **Surface and Edge Guide**.
5. In the **Surface Guide** category of the Details view:
   a. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection**.
   b. Select the face or faces that you wish to define to be the Surface Guide, click in the **Geometry** property's field, and then click the **Apply** button. Or, specify a desired **Named Selection**. You can select multiple faces to define this surface geometry.
   c. Specify the axis for the perpendicular direction from the **Axis** property drop-down list, either **+/- X Axis**, **+/- Y Axis**, or **+/- Z Axis**.
6. In the **Edge Guide** category of the Details view:
   a. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection**.
   b. Select the edge or edges that you wish to define as tangent to the target body, click in the **Geometry** property's field, and then click the **Apply** button. Or, specify a desired **Named Selection**.
   c. Specify the axis for the tangent direction from the **Axis** property drop-down list, either **+/- X Axis**, **+/- Y Axis**, or **+/- Z Axis**.
7. Right-click the **Element Orientation** object and select **Generate Orientations**.

Note:

This feature requires an X, Y, and Z coordinate. You define two of the coordinates, surface and edge, and the application calculates the third required coordinate.

---

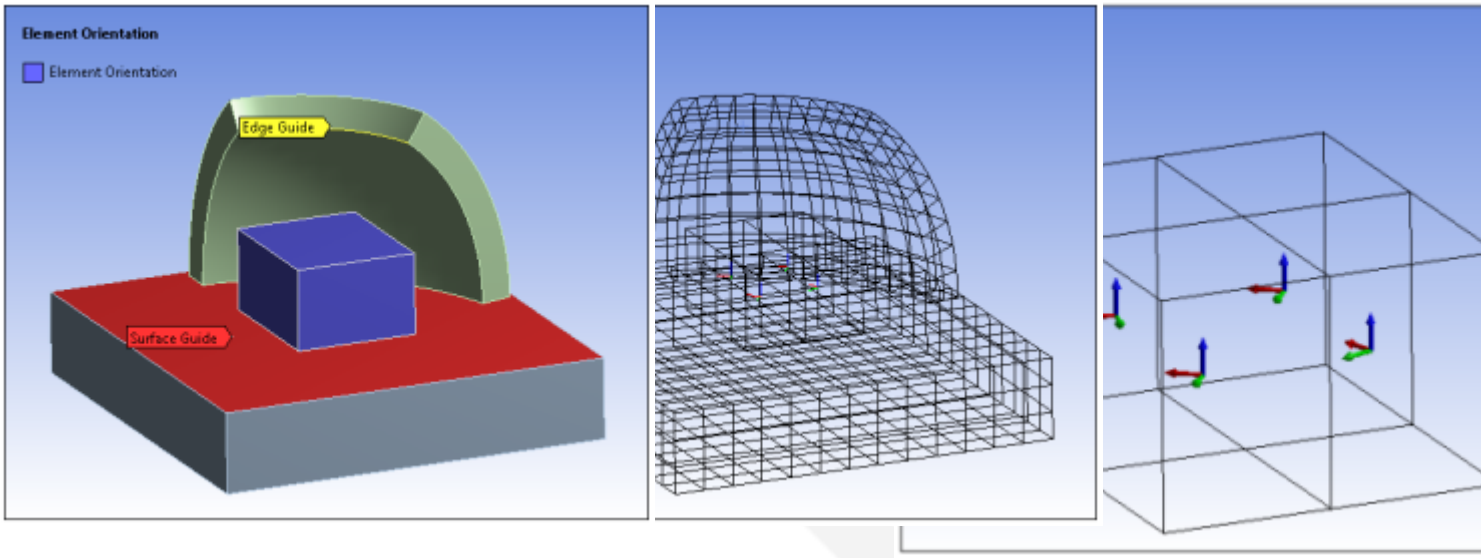
8. Select the **Wireframe** view option to view the new orientations.

Surface and Edge Guide Orientation Example

The following images illustrate the use of the **Surface and Edge Guide** option. The first image shows the defined body (in purple) and the Surface Guide (red face) and Edge Guide (yellow edge) selections. The second image illustrates the internal element orientations that were generated with all bodies displayed and the third image is an enlargement of the target body only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Coordinate System Option

To define your element orientation using the **Coordinate System** option:

1. Select the **Geometry** object in the tree.
2. Select the **Element Orientation** option on the **Geometry** Context tab (p. 56) or right-click and select **Insert**>**Element Orientation** or right-click in the graphics window and select **Insert**>**Element Orientation**. The new object becomes the active object.

   Note:

   You can insert multiple **Element Orientation** objects into the tree. The last object inserted supersedes previous objects that have the same body scoping.

3. In the **Scope** category of the Details view:
   a. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

b. Select the body you wish to apply orientations to and, click in the **Geometry** property's field, and then click the **Apply** button. You can also specify an element or elements (p. 276) as well as a desired geometry- or element-based (p. 276)**Named Selection**.

Note:

The body you select may have an existing coordinate system scoping; however, once defined by an **Element Orientation** system, the feature overwrites any existing coordinate systems.

Note:

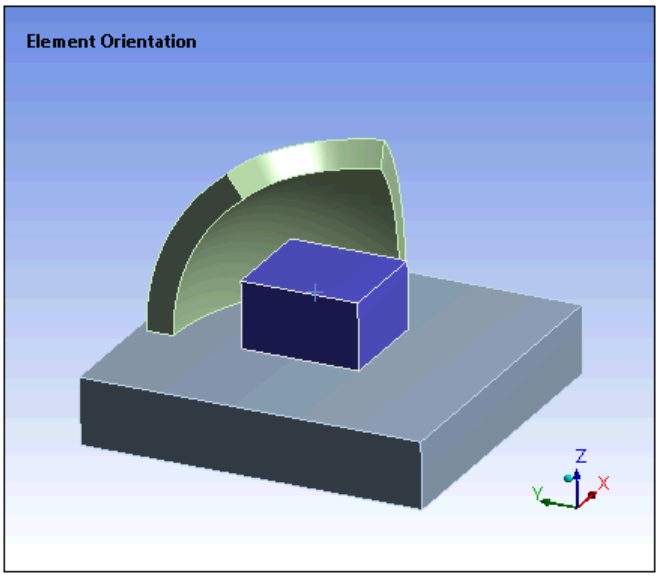
The body you select may have an existing coordinate system scoping; however, once defined by an **Element Orientation** system, the feature overwrites any existing coordinate systems.

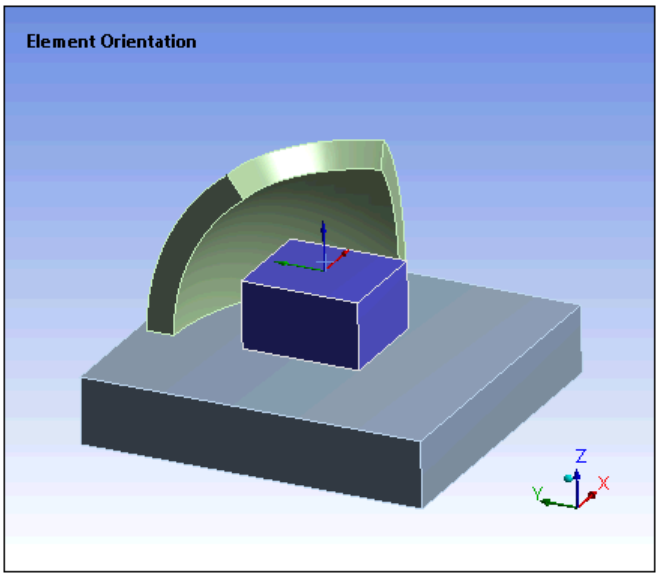
---

4. In the **Definition** category of the Details view specify the **Defined By** property as **Coordinate System**.
5. Set the **Coordinate System** property to the desired coordinate system that you have created. One single triad automatically displays on the applicable region. No other triads are shown, unlike the **Surface and Edge Guide** option.
6. Select the **Wireframe** view option to view the new orientations.

Coordinate System Orientation Example

The following images illustrate the use of the **Coordinate System** option. The first image shows the defined body and the second image illustrates the coordinate-based element orientation.





Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Generation Procedures for Surface and Edge Guide Option

The **Surface Guides** and **Edge Guides** use the following procedure to generate orientations:

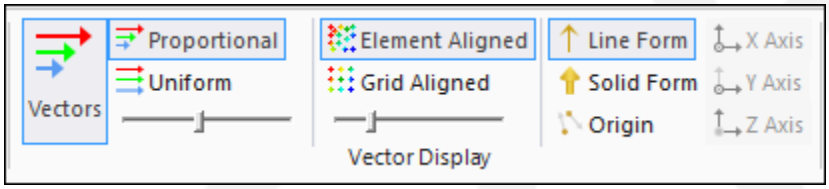
1. Identify one or more surfaces to be the surface guide and one or more edges to be the edge guide.
2. Identify the 1st axis (for example, z-axis) that will align normal to the surface guide.
3. Identify the 2nd axis (for example, x-axis) that will align itself tangential to the edge guide.

For each element in the selected body:

- The application obtains the surface normal direction (N-vector) at a location on the Surface Guide closest to the element's centroid and aligns the specified axis (z-axis) with it.
- The application obtains the tangential direction (T-vector) to the edge at a location on the Edge Guide that is closest to the element's centroid.
- The cross-product of the N-vector and T-vector calculate the 3rd axis (y-axis).
- The tangential, 2nd axis (x-axis), is obtained by taking the cross-product of the N-vector and the 3rd axis.

Display Options

Once orientations are generated, the **Vector Display** group appears in the **Geometry** Context tab.



Note:

Because Element Orientations have no magnitude, vector length is always uniform and therefore, the **Proportional** and **Uniform** options are inactive.

The **Display** options provide the following features.

Option	Illustration	Description
Vectors		Selecting this option activates the associated display options.
Element Aligned		Displays all vectors, aligned with each element.
Grid Aligned		Displays vectors, aligned on a uniform grid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Illustration	Description
Density Slider		rols the relative size of the grid, which determines the quantity (density) of the vectors. Increases are made in uniform steps from 0 [coarse] to 100 [fine] (default = 20), as displayed in the tool tip when you drag the mouse cursor on the slider handle. This control is only available when the **Grid Aligned** option is selected.
Scaling Slider		rols the relative length of the vectors in incremental steps from to 10 (default = 5), as displayed in the tool tip when you drag the mouse cursor on the slider handle.
Line Form		Displays element vectors as line arrows.
Solid Form		Displays element vectors as solid arrows.
Origin		Displays origin of the nodes included in each element orientation.
X Axis		Turns the display of the X Axis of an element on and off.
Y Axis		Turns the display of the Y Axis of an element on and off.
Z Axis		Turns the display of the Z Axis of an element on and off.

These options are similar to the options of the Vector Display (p. 74) for results.

Part Transformations

While setting up your simulation, you can use the **Part Transform** feature to modify the orientation of parts on your model. This can be beneficial if you want to reorient or move certain parts after they have been imported into Mechanical as well as if you wish to simulate different parts orientations. The feature also enables you to parameterize your changes.

Important:

When you use the Part Transform feature on a meshed model that has a crack specified on a part, each time you change the Part Transform, the application clears the mesh on the body with cracks and requires re-mesh of the crack-based body.

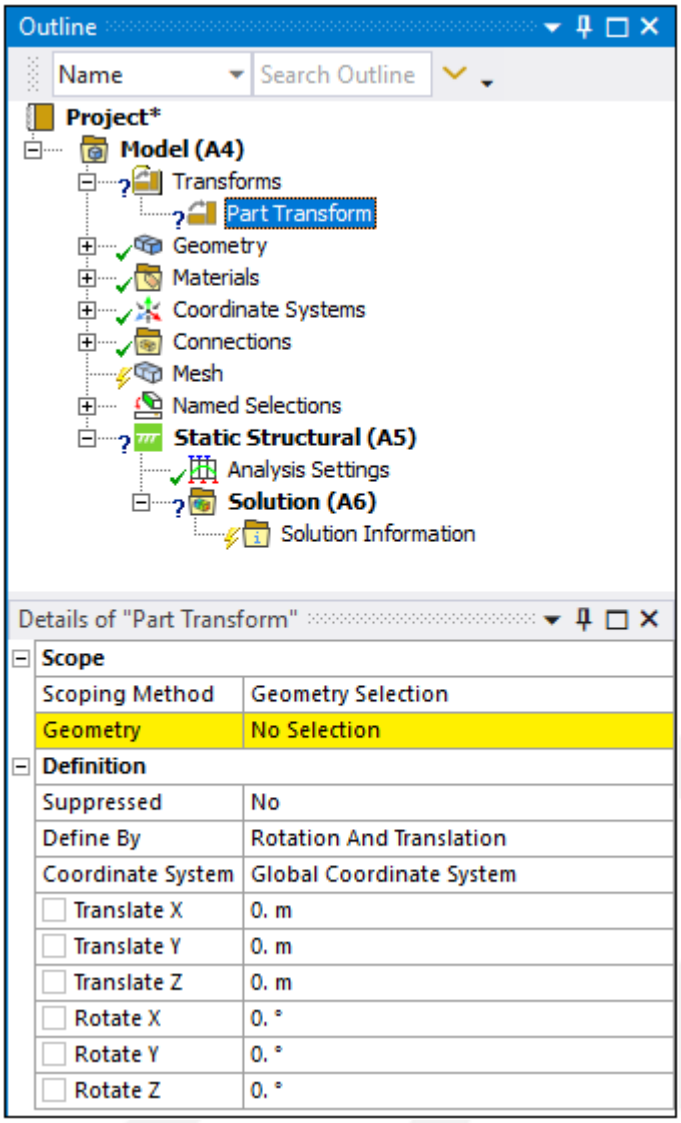
Application

To create part transformations:

1. Select the **Model** object and then select the **Part Transform** option on **Model** context toolbar. As illustrated below, a **Transforms** folder object is placed in the **Outline** and automatically includes a Part Transform object. This folder object houses all of the part transformations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Additional insertion options include:

- Right-click on the **Model** object or in the **Geometry** window and select **Insert** > **Part Transform** to insert these objects.
- Select the parts/bodies you wish to transform from the **Geometry** object or on your model in the **Geometry** window, right-click, and then select **Create Part Transform**.

2. As needed, change the settings of the **Transforms** object properties, including:

   - **Transform Mesh**: Options include **Yes** (default) and **No**. This property controls whether the application automatically transforms the mesh of your transformed parts. Setting this property to **No** instructs the application to clear the mesh of your transformed parts when updated.
   - **Regenerate Contacts**: Options include **Yes** and **No** (default). This property controls whether the application automatically generates contact on your transformed parts.

3. Specify the **Scoping Method** property as either **Geometry** (default) or **Named Selections**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Specify your desired geometry:

   - Select the entry field of the **Geometry** property and, using the **Body** selection filter on the Graphics toolbar (p. 97), make geometry selections on the model, and then select the **Apply** button. Once specified, this field displays the number of parts you have selected (for example, 1 Part, 2 Parts, etc.). An example is illustrated below.

     Or...

   - Select a body-based Named Selection from the **Named Selection** property drop-down list. Once specified, this field displays the specified Named Selection. An example is illustrated below.

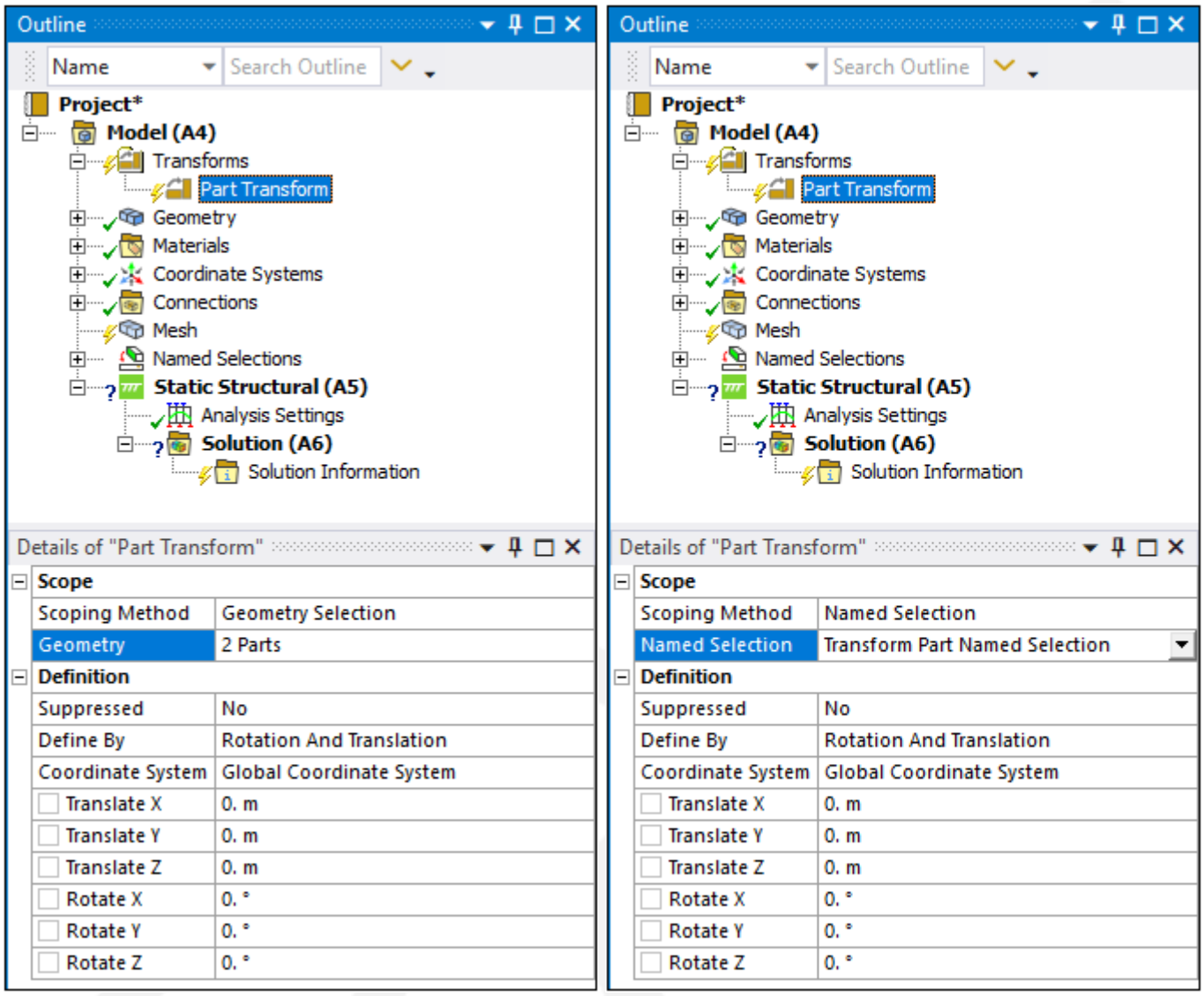
Note:

If you make a geometry selection using either the Face, Edge, or Vertex selection filters or if you specify a Named Selection based on any of these geometric entities, the application automatically selects the entire body associated with the entity. Mesh selection options are not supported.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

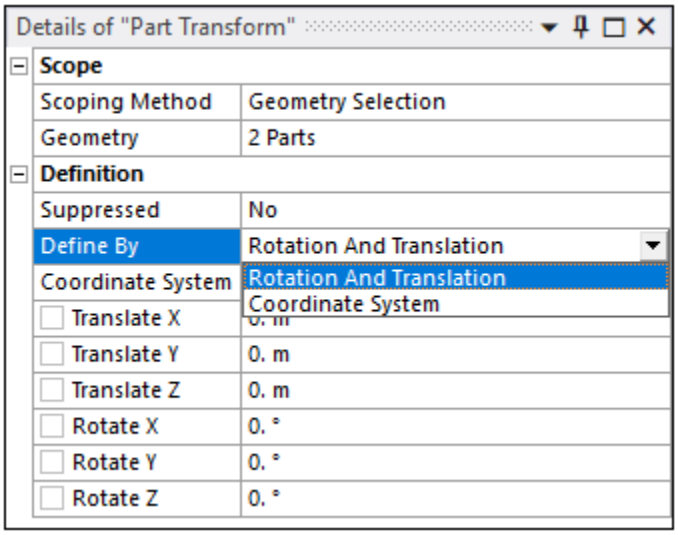
Published: 2023-10-23

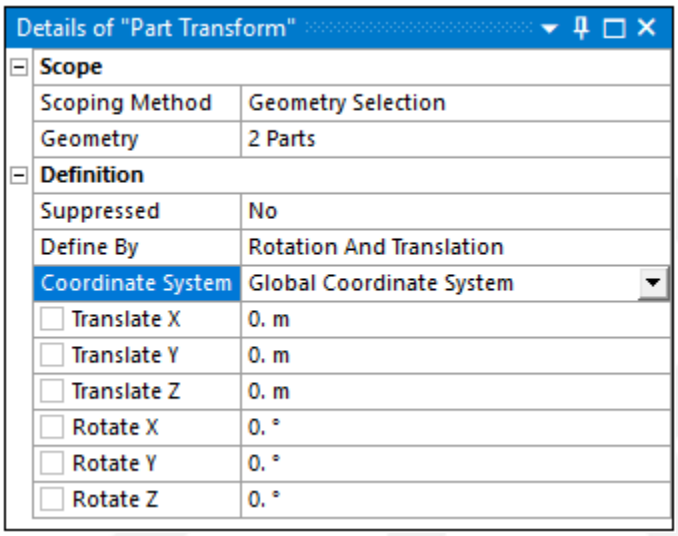


5. Using the **Define By** property, select your transformation method. Options include **Rotation and Translation** (default) and **Coordinate System**.

   - **Rotation and Translation**: You use this option and the associated **Transform X/Y/Z** and **Rotate X/Y/Z** properties to specify the dimensions for moving or reorienting the selected parts. This option also provides an associated **Coordinate System** property that you can use to define a user-defined coordinate system instead of using the default **Global Coordinate System** setting. Note that the application applies the rotational transforms first, followed by translation transformations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

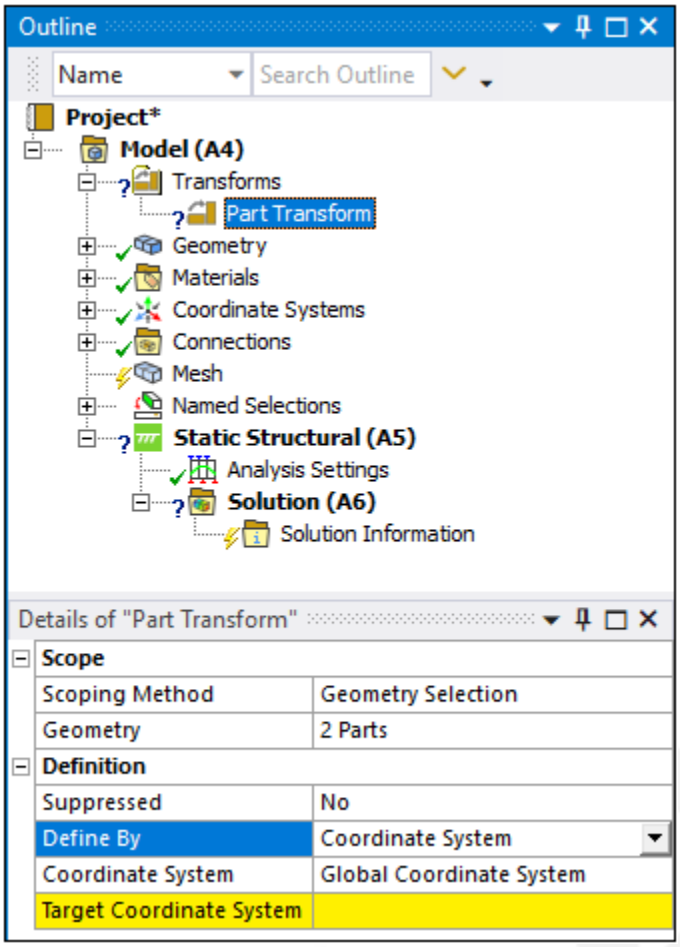




- **Coordinate System**: You use this option and the associated (source) **Coordinate System** and **Target Coordinate System** properties to specify your transformation on the selected parts by coordinate systems. For this option, you need to manually create a **Target Coordinate System** that will be the basis of the transformation. The application reorients your selected parts by aligning the source coordinate system with the user-defined **Target Coordinate System**.

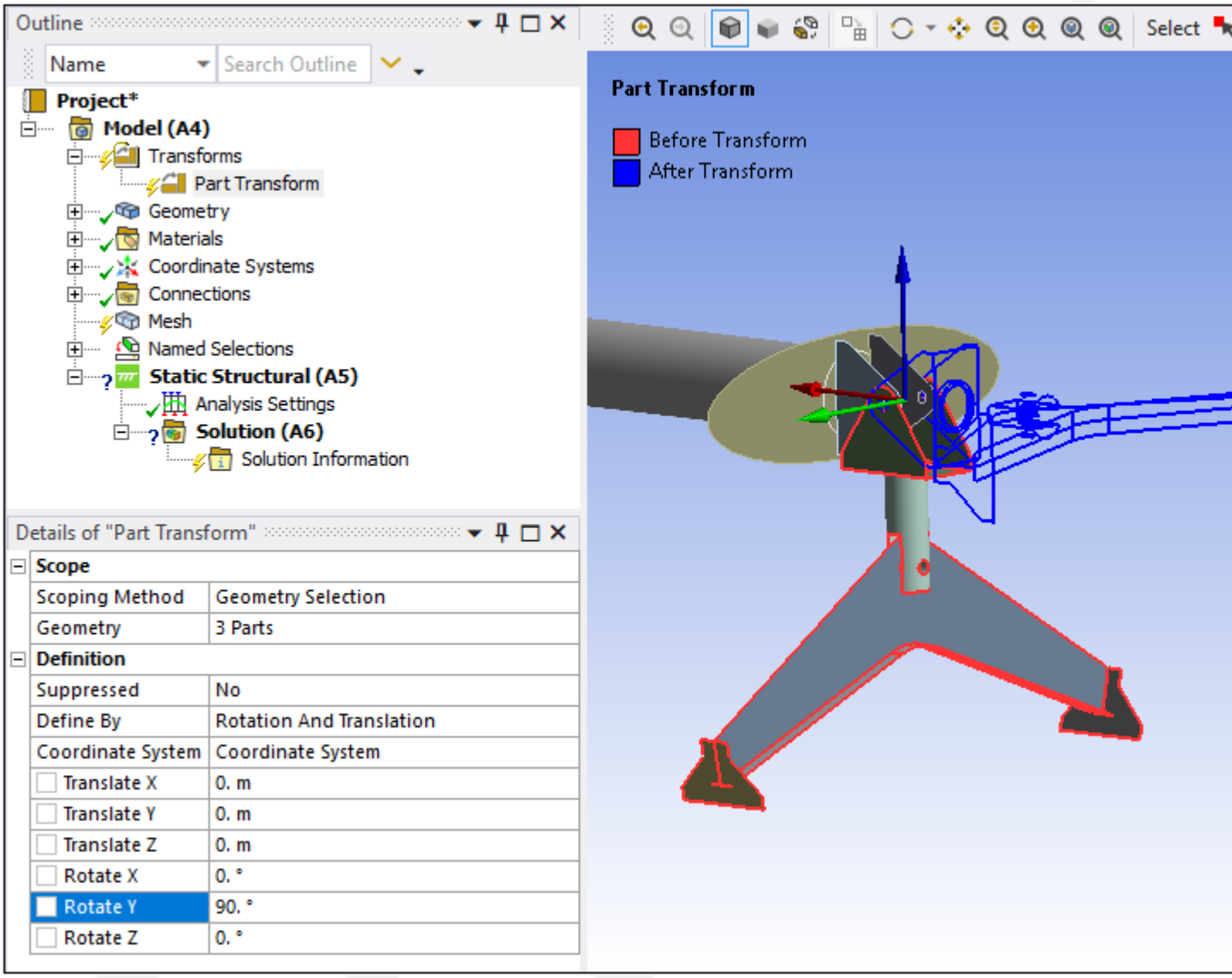
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



6. Specify your desired transformations. Once you complete your entries, the parts highlight as indicated by the annotations **Before Transform**/**After Transform**. The red display shows the current position of your parts and the blue highlight shows the pending location of the transformation. An example is illustrated below. A Coordinate System system is created for the transformation location and the parts are rotated 90° in the +Y direction.

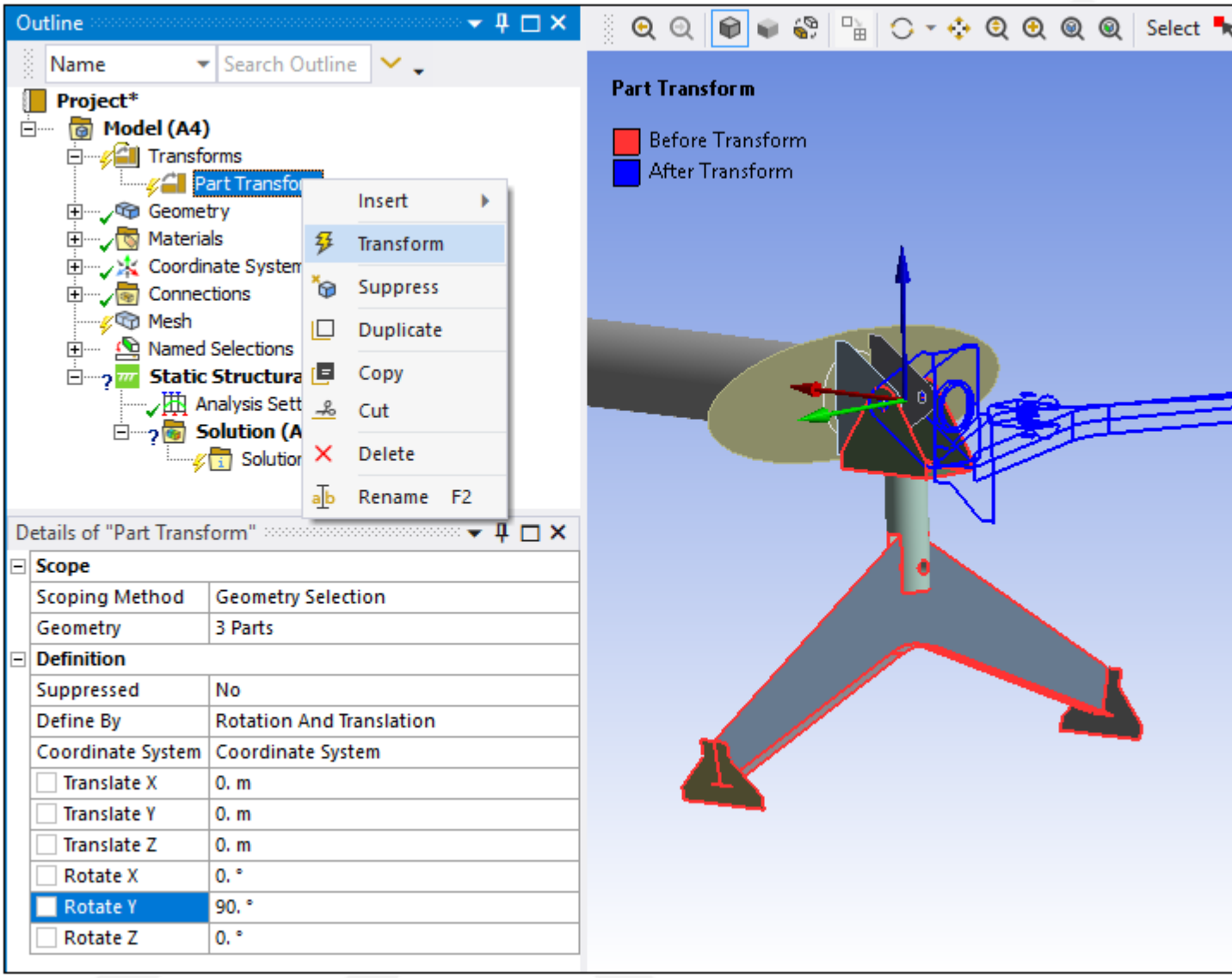
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



7. Right-click on the **Part Transform** object (or the **Transforms** object) and select **Transform**.

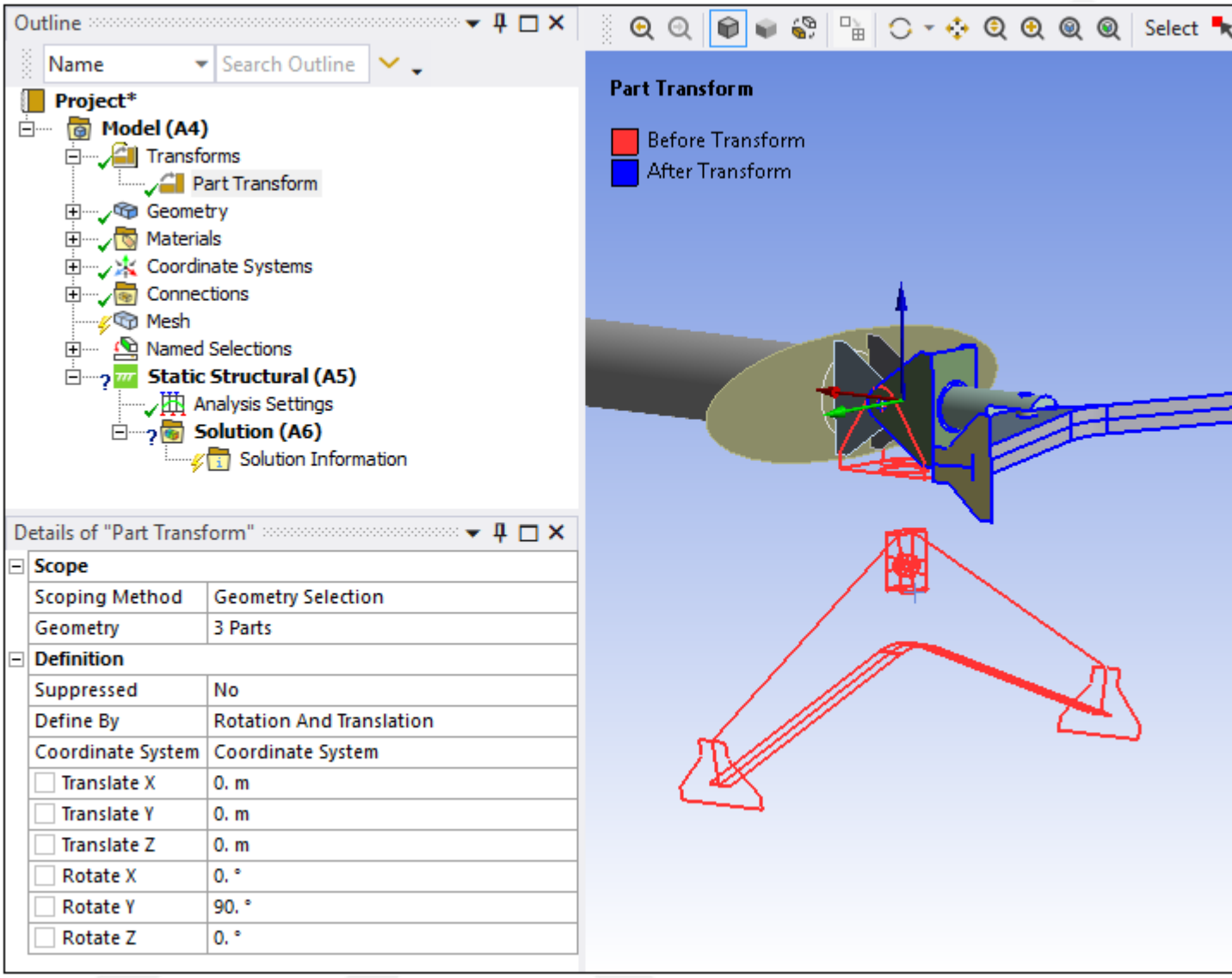
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The parts are moved and/or reoriented as specified. Before and after highlights are also shown following the transformation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Details of "Part Transform"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Parts
Definition	
Suppressed	No
Define By	Rotation And Translation
Coordinate System	Coordinate System
Translate X	0. m
Translate Y	0. m
Translate Z	0. m
Rotate X	0. °
Rotate Y	90. °
Rotate Z	0. °

8. You can perform additional transformations as needed. If you create multiple transformations, you can change their order in the **Outline** by dragging and dropping the objects onto one another.

Geometry from Deformation Results

From the **Project Schematic**, Workbench and Mechanical enable you to link and transfer the deformed geometry and mesh that are based on a Deformation (p. 2018) result, from a Static Structural, Transient Structural, Modal, Eigenvalue Buckling, Explicit Dynamics, or LS-DYNA system to a Mechanical Model system or to (any (p. 359)) Mechanical analysis system.

Important:

Your analysis must use the Mechanical APDL solver or the AUTODYN solver.

This approach is ideal for using a displaced structure as the initial geometry of a subsequent analysis. For example, you might wish to introduce imperfections to an otherwise perfect geometry to overcome convergence issues when running a nonlinear simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifications and Limitations

Note the following:

- Named Selections (face-, node-, and elemental-based) transfer from the upstream system.
- The application creates the deformed geometry using the unit system of the result file, that is, the unit system that was used during the solution of the upstream system.
- Property specifications made to the bodies of the geometry in the upstream systems transfer to the downstream system. These specifications are, by default, read-only in the downstream analysis system. A Details view category, **Transfer Properties**, provides the **Source** name of the upstream system and provides an option to change the object's properties from **Read Only**. If the **Read Only** property for a body is changed to **No**, the property specifications made in the upstream system will not transfer when data is refreshed.
- Property specifications made on the **Geometry** object of the upstream system do not transfer to the downstream system.
- Rigid and Gasket bodies do not transfer.
- Materials transfer only when the upstream and downstream systems share the Engineering Data cell.
- The application does not support the transfer of 1) **Material Assignment** objects or 2) bodies whose material is defined by a **Material Assignment** object.
- This option does not support a Nonlinear Adaptive Region (p. 1707).

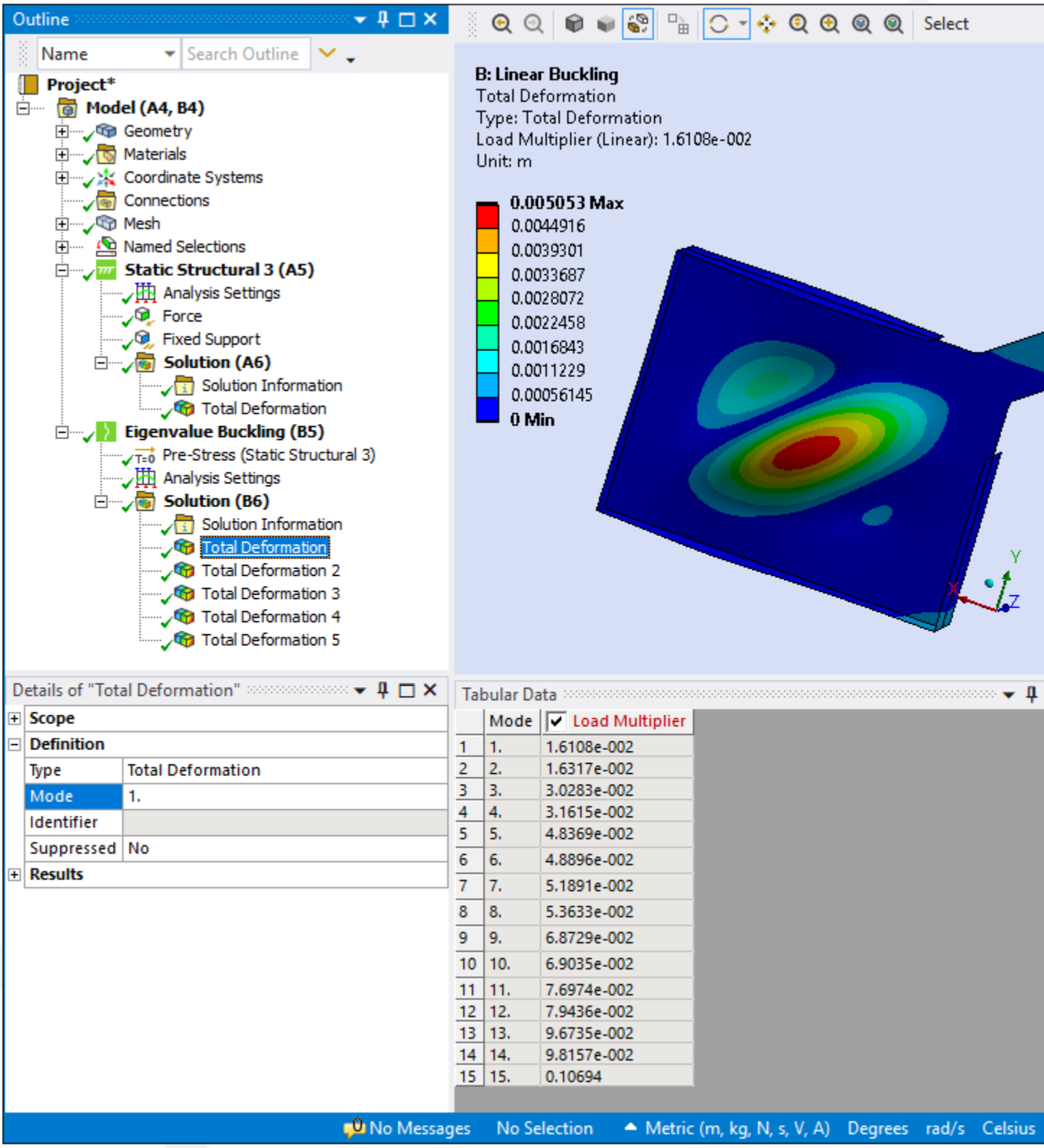
Important:

For line bodies, the nodal coordinates of the orientation nodes are not updated and remain the same as the initial geometry/mesh. Therefore, carefully validate the results of any analysis that uses the deformed geometry if the initial geometry has line bodies.

Application

1. Identify the deformation result within the solved analysis that you wish to use. An example Eigenvalue Buckling analysis is illustrated below. The result to be specified is the first result (**Mode** = **1**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

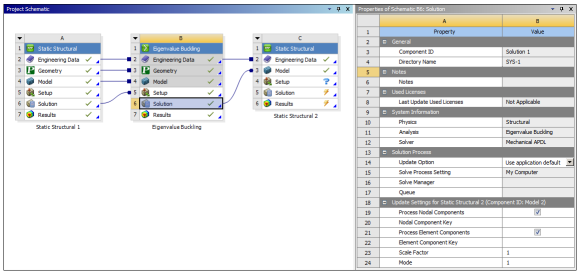


2. Return to the **Project Schematic** and link the Mechanical **Solution** cell to the **Model** cell of the downstream Mechanical system. You can create links from the **Solution** cell to multiple downstream **Model** cells.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

In the continued example shown here, the **Solution** cell is selected. Note the properties under the heading "Update Settings for Static Structural 2", the new downstream system. If multiple links are present, there is an instance of the "Update Settings for *Analysis Type*" in the properties for each linked analysis. If you change a setting in properties for a linked system, this causes the data for **all** of the other downstream systems to become obsolete and you need to refresh the data on each system.



The properties include:

- **Process Nodal Components**: Enables the upstream system to import node-based components defined in the mesh files. The application transfers the data to downstream Mechanical systems as node-based Named Selections. The application renames (p. 1000) the node-based Named Selection objects in Mechanical based on the selection made in the **Object Renaming** property.

- **Nodal Component Key**: This entry field enables you to filter and import only those node-based components that start with a specified name/string value in the mesh files. For example, you want to import only node-based components that start with the prefix string "**`nodal_*`**." Enter that string into this field and the application filters through all component names and returns only the components that begin with this key string value.

- **Process Element Components**: Enables the upstream system to import any element-based components defined in the mesh files. The application transfers data to downstream Mechanical systems as elemental-based Named Selections. The application renames (p. 1000) the element-based Named Selection objects in Mechanical based on the selection made in the **Object Renaming** property.

- **Element Component Key**: This entry field enables you to filter and import only those element-based components that start with a specified name/string value in the mesh files. For example, you want to import only element-based components that start with the prefix string "**`elemental_*`**." Enter that string into this field and the application filters through all component names and returns only the components that begin with this string value.

- **Scale Factor**: this property scales the displacements of the initial configuration. If **Scale Factor** is **`1.0`**, the full displacement value will be added to each node, **`0.5`**, half the displacement value will be added, and so on. Negative Scale Factor values subtract the displacements and reverse the direction of deformation.

- **Time**/**Mode**: this property is based on the analysis system.

  For a Static Structural, Transient Structural, or Explicit Dynamics system, you specify a **Time** setting for the upstream system's results for each downstream **Model** cell. The **Time** property options include **End Time** (default) and **User Defined**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For Eigenvalue Buckling or Modal systems, you specify the desired **Mode**, for each connection to a downstream **Model** cell. In this bucking example, it is **Mode**, and displays the result that you selected in the buckling analysis (**Mode** = 1) to use as your geometry and mesh in this subsequent analysis. You can change the desired/target result at this time (to Mode = 2, 3, etc.).

Important:

- If you enter a **Time** value that is greater than the end time of the upstream analysis, the result generated for the deformed geometry assumes the final time. If specified by **Mode**, and you enter a **Mode** value greater than the maximum number reported by the solution, geometry generation fails.

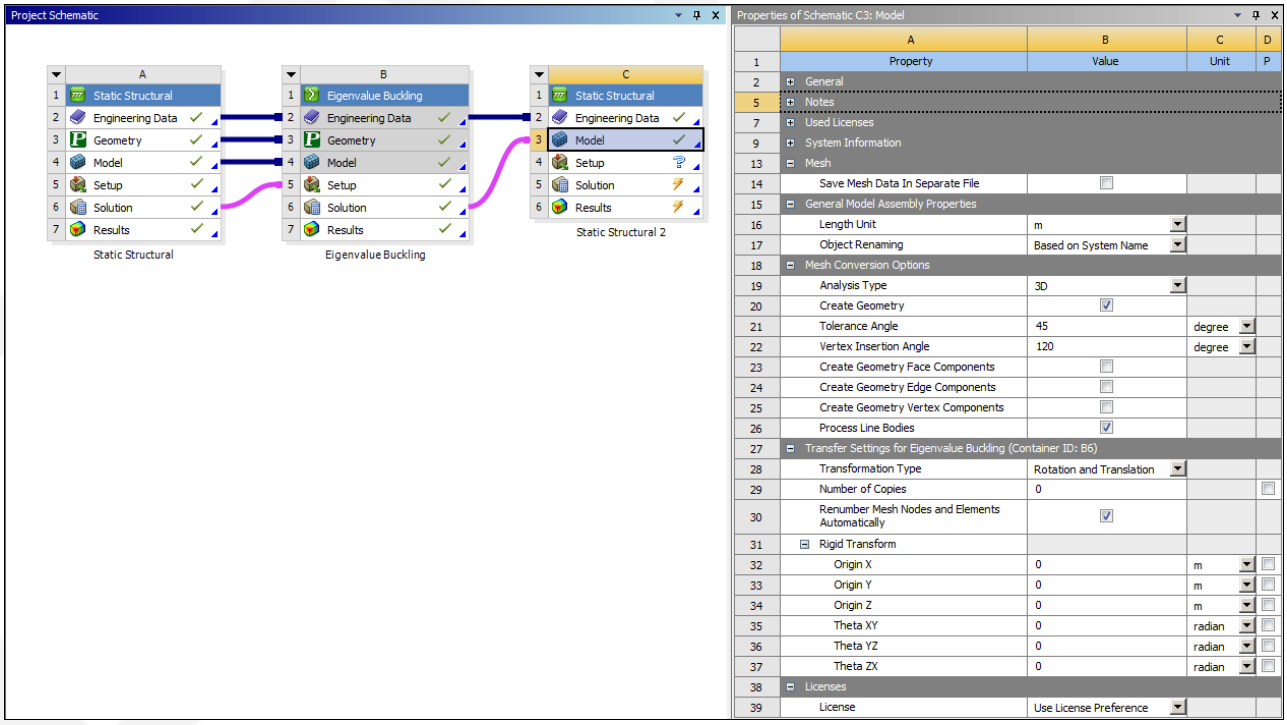
- If you enter a **User Defined** time of **0**, the application generates the geometry at the End Time.

---

3. The **Model** cell is selected in the example shown below for the new system. Specify any Model Assembly or Mesh Conversion Options on the **Model** cell of the downstream system to modify or assemble the new geometry.

Note:

If you are incorporating a deformed geometry into Model Assembly systems, you need to share the **Engineering Data** cell of the deformed geometry system with one or more of the Model Assembly systems in order to have the materials automatically transferred and set for the deformed geometry in the downstream system.



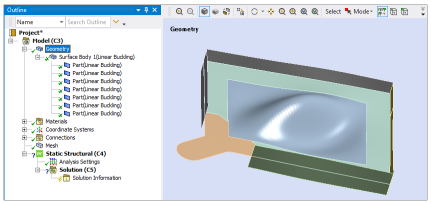
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

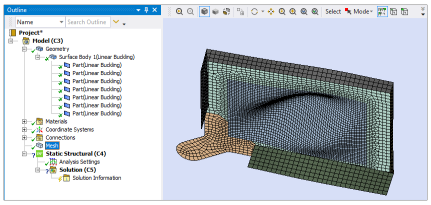
Published: 2023-10-23

4. Open the new system in Mechanical. The new geometry and the mesh are transferred.

Note:

When you are using a deformation result as a geometry, the same behavioral characteristics as well as limitations apply as those associated with importing mesh-based geometries. Refer to the **Behaviors and Characteristics** topic of in the Mesh-Based Geometry (p. 908) section of the Help for specific information.





Geometry From Rigid Dynamics Results

From the **Project Schematic**, Workbench and Mechanical enable you to link and transfer updated geometry and connections that are based on a simulation result from a Rigid Dynamics system to a Mechanical Model system or to any (p. 359) Mechanical analysis system.

This approach is ideal for using a displaced model as the initial geometry of a subsequent analysis. For example, you may wish to convert one body from rigid to flexible and perform a static analysis at a given configuration using the corresponding motion loads. It is also useful when the analysis of one single component is not accurate enough. This can be the case if contact exists on the parts of interest for the stress analysis.

Specification and Limitations

Note the following:

- The application creates the updated geometry using the unit system of the result file (the unit system that was used during the solution of the upstream system).
- Property specifications made to the bodies of the geometry in the upstream system transfers to the downstream system. These specifications are, by default, read-only in the downstream system. A Details view category, **Transfer Properties**, provides the **Source** name of the upstream system and provides an option to change the object's properties from **Read Only**. If the **Read Only** property for a body is changed to **No**, the property specifications made in the upstream system will not transfer when the data is refreshed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Property specifications made on the **Geometry** object of the upstream system do not transfer to the downstream system.
- Joints **Initial Position** property in the downstream system will be set to **Override**.
- Flexible, Line, and Gasket Bodies are not supported.
- Nonlinear spring and bushing properties are identical to those in the upstream properties, and are not re-evaluated around the new operating point.

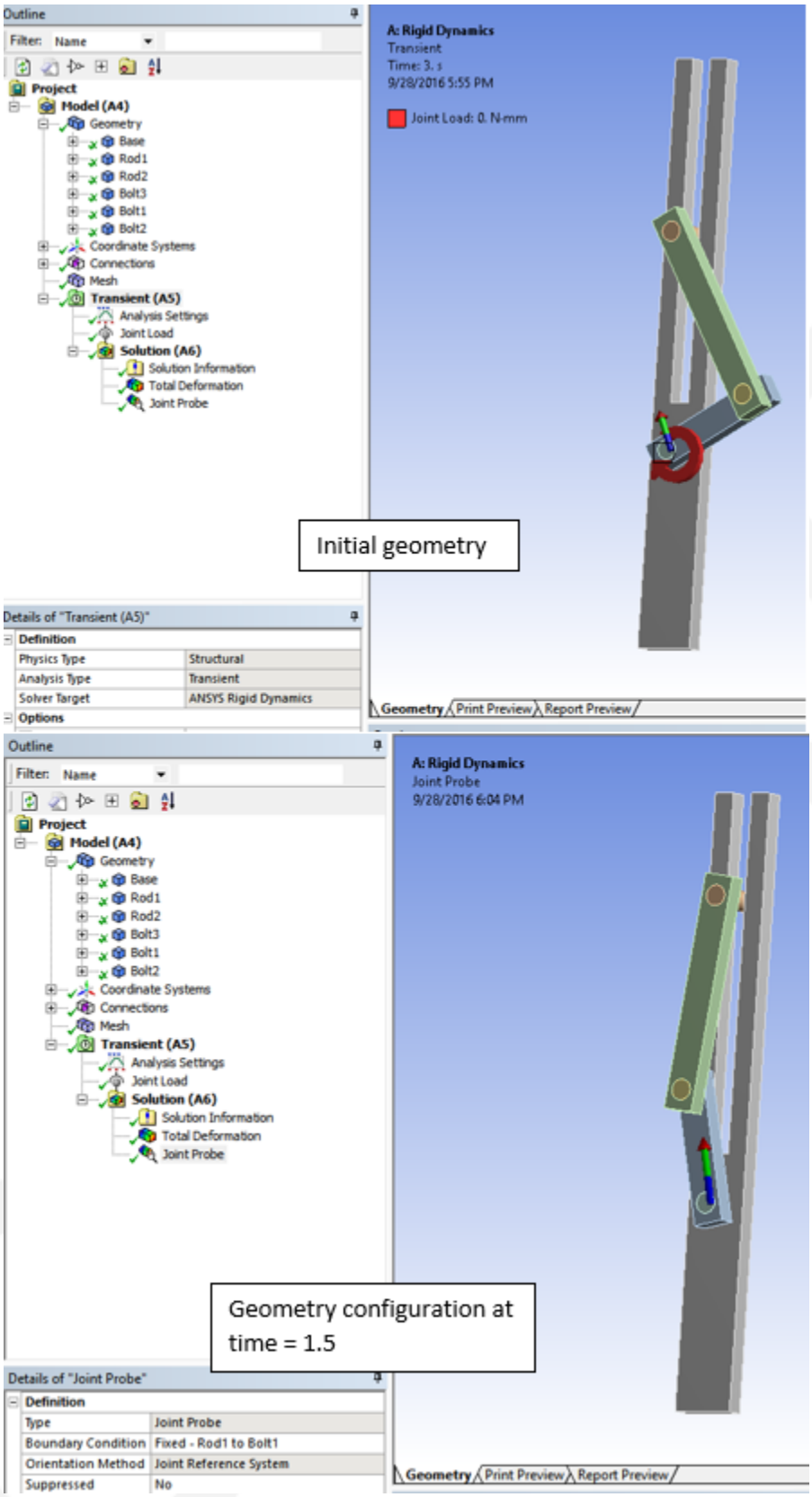
Application

1. Identify the configuration results within the solved analysis that you wish to use. An example is shown below. The configuration to be specified is the result at time = 1.5.

DRAFT

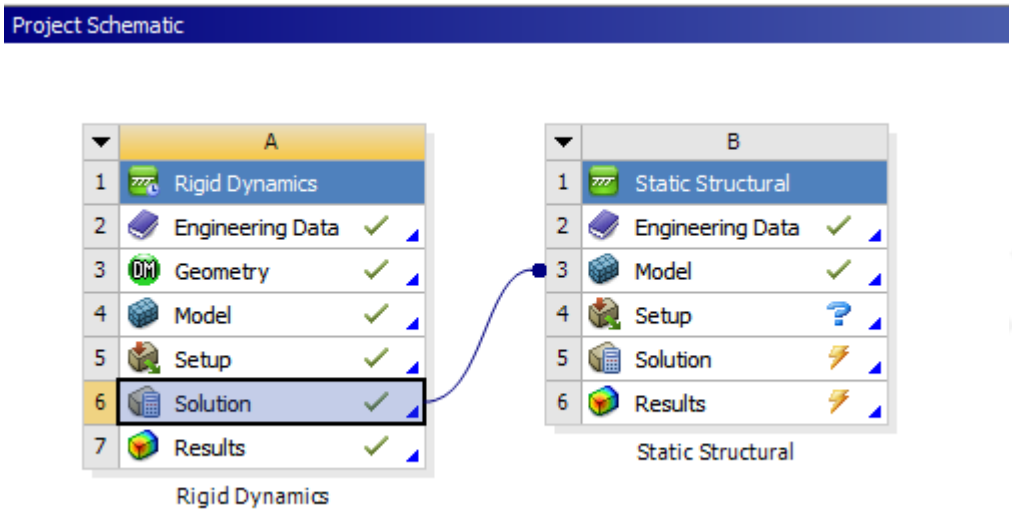
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



2. Return to the **Project Schematic** and link the Mechanical **Solution** cell to the **Model** cell of the downstream Mechanical system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Properties of Schematic B3: Model

	A	B
1	Property	Value
2	General	
3	Component ID	Model 1
4	Directory Name	SYS-1
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	System Information	
10	Physics	Structural
11	Analysis	Static Structural
12	Solver	Mechanical APDL
13	Mesh	
14	Save Mesh Data In Separate File	☐
15	General Model Assembly Properties	
16	Length Unit	m
17	Object Renaming	Based on Cell Id

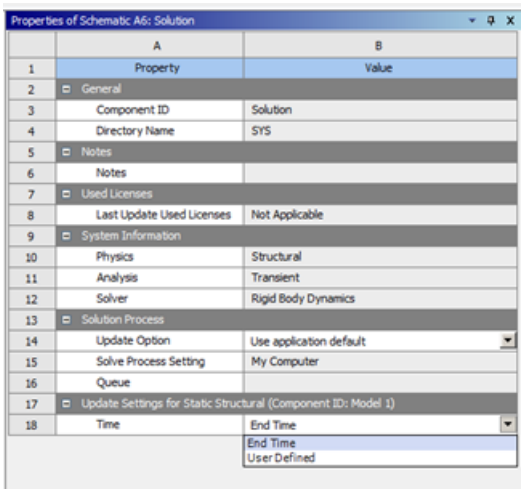
3. If you change a setting in certain properties for the linked system, the data of the downstream system becomes obsolete and you should refresh the data on the system.

   Properties which cause the downstream system to become obsolete include:

   - **Time**: For a Rigid Dynamics system, specify a **Time** setting for the upstream system's results for the downstream **Model** cell. The **Time** property can be set to **End Time** or **User Defined**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Important:

If you enter a **Time** value that is greater than the end time of the upstream analysis, the result generated for the updated geometry uses the end time. If you enter a **User Defined** time of 0, the application also uses the geometry from the end time of the analysis. This differs from the imported geometry in that the model is assembled at time = 0 to satisfy all kinematic constraints (joints, joint conditions, and user-defined constraint equations).

4. Open the new system in Mechanical. The new geometry and the updated connections are transferred.

Bolt Tools Add-on

The Bolt Tools Add-on streamlines the modeling of bolted connections in Ansys Mechanical.

For more information about this add-on, see Bolt Tools Add-on in the *Mechanical Add-ons Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Meshing Features

This section contains meshing tasks you perform in Mechanical. For additional information, see the *Meshing User's Guide*.

Performing Feature Detection

Specifying Mesh Numbering

Creating Mesh Extrusions

Associating Named Selections To Pull Geometries

Performing Feature Detection

Feature Detection is based on the principle of **Feature Based Meshing**. **Feature Based Meshing (FBM)** is a meshing set up that uses set of rules on how to treat certain geometric features while meshing. **Feature Based Meshing** has two parts:

- **Feature Recognition**
- **Mesh Treatment**

FBM performs the following on the geometry features:

1. Recognize the features like holes, fillets and more.
2. Categorize the feature based on geometric parameters or queries.
3. Defines mesh treatment per detected feature.

FBM allows you to minimize the operation performed at geometry level and create an automated or repeatable process to reduce the pre-processing time.

To perform the **Feature Detection**:

In **Mesh Context** tab, click **Feature Detection** under the **Detect** group to open the worksheet for feature detection.

or

1. Right-click **Model** object and click **Insert > Feature Detection** to open the **Worksheet** for feature detection.
2. Select the feature to be detected and provide the required options in **Feature Detection Worksheet**.
3. Right-click **Feature Detection** in the model tree and click **Detect Features** to detect the features in the worksheet.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

or

Click **Detect Features** in the worksheet to detect the selected features.

The **Named Selections** for the features are created in the model tree and the feature controls are available under the mesh object.

The Details view has the following options:

Scope

- **Scoping Method**: Allows you to select **Geometry** or **Named Selection** for scoping of the feature.

The **Feature Detection Worksheet** has the following options:

- **Name**: Allows you to provide name for the feature to be detected
- **Type**: Allows you to select the type of feature to be detected. The available types are **Shell Fillet**, **Shell Hole**, **Solid Fillet**, **Solid Hole**, **Solid Concave Fillet**, **Solid Convex Fillet**.
- **Criteria**: Allows you define the criteria applied for the feature based on their type. The available criteria for **Fillets** and **Holes** are **Radius** and **Variable Radius**.
- **Operator**: Allows you to specify the condition for the selected criteria. The available operators are **Equal**, **Not Equal**, **Less Than**, **Less Than or Equal**, **Greater Than**, **Greater Than or Equal**, **Exclude range**, **Include range**.
- **Value**: Allows you to provide value for selected criteria of the feature.
- **Angle**: Allows you to provide the angle for the selected feature.
- **Min Bound**: Allows you to provide minimum value for the selected feature.
- **Max Bound**: Allows you to provide maximum value for the selected feature.
- **Mesh Treatment**: Allows you to specify the mesh treatment to be applied on the selected feature. **Mesh Treatment** applied varies with the type of the feature. The available mesh treatment for various features are as follows:
  - **Shell Hole**: The available mesh treatments are **None**, **Fill Hole** and **Quad Layer Control**. When **Mesh Treatment** is set to **Quad Layer Control**, the non-circular holes are filtered out.
  - **Shell Fillet**: The available mesh treatments are **Mapped Meshing** and **Deviation Control**. When **Mesh Treatment** is set to **Mapped Meshing**, non-mappable faces are filtered out.
  - **Solid Hole**: The available mesh treatments are **None**, **Mapped Meshing**.
  - **Solid Fillet**: The available mesh treatments are **None**, **Mapped Meshing**. When **Mesh Treatment** is set to **Mapped Meshing**, non-mappable faces are filtered out.
  - **Solid Concave Fillet**: The available mesh treatments are **None**, **Mapped Meshing**. When **Mesh Treatment** is set to **Mapped Meshing**, non-mappable faces are filtered out.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Solid Convex Fillet**: The available mesh treatments are **None**, **Mapped Meshing**. When **Mesh Treatment** is set to **Mapped Meshing**, non-mappable faces are filtered out.

- **Add Feature Detection**: Allows you to add a new row in the feature detection worksheet.
- **Detect Features**: Allows you to detect the features with specific criteria mentioned in the worksheet.

Best Practices

The best practices to be followed while performing **Feature Detection**:

- **Feature Detection** may not detect all the curved faces as fillets or holes. In such cases, you can add or remove the faces from the named selection.
- In the **Feature Detection** worksheet, you must group the fillets or holes of the similar radius under the same **Name Selections** and set the **Mesh Treatment**. This helps you to have more control over the mesh pattern.
- You should always try to provide smaller value as detection radius range. When you provide higher value or use operators such as **Greater Than** or **Greater Than or Equal**, **Feature Detection** may detect faces as fillets or detect the sheet boundary edges as holes.
- When Mesh Treatment is set to **Mapped Meshing**, non-mappable faces are filtered out of the named selection group and provides a warning message.
- When you apply **Mapped Meshing** as **Mesh Treatment** on large number of fillets in a complex geometry with large number of faces, performance issues may occur. This could also reduce robustness of the meshing because mesh based defeaturing is not performed on the edges of the faces with mapped mesh control (except in the case of Explicit Physics Preference Patch Conforming Tet meshing). Hence, it is recommended to limit the number of fillets selected for **Feature Based Meshing** specifying specific radius ranges and focusing on bodies or locations that are most important for analysis accuracy.
- **Feature Detection** is a single operation acting on all or selected bodies. When it is re-run, the **Named Selections** created prior to the execution of **Feature Detection** are removed and new ones are created. If you want to retain the previous **Named Selections**, you can right-click and duplicate the **Named Selections** groups before executing **Feature Detection** again.

Limitations

Feature Detection has the following limitations:

- **Feature Detection** detects only circular bolt holes for solid holes.
- When you **Pull** a 2D surface with circular holes to form a 3D body, the **Feature Detection** may not detect the circular holes on the pulled surface.

Specifying Mesh Numbering

The **Mesh Numbering** object is a **Model**-level object in the tree hierarchy. You can insert this object by selecting the **Mesh Numbering** option on the Mesh Context Tab (p. 58).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The **Mesh Numbering** feature allows you to renumber the node and element numbers of a generated meshed model consisting of flexible parts. The feature is useful when exchanging or assembling models and could isolate the effect of using special elements such as superelements.

The **Mesh Numbering** feature is available for all analysis systems except Rigid Dynamics analyses.

Because this feature changes the numbering of the model's nodes, all node-based scoping is lost when mesh numbering is performed, either in a suppressed or unsuppressed state. If this situation is encountered, a warning message allows you to stop the numbering operation before the node-based scoping is removed.

You can prevent the loss of any node-based scoping by using criteria-based Named Selections, or by scoping an object to nodes after mesh renumbering has taken place. Criteria-based Named Selections scoped to nodes are supported in combination with the **Mesh Numbering** object as long as you have the **Generate on Remesh** property set to **Yes**.

To activate Node Number Compression:

By default node numbers will not be compressed to eliminate gaps in the numbering that can occur from events such as remeshing or suppression of meshed parts. This allows maximum reuse of mesh based Named Selections but can result in node numbers that are higher than required. Node number compression can be turned on by setting **Compress Node Numbers** to **Yes**. If compression is turned on, the compression will occur before any other numbering controls are applied.

To activate Mesh Numbering:

1. Insert a **Mesh Numbering** folder by highlighting the **Model** folder, then:

   a. Select the **Mesh Numbering** toolbar button.

      Or...

   b. Right-click on the **Model** folder and choosing **Insert > Mesh Numbering**.

      Or...

   c. Right-click on the **Geometry** window and choosing **Insert > Mesh Numbering**.

2. In the Details view, set **Node Offset** or **Element Offset** values for the entire assembly, as needed. For example, specifying a **Node Offset** of 2 means that the node numbering for the assembly will start at 2.

   Note:

   The **Node Offset** value cannot exceed a value that results in a node number having a magnitude greater than one (1) billion. Mesh numbering of this magnitude requires considerable processing power.

3. Insert a **Numbering Control** object by highlighting the **Mesh Numbering** folder (or other **Numbering Control** object), then:

   a. Select the **Numbering Control** toolbar button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Or...

b. Right-click on the **Mesh Numbering** folder (or other **Numbering Control** object) and choosing **Insert> Numbering Control**.

Or...

c. Right-click on the **Geometry** window and choosing **Insert > Numbering Control**.

4. Specify a part, a vertex, or a Remote Point in the model whose node or element numbers in the corresponding mesh are to be renumbered.

   a. To specify a part:

      i. Select the part.

      ii. In the Details view, set **Scoping Method** to **Geometry Selection**, click the **Geometry** field and click **Apply**.

      iii. Enter numbers in the **Begin Node Number** and/or **Begin Element Number** fields. Also, if needed, change the **End Node Number** and **End Element Number** from their default values.

   b. To specify a vertex:

      i. Select the vertex.

      ii. In the Details view, set **Scoping Method** to **Geometry Selection**, click the **Geometry** field and click **Apply**.

      iii. Enter the **Node Number**.

   c. To specify a Remote Point that has already been defined:

      i. In the Details view, set **Scoping Method** to **Remote Point**, click the **Remote Points** field and choose the specific Remote Point in the drop-down menu.

      ii. Enter the **Node Number**.

5. Right-click the **Mesh Numbering** folder, or a **Numbering Control** object, and choose **Renumber Mesh**. If the model is not meshed, it will first generate a mesh and then perform mesh numbering. The nodes and elements are numbered based on the values that you specified.

Note:

During the mesh numbering process, the user interface enters a waiting state, meaning you cannot perform any actions such as clicking objects in the tree. In addition, you cannot cancel the process once it is started and must wait for its completion. However, a progress dialog box appears to report status during the operation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mesh Numbering Characteristics

- The Mesh Numbering feature is available in both the Mechanical application and the Meshing applications.
- The **Node Offset** value cannot exceed a value that results in a node number having a magnitude greater than one (1) billion. Mesh numbering of this magnitude requires considerable processing power.
- Geometry selection is part-based, not body-based.
- Selecting **Update** at the **Model** level in the **Project Schematic** updates the mesh renumbering.
- The Solve is aborted if mesh renumbering fails.
- Whenever a control is changed, added, or removed, the mesh renumbering states are changed for all controls where mesh numbering is needed.
- When exporting mesh information to Fluent, Polyflow, CGNS, or ICEM CFD format, the last status is retained at the time of export. If renumbering has been performed, the mesh is exported with nodes and elements renumbered. If not, the original mesh numbering is used.
- Mesh renumbering of a Point Mass is not supported.
- The **Convergence** (p. 1852) object is not supported with **Mesh Numbering** folder.

Note:

Be cautious when deleting the **Mesh Numbering** folder. Deleting this folder leaves the mesh in the numbered state that you specified. There is no way to know that the existing mesh has been renumbered.

Mesh Numbering Suppression Characteristics

For **Mesh Numbering**, the suppression feature operates differently. Rather than excluding the object when the **Mesh Numbering** object is suppressed, the mesh numbering instead returns to the original numbering. That is, it resets and updates the input deck's contents. This change can affect analysis operations.

As a result, restrictions have been implemented, and Mechanical no longer supports suppression of the **Mesh Numbering** object. For legacy (v14.5 and earlier) files, an error is generated in the **Message Window** if suppressed **Mesh Numbering** objects are present. You can continue your analysis by manually changing the **Suppressed** property setting to **No**, but the change is then permanent. The application will not allow you to return this setting to **Yes**.

Creating Mesh Extrusions

The **Pull** feature enables you to extrude and revolve element faces, geometric faces, or scoped (surface only) bodies from a surface or a solid mesh. The feature also enables you to create a surface coating on solid bodies, the faces of 3D bodies, element faces (requires mesh), and the edges of 2D axisymmetric surface bodies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Jump to a section topic using the following links:

Overview (p. 1039)
Background (p. 1041)
Application (p. 1042)
Limitations (p. 1042)

Overview

The menu for the feature is displayed when you use the context (right-click) menu of the Mesh or Mesh Edit objects or using the drop-down menu on the Mesh Context Tab (p. 58). **Pull** menu options include:

- **Extrude**: Create specified layers of solid elements based on height input. Supported scoping includes body (surfaces only), faces, or element faces.
- **Revolve**: Create solid element based on angle of revolution. Supported scoping includes body (surfaces only), faces, or element faces.

Note:

For the **Extrude** and **Revolve** options, you can automatically apply your extrusion or revolution to Named Selections scoped to the **Pull** body using the **Associate To Pull Geometry** property of the **Mesh Edit** object. See the next section, Associating Named Selections To Pull Geometries (p. 1043), for the steps to use this feature.

- **Surface Coating**:

  3D Body

  For a three-dimensional body, you use this option to create a coating of shell elements on top of a face of an existing body. This can include a body created by the **Extrude** and **Revolve** options. The mesh of the new surface body matches the underlying solid body. You can then use this body in an analysis as you would any body. For example, you could scope a result to it.

  2D Surface Body

  You can scope the **Surface Coating** option of the Pull object to the edges of a two-dimensional axisymmetric surface body to create a line body from the underlying mesh. You can use the generated line body to study the influence of coatings (such as paint) on these axisymmetric structures. These line coatings are represented by the elements SHELL208 (lower order) and SHELL209 (higher order). For the surface body (under Geometry object) that you scope to, the following property settings are required:

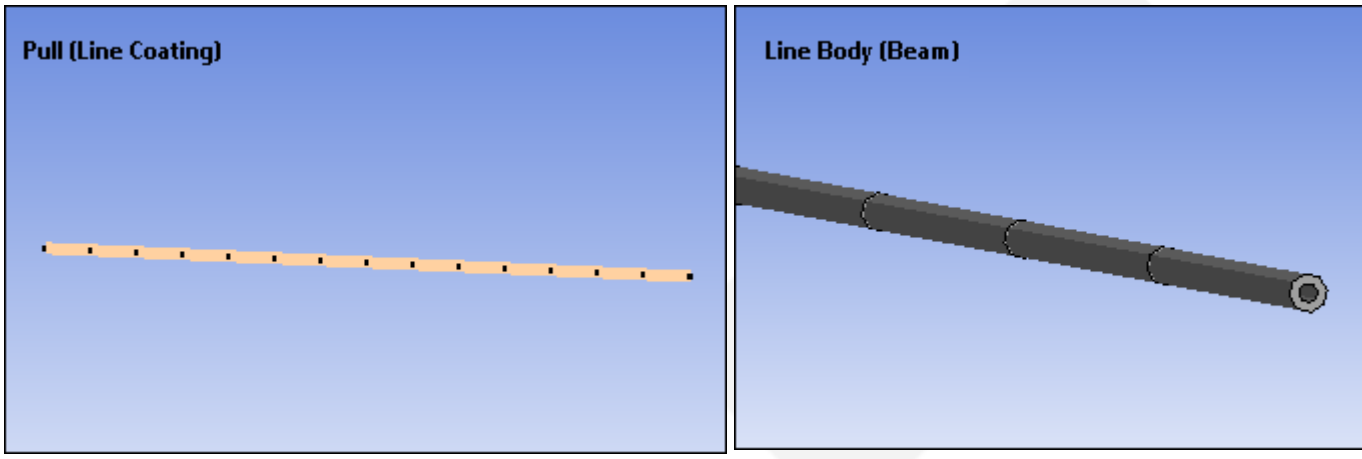
  – The **Dimension** property must be set to **2D**.

  – The **2D Behavior** property must be set to **Axisymmetric**.

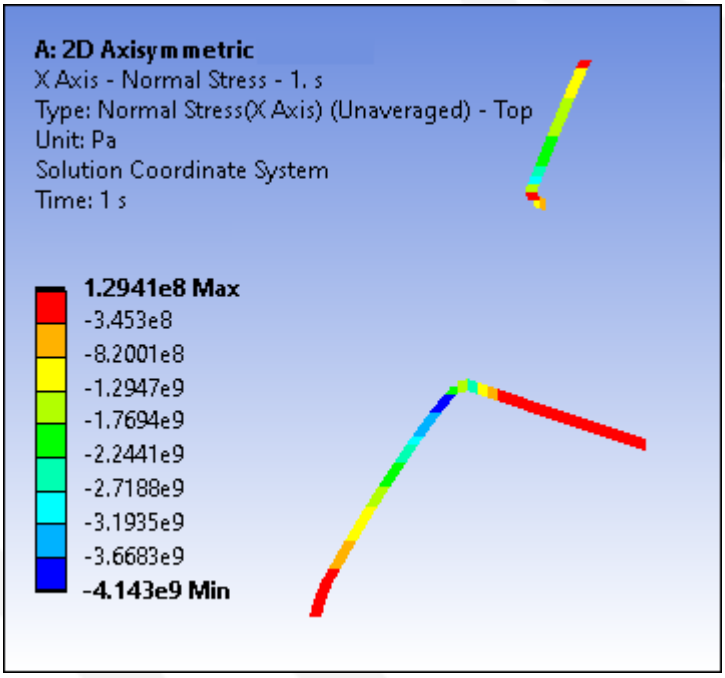
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Line Coating Display

As illustrated, unlike beams, a **Line Coating** has no visible element expansion (no visible cross section). It displays as a line only.



As show, this also applies to stress and strain results (p. 2026) scoped to a **Line Coating**. The result displays as a colored line.



In addition, the **Position** property displays for stress and strain results scoped to a **Line Coating**. For the option **Top/Bottom**, the application only displays Bottom results and generates an associated Message.

Important:

- For the new body created under the **Geometry** folder, the application ignores the value of the **Thickness** property when the **Stiffness Option** property is set to **Stress Evaluation Only**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

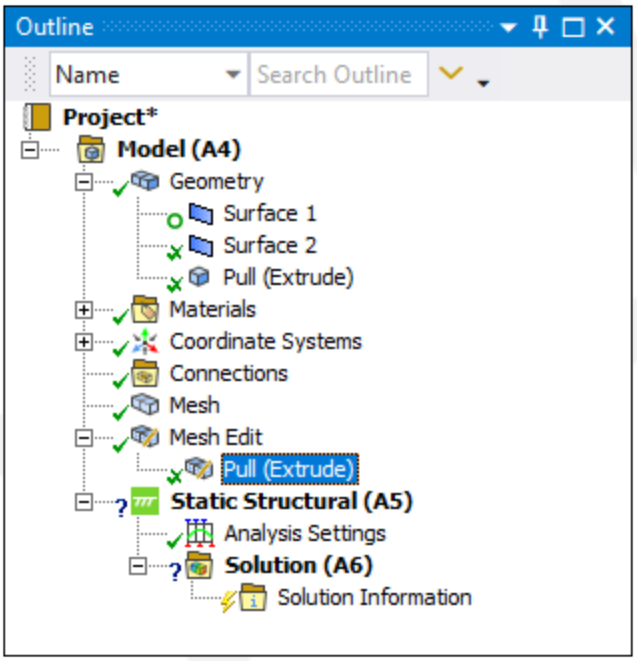
– If you use the **Clear Generated Data** option on a surface or line coating, the application deletes the generated pull body but retains (but does not reset) the specified value of the **Thickness** property and applies it when you regenerate the Pull body. Note that this retention happens only in the current active Mechanical session.

When you have defined the desired type of pull, you select the context (right-click) menu option **Generate**. This action creates the mesh of the corresponding extrude/revolve/coating. This action also creates a corresponding body, based on the pull type, under the **Geometry** object. You can then use this body as you would use any body for your analysis, however, because it was generated from mesh data, certain limitations exist. For example, if you encounter a scoping issue, try using mesh-based scoping, such as Element Face selection, to address such issues. See the Limitations (p. 1042) topic below for more information.

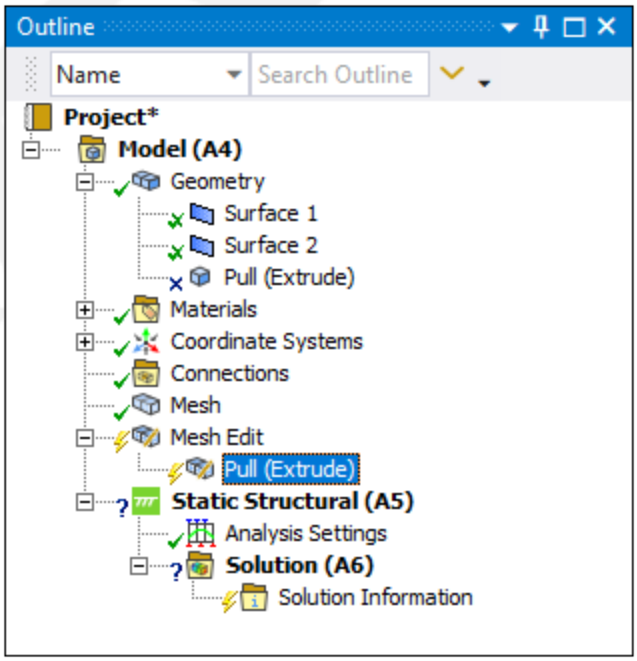
Note:

When you scope a **Pull** object to a surface body and generate a 3D mesh using either the **Extrude** or **Revolve** method, the original surface body is marked as inactive using a circle symbol as shown below. This body becomes hidden in the Geometry window. The circle symbol also indicates an inactive body that does not participate in the analysis. The application does not write the body's definition, or any Named Selections scoped to it, to the input file. In addition, the body does not contribute to the mass or volume of the geometry.

Generated Pull



Cleared Pull



Background

The ability to create a volume mesh from a surface mesh is beneficial for the design and simulation of structures such as turbo machines and Printed Circuit Boards (PCB). It is common to first design a base surface, as it is easier to generate a surface mesh, and then to extrude a small layer of volume elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

from this base mesh. Another example would be creating a 3D sector from a 2D surface body using the **Revolve** option and then perform a cyclic symmetry analysis. And you can scope the **Surface Coating** option to the face of a solid body and create a shell body whose mesh matches with the underlying solid body.

Application

See the Pull section of the *Meshing User's Guide* for a more detailed description about how to use this feature, its properties, and capabilities.

Limitations

The **Pull** feature is a mesh operation that generates volume mesh. Once created, you can scope the faces, edges, and vertices of the volume to other objects, such as loads, supports, and results. However, there are certain limitations associated with the pull-generated body as compared to an actual 3D body. Review the pull feature limitations listed below. You should also see the Pull section of the *Meshing User's Guide* for additional information.

Review the following feature limitations:

- When you regenerate a **Pull** object, any other object scoped to geometric entities (face/vertex etc.) of the pull-generated body are reassociated. But any mesh-based scoping, such as element face or nodes, are automatically cleared.
- Linear Periodic (p. 1128) and Periodic Region (p. 1131) objects are not supported when scoped to a body generated from the **Pull** operation.
- If you scope a **Pull** body to a Cyclic Region (p. 1134), the **Low Boundary** property must be scoped to the face(s) adjacent to the original shell bodies and the **High Boundary** property must be set to the faces opposite to the Low Boundary in the direction of the Pull. Selecting any other faces may lead to a solver error.
- You cannot scope Symmetry objects to **Pull** generated bodies in combination with the other bodies of your model.
- Joint coordinate systems are not automatically oriented when scoped to **Pull** bodies.
- Part Transformations (p. 1014) are not supported for pull-generated bodies or the bodies scoped to a **Pull** object after it is generated. You must perform all transformations involving pull bodies prior to generation.
- If you scope a load to a **Pull** body and need to specify a vector direction, use the Element Face selection to set the direction. Geometric entities are not supported for this capability.
- Convergence is not supported when **Pull** objects are present in the model.
- For a shell body generated using the **Pull** > **Surface Coating** option, note the following limitations:
  - By default, when the application is creating the input file, the element adopts a full-integration scheme (KEYOPT(3) = 2) for low order mesh (element with no mid-side nodes). This is different than the Surface Coating (p. 903) feature specified via the **Geometry** object where (input file entries for) the element use the reduced-integration scheme (KEYOPT(3)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

= 0) for low order mesh. Therefore, you may see a difference between the results based on a Surface Coating created via Pull versus the Surface Coating feature specified via the Geometry object.

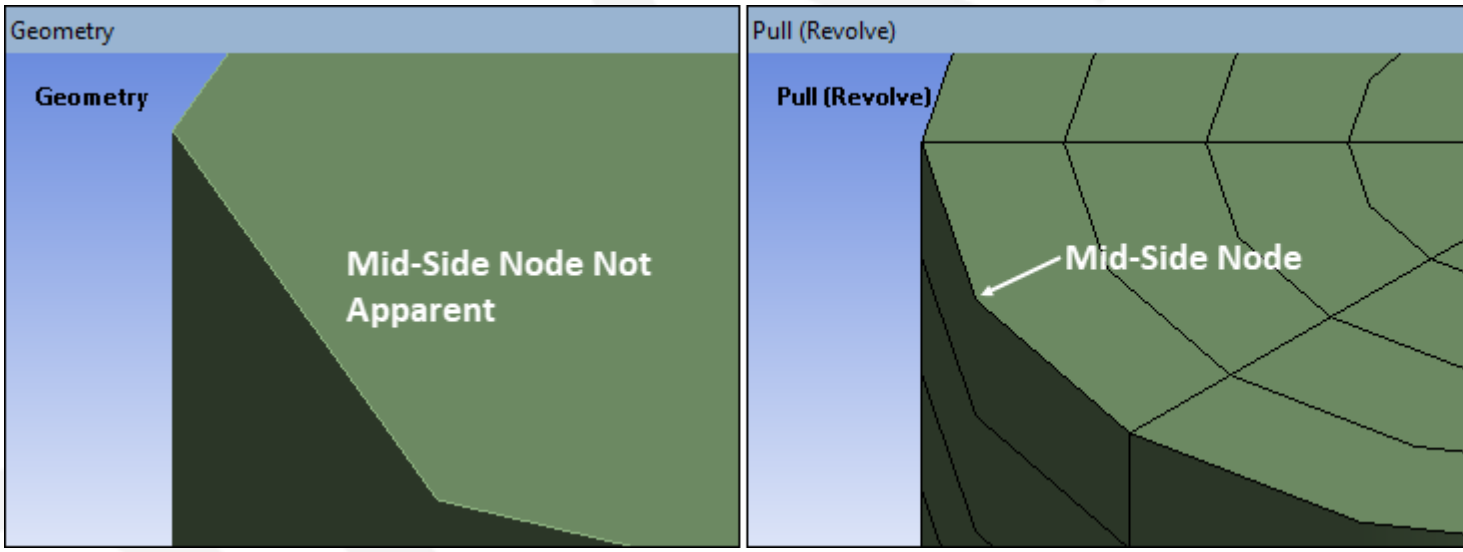
See the SHELL181 element section for more information regarding default KEYOPT(3) settings for shells.

– The surface added using the Surface Coating is a separate part in the Geometry group rather than a multibody part with the solid body it was generated from. As a result, you cannot apply loads to vertices. You should apply the loads to the vertex of the underlying solid body.

– If you set the **Stiffness Option** property of a Pull generated Surface Coating body to **Stress Evaluation Only** (default setting), loads using the **Direct** setting for the **Applied By** property are not supported.

- Because the surfaces of **Pull** generated bodies are constructed from the underlying mesh, you cannot scope spherical- and cylindrical-based boundary conditions, such as a **Cylindrical Support** or an **Absorption Element**, to these bodies.

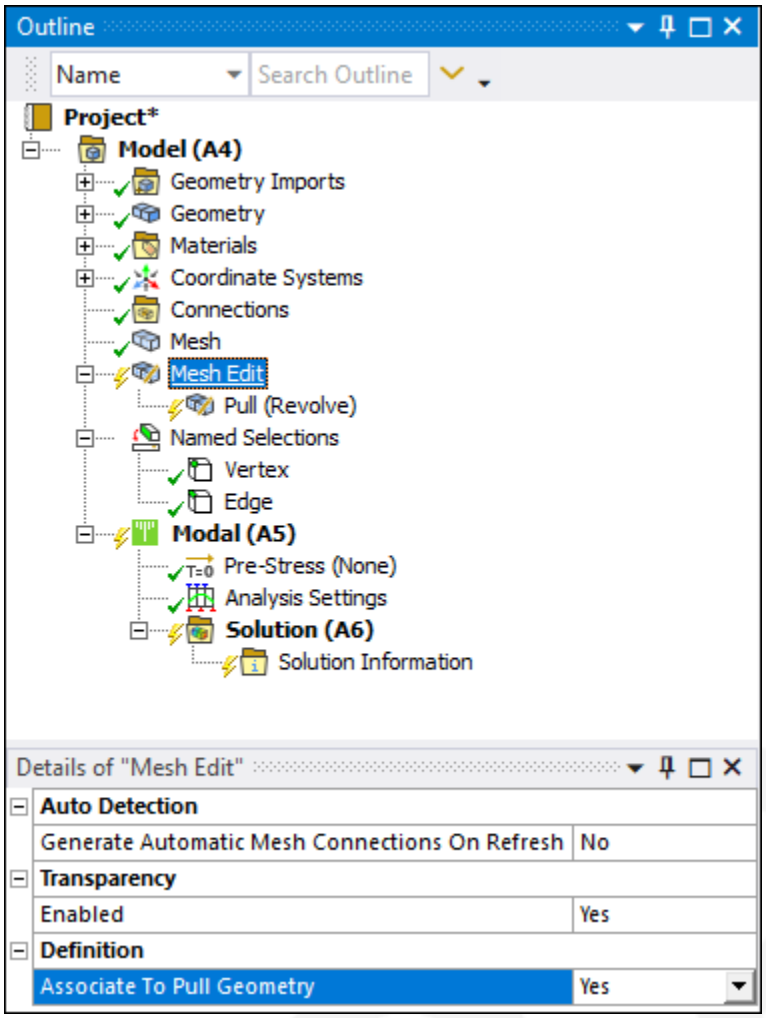
- There is currently a display limitation for bodies generated using the **Pull** > **Revolve** feature for higher order elements. You may notice when viewing a Pull Revolve generated body that it's geometric outlines does not appear to pass through the mid-side nodes of the generated mesh. This is a display issue and does not affect your results as illustrated below.



Associating Named Selections To Pull Geometries

The **Mesh** (or **Mesh Edit**) object **Pull** feature enables you to extrude and revolve element faces, geometric faces, or scoped (surface only) bodies from a surface or a solid mesh. In addition, you can automatically apply your extrusion or revolution to Named Selections scoped to the **Pull** body using the **Associate To Pull Geometry** property of the **Mesh Edit** object. Options for this property include **Yes** and **No** (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Go to a section topic:

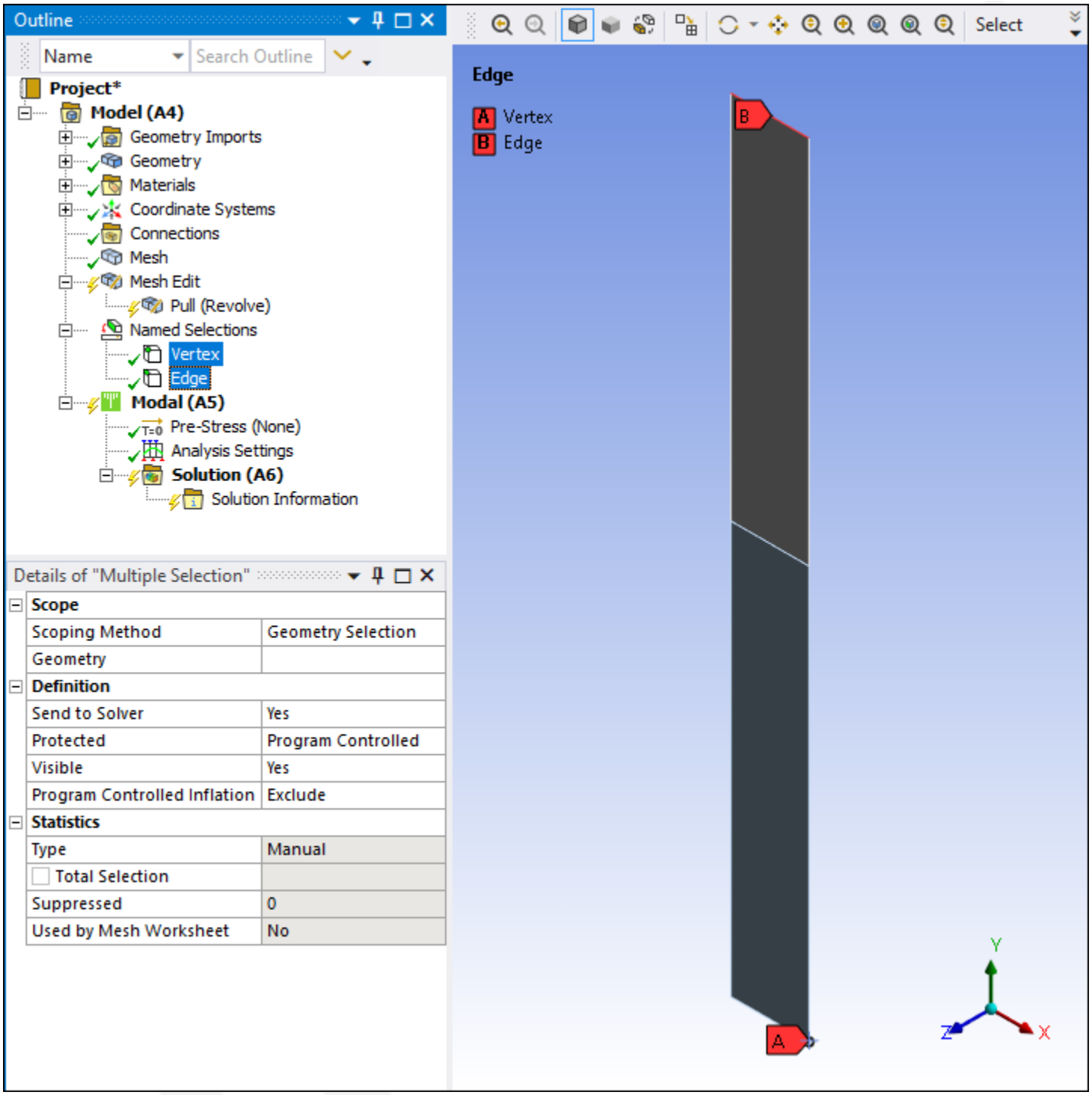
- Vertex and Edge Transformations (p. 1044)
- Shell Body Transformation (p. 1047)
- Limitations (p. 1048)

Vertex and Edge Transformations

Consider the system shown below that includes a vertex- and edge-based Named Selection.

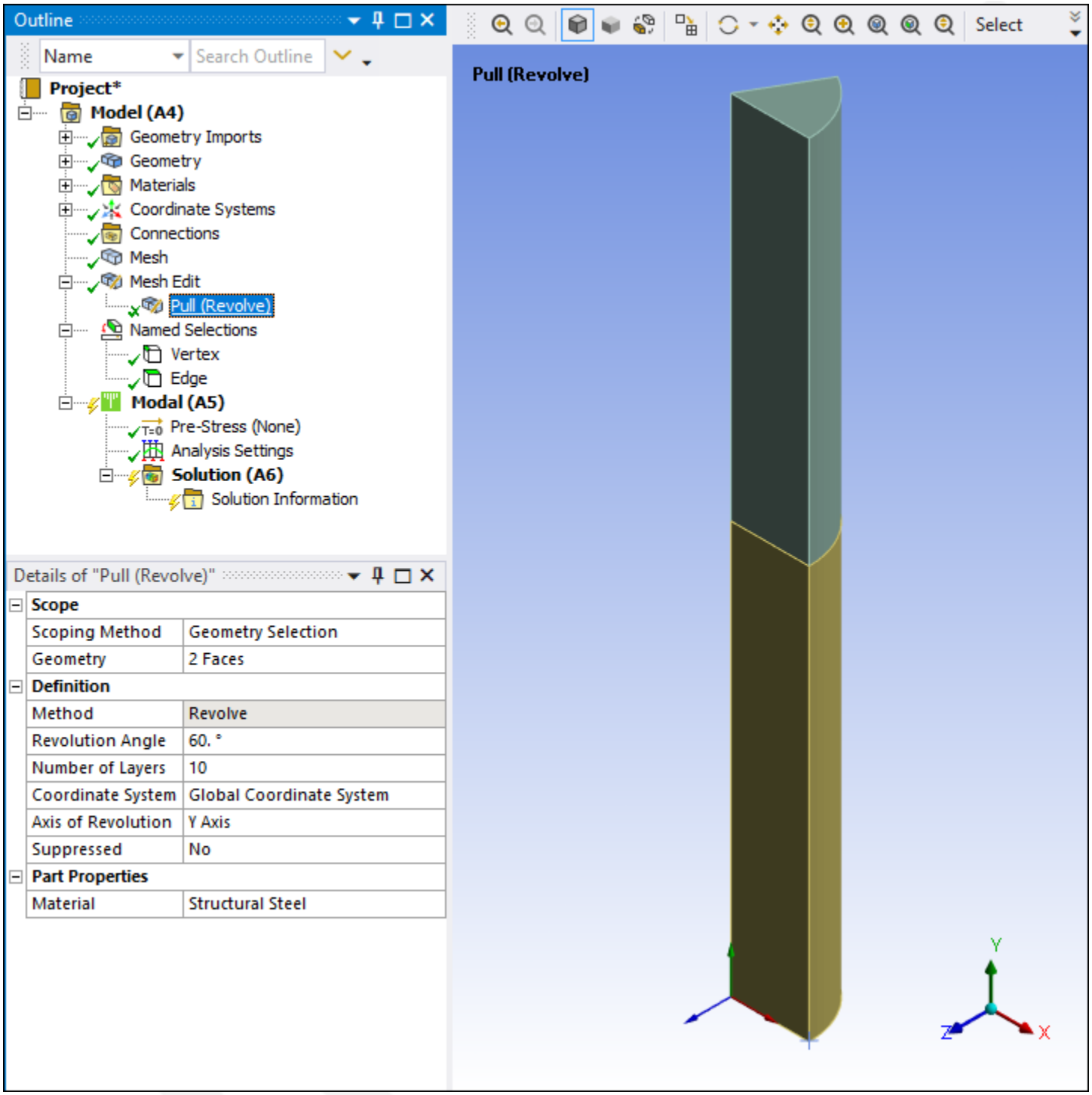
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Assume that you have set the **Associate To Pull Geometry** property set to **Yes** and generated a **Pull Revolve**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

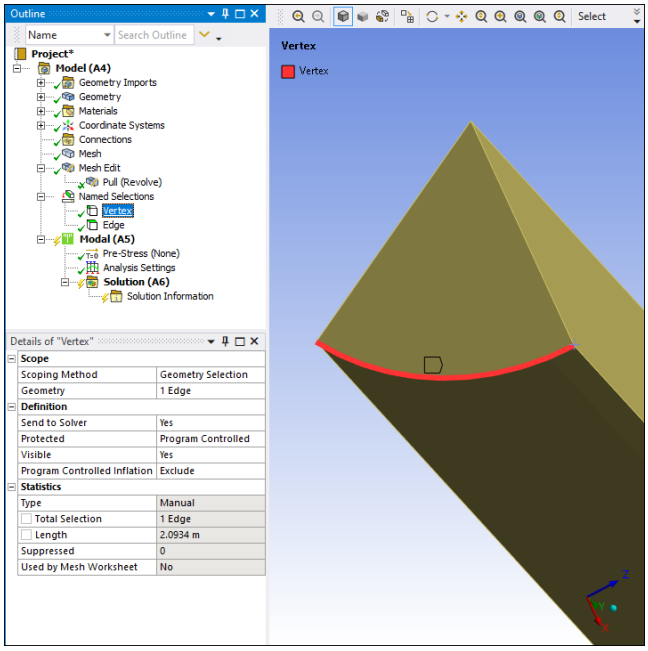


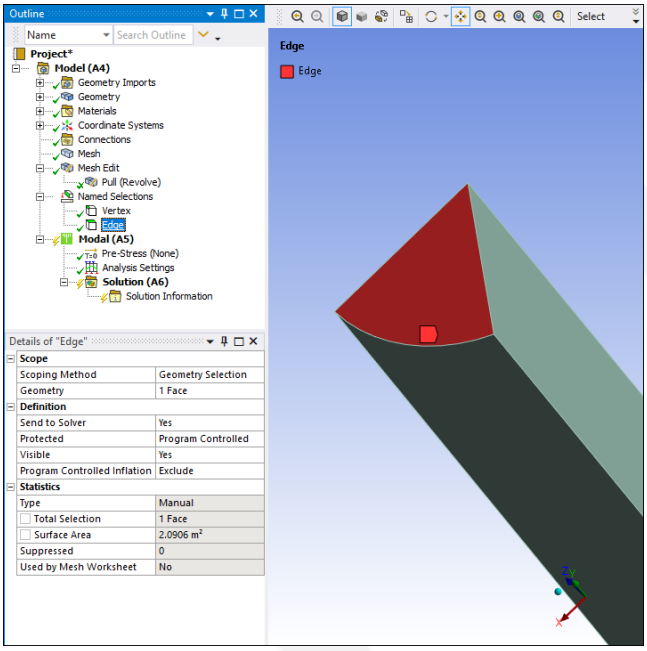
As illustrated below, the application automatically applies the transformation to the Named Selections. The vertex Named Selection is converted to an edge and the edge Named Selection is converted into a face.

Transformed Vertex **Transformed Edge**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

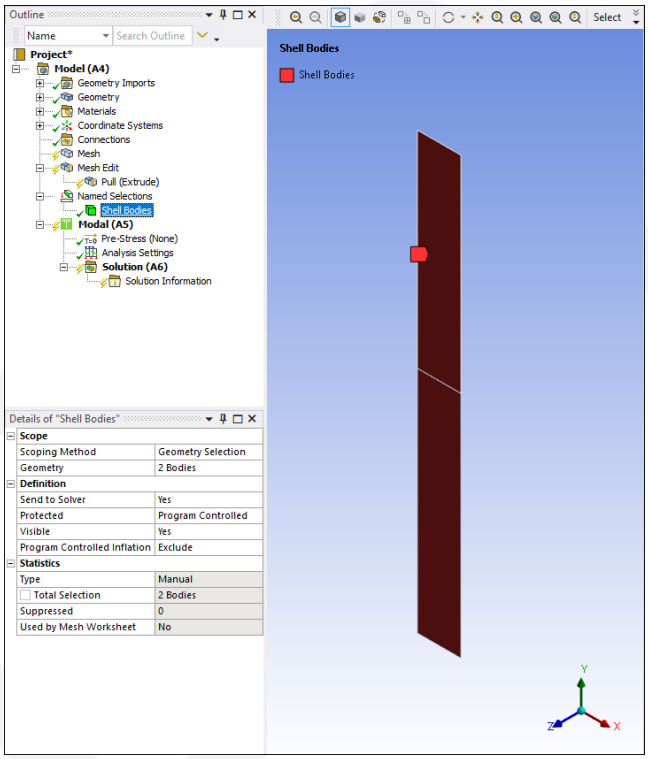




Shell Body Transformation

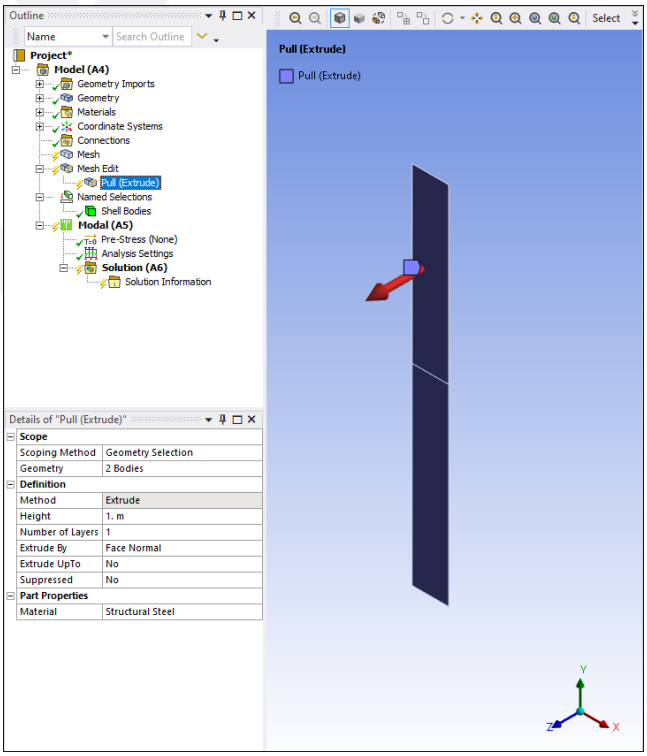
In addition to the vertex and edge transformations above, you can extrude and revolve surface bodies. This transforms surface body Named Selections into a solid body, as illustrated below.

Pull Scoped to Surface Bodies



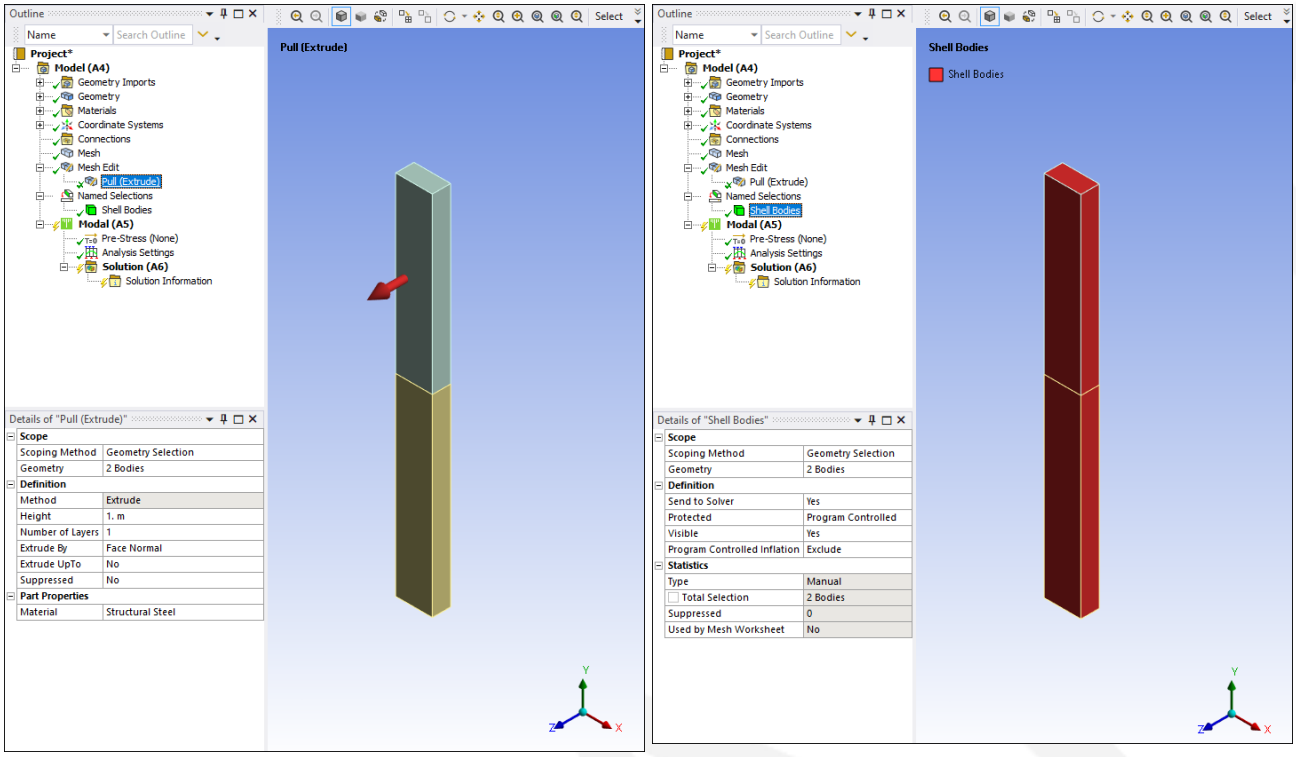
Generated Pull

Named Selection Scoped to Surface Bodies



Transformed Named Selection

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Limitations

If the **Material** property on the **Pull** object is set to **None** and the **Associate To Pull Geometry** property of the **Mesh Edit** object is set to **Yes**, the application automatically assigns the material of the original profile bodies to the pull geometry generated under the **Geometry** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifying Materials

The following topics explain the use of material features:

- Materials (Group)
- Material
- Material Assignment (p. 1049)
- Material Plot (p. 1051)
- Material Combination (p. 1052)
- Imported Material Fields (p. 1055)
- Unique Material-Based MAPDL Commands (p. 1056)

Material Assignment

The **Material Assignment** feature provides a convenient way to assign a material to multiple bodies and control its behavior, like nonlinear effects, thermal strain calculation, reference temperature etc. It also allows for a convenient way for users to edit material properties through a Commands (APDL) object.

Important:

- If you scope a **Material Assignment** object to elements, the application does not validate that the assigned material includes all required material properties to assure a successful solution. As a result, the following results and result tools that use these material properties, do not produce correct result values. These result types only support body scoping.
  - Error (Structural, Thermal, and Magnetic)
  - Strain results
  - Stress Tool results
  - Fatigue Tool results
- The **Material Assignment** feature is not supported by LS-DYNA.

To insert and apply a **Material Assignment** object:

1. Select either the Materials object or one of its material child objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. Select the **Material Assignment** option from the **Materials** Context tab or, right-click the **Materials** object and select **Insert** > **Material Assignment**. A **Material Assignment** object is placed in the **Outline**.

3. Select your **Scoping Method**: either **Geometry Selection** or **Named Selection**.

4. Select the desired Bodies or Elements or select an appropriate user-defined Named Selection.

5. Specify the desired material using the **Material Name** property. This property behaves just like the Material Assignment (p. 345) property of a body or part. And, it can be designated as a parameter.

6. Specify the following as needed:

   - **Nonlinear Effects** (p. 345)
   - **Thermal Strain Effects** (p. 346)
   - **Reference Temperature** (p. 344)

7. As needed, modify the **Field Coordinate System** setting. Using the **Default Coordinate System** setting, all of the specified bodies will have the same coordinate system.

The use of material assignment also affects how materials are sent to the solver. By default, when Mechanical creates an input file, the application assigns a unique material identifier (**`matid, #`**) to each body of the model regardless of the material assigned to the body/part. Here is an example input file. Each part is assigned the material Structural Steel and has its own material identifier.

```
!*********** Model Summary ********************
!Part 12,     Structural Steel,   matid,    1
!Part 13,     Structural Steel,   matid,    2
!Part 14,     Structural Steel,   matid,    3
!Part 133,    Structural Steel,   matid,    4
!Part 134,    Structural Steel,   matid,    5
!Part 135,    Structural Steel,   matid,    6
!********************** End Model Summary *****************
```

When material assignment object is used, all the scoped bodies are assigned the same material identifier, as illustrated below (**`mat id, 43`**). For this example, you can see the default behavior of material assignment as well as the **Material Assignment** feature.

```
!*********** Model Summary ********************
!Part 12,     Structural Steel,   matid,    1
!Part 13,     Structural Steel,   matid,    2
!Part 14,     Structural Steel,   matid,    3
!Part 133,    Titanium Alloy,     matid,    43
!Part 134,    Titanium Alloy,     matid,    43
!Part 135,    Structural Steel,   matid,    6
!********************** End Model Summary *****************
```

Note:

- When specifying the same material to multiple bodies using Material Assignment object, the application can no longer identify the bodies using the material identifier (**`matid`**) in the solver input file. In this case, you can use the **`typeids`** list to identify a body. The identifier **`typeids`** is a one-dimension array parameter that you can use to access the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

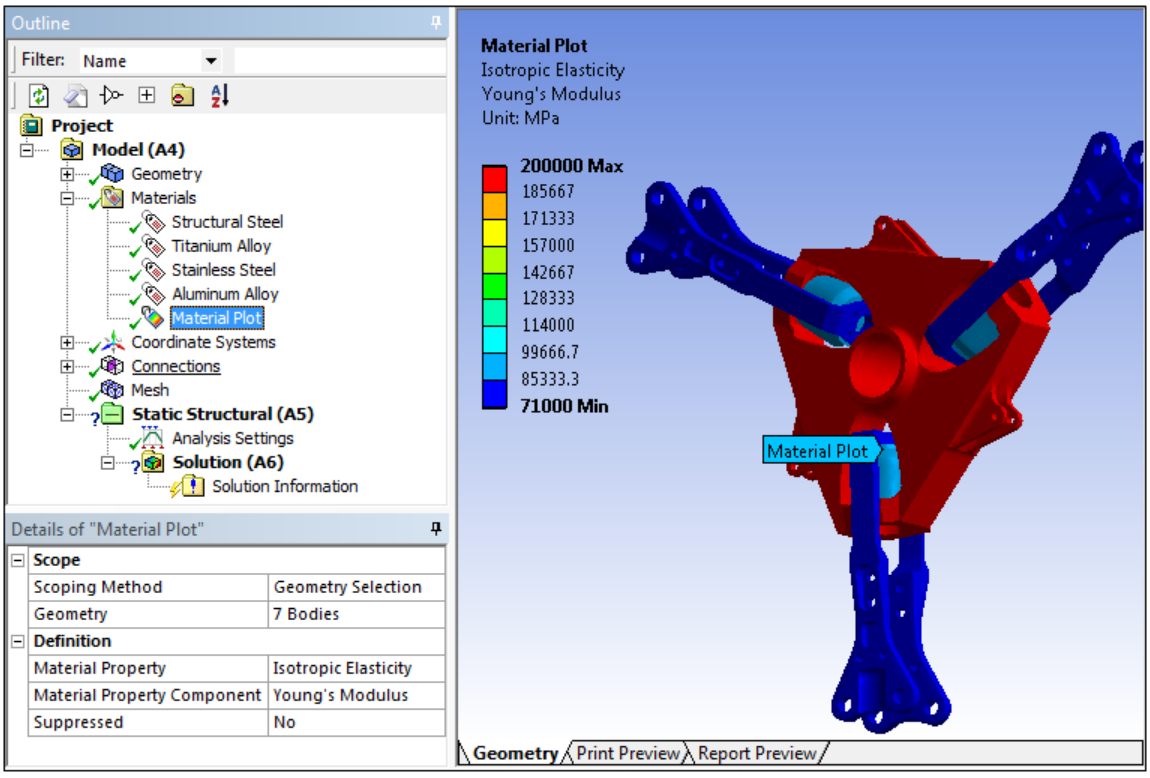
type numbers for a body. You can access type numbers using a subscript (enclosed in parentheses) to identify the required item of the array. For example, to access the first type number for the body use **`typeids(1)`**.

- The Material Assignment feature cannot be used with either Layered Section or Imported Trace.

Material Plot

The **Material Plot** feature enables you to plot the materials of your model as contours using geometry or Named Selection scoping. The supported geometric entities include Body, Face, Edge, and Element. Once you specify your geometry selections, right-click the object and select **Generate Material Plot**. The **Geometry** window displays contours on your model that correspond to the material (properties) of the specified geometry/geometries/named selections. An example is illustrated below that includes Isotropic Elasticity, Young's Modulus, and the unit of measure.

You insert this object from either the **Materials** Context tab, or by right-clicking the **Materials** object and selecting **Insert** > **Material Plot**.



See the Material Plot object reference page for a description of the object's properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Temperature Dependent Properties

If a material property depends on temperature, then the values are calculated at the reference temperature of the body. If the reference temperature of the body is set per environment, then a default value of 22 °C is chosen.

Properties Dependent on Field Variables

Material properties can be dependent on user-defined field variables. User-defined Field Variables are either imported through **Imported Material Field** objects or a single default value is defined in the Engineering Data Workspace. As such, your material property values are calculated based on the definition of one of these user-defined field variables. Field Variable interpolation algorithms (**Algorithm Type**) can be modified in the Engineering Data Workspace and the change is reflected in your contour plot.

Material Combination

Using the materials available in the Engineering Data workspace, the **Material Combination** feature enables you to assign a combination of different materials, specifically their material properties, to a body or part.

Go to a section topic;

- Guidelines (p. 1052)
- Temperature Dependence (p. 1053)
- Application (p. 1053)

Guidelines

The material properties of combined materials are used to determine effective material properties. The effective material properties are calculated as weighted averages based on the specified percentage of the combined Materials. Additionally:

- For a material property to be used in combination, it must be defined for each Material specified in the Material category and the combined materials must share at least one such material property. Currently, the following material properties are available for combination:
  - Thermal Conductivity
  - Density
  - Specific Heat
  - Young's Modulus
  - Poisson's Ratio
  - Coefficient of Thermal Expansion
  - Orthotropic Elasticity
  - Orthotropic Conductivity

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Only linear materials are supported.

Temperature Dependence

The material combination feature supports temperature-dependent material properties. When you create a combination, the application combines the temperature (range) values of each material linearly. This means that, for the materials you select, the application uses a temperature range for the temperature values that all of the selected materials share.

Each material property listed under the **Common Material Properties** category in the Details of the **Material Combination** object, such as **Density**, must include an associated temperature range. If this data is not available, the application does not specify a value. Furthermore, if the temperature values produced during the simulation go outside of the ranges for a material property, as specified in the **Engineering Data** workspace, the application treats these temperature values as a constant value.

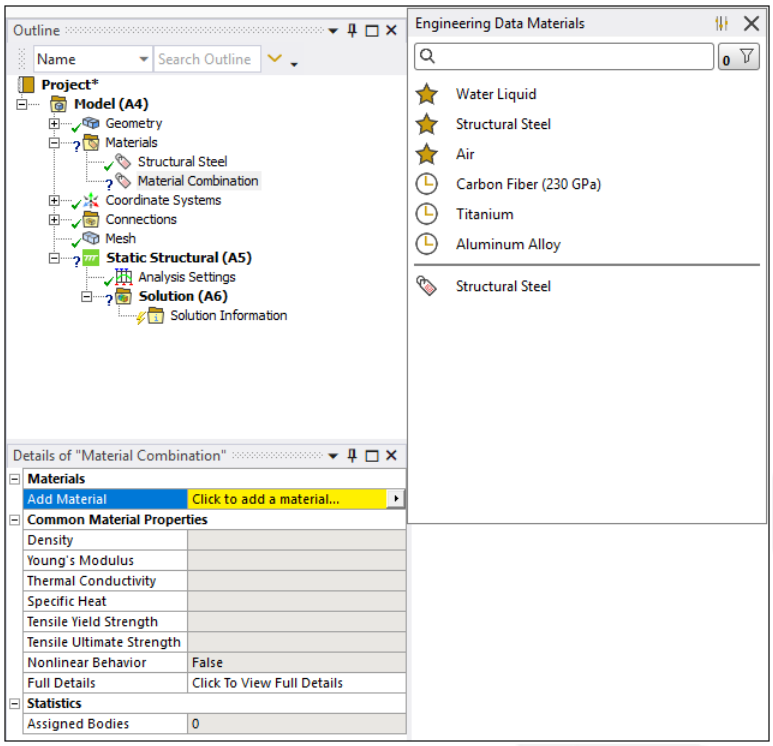
Application

To add a Material Combination object:

1. Right-click the **Materials** folder and select **Insert** > **Material Combination**. Or, use the option available on the Materials (p. 57) Context tab.

2. Open the flyout menu of the **Add Material** property to display the **Engineering Data Materials** pane (p. 156). Using this pane, you can select from existing materials and/or search for additional materials, that you wish to combine.

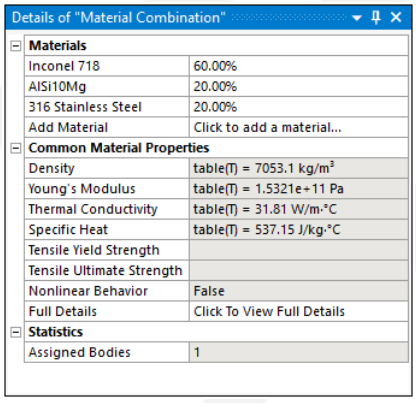
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

You can select multiple materials from the group folder and then either right-click and select **Create Material Combination** or select the **Material Combination** option on **Materials** Context tab to scope the **Materials Combination** object.

3. Once added, the material displays in the **Details** view and is assigned a percentage. As you add materials you can modify the percentages as desired. The total percentage cannot exceed 100%. However, you can have a total percentage of less than 100%. In this instance, any percentage un-accounted for is considered to be empty space with no material properties.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Once your **Material Combination** object is complete, you can use it to scope the Material **Assignment** (p. 345) of parts and bodies.

Note:

- You can create multiple **Material Combination** objects.
- You can delete an added material by selecting its row in the **Details** view and selecting the delete key or you can use the **Delete** option available on the **Materials** Context tab.

Imported Material Fields

The **Imported Material Field** object enables you to import user-defined Field Variable values, created in the Engineering Data workspace, from External Data.

Preparing Required Data

1. On the Workbench Project page select the desired analysis type, select your model using the **Geometry** cell, and then open the **Engineering Data** Workspace.
2. Using the Field Variables feature in the Engineering Data Workspace, create a user-defined variable that contains your desired material properties.
3. Create a comma-separated values (CSV) file that defines the mapping coordinates based upon the user-defined variable specified in Engineering Data. This file type enables the transfer of data between External Data and Mechanical. Save the file to a known location.
4. Insert an External Data system.
5. Open your CSV file in the **External Data** system and check (make active) the **Material Data Field** property in the **Properties of File** pane.
6. In the **Table for File** and **Preview of File** panes, specify and verify your mapping.
7. Return to the Workbench Project page and link your **External Data** system with the desired analysis type that includes the corresponding model in the Project Schematic.
8. Launch Mechanical.

Mapping Material Values in Mechanical

1. In Mechanical, open the **Materials** folder as well as the Imported Material Fields object.
2. Select the Imported Material Field child object.
3. Using the **Apply To** property, specify whether to map values to the **Elements** (default) or **Corner Nodes** of your model.
4. As necessary, review and modify the remaining default settings of the properties in the Details view.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

5. Right-click the **Imported Material Field** object and select the **Import Material Field** option from the context menu. This action maps the values of your CSV file to the nodes or elements of your model.

See Data Transfer Mapping and Validation (p. 2389) section for additional information about node and element mapping from external data sources.

Unique Material-Based MAPDL Commands

When Mechanical creates the input file for the Mechanical APDL solver, there are certain commands, described below, the application uses to provide additional information, capabilities, or support for legacy data.

Material Identification

When the Mechanical application creates the input file for the Mechanical APDL solver, the material property labels shown below are used on the **MP** command. The application stores these identifiers in the result file along with other material data. You can use these material identifiers to query the Engineering Data (or Granta) application for complete material data (not available in Mechanical APDL). You would typically use these identifiers when creating a data processing framework.

- **UMID**: This material identifier is assigned to a material when it is created. For Granta materials, this is the identifier of the material in the database.
- **UVID**: This material identifier is assigned when a material is being transferred from Engineering Data to another component, such as the **Model** object of an analysis system in Mechanical.

Legacy Total Strain Input for Bilinear Isotropic or Kinematic Hardening

When Engineering Data imports data for these models prior to Ansys 2024 R2, the data is using total strain. The data should be converted or modified to use plastic strain if possible. If the total strain input is used then when the Mechanical application creates the input file for the Mechanical APDL solver, the material property labels shown below are used on the **TB** command. These options are deprecated.

TB,BISO

TB,BKIN

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Working with Named Selections

The **Named Selection** object is a **Model**-level object in the tree hierarchy. The **Named Selection** feature allows you to create groupings of similar geometry or meshing entities.

The following sections describe the steps to create Named Selections objects and prepare them for data definition. Subsequent sections further define and build upon these techniques.

Create a Named Selection Object
Defining Named Selections
Specifying Criteria for Geometry-Based Named Selections
Understanding the Named Selections Worksheet
Applying Named Selections via the Ribbon
Promoting Scoped Objects to a Named Selection
Displaying Named Selections
Displaying Interior Mesh Faces
Scoping Analysis Objects to Named Selections
Sending Named Selections to the Solver
Protecting Named Selections
Including Named Selections in Program Controlled Inflation
LS-DYNA Named Selection IDs
Importing Named Selections
Exporting Named Selections
Merging Named Selections
Converting Named Selection Groups to Mechanical APDL Application Components

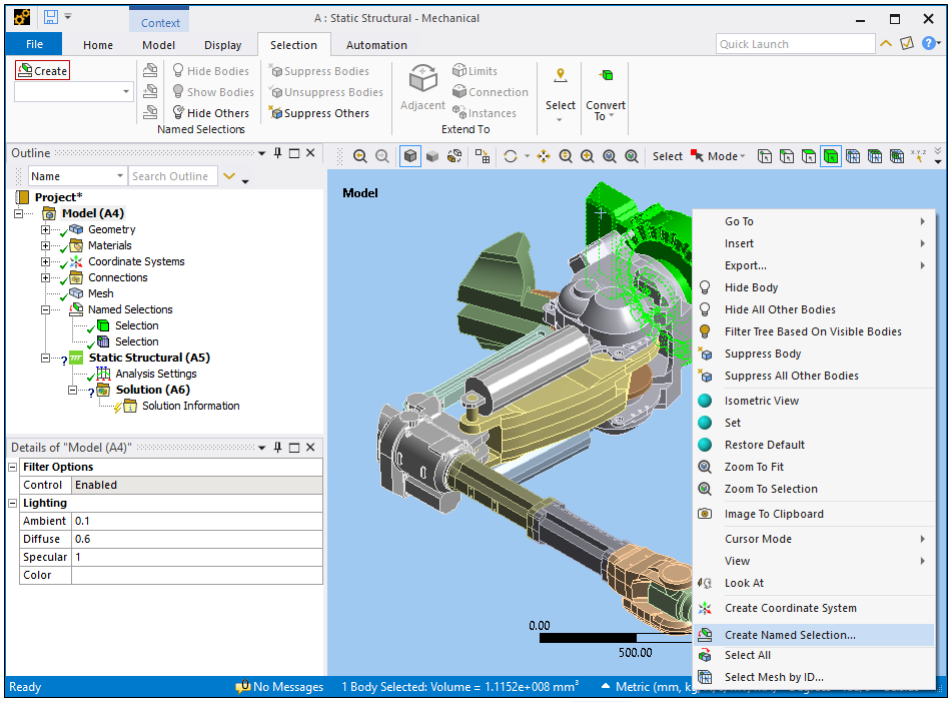
Create a Named Selection Object

Creating **Named Selections** objects is easy and can be accomplished using different methods, including:

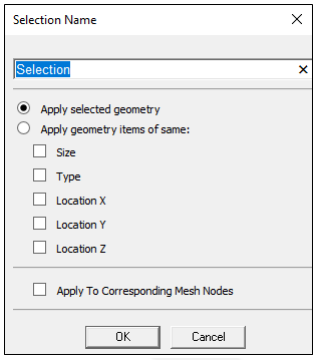
- Select desired geometry or mesh entities in the **Geometry** window (bodies, faces, nodes, etc.), right-click the mouse, and then select **Create Named Selection**. Or you can the select the **Create** option in the **Named Selections** group (p. 89) of the **Selection** tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



A **Selection Name** dialog appears so that you can enter a specific name for the Named Selection as well as specify criteria based on the selected geometry (p. 1071).



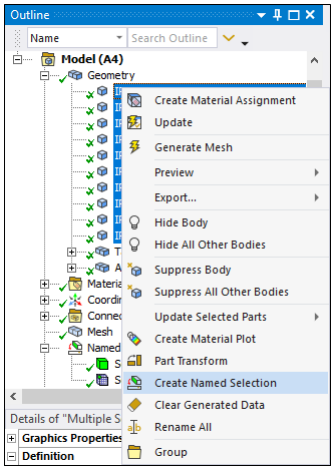
Note:

When the Mechanical APDL solver encounters a du
Options dialog preference, **Unique Names (MAPD**
named selections as a **Warning** (default) or an **Err**
address the naming issue.

- Select desired parts/bodies from the **Geometry** object, right-click the mouse, and then select **Create Named Selection**. A **Selection Name** window appears so that you can enter a specific name for the Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



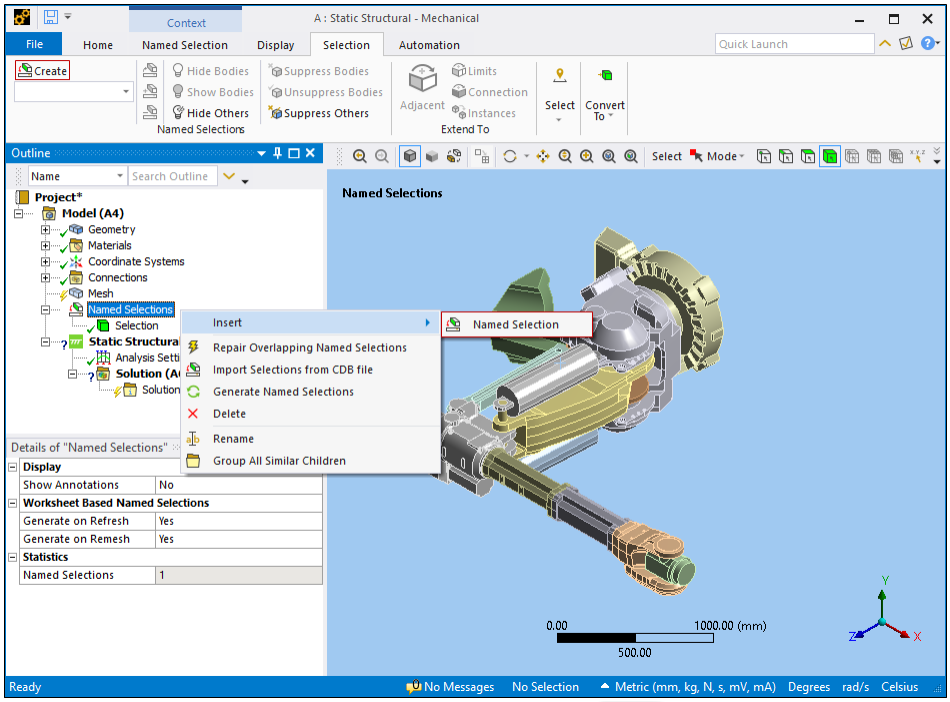
Once you have created a Named Selection, a **Named Selections** folder object is placed into the tree and includes your new child object titled **Selection** or titled with the name you have given it. This new object, and any subsequent Named Selection objects that are inserted into the parent folder, require geometry or mesh entity scoping. If a direct selection method (via Geometry object or graphical selection) was used, the Geometry entities may already be defined.

The **Selection** objects are the operable "Named Selections" of your analysis. You may find it beneficial to rename these objects based on the entities to which they are scoped or the purpose that they will serve in the analysis. For example, you may wish to rename a Named Selection containing edges to "Edges for Contact Region".

Adding Named Selection Objects

If a **Named Selections** folder exists in the tree, insert additional Named Selection objects using the same general methods as above: (1) select the **Create** option in the **Named Selections** group (p. 89) of the **Selection** tab (available once the **Named Selection** folder is generated) or (2) when either the **Named Selections** parent folder object or another **Selection** object is highlighted, right-click the mouse and select **Insert**>**Named Selection**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Defining Named Selections

Once you create **Named Selections**/**Selection** objects, you need to define the geometry or node-based meshing entities that you would like to scope to the object. Scoping method options include:

- **Geometry** (p. 1060): geometry-based entries/selections.
- **Worksheet** (p. 1061): criteria-based entries/selections.

For the steps to scope your Named Selection to nodes or elements, see one of the following sections:

- Specifying Named Selections by Direct Node Selection (p. 275)
- Specifying Element and Element Face-Based Named Selections (p. 276)

Specifying Named Selections by Geometry Type

Use the steps shown below to define the Details of your Named Selections based on geometry types (body, face, edge, or vertex).

To define geometry-based named selections:

1. Highlight the **Selection** object in the tree. In the Details view, set **Scoping Method** to **Geometry Selection**.
2. Select the geometry entities in the graphics window to become members of the Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. Click in the **Geometry** field in the details view, then click the **Apply** button. The named selection is indicated in the graphics window. You can rename the object by right-clicking on it and choosing **Rename** from the context menu.

Tip:

To allow the Named Selection criteria to be automatically generated after a geometry update, highlight the **Named Selections** folder object and set **Generate on Refresh** to **Yes** (default). This setting is located under the **Worksheet Based Named Selections** category in the **Details** view.

Note:

- If you change the **Scoping Method** from **Geometry Selection** to **Worksheet**, the original geometry scoping remains until you select the **Generate** button.
- For geometric entity Named Selections, the status of a Named Selection object can be fully defined (check mark) only when a valid geometry is applied, or suppressed ("x") if either no geometry is applied or if all geometry applied to the Named Selection is suppressed.
- For a Named Selection created using the Graphics Viewer, the selections must be manually updated after you change the geometry.

Specifying Named Selections using Worksheet Criteria

You can specify **Worksheet** as your **Scoping Method** for a Named Selection.

Doing so displays the **Worksheet** window. By default, this window displays below the **Geometry** pane in Mechanical. **Worksheet** data entries define the criteria for Named Selections based on geometric or meshing entities. Each row of the worksheet performs a calculation for the specified criteria. If multiple rows are defined, the calculations are evaluated and completed in descending order.

Named Selections Defined by Worksheet Criteria

1. Highlight the **Selection** object. In the Details view, set **Scoping Method** to **Worksheet**.
2. As needed, right-click the mouse and select **Add Row**.
3. Enter data in the worksheet for specifying the criteria that will define a Named Selection. See the **Worksheet Entries and Operation** section below for specific entry information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

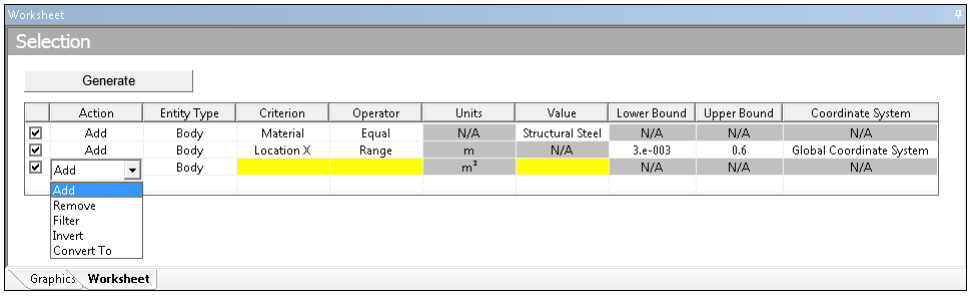
4. Click the **Generate** button located on the Worksheet to create the Named Selection based on the specified criteria. Alternatively, you can right-click the Named Selection *object* and choose **Generate Named Selection** from the context menu.

Note:

- If you change the **Scoping Method** from **Geometry Selection** to **Worksheet**, the original geometry scoping will remain until you select **Generate**.
- When you select **Generate** and the generation fails to produce a valid selection, any prior scoping is removed from the Named Selection.
- If there is no indication that the worksheet has been changed and the Named Selection should be regenerated, you still may want to select **Generate** to ensure that the item is valid.
- If a row inside the worksheet has no effect on the selection, there are no indications related to this.
- Named Selections require valid scoping. If the application detects a criterion that is not properly scoped, it becomes highlighted in yellow to alert users of a possible problem. A highlighted criterion does not effect on the overall state of the object.
- Named Selections created using the **Worksheet** may not support virtual entities.

Worksheet Entries and Operation

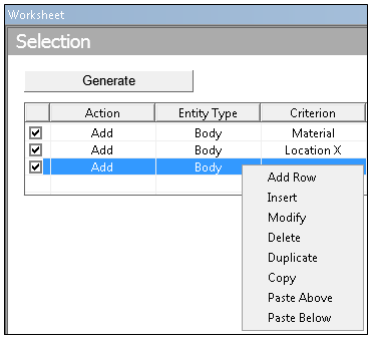
A sample worksheet is illustrated below. Criteria of the **Worksheet** is defined by making selections in the drop-down menus of the columns for each row. Certain values are read-only or they are only available as the result of other criterion being specified.



As illustrated here, when you add a row into the **Worksheet**, right-click context menu options become available. When using these options, you need to also click the **Generate** button to have these changes applied to the Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The content of each Worksheet column is described below.

Action column:

- **Add**: Adds the information defined in the current row to information in the previous row, provided the item defined in the **Entity Type** column is the same for both rows.
- **Remove**: Removes the information defined in the current row from information in the previous row, provided the geometry defined in the **Entity Type** column is the same for both rows.
- **Filter**: Establishes a subset of the information defined in the previous row.
- **Invert**: Selects all items of the same **Entity Type** that are not currently in the named selection.
- **Convert To**: Changes the geometric **Entity Type** selected in the previous row. The change is in either direction with respect to the topology (for example, vertices can be converted "up" to edges, or bodies can be converted "down" to faces). When going up in dimensionality, the higher level topology is selected if you select any of the lower level topology (for example, a face will be selected if any of its edges are selected).

  Mesh Nodes: You can convert a geometry selection (bodies, edges, faces, vertices) to mesh nodes. The nodes that exist on the geometry (that is, the nodes on a face/edge/vertex or nodes on and within a body) will be selected. In addition, node-based Named Selections can be converted to elements and element-based Named Selections can be converted to nodes using this action.

  Element Faces: You can convert a face-based Named Selection, a node-based Named Selection, or a body-based Named Selection to an Element Face Named Selection. Face- and node-based conversion does not support element faces that share two (or more) bodies. However, for body-based conversion, you can specify the **Criterion** options **Exclude Shared Faces** or **Include Shared Faces** to specify whether to include shared element faces.

  Note:

  The conversion from geometry selection to mesh nodes is analogous to using Mechanical APDL commands **NSLK**, **NSLL**, **NSLA**, and **NSLV**. The conversion from elements to mesh nodes uses **NSLE** and conversion from mesh nodes to elements uses **ESLN**.

- **Diagnostics**

The **Entity Type** column:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Body**
- **Face**
- **Edge**
- **Vertex**
- **Mesh Node**
- **Mesh Element**
- **Element Face**: This option does **not** support element faces defined on shell bodies.

Criterion column:

- **Size** - available when **Entity Type** = **Body**, **Face**, or **Edge**.
- **Type** - available when **Entity Type** = **Body**, **Face**, **Edge**, or **Mesh Node**, or **Mesh Element**.
- **Location X/Y/Z**

  Important:

  For **Location X/Y/Z** criterion, the calculation of the centroid is not supported for Line Bodies.

- **Within Body**: available when **Entity Type** = **Mesh Node**, **Mesh Element**, or **Element Face**, and the **Value** column is a Body with the **Treatment** property set to **Construction Body**.
- **Face Connections** - available when **Entity Type** = **Edge**.
- **Radius** - available when **Entity Type** = **Face** or **Edge**. Applies to faces that are cylindrical and edges that are circular.
- **Distance**

  Important:

  Line Bodies are not supported for the **Distance** Criterion option.

- **Named Selection**
- **Normal** - available when **Entity Type** = **Element Face**. The **Normal** option requires you to specify an axis entry, either positive or negative X, Y, or Z in the **Value** column. Based on that selection, the application creates a Named Selection that includes all element faces whose normal direction is same as the selected axis (for the selected Coordinate System, global or user-defined).

  When using the **Normal** selection for the **Criterion** and you have defined a cylindrical coordinate system:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - The (positive) **X Axis** option selects element faces that have their normal pointing radially outward in the specified cylindrical coordinate system.
  - The (negative) **-X Axis** option selects element faces that have their normal pointing radially inward in the selected cylindrical coordinate system.
  - The application does not support the **Y Axis/-Y Axis** options for cylindrical coordinate systems.

- **Shared Across Bodies** - available when **Entity Type** = **Face**, **Edge**, **Vertex**. This option is useful when you want to create a Named Selection of faces, edges, or vertices shared across bodies.
- **Shared Across Parts** - available when **Entity Type** = **Mesh Node**. This option is useful for examining and working with shared nodes that were generated by Node Merge or Mesh Connection operations.
- **Element Connections** - available when **Action** = **Add** and **Entity Type** = **Mesh Node**. This option requires a **Value** entry. The **Value** entry defines the number of elements to which a node is connected. For example, if you enter a **Value** of **4**, the application selects every node in the model that is connected to four elements.

  This option supports 2D and 3D analyses for all element types.
- **Material** - available when **Entity Type** = **Body**.
- **Name** - available when **Entity Type** = **Body**.
- **Thickness** - available when **Entity Type** = **Body**. This option is supported for surface/shell bodies only.
- **Cross Section** - available when **Entity Type** = **Body**.
- **Seam Weld** when **Entity Type** = **Body**.
- **Node ID** - available when **Entity Type** is **Mesh Node**.
- **Any Node** - available when:
  - **Action** = **Convert To**
  - **Entity Type** = **Mesh Element** or **Element Face** (node-based conversion only)
- **All Nodes** - available when:
  - **Action** = **Convert To**
  - **Entity Type** = **Mesh Element** or **Element Face** (node-based conversion only)
- For **Entity Type** = **Mesh Element**.
  - **Type**
  - **Element ID**
  - **Volume**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Area**
- **Location X**
- **Location Y**
- **Location Z**
- **Element Quality**
- **Aspect Ratio**
- **Jacobian Ratio [MAPDL]**
- **Jacobian Ratio [Corner Nodes]**
- **Jacobian Ratio [Gauss Points]**
- **Warping Factor**
- **Parallel Deviation**
- **Skewness**
- **Orthogonal Quality**
- **Named Selection**
- **Ply**
- **Maximum Corner Angle**
- **Within Body**
- **Seam Weld HAZ1**
- **Seam Weld HAZ2**
- **Seam Weld HAZ3**
- **Seam Weld All HAZ**
- **Seam Weld Normal**
- **Seam Weld Angle**
- **Minimum Tri Angle**
- **Maximum Tri Angle**
- **Minimum Quad Angle**
- **Maximum Quad Angle**
- **Warping Angle**

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Tet Collapse**
  - **Aspect Ratio [Explicit]**
  - **Minimum Element Edge Length**
  - **Maximum Element Edge Length**
  - **Characteristic Length [LS-DYNA]**

  You may wish to refer to the Mesh Metric section of the *Meshing User's Guide* for more information about these **Criterion** options.

Operator column:

- **Equal**
- **Not Equal**
- **Contains** - available when **Criterion** = **Name** or **Name Selection**.
- **Less Than**
- **Less Than or Equal**
- **Greater Than**
- **Greater Than or Equal**
- **Range** includes **Lower Bound** and **Upper Bound** numerical values that you enter.
- **Smallest** - only available when **Action** column = **Add**
- **Largest** - only available when **Action** column = **Add**
- **Yes**
- **No**

Note:

The **Smallest** and **Largest** options apply to the entire model. They do not function with any other worksheet criteria entry. Therefore, they are only available when the **Action** column is set to **Add**.

Units column: read-only display of the current units for **Criterion** = **Size** or **Location X, Y, or Z**.

Value column:

- For **Criterion** = **Size**, enter positive numerical value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For **Criterion** = **Location X, Y,** or **Z**, enter numerical value.

  Note:

  Selection location is at the centroids of edges, faces, bodies, and elements.

- For **Entity Type** = **Body** and **Criterion** = **Type**:
  - **Solid**
  - **Surface**
  - **Line**
- For **Entity Type** = **Body** and **Criterion** = **Cross Section**: select the desired cross section name from the drop-down list.
- For **Entity Type** = **Face** and **Criterion** = **Type**:
  - **Plane**
  - **Cylinder**
  - **Cone**
  - **Torus**
  - **Sphere**
  - **Spline**
  - **Faceted**
- For **Entity Type** = **Edge Criterion** = **Type**:
  - **Line**
  - **Circle**
  - **Spline**
  - **Faceted**
- For **Entity Type** = **Mesh Node** and **Criterion** = **Type**:
  - **Corner**
  - **Midside**
- For **Entity Type** = **Body** and **Criterion** = **Name**, enter a valid body name. When:
  - **Operator** = **Contains**, you can enter a partial body name. The application matches name characters at the beginning, middle, or at the end of the name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Operator** = **Equal**, you must enter the body name exactly as it appears in the Geometry folder.
  - **Operator** = **Not Equal**, you enter the body name that you do not wish to include in your named selection. The application will select all other bodies from the Geometry folder.

- For **Entity Type** = **Mesh Element** and **Criterion** = **Type**:
  - **Tet10**
  - **Tet4**
  - **Hex20**
  - **Hex8**
  - **Wed15**
  - **Wed6**
  - **Pyr13**
  - **Pyr5**
  - **Tri6**
  - **Tri3**
  - **Quad8**
  - **Quad4**
  - **High Order Beam**
  - **Low Order Beam**
  - **Line2**
  - **Line3**
- For **Entity Type** = **Element Face** and **Criterion** = **Normal**: the **Value** field options include **X Axis**/**Y Axis**/**Z Axis** and **-X Axis**/**-Y Axis**/**-Z Axis**. This axis selection is based on the coordinate system defined in the **Coordinate System** column.
- For **Entity Type** = **Edge** and **Criterion** = **Face Connections**, enter the number of shared edge connections. For example, enter **Value** = 0 for edges not shared by any faces, enter **Value** = 1 for edges shared by one face, and so on.
- For **Criterion** = **Named Selection**, you can include a previously defined named selection from the **Value** field.

  For **Criterion** = **Named Selection** and **Operator** = **Contains**, you can specify the name of multiple Named Selections, or, as desired, only a portion of the name of multiple Named Selections in the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Value field. For example, for NS1, NS2, and NS3, you can create a new Named Selection that includes all three by specifying "NS" in the **Value** field.

Note:

Whenever you create a Named Selection that references an existing Named Selection using the **Criterion** option, it is recommended that you use the **Generate Named Selections** context (right-click) menu option from the Named Selections folder to generate your Named Selections. This action calculates the relationship between all the Named Selections contained in the folder to determine the correct order for generation. Using the Worksheet **Generate** option updates only the selected/active Named Selection. Therefore, if that Named Selection has a reference to another Named Selection, the associated Named Selection is not regenerated.

Furthermore, the **Generate Named Selections** option makes sure that the application updates all child objects of the Named Selection folder in the appropriate order while making sure that the application does not create any cyclic dependencies (also called a closed loop reference) between Named Selections. A cyclic dependency occurs when you have one or more named selections that depend upon one another to generate results. If this is detected, the application generates an error.

- For **Criterion** = **Ply**, you can include a previously-defined named selection in the **Value** field. You can select from the named selections provided in the list only.
- For **Criterion** = **Material**, select the desired material from the drop-down list. See the Material Assignment (p. 345) topic for more information.
- For **Criterion** = **Distance**, enter a positive numerical value. The distance is the difference between the origin of the selected coordinate system and the centroid of the scoped geometry (body, face, edge).

Important:

Line Bodies are not supported for the **Distance** Criterion option.

- For **Criterion** = **Thickness**, enter a positive numerical value or values based on the selection made in the **Operator** column.

Lower Bound column: enter numerical value.

Upper Bound column: enter numerical value.

Coordinate System column:

- **Global Coordinate System**
- Any defined local coordinate systems

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Adjusting Tolerance Settings for Named Selections by Worksheet Criteria

Tolerance settings are used when the **Operator** criterion is defined as an "equal" comparison. Tolerances are not used when doing greater than or less than operations. Tolerance values apply to the *entire* worksheet. If you wish to adjust the tolerance settings for worksheet criteria, use the **Tolerance** settings in the chosen Named Selection's Details view.

By default, the **Zero Tolerance** property is set to 1.e-008, the **Relative Tolerance** value is set to 1.e-003, and the **Angular Tolerance** is set to 1°. As a result of the significant digit display, the value used for calculations and the display value may appear to be different. The **Zero Tolerance** property's value is past the number of significant digits that Mechanical shows by default. The application's default setting for significant digits is 5 (the range is 3 to 10). This setting affects only the numbers that are displayed, any calculation or comparison uses the actual values when processing. In addition, it is important to note that most values (including selection values seen in the status bar and the Selection Information window) in Mechanical display in a significant digit format. See the Appearance option in the Setting Ansys Workbench section of the Help for information about changing default display settings.

Setting the tolerance values manually can also be useful in meshing, when small variances are present in node locations and the default relative tolerance of .001 (.1%) can be either too small (not enough nodes selected) or too big (too many nodes selected).

1. In the Details view, set **Tolerance Type** to **Manual**.
2. Specify the properties as desired: **Zero Tolerance**, **Relative Tolerance**, and **Angular Tolerance**. The **Zero Tolerance** and **Relative Tolerance** values are dimensionless. **Relative Tolerance** is a multiplying factor applied to the specified worksheet value. For example, if you want a tolerance of 1%, enter .01 in the Relative Tolerance field. **Angular Tolerance** is specified in degrees or radians. The **Angular Tolerance** setting determines when the normal direction of an element face is equal to the direction of your specified coordinate system.

Note:

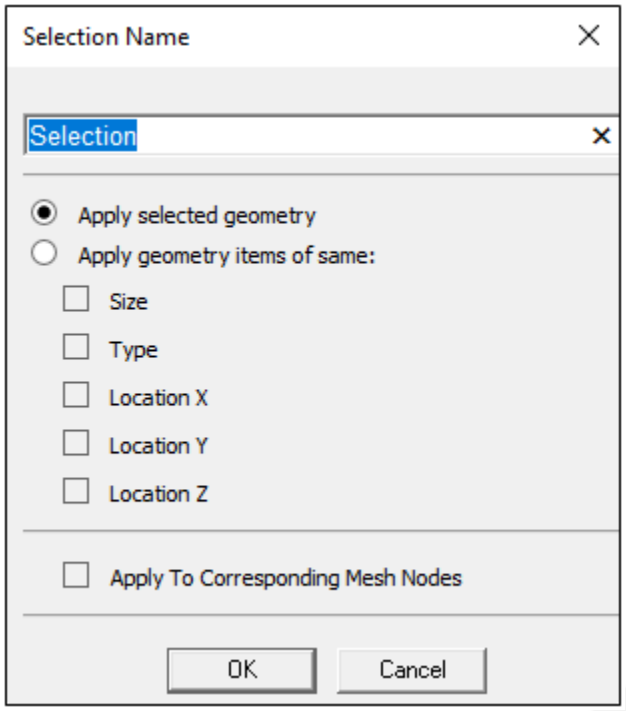
Angular tolerance is available for Worksheet-based Named Selections only. It is applicable once you set the **Criterion** to **Normal** and the **Entity Type** to **Element Face**.

All comparisons are done in the CAD unit system. If your current unit system is not the same as the CAD unit system, the values entered will first be converted to the CAD unit system before searching for criteria.

Specifying Criteria for Geometry-Based Named Selections

Mechanical enables you to specify certain criteria using the context menu (right-click) option when you make geometry selections in the **Geometry** window. When you select a geometric entity (face, edge, etc.) in the **Geometry** window, right-click, and then select the option **Create Named Selection (N)**, the following dialog box displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The two control options on the dialog include:

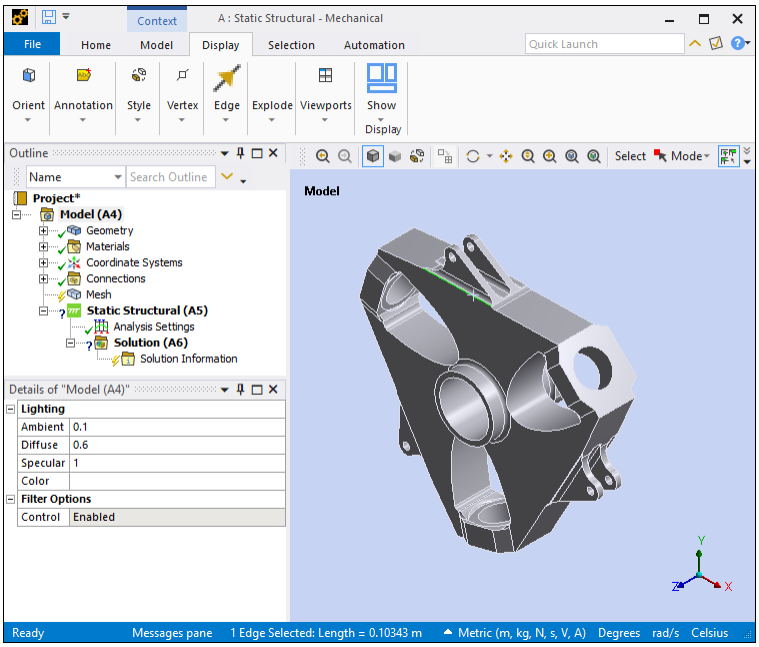
- **Apply selected geometry**: this option simply creates a Named Selection based on the geometry you select.
- **Apply geometry items of same**: this option enables you to make certain criteria-based specifications (**Size**, **Type**, **Location X**, **Y**, **Z**) for geometry selections. If you select only one geometric entity (one face, edge, etc.) you can choose multiple criteria options. However, when you select more than one geometric entity (multiple faces, edges), you are limited to choosing only one criteria option.

  In addition, there is the **Apply To Corresponding Mesh Nodes** option. You can also select this option when using the available criteria in order to include the nodes that correspond to the geometry. This option requires that you generate the mesh.

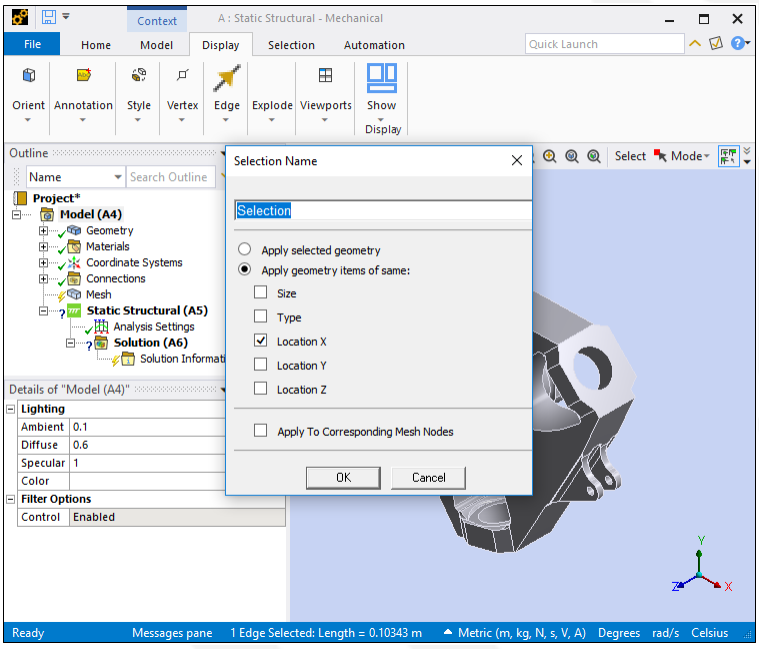
Let's look at an example. A singular edge is selected on the following model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

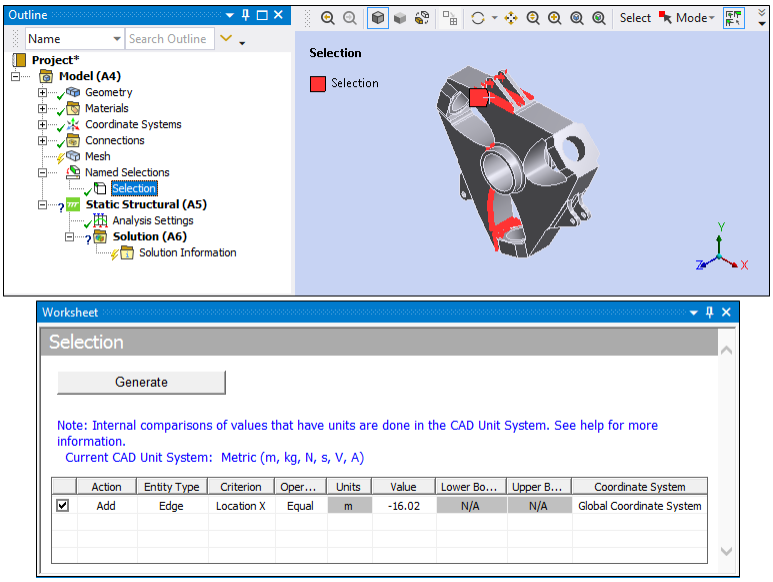


Here we display the **Selection Name** dialog (via right-click), select the option **Location X**, and then click **OK**.



As illustrated, you need to select the new named Selection object in the tree. The application specified the Named Selection as a **Worksheet** entry and now includes eight (8) edges (all of which share the X coordinate).

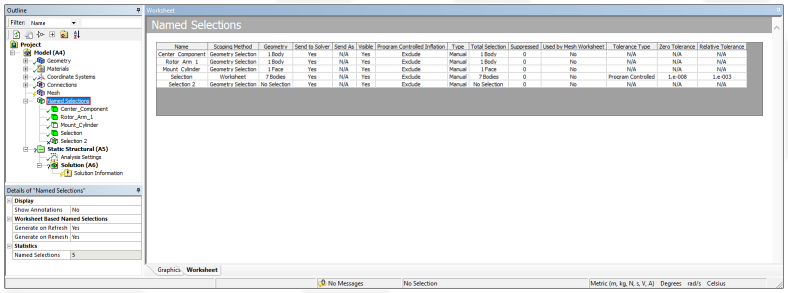
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note that the supported selections may differ based on the type of geometry that you select. For example, the options **Size** or **Type** are not supported for a vertex selection.

Understanding the Named Selections Worksheet

When you select the **Named Selections** parent folder object and then select the **Worksheet** button on the **Home** tab (p. 40), the **Worksheet** displays as shown here and lists all existing Named Selections. This view provides a summary of all the defined Named Selections. Table headings correspond to the **Details** view properties of the user-defined Named Selections. See the Named Selections object reference page for a description of the properties.

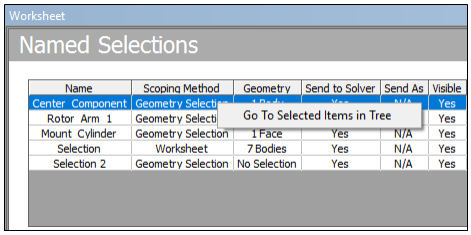


Worksheet Tools

You can single-click a table heading to sort the table content according to that heading type. You can also automatically select a Named Selection object by selecting the appropriate row in the table, right-clicking, and selecting the **Go To Selected Items in Tree** option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Applying Named Selections via the Ribbon

The **Named Selections** group of the **Selection** tab enables you to select and modify user-defined named selections.

To use a Named Selection options:

1. Select a Named Selection from the drop-down list. This list matches the Named Selections contained in the **Named Selections** folder object.

2. Choose from the following options provided by toolbar:

Option	Description
Selection drop-down menu (or in context menu from right-clicking the mouse button on individual Named Selection object)	Controls selection options on items that are part of the group whose name appears in the **Named Selection** display. Available options are: • **Select Items in Group**: selects only those items in the named group. • **Add to Current Selection**: Picks the scoped items defined by the Named Selection that you have highlighted and adds those items to the item or items that you have selected in the geometry window. This option is grayed out if the selections do not correspond, such as selecting trying to add a faces to vertices. • **Remove from Current Selection**: Removes the selection of items in the named group from other items that are already selected. Selected items that are not part of the group remain selected. This option is grayed out if the entity in the Named Selection does not match the entity of the other selected items. • **Create Nodal Named Selection**: Automatically converts the geometry specified by the Named Selection to mesh nodes. A corresponding **Convert To** row is added to the Worksheet **Note:** Choosing any of these options affects only the current selections in the **Geometry** view. These options have no effect on what is included in the Named Selection itself.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	Description
Visibility drop-down menu	Controls display options on bodies that are part of the group whose name appears in the **Named Selection** display. Available options are: • **Hide Bodies in Group**: Turns off display of bodies in the named group (toggles with next item). Other bodies that are not part of the group are unaffected. • **Show Bodies in Group**: Turns on display of bodies in the named group (toggles with previous item). Other bodies that are not part of the group are unaffected. • **Show Only Bodies in Group**: Displays only items in the named group. Other items that are not part of the group are not displayed. You can also hide or show bodies associated with a Named Selection by right-clicking the **Named Selections** object and choosing **Hide Bodies in Group** or **Show Bodies in Group** from the context menu. You can hide only the Named Selection by right-clicking on the **Named Selections** object and choosing **Hide Face(s)**.
Suppression drop-down menu	Controls options on items that affect if bodies of the group whose name appears in the **Named Selection** display are to be suppressed, meaning that, not only are they not displayed, but they are also removed from any treatment such as loading or solution. Available options are: • **Suppress Bodies in Group**: Suppresses bodies in the named group (toggles with next item). Other bodies that are not part of the group are unaffected. • **Unsuppress Bodies in Group**: Unsuppresses bodies in the named group (toggles with previous item). Other bodies that are not part of the group are unaffected. • **Unsuppress Only Bodies in Group**: Unsuppresses only bodies in the named group. Other bodies that are not part of the group are suppressed. You can also suppress or unsuppress bodies associated with a Named Selection by right-clicking the particular **Named Selections** object and choosing **Suppress Bodies In Group** or **Unsuppress Bodies In Group** from the context menu. The **Suppress Bodies In Group** and **Unsuppress Bodies In Group** options are also available if you select multiple Named Selection items under a **Named Selections** object. The options will not be available if your multiple selection involves invalid conditions (for example, if you want to suppress multiple items you have selected and one is already suppressed, the **Suppress Bodies In Group** option will not be available from the context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The status bar shows the selected group area only when the areas are selected. The group listed in the toolbar and in the Details Pane (p. 119) provides statistics that can be altered.

Promoting Scoped Objects to a Named Selection

In addition to creating Named Selections, you can also use the promotion feature to create a named selection from an existing object that is scoped to geometry or mesh. Objects that support the promotion feature include:

- Remote Points
- Contact Regions
- Springs
- Joints
- Boundary Conditions
- Results and Custom Results

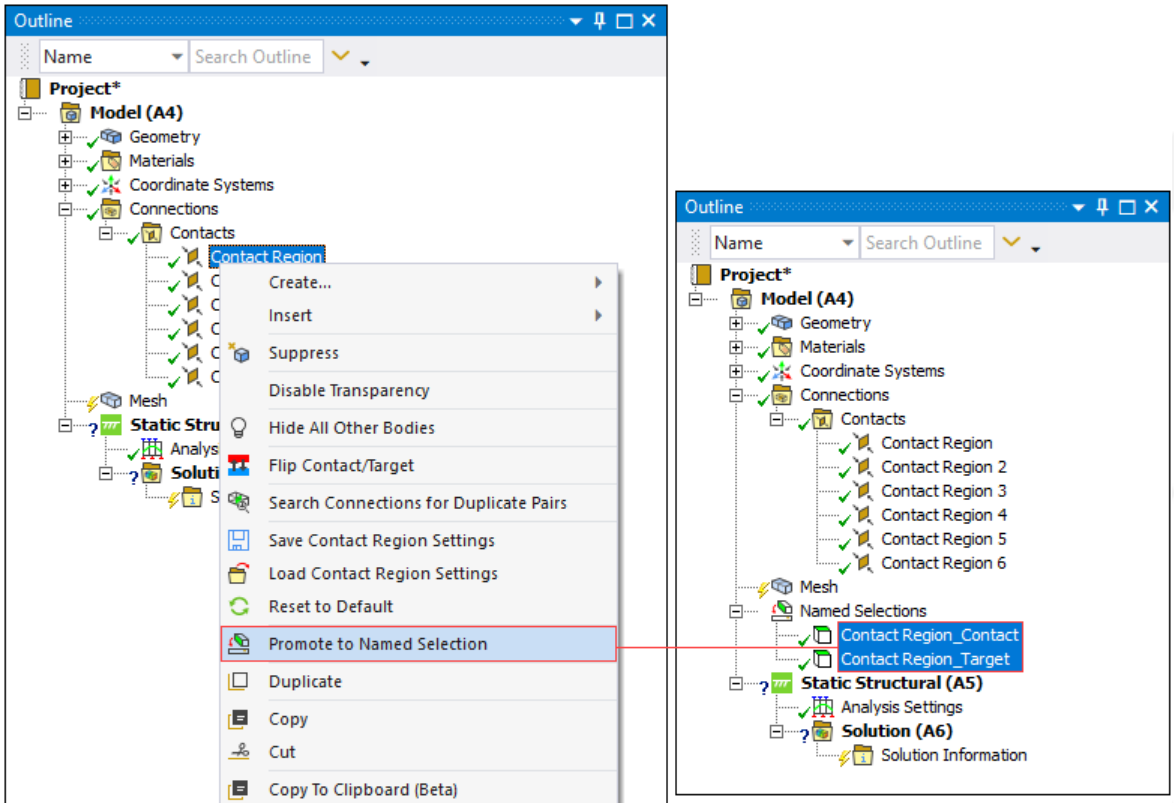
All of these objects have one thing in common when using the promotion feature, they are first scoped to geometry or mesh. This is the specification basis for the promoted Named Selections. Each promoted Named Selection inherits the geometry or mesh scoping of the object used. In addition, the Scoping Method property automatically updates to Named Selection and specifies the corresponding scoping.

Note:

- This action changes the scoping of the corresponding object and may, as a result, cause up-to-date states to become obsolete. For example, promoting a Fixed Support from a completed solution would cause the solution to become obsolete and require it to be re-solved.
- In order to promote objects scoped to the mesh, you need to make sure that the **Show Mesh** option (p. 80) is selected.

By highlighting one of the above objects and right-clicking, such as the **Contact Region** example illustrated below, the context menu provides the option **Promote to Named Selection**. Once selected, the feature automatically adds a **Named Selections** folder to the tree that includes two new Named Selections based on the existing name of the contact object as well as its geometry scoping, Contact and Target. You can promote an object to a Named Selection only once. Deleting the corresponding Named Selection makes the option available again. However, deleting the Named Selection also invalidates the corresponding source object, such as the Contact Region shown in the example below. As a result, you must re-scope the source object to geometry or mesh for the feature to be available. A Contact Region example is slightly different in that it has Contact and Target scoping and that this feature creates two Named Selections. Springs and Joints also create two Named Selections if they are defined as Body-Body. The other object types create one Named Selection. Also note that result objects can be promoted before or after the solution process.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Displaying Named Selections

You can use geometry-based Named Selections to inspect only a portion of your model's total mesh. Although this feature is available regardless of mesh size, it is most beneficial when working with a large mesh (greater than 5 - 10 million nodes).

After you have designated a Named Selection group, you can use any of the following features to assist you:

- Showing the Mesh (p. 1079)
- Showing Annotations (p. 1079)
- Displaying Individual Named Selections in Different Colors (p. 1080)
- Setting Visibility (p. 1080)
- Displaying an Enhanced View of Meshed Items (p. 1081)
- Important Characteristics and Notes (p. 1082)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

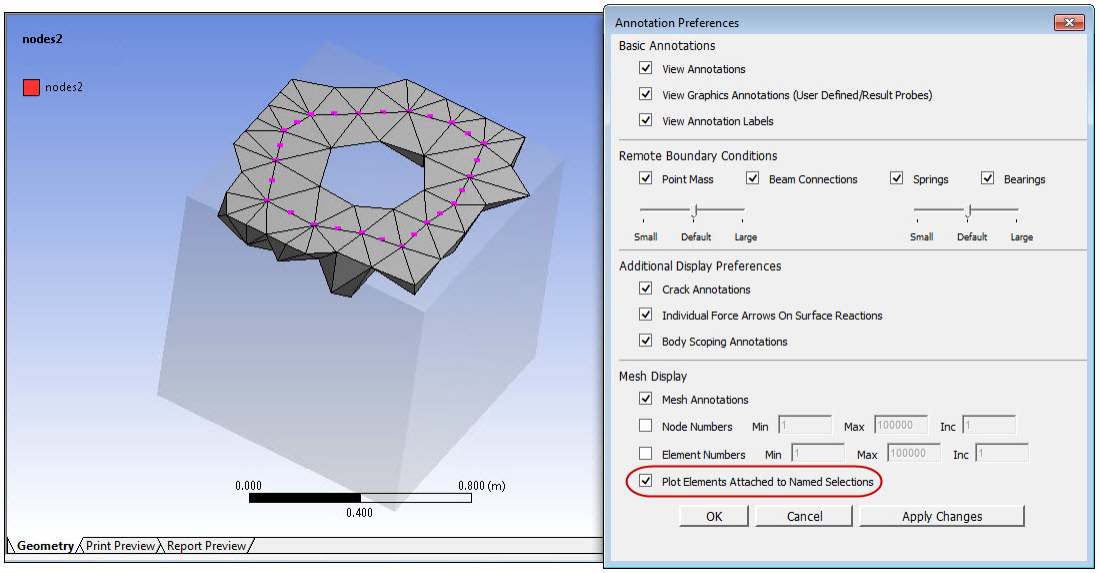
Showing the Mesh

By setting the **Plot Elements Attached to Named Selections** option in the **Annotation Preferences** (p. 310), you can view the elements for all items in the Named Selection group. For node-based Named Selections, this option shows the full elements, while for face or body Named Selections, this option shows just the element faces.

Note:

This option does not affect Line Bodies, and you must have the **Show Mesh** button toggled off to view the elements in the Named Selection.

An example is shown below of a node-based Named Selection.

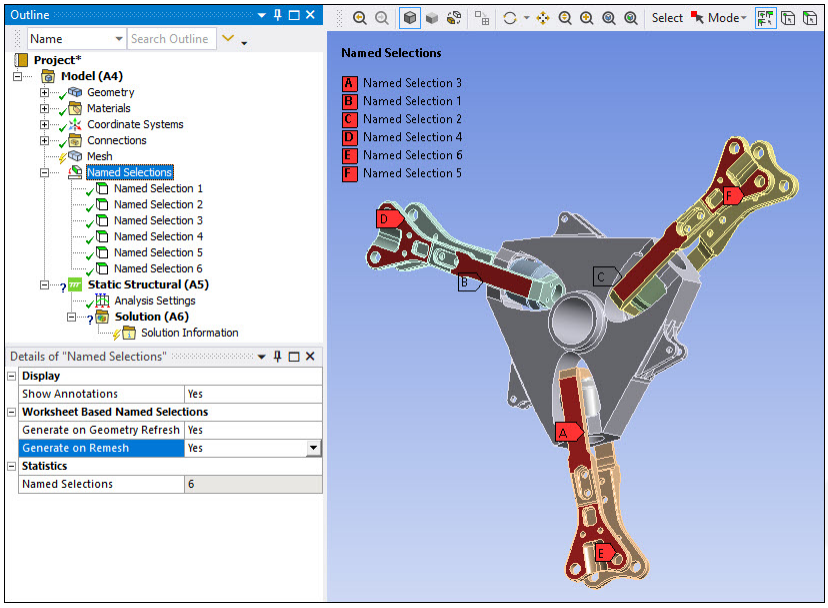


Showing Annotations

As illustrated below, selecting the Named Selection folder displays **all** of the user-defined Named Selection annotations in the Geometry window. This display characteristic can be turned **On** or **Off** using the **Show Annotations** category in the **Named Selections** Details view.

Selecting an individual Named Selection displays the annotation specific to that Named Selection in the Graphics pane.

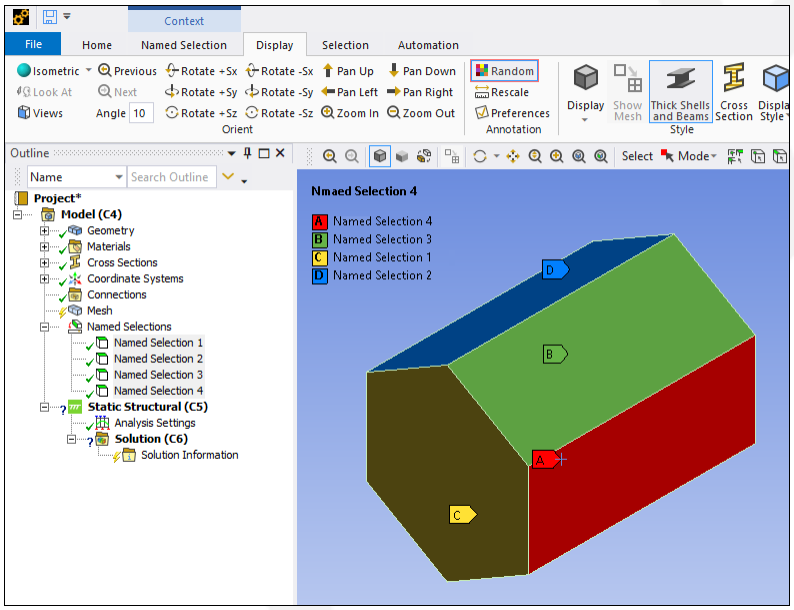
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



You can also toggle the visibility of mesh node annotations and numbers in the annotation preferences. For more information, see Specifying Annotation Preferences (p. 310).

Displaying Individual Named Selections in Different Colors

By default, Named Selections are shown in red. You can use the **Random Colors** button on the **Display** tab to display each named selection with a random color at each redraw.

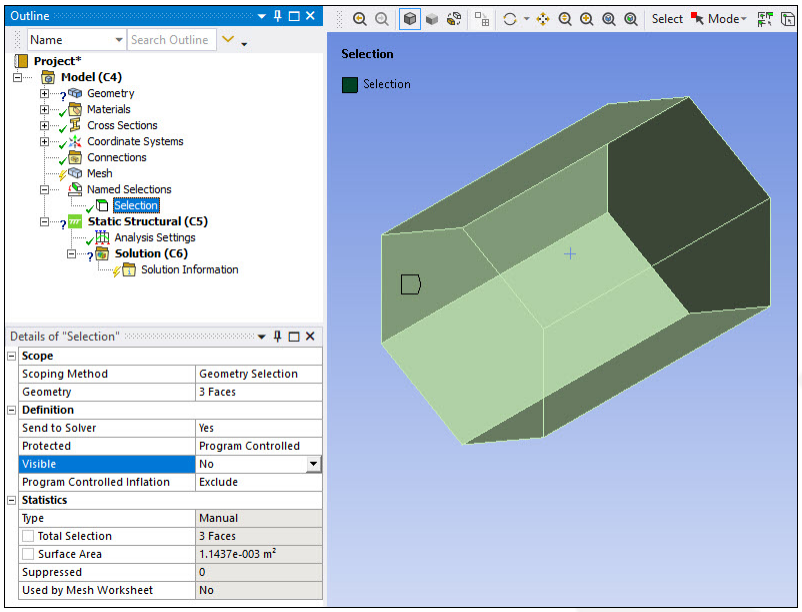


Setting Visibility

By setting the **Visible** object property in the Details view of an individual Named Selection object to **No**, the Named Selection can be made invisible, meaning it will not be drawn and, more importantly, not taken into consideration for picking or selection. This should allow easier inspection inside complicated models having many layers of faces where the inside faces are hardly accessible from the outside.

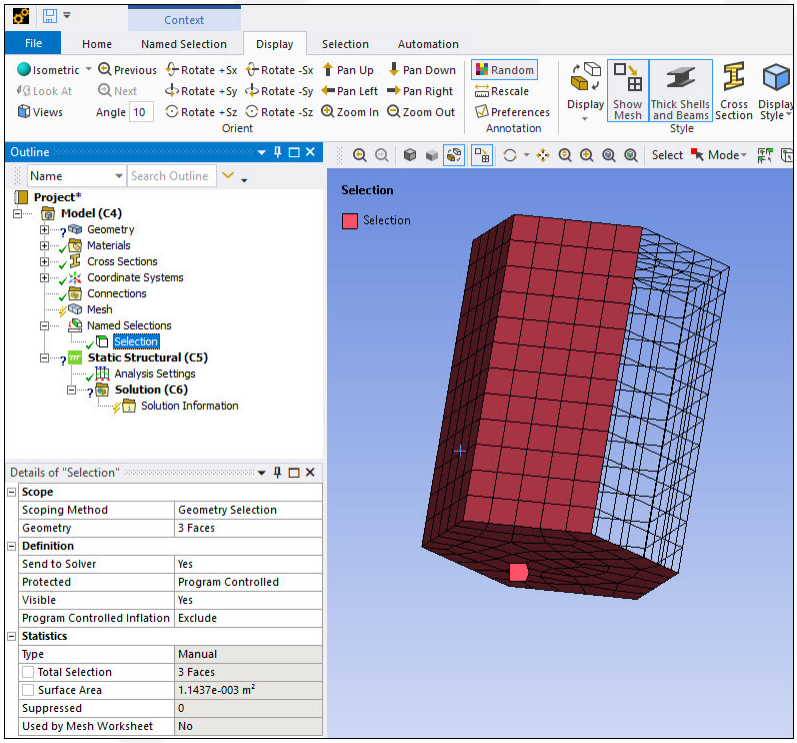
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You can define Named Selections and make them invisible as you progress from outside to inside, similar to removing multiple shells around a core. The example shown below displays the Named Selection **3 Faces** with the **Visible** property set to **No**.



Displaying an Enhanced View of Meshed Items

Display your model in Wireframe mode by selecting the **Wireframe** option in the **Display** (p. 80) drop-down menu on the **Display** tab or by selecting the **Wireframe** option on the Graphics Toolbar (p. 97). Then, open the Annotation Preferences (p. 310) dialog box by selecting the **Preferences** option in the **Annotations** group (p. 79). Check the **Plot Elements Attached to Named Selections** option. This feature displays the meshed entities of your Named Selection only, as illustrated below.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Characteristics and Notes

- The **Visible** object property is the same as the **Hide Face(s)** option in the right mouse button context menu. These options will hide *only* the specified Named Selection. This behavior differs from that of the **Hide Bodies in Group** and **Suppress Bodies in Group** options, which hide or suppress the full body containing a given Named Selection.
- When a Named Selection's **Visible** setting is set to **No**:
  - Only the faces from that Named Selection are not drawn; the edges are always drawn.
  - The Named Selection will not appear in any drawing of the geometry (regardless of which object is selected in the tree).

    Unless...

  - The Named Selection is displayed as meshed, it displays the mesh, but **only** if you have the Named Selection object or the **Named Selections***folder* object is selected in the tree. This behavior is the same as the behavior of the red annotation in the **Geometry** window for Named Selections (that is, the annotation appears only when the current selected object is the specific Named Selection object or the **Named Selections***folder* object).
- After at least one Named Selection is hidden, normally you can see the inside of a body, so displaying both sides of each face is enabled (otherwise displaying just the exterior side of each face is enough). But if a selection is made, the selected face is always displayed according to the setting in the **Options** preference under **Mechanical** > **Graphics** (p. 228) > **Highlight Selection**. **Single Side** is the default setting. It can be one side or both sides.
- If the **Wireframe** display option is used and **Show Mesh** is **Yes**, any face selected is displayed according to the **Highlight Selection** property of the **Options** dialog under **Mechanical** > **Graphics** (p. 228). **Single Side** is the default setting. It can be one side or both sides.

Displaying Interior Mesh Faces

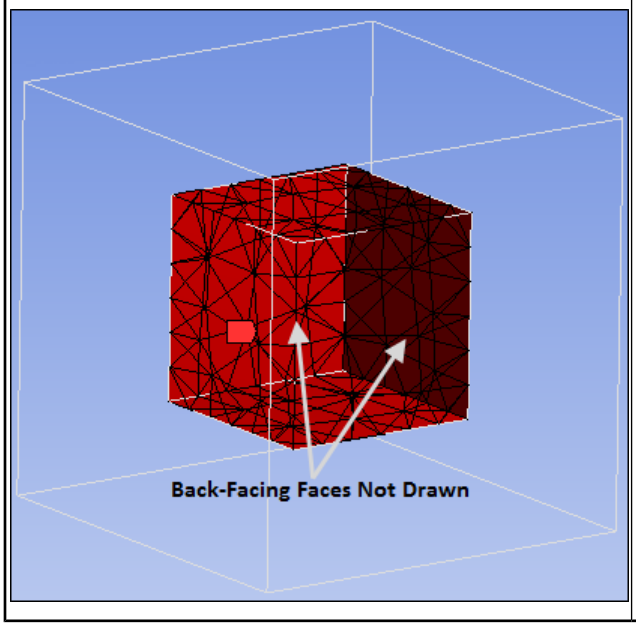
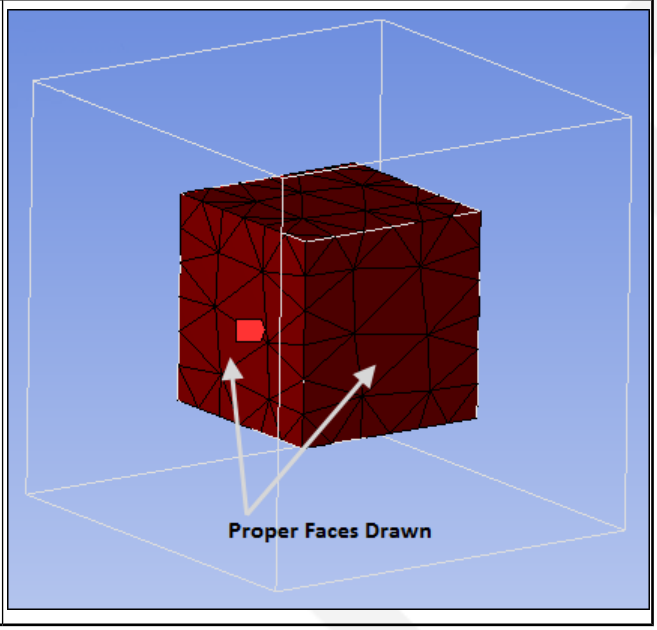
There are special instances when a Named Selection is an interior "back-facing face". This is a unique case that occurs when the external faces of the geometry are hidden allowing interior faces to become visible.

To display the faces of the mesh, the **Named Selections** object must be highlighted in the tree and the **Plot Elements Attached to Named Selections** option in the **Annotation Preferences** (p. 310) must be selected. Then, to correct the display, use the **Draw Face Mode** menu options from the **Show** drop-down menu of the **Display** group (p. 87) (**Display** tab). The **Draw Face Mode** menu options include:

- **Auto Face Draw** (default): Turning back-face culling on or off is program controlled. Using Section Planes is an example of when the application would turn this feature off.
- **Draw Front Faces**: Face culling is forced to stay on. Back-facing faces will not be drawn in any case, even if using Section Planes.
- **Draw Both Faces**: Back-face culling is turned off. Both front-facing and back-facing faces are drawn.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Incorrect Display	Correct Display using Draw Face Mode
	

Scoping Analysis Objects to Named Selections

Many objects can be scoped to Named Selections. Some examples are contact regions, mesh controls, loads, supports, and results.

To scope an object to a Named Selection:

1. Insert or select the object in the tree.
2. Under the Details view, in the **Scoping Method** drop-down menu, select **Named Selections**.
3. In the **Named Selection** drop-down menu, choose the particular name.

Notes on scoping items to a Named Selection:

- Only Named Selections valid for the given analysis object are displayed in the **Named Selection** drop-down menu. If there are no valid Named Selections, the drop-down menu is empty.
- No two **Named Selections** branches can have the same name. It is recommended that you use unique and intuitive names for the **Named Selections**.
- Named Selection modifications update scoped objects accordingly.
- Deleting a Named Selection causes the scoped object to become underdefined.
- If all the components in a Named Selection cannot be applied to the item, the Named Selection is not valid for that object. This includes components in the Named Selection that may be suppressed. For example, in the case of a bolt pretension load scoped to cylindrical faces, only 1 cylinder can be selected for its geometry. If you have a Named Selection with two cylinders, one of which is suppressed, that particular Named Selection is still not valid for the bolt pretension load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Sending Named Selections to the Solver

Named Selections objects provide the property **Send to Solver**. This property enables you to choose (**Yes** by default or **No**) whether a Named Selection is sent to the solver. Named Selections scoped to edges/vertices/faces are sent to the solver as nodes and Named Selections scoped to bodies are sent to the solver as elements. If your Named Selection is scoped to an element face, an additional property becomes available: the **Send As** property. Using this property, you can send the element face-based Named Selection to the solver as nodes (default) or you can specify that it is sent as **Mesh200** elements that converts the scoped element faces to **Mesh200** elements and sends those elements as a component to the solver.

Note:

When the application sends a named selection scoped to bodies to the Mechanical APDL solver, elements with the element types equal to those associated with the scoped bodies are selected using the **ESEL** command. If element types are modified for these elements using a command snippet, the named selection scoped to these bodies may not select the correct set of elements. In this case, Ansys recommends that you convert the bodies into elements in Mechanical and scope the named selections directly to the elements.

Protecting Named Selections

Named Selections objects provide the property **Protected**. Setting this property instructs the mesher to respect the geometry features (bodies, faces, edges, or vertices) that the Named Selection object is scoped to.

You can choose **Program Controlled** (default), **Yes**, or **No** to determine whether a Named Selection is to be protected. When **Program Controlled** is selected, the scoped object(s) will be protected if the Named Selection is used for boundary conditions, symmetry, other types of loads for the solver, as well as match controls and hard sizing controls for meshing. The scoped objects will not receive additional protection by the mesher if the Named Selection is not used for one of these controls.

Including Named Selections in Program Controlled Inflation

By default, faces in Named Selections are not selected to be inflation boundaries when the **Use Automatic Inflation** control is set to **Program Controlled**. However, you can select specific Named Selections to be included in **Program Controlled** inflation. To do so:

1. Create a Named Selection. (p. 1057)
2. Click the desired Named Selection in the tree and then in the Details view, set the **Program Controlled Inflation** option to **Include**.
3. In the mesh controls, set the **Use Automatic Inflation** control to **Program Controlled**. As a result, the Named Selection you chose in step 2 is selected to be an inflation boundary, along with any other faces that would have been selected by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

LS-DYNA Named Selection IDs

The **LS-DYNA Named Selection User ID** field is only visible when a Named Selection is created while performing an analysis using an LS-DYNA system in Workbench. This field allows you to define an id that will be used by the LS-DYNA solver for the Named Selection that you created. For more information, see Defining Named Selections in the *LS-DYNA User's Guide*

Importing Named Selections

You can import Named Selections that you define in your CAD application, SpaceClaim, or DesignModeler. Named Selections are often imported to scope specific geometric or mesh entities to loading and boundary conditions.

SpaceClaim or DesignModeler

Mechanical automatically imports Named Selections created in SpaceClaim or DesignModeler.

CAD Applications

To import a Named Selection from a CAD system:

1. Create an analysis system and display the properties pane for the **Geometry** cell.
2. Under the **Basic Geometry Options** category, select the checkbox for the **Named Selections** property. The **Named Selections Key** property displays automatically. The property contains the default key from the global preferences, originally set to "NS." This key can include multiple entries, with each separated by a semicolon (i.e. entry1;entry2;entry3). Named selections in the CAD will be transferred if it starts with the same character set as any of the entries in the key. There are also several additional keywords that can be included (e.g. Color, Layer) that will transfer named selection data from CAD. See individual geometry interface support pages for the supported keywords.
3. Launch Mechanical. A **Named Selections** folder is automatically inserted into the **Outline** pane and includes all of the imported named selections.

Note:

There is an **Options** dialog preference for the default behavior between Named Selections imported from your CAD application and how Mechanical treats the Named Selection if it is deleted in the upstream CAD system. The **Delete Missing Named Sections from CAD** setting enables you to maintain a reference between upstream Named Selections and the downstream scoping specified in Mechanical. See the **Import** category of the Geometry (p. 225) preference group for more information.

Exporting Named Selections

You can export the Named Selection that you create and save the contents to a text or Microsoft Excel file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To export the Named Selection object:

1. Right-click the desired Named Selection object and select **Export**.
2. Name and save the file.

The text or Microsoft Excel file you export includes a list of generated node ids, by default. You can also include the location information of the generated node ids in the exported file. To include node id location information in the exported file:

1. Open the **Options** dialog.
2. Select the **Export** option.
3. Under **Export**, click the **Include Node Location** drop-down list, and then select **Yes**.

Note:

- The Named Selection **Export** feature is available only for node-based and element-based Named Selection objects.
- Node Numbers are always shown in the exported text or Microsoft Excel file irrespective of setting for **Include Node Numbers** in **Options** > **Export**.

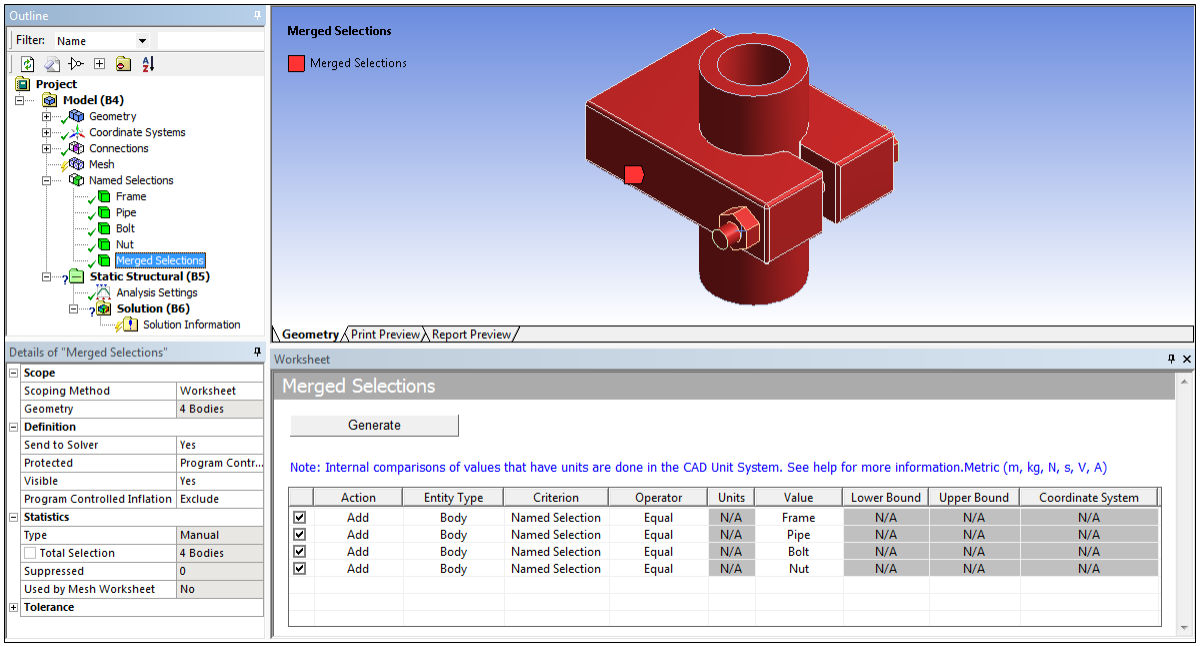
Merging Named Selections

Selecting more than one Named Selection in the tree and then right-clicking makes the **Merge Selected Named Selections** option available. This option creates a new Worksheet-based Named Selection (p. 1061) for the chosen Named Selection objects. The **Worksheet** displays automatically below the **Geometry** window when you use this option or when the merged Named Selection is picked in the tree.

A completed merge is illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Converting Named Selection Groups to Mechanical APDL Application Components

When you write a Mechanical APDL application input file (p. 1858) that includes a Named Selection group, the group is transferred to the Mechanical APDL application as a component provided the name contains only standard English letters and numbers, spaces, underscore, hyphen, and/or parenthesis characters. The Named Selection will be available in the input file as a Mechanical APDL component for use in a **Commands** (p. 2303) object. Geometry scoping to bodies will result in an element-based component. All other scoping types will result in a nodal component. The following actions occur automatically to the group name in the Mechanical application to form the resulting component name in the Mechanical APDL application:

- A name exceeding 32 characters is truncated.
- A name that begins with a number, space, hyphen, or parenthesis character, is renamed to include "C_" before the character.
- Spaces, hyphens, or parentheses between characters in a name are replaced with underscores.

Example: The Named Selection group in the Mechanical application called `1 Edge` appears as component `C_1_Edge` in the Mechanical APDL application input file.

Note:

Named selections starting with ALL, STAT, or DEFA will not be sent to the Mechanical APDL application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Specifying Remote Points

You use a **Remote Point** as a scoping (p. 1497) mechanism for remote boundary conditions (p. 1735). Remote points are a way of abstracting a connection to a solid model, be it a vertex, edge, face, body, node, or element face to a point in space (specified by the **Location** property). The solver uses multipoint constraint (MPC) equations to make these connections.

The following sections describe how to create and define a Remote Point as well as the characteristics and limitations associated with this scoping tool.

Remote Point Application

Geometry Behaviors

Support Specifications

Remote Point Features

Background

Remote Points are akin to the various remote loads available in the Mechanical application. Remote boundary conditions create remote points in space behind the scenes, or, internally, whereas the Remote Point objects define a specific point in space only. As a result, the external Remote Point can be associated to a portion of geometry that can have multiple boundary conditions scoped to it. This single remote association avoids overconstraint conditions that can occur when multiple remote loads are scoped to the same geometry. The overconstraint occurs because multiple underlying contact elements are used for the individual remote loads when applied as usual to the geometry. When the multiple remote loads are applied to a single remote point, scoped to the geometry, the possibility of overconstraint is greatly reduced.

Remote Points are a powerful tool for working with and controlling the Degrees of Freedom (DOF) of a body. Remote Points provide a property, **DOF Selection**, which gives you a finer control over the active DOFs used to connect the Remote Point location to the body.

Furthermore, Remote Points can be can be used independently, without being scoped to a boundary condition. Remote Point create MPC equations and therefore can be used to model phenomena, such as coupling a set of nodes so that they have the same DOF solution.

Another capability of Remote Points is that they are also a scoping mechanism for the Constraint Equation (p. 1702) object. The equation relates the degrees of freedom (DOF) of one or more remote points

A Remote Point or multiple remote points work in tandem with the following boundary conditions:

- Point Mass (p. 901)
- Thermal Point Mass (p. 905)
- Joints (p. 1292)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Spring (p. 1395)
- Bearing (p. 1413)
- Beam Connection (p. 1404)
- Remote Force (p. 1552)
- Moment (p. 1574)
- Remote Displacement (p. 1674)

These objects acquire data from remote points and eliminate the need to define the objects individually. You can scope one or more of the above objects to a defined Remote Point. This provides a central object to which you can make updates that will affect the scoping of multiple objects.

Note:

Following are important points to keep in mind when using **Remote Points**:

- A **Remote Point** can reference only one **Remote Force** and one **Moment**. If you scope a **Remote Point** to multiple remote forces or moments, duplicate specifications are ignored and a warning message is generated.
- A **Remote Point** with **Deformable** behavior should not be used on surfaces that are modeled with symmetry boundary conditions. The internally generated weight factors only account for the modeled geometry. Therefore, remote points with deformable behavior should only be used on the "full" geometry.
- **Remote Points** create MPC equations. If a model has constraint equations with too many terms, it is ill-posed, requires significantly higher memory requirements and solution times, and may never converge. The program automatically terminates the solution of such models with an error message stating "If you wish to bypass this message to allow the solution to continue with increased solution time and memory requirements, issue the **CEFILTER**,OFF command."

For additional Mechanical APDL specific information, see the Multipoint Constraints and Assemblies section as well as KEYOPT(2) in the Mechanical APDL Contact Technology Guide.

Remote Point Application

To specify a **Remote Point**:

1. Select the **Model** object and then select the **Remote Point** option from the **Model** (p. 48) Context tab.

   Or, you can right-click and select **Insert** > **Remote Point**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. Specify the scoping. Options include **Geometry Selection** (default), **Named Selection**, **Remote Point**, **Remote Points and Nodes**, or **Free Standing**. See the descriptions below.

Note:

- To select nodes, you first need to generate the mesh.
- This scoping option is not supported for the LSDYNA Solver.

Mechanical APDL Reference

When you scope your **Remote Point** to a single node or multiple nodes, a point-to-surface contact algorithm is used (using contact element CONTA175). This process can produce slightly different result at the area of application compared to face scoping of the same topology. Geometry scoping to 3D faces and 2D edges uses a constant traction contact application (contact elements CONTA171 through CONTA174). However, a Remote Point scoped to one or more vertices of a 2D or a 3D solid does not use MPC-based contact. Instead, beam elements are created by the solver to connect the vertex to the Remote Point. Furthermore, when scoped to the vertex of a line body, the application uses the element TARGE170 in (rigid) line form.

Note:

Be very careful when you scope remote points to nodes if the nodes are collinear. A rigid Formulation avoids issues when you scope to Surface or Line bodies. However for solids, you should not scope collinear nodes for any Formulation.

Details Properties

Remote Point definable properties are listed and described below:

- **Scoping Method**: Options include **Geometry** (default), **Named Selection**, **Remote Point**, **Remote Points and Nodes**, or **Free Standing**.

Note:

When you specify **Free Standing**, the **Geometry**, **Pinball Region**, **Behavior**, and **DOF Selection** properties **do not** appear in the Details. Free Standing Remote Points can be used to model structures such as Tuned Mass Dampers by directly connecting pieces of your model together.

- **Geometry**/**Named Selection** (geometry or node-based) selection. These options support face, edge, vertex, node, or element face selection/specification.
- **Outline Selection**: When you specify **Remote Point** or **Remote Points and Nodes** as the **Scoping Method**, this property displays. It is scoped to an existing **Remote Point**.

1. Select the entry field of the property and then select the desired **Remote Point** objects from the **Outline**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. Click the **Apply** button that displays automatically. Once specified, this field displays the number of **Remote Point** objects you have selected (for example, **1 Object**, **2 Objects**, etc.).

Note:

During Transient Thermal analyses, an error condition could arise for a **Remote Point** scoped to multiple **Remote Point** objects.

- **Nodes**: When you specify **Remote Points and Nodes** as the **Scoping Method**, this property displays. This option enables you perform direct node selection for the Remote Point in addition to the scoping of one or more **Remote Point** objects.
- **Coordinate System**: the Coordinate System based on the original location of the remote point. This property does not change if you modify the remote point's position with the **Location** property.

Note:

When you scope a **Remote Point** to a load or a result (**Scoping Method** property is set to **Remote Point**), the direction used to calculate the load or to evaluate the result is based on the **Coordinate System** property of the **Remote Point** object. This also applies to when Remote Points like this are used with a Commands (APDL) object.

- **X Coordinate**: the distance from the coordinate system origin on the x axis.
- **Y Coordinate**: the distance from the coordinate system origin on the y axis.
- **Z Coordinate**: the distance from the coordinate system origin on the z axis.
- **Location**: the location in space of the remote point. This property enables you to manually modify the remote point's original position. Changing the **Location** property re-plots the x, y, and z coordinate locations, but does **not** establish a new coordinate system, reflected by the **Coordinate System** property.

  For a **Free Standing** remote point, use this property to define the remote point's position in space.

Important:

When you first scope a **Remote Point** and you **do not** also define the **Location** property, the application sets the position of the remote point to the centroid of the scoped geometry selection(s) or if scoped to multiple Remote Points, the centroid of those Remote Points. Any subsequent scoping changes **will not** change this position. You must update the Remote Point's location as needed.

- **Behavior** (p. 1095). Specify the behavior of the connection of the Remote Point to the model. Options include **Deformable**, **Rigid**, **Coupled**, or **Beam**.
- **Formulation**: This property enables you to specify the contact algorithm the application uses for a particular remote point computation. Property options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **MPC** (default): This option creates multipoint constraint equations internally during the Mechanical APDL solution to tie the bodies together.

– **Lagrange Multiplier**: This option enforces zero penetration when the contact is closed, making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. This formulation helps to overcome over-constraint problems better than the MPC formulation.

Note:

– The **Formulation** property is not supported when the **Behavior** property is set to **Beam**.

– The **Lagrange Multiplier** option is not supported when the **Behavior** property is set to **Coupled**.

– The **Lagrange Multiplier** option is not supported when you deactivate any DOF for target elements using the **DOF Selection** property (KEYOPT(4)).

For more information, see the Selecting a Contact Algorithm (KEYOPT(2)) topic in the *Mechanical APDL Contact Technology Guide*.

- **Relaxation Method**: This property is only available when the **Formulation** property is set to **MPC**. Property options include **No** (default) and **Yes**. Setting this property to **Yes** eliminates overconstraints. When MPC-based surfaced-based constraints or rigid bodies are subjected to overconstraint, this method relaxes the constraint between contact-generated internal constraint equations and other constraint equations or Lagrange multipliers.

  For more information, see the Overconstraint Detection and Elimination section of the *Mechanical APDL Contact Technology Guide*.

- **Pinball Region**: The **Pinball Region** for a **Remote Point** is a radius value (length unit) that defines a region for selecting elements to be used by the solver for the Remote Point's scoping.

- **DOF Selection**: Options include **Program Controlled** (default) or **Manual**. This offers an opportunity for better control of which DOF's will activate for corresponding constraint equations. If you specify Manual for **DOF Selection** in an Explicit Dynamics system, the active/inactive settings of the X, Y, and Z Components and the X, Y, and Z Rotations are ignored, because these settings do not play a role in the constraint equations for an Explicit Dynamics system.

Important:

When the **Behavior** property of a Remote Point is set to:

– **Rigid**: The **DOF Selection** property references the active DOF on the scoped geometry. The **Rigid** setting always utilizes six (6) DOF on the Remote Point location. As a result, the **DOF Selection** setting will not necessarily remove entire constraint equations. For these cases, consider setting the **Behavior** property to **Coupled** or use a General Joint. In addition, when the **Coordinate System** property is set to a local coordinate system, make sure that you either

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1) set **DOF Selection** property to **Program Controlled** or 2) set all the DOFs as active when using the **Manual** setting.

Note:

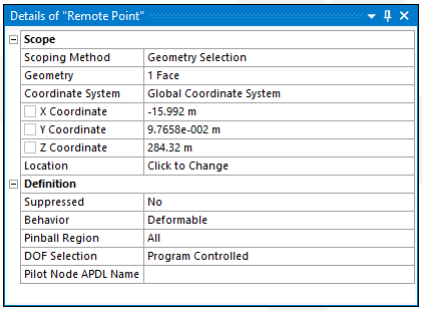
For your selected geometry, active DOFs use the **Global Coordinate System**, not the system specified by the **Coordinate System** property. As a result, you could see unexpected results unless all DOFs are set to **Active**. To achieve Rigid behavior without setting all DOFs as **Active**, you can specify a **Joint** instead of a **Remote Point**.

– **Deformable**: The DOF Selection property references the active DOF on the Remote Point location.

Use caution when using the **Manual** setting of the property. The difference between the location of active DOF Selection based on the **Behavior** property setting causes the Remote Point to behave differently.

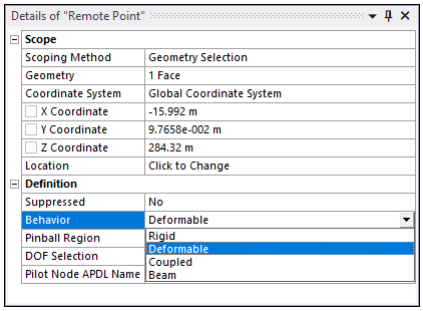
- **Pilot Node APDL Name**: This optional property enables you to create an APDL parameter (in the input file) and assign its value to the pilot node number of the Remote Point. This facilitates easy programmatic identification of the Remote Point's pilot node for later use/reference in a Command object (p. 2303).
- **Material**: This property is available when the **Behavior** property is set to **Beam**. Select a material to define material properties for the beam connection of the Remote Point. Density is excluded from the material definition.
- **Radius**: This property is available when the **Behavior** property is set to **Beam**. Specify a radius to define the cross section dimension of the circular beam used for the beam connection of the Remote Point.

Common Properties



Deformable Behavior and Manual DOF Selection Specified

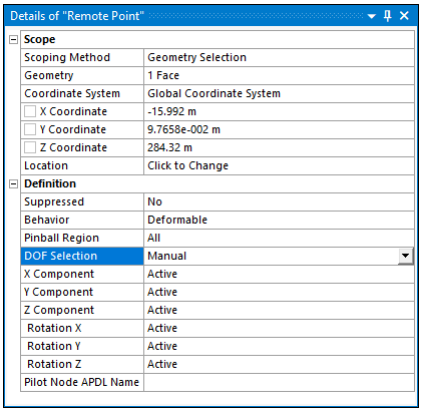
Behavior Options

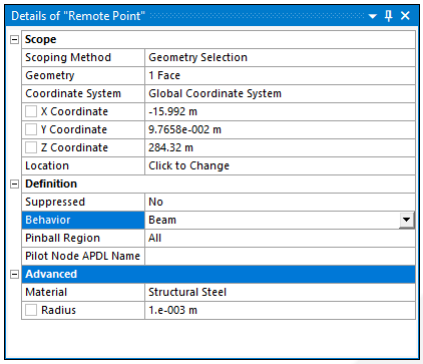


Beam Behavior Specified - Advanced Category Displays

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Geometry Behaviors

The **Behavior** option dictates the behavior of the attached geometry. You can specify the **Behavior** of the scoped geometry for a remote boundary condition in the Details as either **Rigid**, **Deformable**, **Coupled**, or **Beam**.

Note:

The Ansys Explicit Dynamics solver only supports Rigid Behavior.

- **Deformable**: The geometry is free to deform. This is a general purpose option used when applying boundary conditions such as a force or mass through "abstract" entities not explicitly represented as geometry inside Mechanical. This formulation is similar to the Mechanical APDL constraint defined by the **RBE3** command.
- **Rigid**: The geometry will not deform (maintains the initial shape). This option is useful when the "abstracted" object significantly stiffens the model at the attachment point. Note that thermal expansion effects cause artificially high stresses because the geometry cannot deform where the load is applied. This formulation is similar to the Mechanical APDL constraint defined by the **CERIG** command.
- **Coupled**: The geometry has the same DOF solution on its underlying nodes as the remote point location. This is useful when you want a portion of geometry to share the same DOF solution (such as UX) that may or may not be known. For example, to constrain a surface to have the same displacement in the X direction, simply create a remote point, set the formulation to Coupled, and activate the X DOF. Because the DOF is known, you can specify an additional Remote Displacement. This formulation is similar to the Mechanical APDL constraint defined by the **CP** command.
- **Beam**: This option specifies that the Remote Point is connected to the model using linear massless beam elements (BEAM188). This approach is more direct than using Constraint Equations and can help prevent over-constraint issues that can occur with CE's. The following two user-defined properties are available to define the connection:
  - **Material**: Specifies the material properties, except density, that will be used for the beam connections. The application does not include the material properties for the Coefficient of Linear Thermal Expansion by default. As a result, this may develop thermal differential strains when thermal conditions are applied to the model. To address this issue, use Line Bodies with the necessary user-defined material properties instead of a Remote Point.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– **Radius**: Defines the cross section dimension of the circular beam (**CSOLID**) and is sent to the Mechanical APDL solver via the **SECDATA** command.

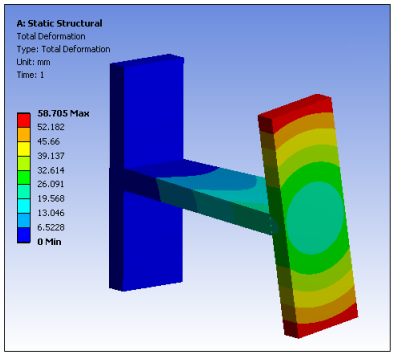
The **Beam** formulation can be useful when working with shells. For example, when you are trying to model Spot Welds (p. 1407) between two sheet bodies with holes.

Important:

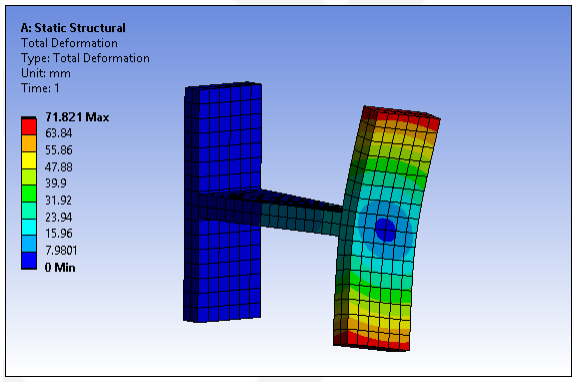
When you apply a thermal load to your model, thermal differential strains can develop when using the **Beam** option. This is because the thermal loading does not become properly associated to the automatically generated beam elements created to make the connection.

You must determine which **Behavior** best represents the actual loading. Note that this option has no effect if the boundary condition is scoped to a rigid body in which case a **Rigid** behavior is always used. Presented below are examples of the **Total Deformation** resulting from the same **Remote Displacement**, first using a **Rigid** formulation, then using a **Deformable** formulation, and finally the **Coupled** formulation.

Rigid Behavior



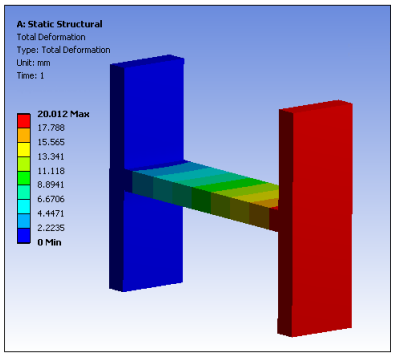
Deformable Behavior



Coupled Behavior

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Support Specifications

Note the following when using the Remote Point feature.

- Mechanical APDL solver logic is based on MPC-based contact. See the Surface-Based Constraints section of the *Mechanical APDL Contact Technology Guide* for more information. However, a Remote Point scoped to a vertex (or vertices) of a 2D or 3D solid does not use MPC-based contact. Instead, beam elements are created by the solver to connect the vertex to the Remote Point.
- The MPC equations are generated from the definition of a Remote Point are based on the underlying element shape functions. In a large deflection analysis, these element shapes functions are reformed at each substep. As a result, MPC equations are superior to the **RBE3**, **CERIG**, and **CP** commands.
- For Remote Boundary Conditions (p. 1735) applied to an edge or edges of a line body that are colinear, the deformable behavior is invalid. As such, the scoped entities exhibit rigid behavior even if a deformable formulation is specified, and a warning is issued in the **Message** Window.
- All remote boundary conditions are associative, meaning they remember their connection to the geometry. Their location however does not change. If you want the location to be associative, create a coordinate system on the particular face and set the location to 0,0,0 in that local coordinate system.
- If the geometry to which a Remote Point is scoped becomes suppressed, the Remote Point also becomes suppressed. Once the geometry is Unsuppressed, the Remote Point becomes valid again.
- Remote boundary conditions scoped to a large number of elements can cause the solver to consume excessive amounts of memory. Point masses in an analysis where a mass matrix is required and analyses that contain remote displacements are the most sensitive to this phenomenon. If this situation occurs, consider modifying the **Pinball** setting to reduce the number of elements included in the solver. Forcing the use of an iterative solver may help as well. Refer to the troubleshooting (p. 2370) section for further details.
- If a remote boundary condition is scoped to rigid body, the underlying topology on which the load is applied is irrelevant. Since the body is rigid, the loading path through the body will be of no consequence; only the location at which the load acts.
- For the explicit dynamics solver, when remote points are scoped to a vertex they will always have a rigid behavior. For other solvers, remote points scoped to a vertex will always have a deformable behavior.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For additional Mechanical APDL specific information, see the Multipoint Constraints and Assemblies section as well as KEYOPT(2) in the *Mechanical APDL Contact Technology Guide*.

Note:

To apply a remote boundary condition scoped to a surface more than once (for example, two springs), you must do one of the following:

- Set scoped surface **Behavior** to **Deformable**.
- Change scoping to remove any overlap.
- Leverage the **Pinball Region** option.

Remote Point Features

Use the following tools to get the most out of the Remote Point feature.

- View Remote Point Connection Lines (p. 1098)
- Promote Remote Points (p. 1099)
- Program Remote Points with Commands Objects (p. 1101)

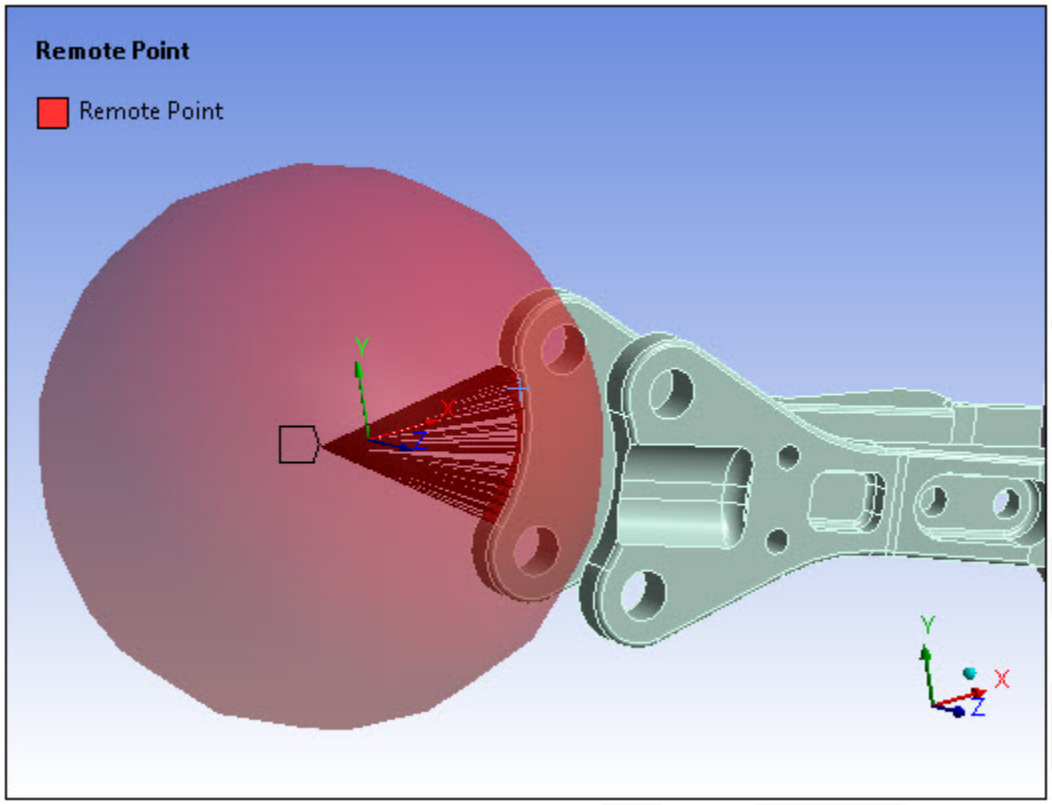
View Remote Point Connection Lines

You can visualize the connection between the underlying geometry associated with a remote point and the remote point itself using the **Remote Point Connections** option of the Style (p. 80) group on the Display tab.

Once you have generated the mesh, the connection lines are drawn between the remote point and the nodes on the corresponding mesh of the underlying geometry. The connection lines take the Pinball radius into account, and only those nodes that are inside that radius will be connected with the remote point. Any remote loads that have been promoted to reference remote points will have these lines drawn when their object is selected as well. An example of connection lines is shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



See the Viewing and Exporting Finite Element Connections (p. 1840) topic in the *Solution Information Object* section of the Help for additional information about the ability to view and work with connection lines.

Promote Remote Points

The **Promote Remote Point** feature is a contextual menu option that enables you to generate **Remote Point** objects from existing, and currently defined, boundary conditions and connection features.

Important:

The promotion action changes the scoping of the corresponding object and may, as a result, cause up-to-date states to become obsolete. Promoted objects associated with a completed solution would cause the solution to become obsolete and require it to be re-solved.

Note:

Automatic Remote Point promotion is available in Harmonic Response or Transient Structural analyses that use a linked Modal system to implement the Mode Superposition (MSUP) solver method for the **Remote Force** and **Moment** loads. To use this feature:

- Select a supported topology (geometric or mesh entity) on your model, and
- Insert a **Remote Force** or a **Moment** load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The application automatically creates (promotes) a corresponding Remote Point for the specified load and scopes the load to that Remote Point. Automatic Remote Point creation improves processing performance by eliminating the need to create internal elements for the load.

The following objects provide the **Promote Remote Point** option:

- Point Mass (p. 901)
- Thermal Point Mass (p. 905)
- Joint (p. 1292)
- Spring (p. 1395)
- Beam Connection (p. 1404)
- Bearing (p. 1413)
- Remote Force (p. 1552)
- Moment (p. 1574)
- Remote Displacement (p. 1674)

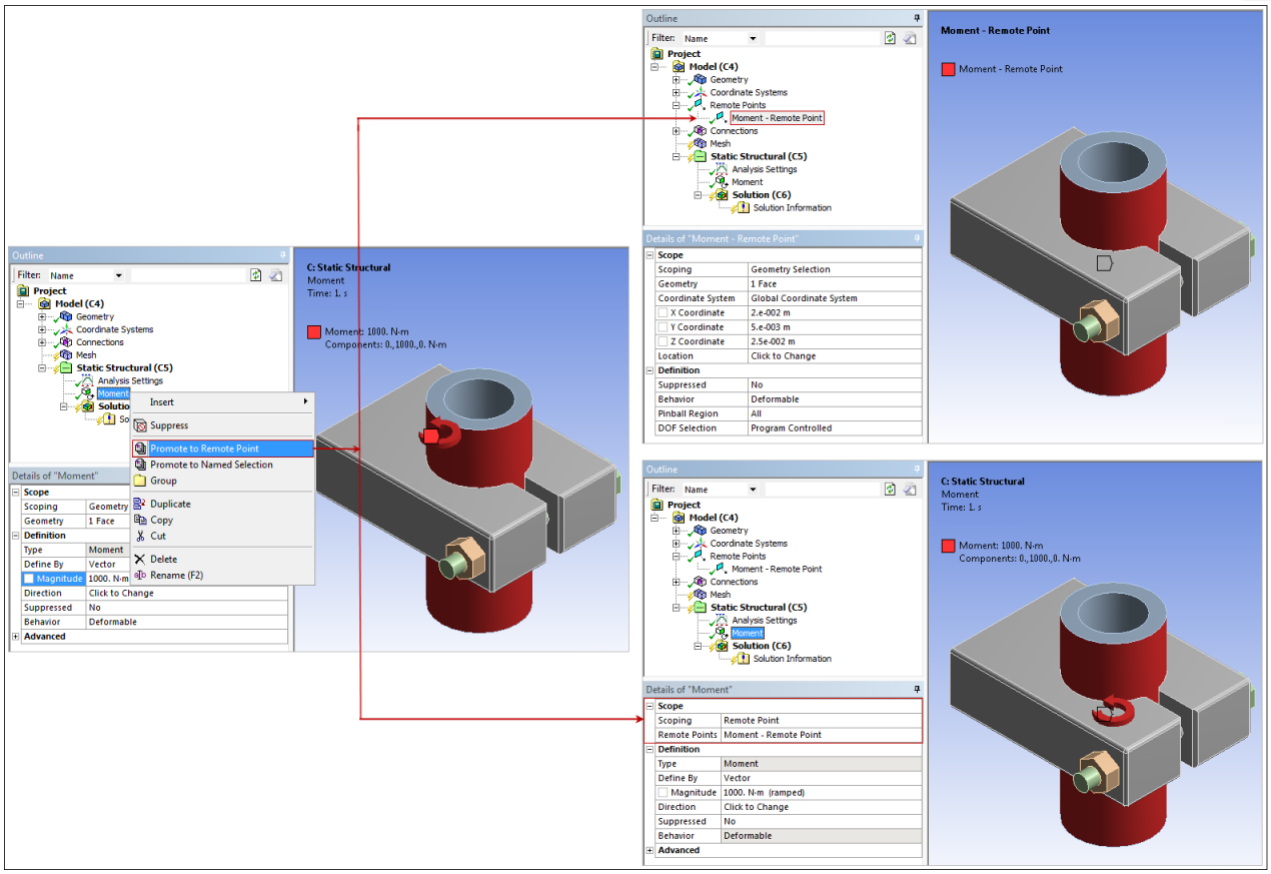
To create a remote point by promotion:

1. Select the object you wish to use to generate a new Remote Point, right-click, and select **Promote Remote Point**. The application adds a new **Remote Point** object to the tree. This new object contains the same scoping as the object used for promotion.

   In addition and as illustrated in the example shown below, it is important to note that the scoping of the source object, the object used to generate the promotion, also changes. The application automatically updates the properties of the **Scoping** category of the source object and specifies the scoping to the new **Remote Point**. Furthermore, multiple **Remote Point** object can be generated based on the scoping of the source object, such as a Body-To-Body **Spring**.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



2. As necessary, you can modify the specifications of the new object.

Note:

This option is not available for objects scoped as a **Direct Attachment**, such as Springs, Joints, Beams, a Point Mass, or a Thermal Point Mass.

Program Remote Points with Commands Objects

A **Commands** (p. 2303) object can be placed in the tree as a child object of a **Remote Point** providing you programmable access to the **Remote Point** pilot node. This is useful if you wish to apply conditions to the **Remote Point** that are not supported in Mechanical, such as beam or constraint equations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Specifying Tables

Overview

Use the **Tables** feature to create multi-variable tables of numeric data. You can then assign tables to specify the following types of loads:

- Pressure (p. 1521)
- Temperature (p. 1598)
- Thermal condition (p. 1594)

These loads represent physical quantities that depend on the values of variables such as spatial coordinates or time. Variables are represented by columns of data in a table. Their values are entered in the rows of the table. Table assignments can optionally be parameterized to explore different design options.

Tables can be created in third-party software such as logging packages, spreadsheets, or mathematical analysis packages, then imported into Mechanical. Table data can also be entered manually. Once a table is defined, the application presents your data in table-array format and plots it in a graph.

To learn more about tables, see the following sections:

Table Requirements and Limitations
Create a Table
Enter and Edit Data in a Table
Work with Tables in Free View
Work With Tables in Unified View
View Table Details
Rename a Table
Delete a Table

Mechanical APDL Notes

Tables are a special type of numeric array that enables the Mechanical APDL solver to calculate (through linear interpolation) the values between the entries in your table The solver applies these interpolated values across the selected geometry when it computes the solution. See the discussion in Array Parameters.

Table Requirements and Limitations

Keep the following in mind when working with tables.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Requirements for Tables

- Table data must be numeric. Your localization settings determine the format of this numeric data.
- A table must include at least one *independent variable*. These variables correspond to **Field** variables in Mechanical APDL and represent independent, measurable quantities such as coordinates (X, Y or Theta, Z), time, and temperature.
- A table must include at least one *dependent variable*. These variables represent real physical quantities (such as pressure and temperature) whose values depend on the independent variables in the table.

Note:

Temperature can be either an independent or dependent variable, depending on whether its value is tied to that of another variable (such as x, y, and z-coordinates).

- Tables can include either the independent variable **Y Coordinate** (that is, cartesian coordinates) or the independent variable **Theta Coordinate** (that is, cylindrical coordinates), but not both.
- The order of columns in a table is not important. Independent and dependent variables can be specified in any order.
- Table rows do not need to be sorted. The solver automatically sorts the data before applying the boundary condition. You can optionally sort table data for display as described in Sort Data by Column Value or Row Number (p. 1116).

Limitations on Tables

- The maximum size of tables depends on your computing resources. You might experience performance issues with very large tables that contain thousands of rows.
- When you assign a table as a boundary condition load, the spatial coordinate system defines the origin for the coordinates in the table. The coordinate system of the table must match the spatial coordinate system. Tables that include **Y Coordinate** as an independent variable (that is, tables with cartesian coordinates) cannot be used with cylindrical spatial coordinate systems. Similarly, tables that include **Theta Coordinate** as an independent variable (that is, tables with cylindrical coordinates) cannot be used with cartesian spatial coordinate systems.
- Mechanical automatically converts table data to the unit system that is selected for the model. If you change the model's unit system, all table data is converted to its equivalent values in the new units of measurement.
- The solver converts all table values to SI units during the solution, regardless of the units assigned in Mechanical.
- If you are performing a two-dimensional analysis, Mechanical ignores the independent variable **Z Coordinate**. If you specify pressure components as dependent variables for a two-dimensional analysis, the application ignores **Pressure_k**. However, both **Pressure_i** and **Pressure_j** must be included in the table.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Create a Table

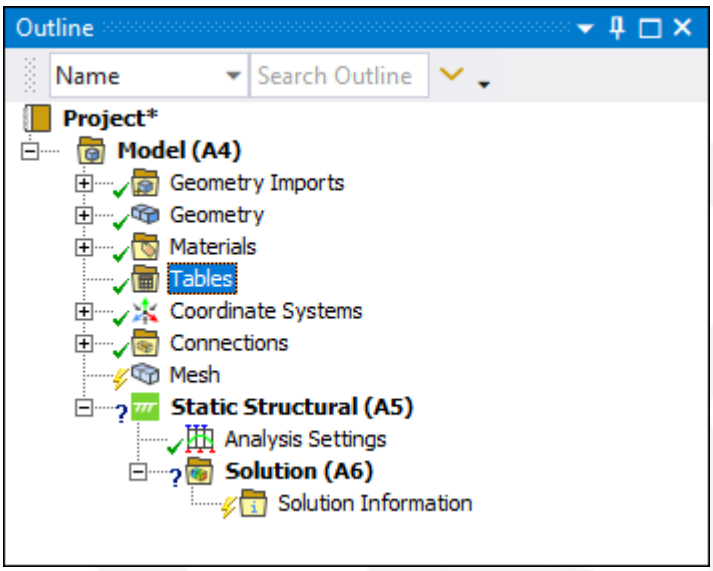
To create a table:

1. Add a **Table** object to the model. Do one of the following:

   - Select the **Model** object and then click the **Tables** icon

 on the Model (p. 48) Context tab.
   - Alternatively, right-click the **Model** object or in the **Geometry** window and select **Insert** > **Tables**.

   The application inserts a Tables folder object in the Outline (as shown below). This folder object houses all Table objects.

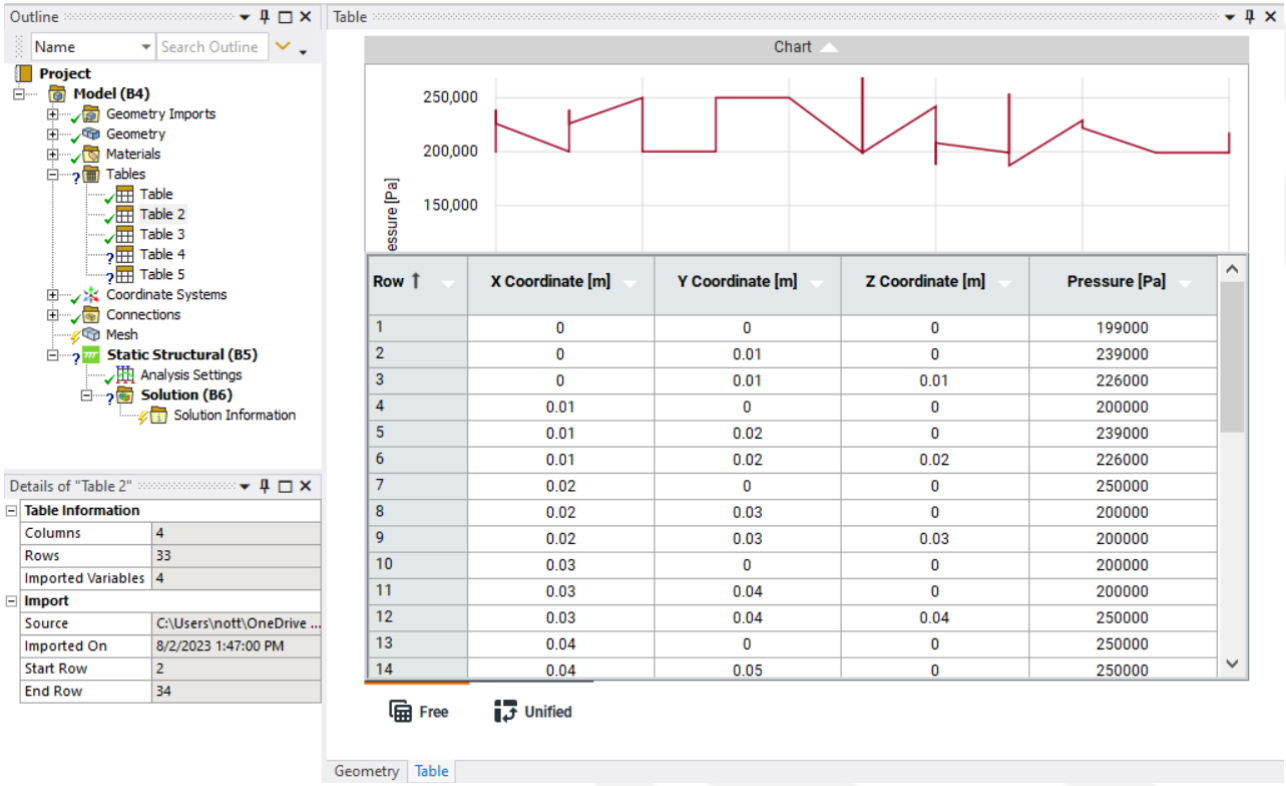


2. The **Tables** object supports two ways to create a table:

   - Create a table from data that is stored in a file. See Import a Table from a Data File (p. 1107) for supported file formats and instructions.
   - Create a table by manually entering data in the application. See Manually Enter a Table (p. 1109) for instructions.

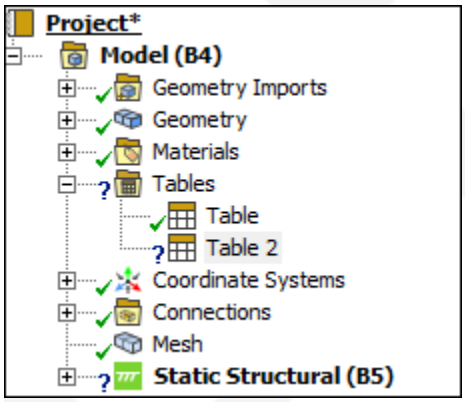
After a table is created, the application displays it as shown below. It also plots a two-dimensional (2D) graph of data in the table. Table data is automatically converted to the equivalent values in the unit system that is selected for the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The new table is listed under the **Tables** object.

- A question mark next to the **Tables** object indicates that one or more tables are incomplete. Required variable values are missing from at least one table.
- A check mark next to the **Table** object indicates that all tables are complete. No table is missing any of its required variable values.
- A question mark next to a table name indicates that the table is incomplete. One or more variable values is missing from that table..
- A check mark next to a table name indicates that the table is complete. Values have been entered for all variables in the table.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Import a Table from a Data File

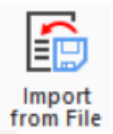
You can create or record a table of data in another application (such as a spreadsheet or logging program), save it to a delimited text file, and import the data file into the Mechanical application.

Table Data File Requirements

Table data files must comply with Table Requirements and Limitations (p. 1103). In addition, they must be in the following format:

- Table data is stored in a delimited text file. Valid delimiters are commas (,), tabs, spaces, and semi-colons (;).
- Data files can optionally contain text header rows at the beginning of the file and text footer rows at the end of the file. However, the table import function ignores these headers and footers. Skip these rows when you import a file (as described below). Otherwise, these rows will be left blank.
- When you import a table from a file, you must specify the units of measurement for its independent and dependent variables. These units can be from a single system of measurement or a mixture different systems of measurement. However, Mechanical automatically converts all numeric values in the table to their equivalent values in the model's current units of measurement.

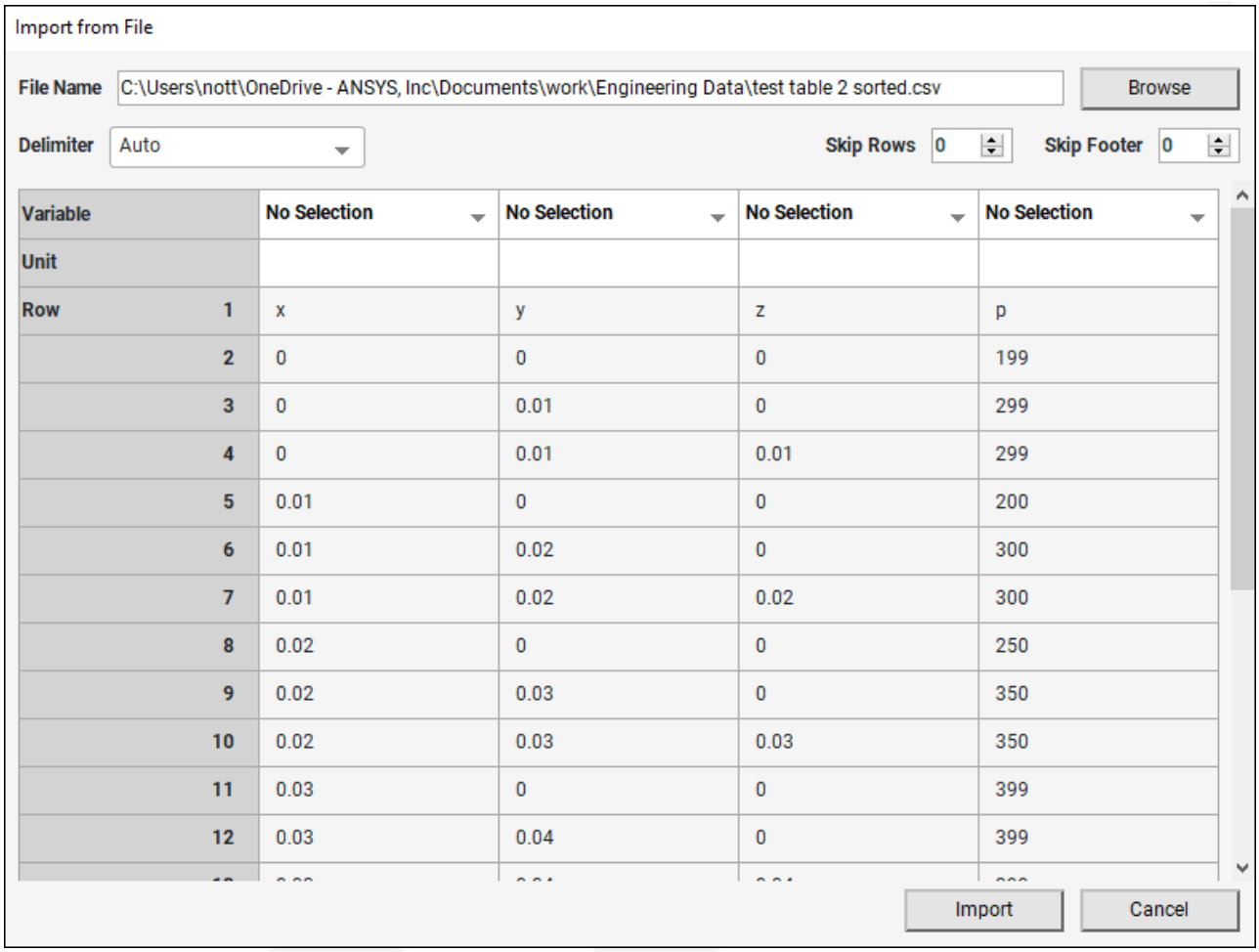
Import a Data File

1. Click the **Import from File** icon 

on the **Tables** Context tab. Alternatively, you can right-click the **Tables** object and select **Table** > **Import from File**.
2. An **Open** dialog appears for you to select a file. Data files must be formatted as described inTable Data File Requirements (p. 1107).

   After you open a data file, the **Import From File** dialog displays as shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Choose the **Delimeter** that separates the entries in your data file.

   - Select **Auto** (the default) to have the table import function automatically identify the delimeter character.

   - Select one of the following to manually identify the delimeter character: **Comma**, **Semicolon**, **Space**, or **Tab**.

4. If your table has one or more rows of header text at the beginning of the file, set the **Skip Rows** option to skip these lines. The table import dialog highlights the skipped header rows in red.

5. If your table has one or more rows of footer text at the end of the file, set the **Skip Footer** option to skip these lines. The table import dialog highlights the skipped footer rows in red.

6. Select the **Variable** type for each column of data from the drop-down menu at the top of the column. Use the search box to filter variable types by name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

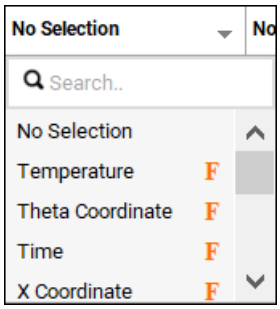
Published: 2023-10-23

- **F** next to the variable name indicates an independent variable. A table must have at least one independent variable.

  Note:

  Independent variables can include either cartesian coordinates or cylindrical coordinates, but not both. If you select **Y Coordinate** as an independent variable, you cannot select **Theta**, and vice versa.

- **R** next to the variable type indicates a dependent variable. A table must have at least one dependent variable.



See Table Requirements and Limitations (p. 1103) for a discussion of independent and dependent variables in tables.

7. Select the **Unit** for each variable from the drop-down menu.

   The available units correspond to the type of variable you selected. For instance, if you picked **Temperature** as the variable type, the **Unit** menu displays units of temperature: **C** (Celsius), **K** (Kelvin), **F** (Fahrenheit), and **R** (Rankine).

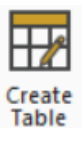
   Note:

   Imported tables can have units from multiple unit systems. Mechanical automatically converts all table values to the unit system that is currently selected for the model.

8. When you finish, click the **Import** button. The application imports the table data and displays it in the Table worksheet.

Manually Enter a Table

1. Do one of the following to add a table to your model:

   - Click the **Create Table** icon 

on the **Tables** Context tab.
   - Right-click the **Tables** object and select **Table** > **Create Table**.

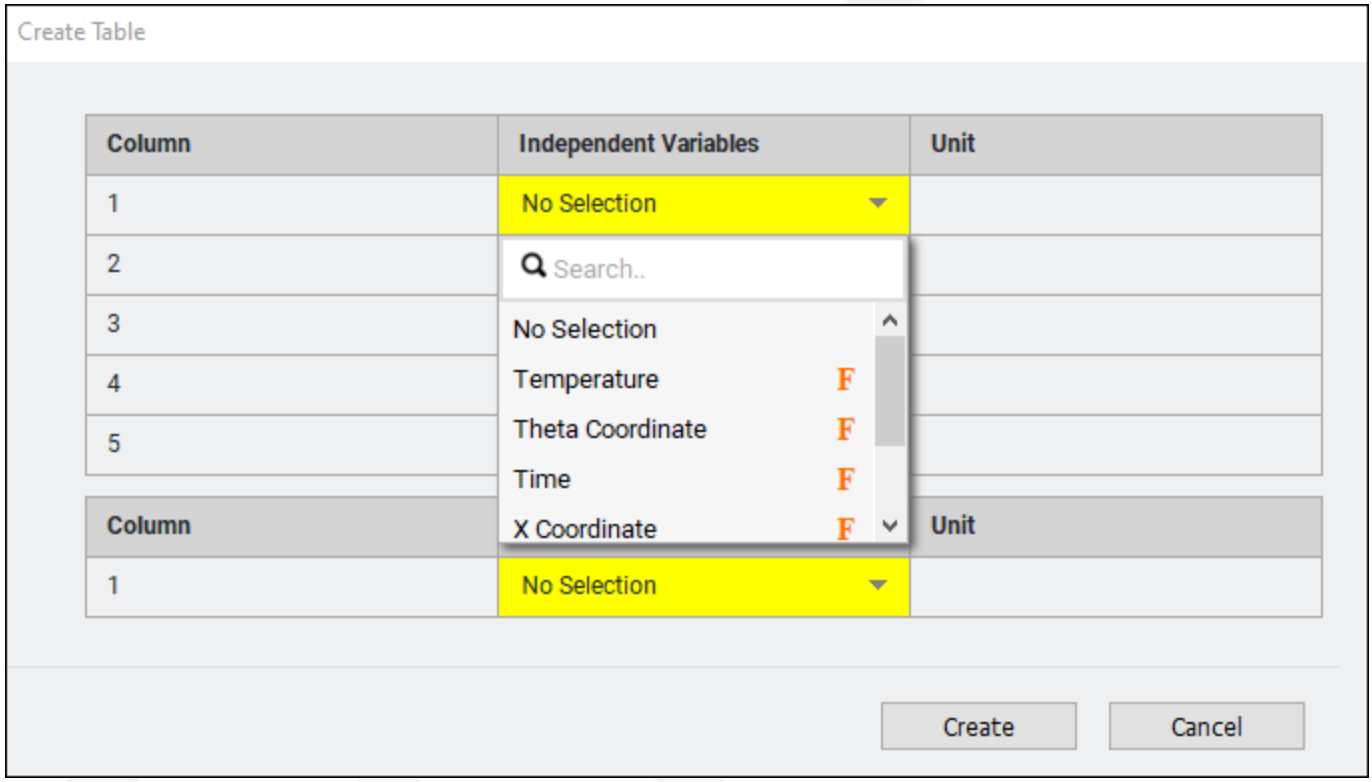
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- When using a table to specify a load, select **Add Table** from the drop-down menu (see Specifying Boundary Conditions with Tables (p. 1521)).

The **Create Table** dialog then displays as shown below.

2. Using the drop-down menus, select the desired independent and dependent variables. Use the search box to filter variables by name.

   - **Independent Variables** are identified with an **F** in the drop-down menu. Your table must contain at least one independent variable.



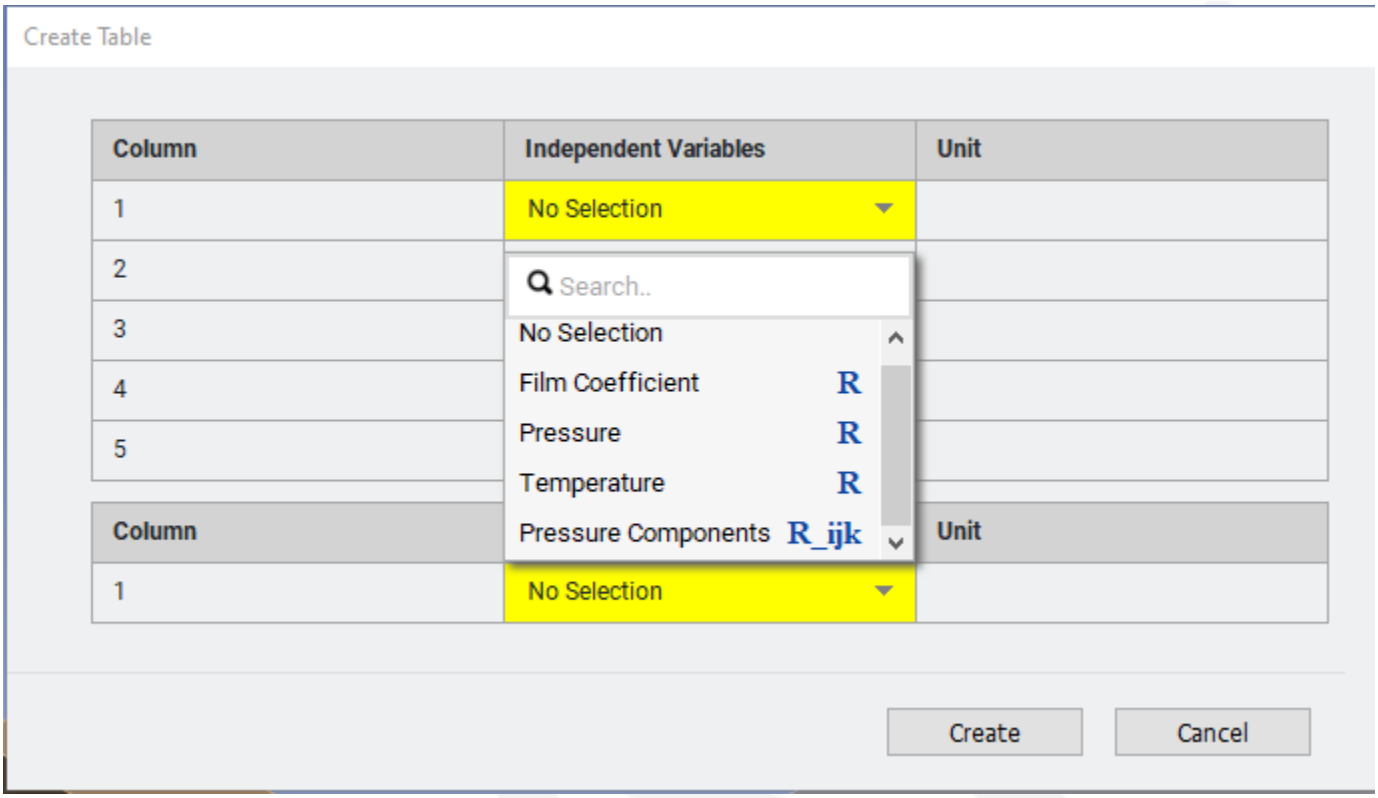
Note:

Independent variables can include either cartesian coordinates or cylindrical coordinates, but not both. If you select **Y Coordinate** as an independent variable, you cannot select **Theta**, and vice versa.

   - **Dependent Variables** are identified with an **R** in the drop-down menu. Your table must contain one dependent variable.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

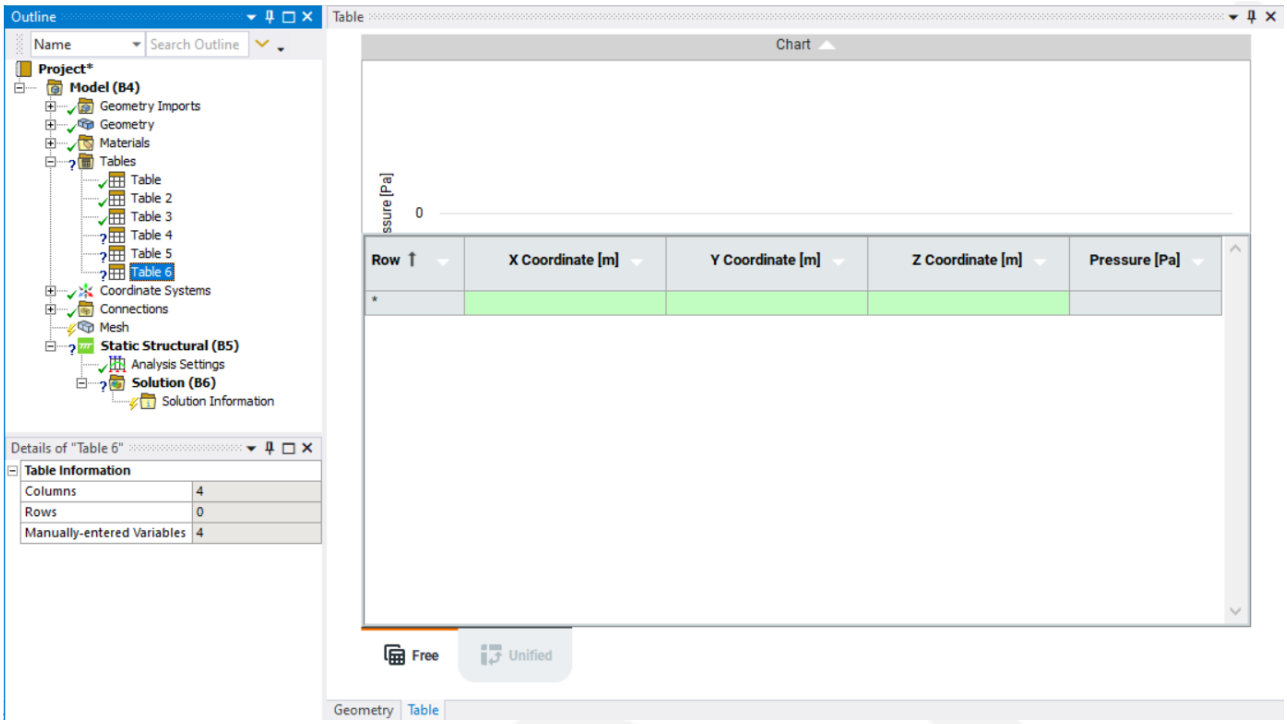
Published: 2023-10-23



When you manually enter a table, Mechanical automatically assigns the appropriate **Unit** to each variable according to the currently-selected unit system (p. 138). For example, if you specified **X Coordinate** as an independent variable and selected **Metric (m, kg, N, s, mV, mA)** as the unit system, the **Unit** is automatically set to meters (**m**).

3. When you finish setting up the independent and dependent variables for the table, click the **Create** button. The Table worksheet appears with the variables you specified. Initially, the table is blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



4. Enter numeric data into the cells of your table as described in Enter and Edit Data in a Table (p. 1112).

Note:

Although you can define only one dependent variable when manually specifying a table, you can enter additional dependent variables after the table is created. To manually add dependent variables to a table:

1. Create the table as described above.
2. For each additional dependent variable, add a new column to the table as described in Add a Column to a Table (p. 1118).
3. Assign dependent variables to those columns as described in Change a Variable for a Column (p. 1117).
4. Enter values for the dependent variables as described in Enter and Edit Data in a Table (p. 1112).

Enter and Edit Data in a Table

After you create a table, you can enter or edit values for its dependent and independent variables. This applies to tables that you imported from a file as well as tables that you created within the application.

Click the mouse in a table cell and enter numeric data.

- Press the **Tab** key to move to the column to the right.
- Press the **Shift** + **Tab** keys to move to the column to the left.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Press the **Enter** key to move to the next row of the same column.

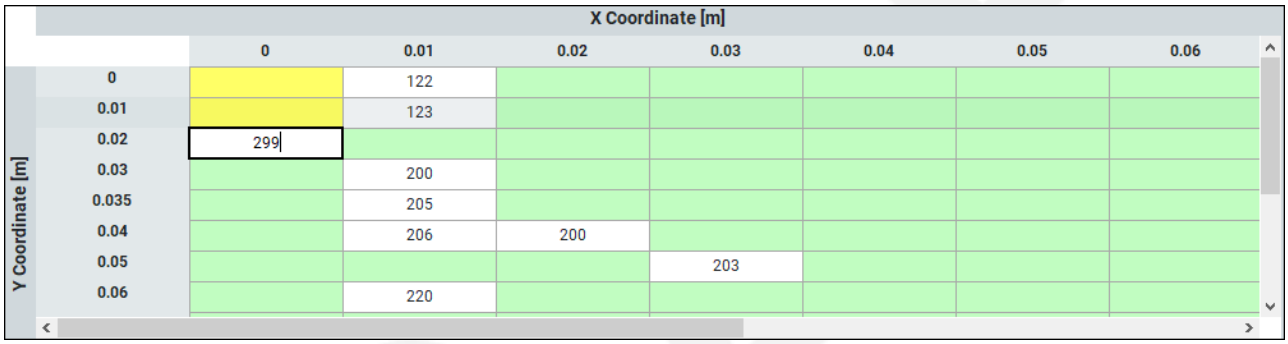
These keys automatically add a new row when the cursor reaches the last row in the table.

Table Cell Colors and Content Status

The color of a table cell indicates the status of its content.

- Green cells show where you can optionally enter new data. This data is added to the table and is not required to complete it.
- Yellow cells show where required data is missing. You must fill in these cells to complete the table.
- Cells that display numerical values and are not highlighted contain valid data.

In the example below, you can add data to the table in the green cells. The yellow cells show missing data. You will need to enter values into these cells to complete the table.



Work with Tables in Free View

Click the **Free** icon

at the bottom of the table window to display a table in *free view* (as illustrated below). The application displays the rows and columns of the table and plots a 2D graph of the first column of independent variables versus the first column of dependent variables.

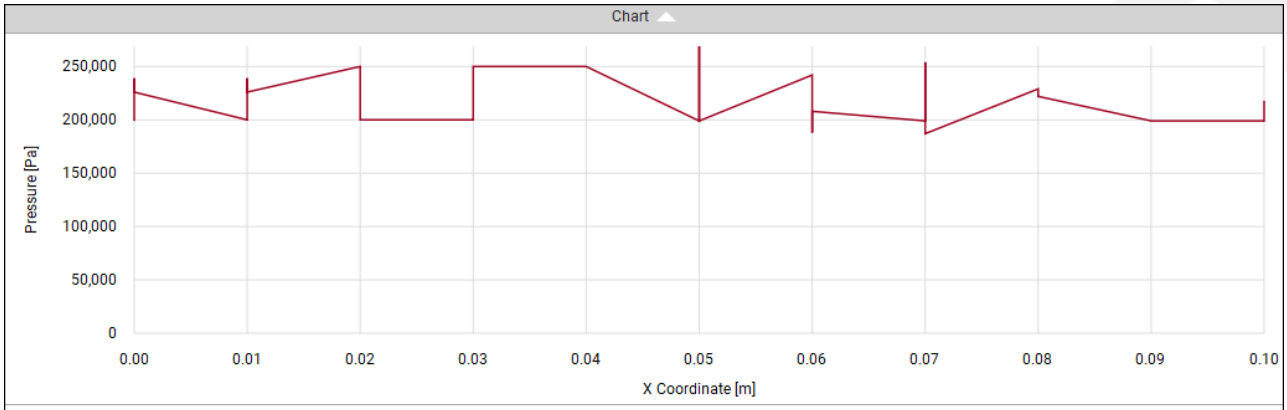
The application remembers which view you selected and will load the table in that view the next time you open it.

You can perform the following operations on a table in free view:

- Enter and Edit Data in a Table (p. 1112)
- Plot Dependent Versus Independent Variables in Free View (p. 1114)
- Hide Plot in Free View (p. 1115)
- Sort Data by Column Value or Row Number (p. 1116)
- Change a Variable for a Column (p. 1117)
- Add a Column to a Table (p. 1118)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Delete a Column from a Table (p. 1119)
- Add a Row to a Table (p. 1119)
- Delete a Row from a Table (p. 1119)

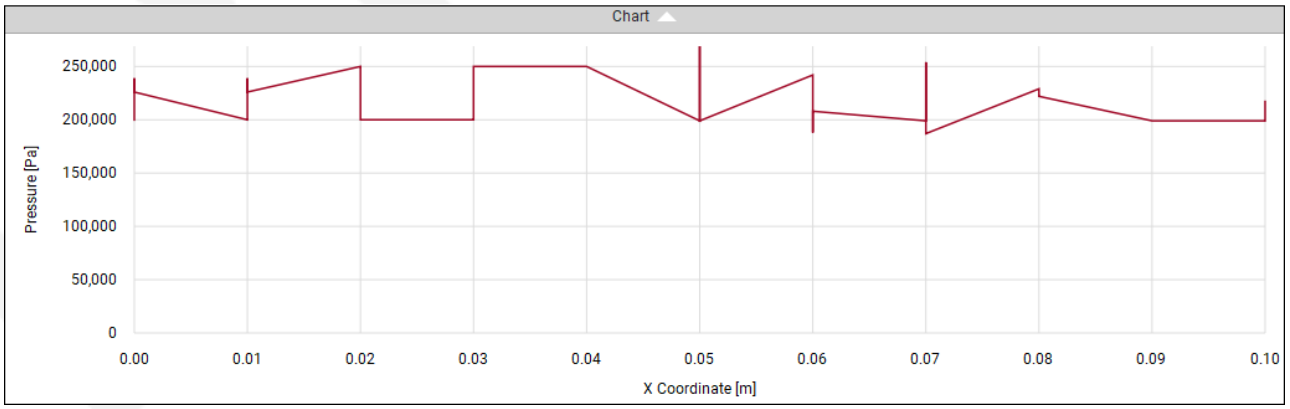


Row	X Coordinate [m]	Y Coordinate [m]	Z Coordinate [m]	Pressure [Pa]
1	0	0	0	199000
2	0	0.01	0	239000
3	0	0.01	0.01	226000
4	0.01	0	0	200000
5	0.01	0.02	0	239000
6	0.01	0.02	0.02	226000
7	0.02	0	0	250000
8	0.02	0.03	0	200000
9	0.02	0.03	0.03	200000
10	0.03	0	0	200000
11	0.03	0.04	0	200000
12	0.03	0.04	0.04	250000
13	0.04	0	0	250000
14	0.04	0.05	0	250000
15	0.04	0.05	0.05	250000
16	0.05	0	0	199000
17	0.05	0.06	0	269000
18	0.05	0.06	0.06	199000
19	0.06	0	0	242000

Free Unified

Plot Dependent Versus Independent Variables in Free View

In free view, you can display a plot of the values of a dependent variable versus those of an independent variable.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

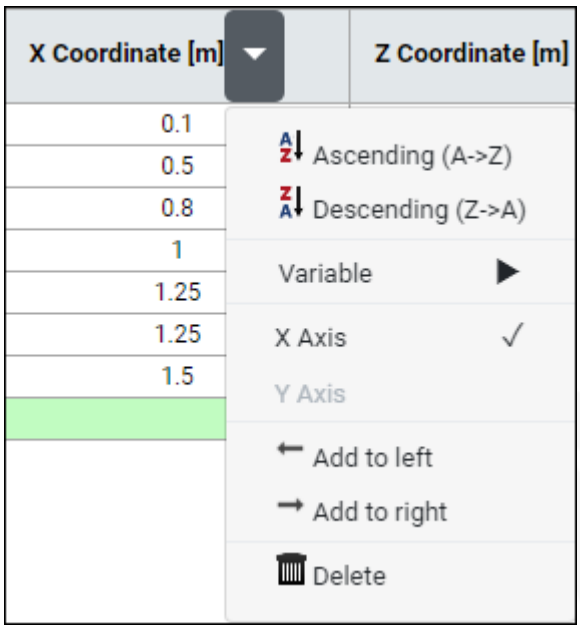
Published: 2023-10-23

Change X and Y-Axis Variables

Initially, the application plots the first column of independent variables on the x-axis and the first column of dependent variables on the y-axis. The plotted variables are labeled on the axes of the plot.

To change which variables are plotted on the x or y-axis:

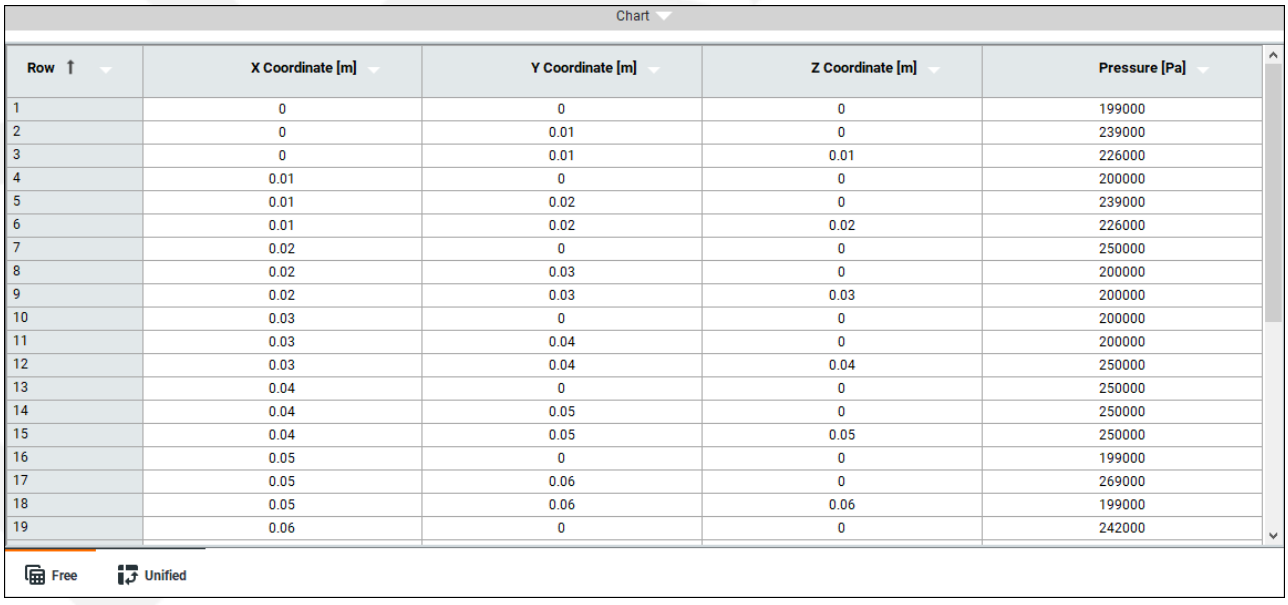
1. Click the drop-down menu arrow next to the variable you want to plot.



2. Select whether you want to plot the variable on the **X Axis** or the **Y Axis**. A check mark appears next to the selected axis.

Hide Plot in Free View

To hide the plot in free view, click the **Chart** arrow at the top of the plot. Only table data is displayed.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To display the plot, click the **Chart** arrow again.

Sort Data by Column Value or Row Number

Tables do not need to be sorted. The solver automatically sorts the data in a table before applying the boundary condition where it is used. However, you can optionally sort table data by row number or column value to make this data easier to visualize and understand.

Select the sort order from the drop-down menu at the top of the desired column.

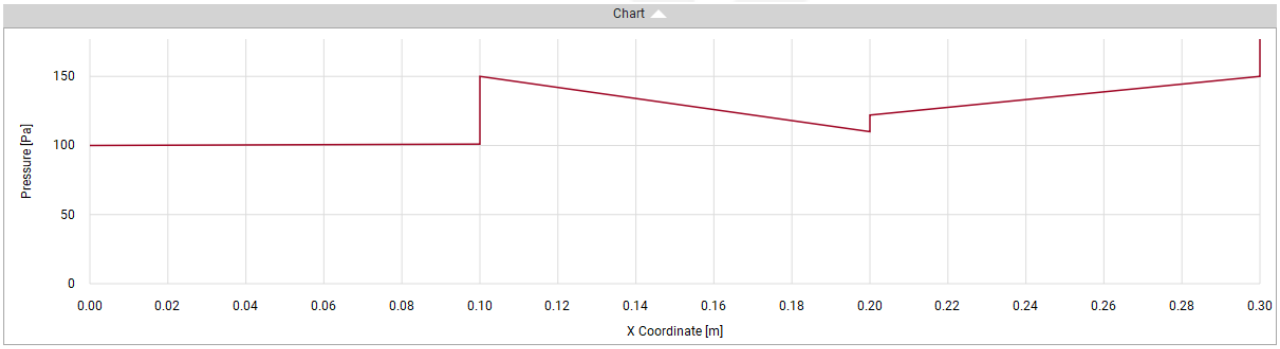
- **Ascending (A->Z)** sorts from lowest to highest values.
- **Descending (Z->A)** sorts from highest to lowest values.
- **None** cancels the sort and returns the data to its original order.

The application remembers the most recent sort order that you specified.

How Sorting Affects Plots

In free view, variable values are plotted in the same order as they are listed in the table. Sorting this data changes how it is plotted, giving you different views into the relationships between the variables in the table.

The plot below shows how pressure varies according to the x-coordinate. This data is plotted according to the original row numbering, which is shown in the leftmost column of the table. Pressure is graphed on the y-axis and the x-coordinate is graphed on the x-axis.

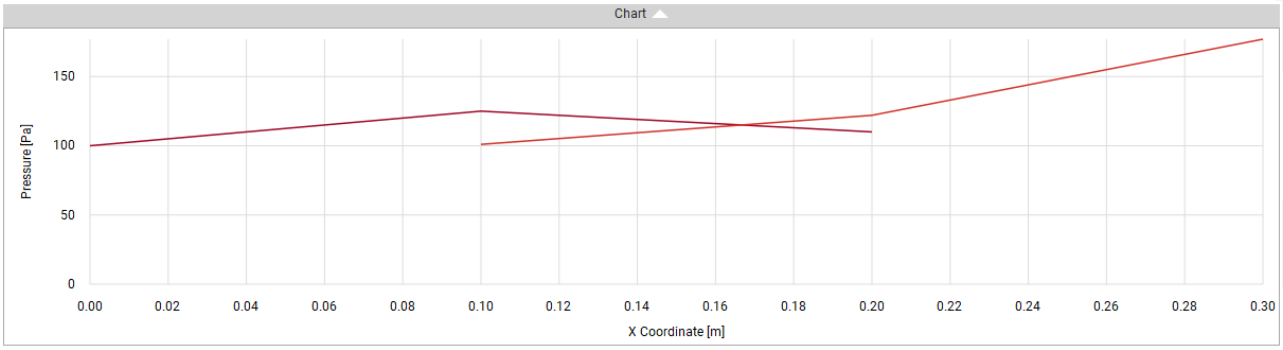


Row	X Coordinate [m]	Time [s]	Pressure [Pa]
1	0	0	100
2	0.1	0.01	101
3	0.1	0	125
4	0.1	0.03	150
5	0.2	0	110
6	0.2	0.01	122
7	0.3	0.03	150
8	0.3	0.02	177
*			

When this data is sorted by time, a very different looking plot appears. The pressure and x-coordinte are still plotted on the graph below, but the table is now sorted by time. This provides insight into how the pressure at different x-coordinates varies over time. For instance, the pressure at x=0.1 m has two different values at two different times.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Row	X Coordinate [m]	Time [s]	Pressure [Pa]
1	0	0	100
3	0.1	0	125
5	0.2	0	110
2	0.1	0.01	101
6	0.2	0.01	122
8	0.3	0.02	177
4	0.1	0.03	150
7	0.3	0.03	150
*			

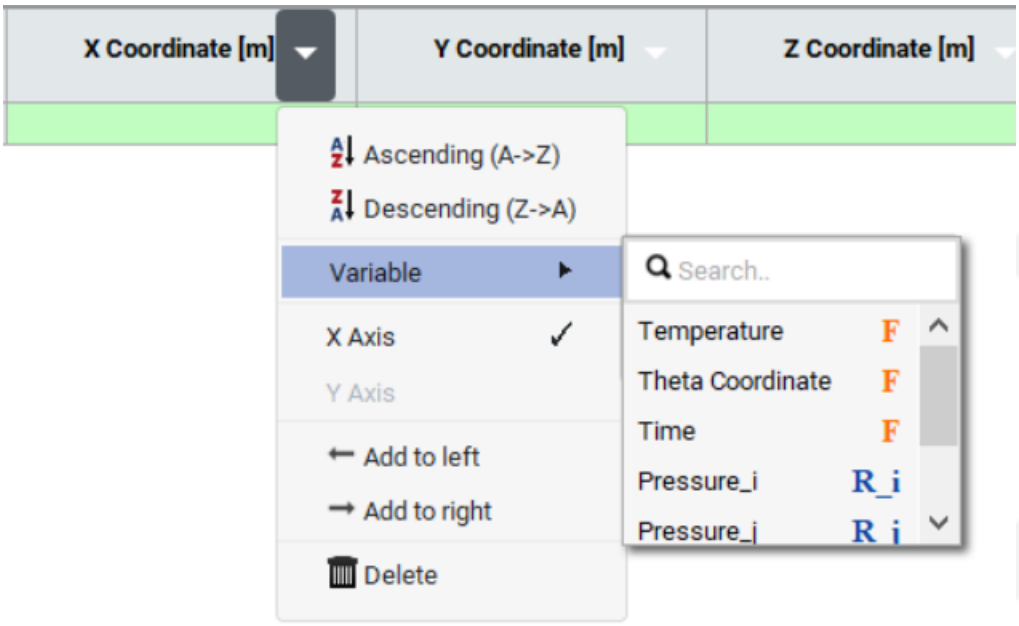
Change a Variable for a Column

The variable that is currently assigned to a column is listed at the top of the column in free view. You can change this variable assignment. This is useful if you add columns to a table (see Add a Column to a Table (p. 1118)) or accidentally assigned the wrong variable to a column when creating a table.

To change the variable assigned to a column:

- Select **Variables** from the column's drop-down menu, then select the new variable from the drop-down list. Use the search box to find variables by name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

This menu only lists variables that have not already been added to the table. The new variable is automatically assigned the appropriate units of measurement for the selected unit system.

Add a Column to a Table

You can add more columns to a table to store additional variables. To add a column to a table:

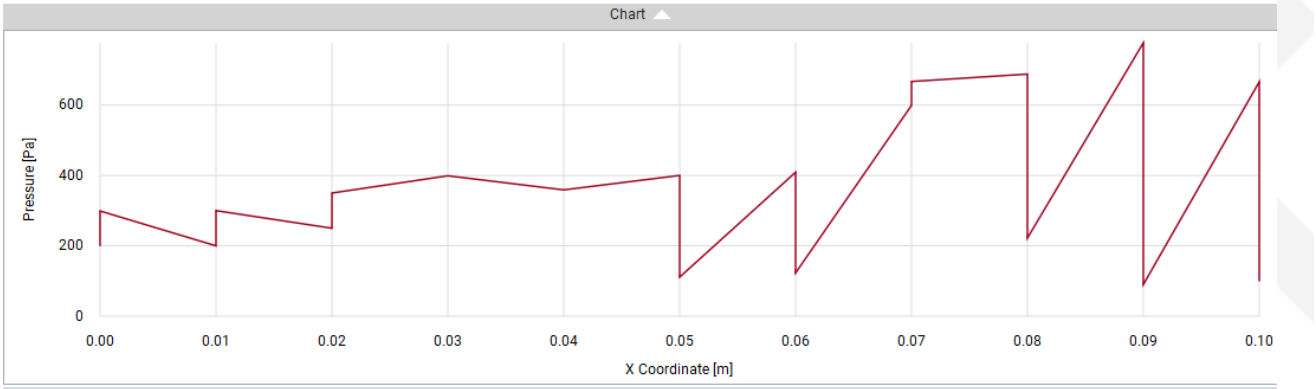
- Click **Add to left** on the column drop-down menu to add a new column to the left of the column.
- Click **Add to right** on the column drop-down menu to add a new column to the right of the column.

The application adds the new column to the table. The cells in the new column are yellow, indicating that their values are required. Enter data into these cells to complete the table.

When you add a column, the application automatically assigns the next independent variable in the list of variables. To change which variable is assigned to the new column, see Change a Variable for a Column (p. 1117). When no more independent variables are available, you cannot add more columns.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Row	X Coordinate [m]	Temperature [C]	Y Coordinate [m]	Z Coordinate [m]	Pressure [Pa]
1	0		0	0	199
2	0		0.01	0	299
3	0		0.01	0.01	299
4	0.01		0	0	200
5	0.01		0.02	0	300
6	0.01		0.02	0.02	300
7	0.02		0	0	250
8	0.02		0.03	0	350
9	0.02		0.03	0.03	350
10	0.03		0	0	399
11	0.03		0.04	0	399
12	0.03		0.04	0.04	399
13	0.04		0	0	359
14	0.04		0.05	0	359

Delete a Column from a Table

To delete a column from a table in free view:

- Click **Delete** on the drop-down menu of the column.

Note:

You cannot delete the last remaining independent and dependent variables in a table.

Add a Row to a Table

You can append rows to the end of a table in free view, allowing you to manually enter additional data. Enter numeric values in the green cells marked with an asterisk (*), which are located at the bottom of the table. Pressing the **Enter** or **Tab** key adds another row to the table.

Row	X Coordinate [m]	Y Coordinate [m]	Z Coordinate [m]	Time [s]	Pressure [Pa]
1	0.01	0	0	0.1	1000
2	0.015	0.011	0	0.2	2000
3	0.02	0.02	0.01	0.3	3000
4	0.03	0.03	0.01	0.4	4000
*					

Delete a Row from a Table

To delete a row from a table in free view:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Right-click the row and select **Remove Row** from the menu.

Work With Tables in Unified View

Click the **Unified** icon to display a table in *unified view*. This view slices table data into a 2D grid according to the value of the variable you select.

In unified view, you can step through slices (or cross-sections) of table data according to the value of the selected variable. Unified view displays a grid of the dependent variable versus other independent variables at each value of the selected variable.

Unified view shows where you may need to add dependent variable values to complete your table and fully model the load that it defines. The order of data in the table does not matter. Unified view shows all dependent variable values that are associated with the selected independent variable values, regardless of their row number or sort order.

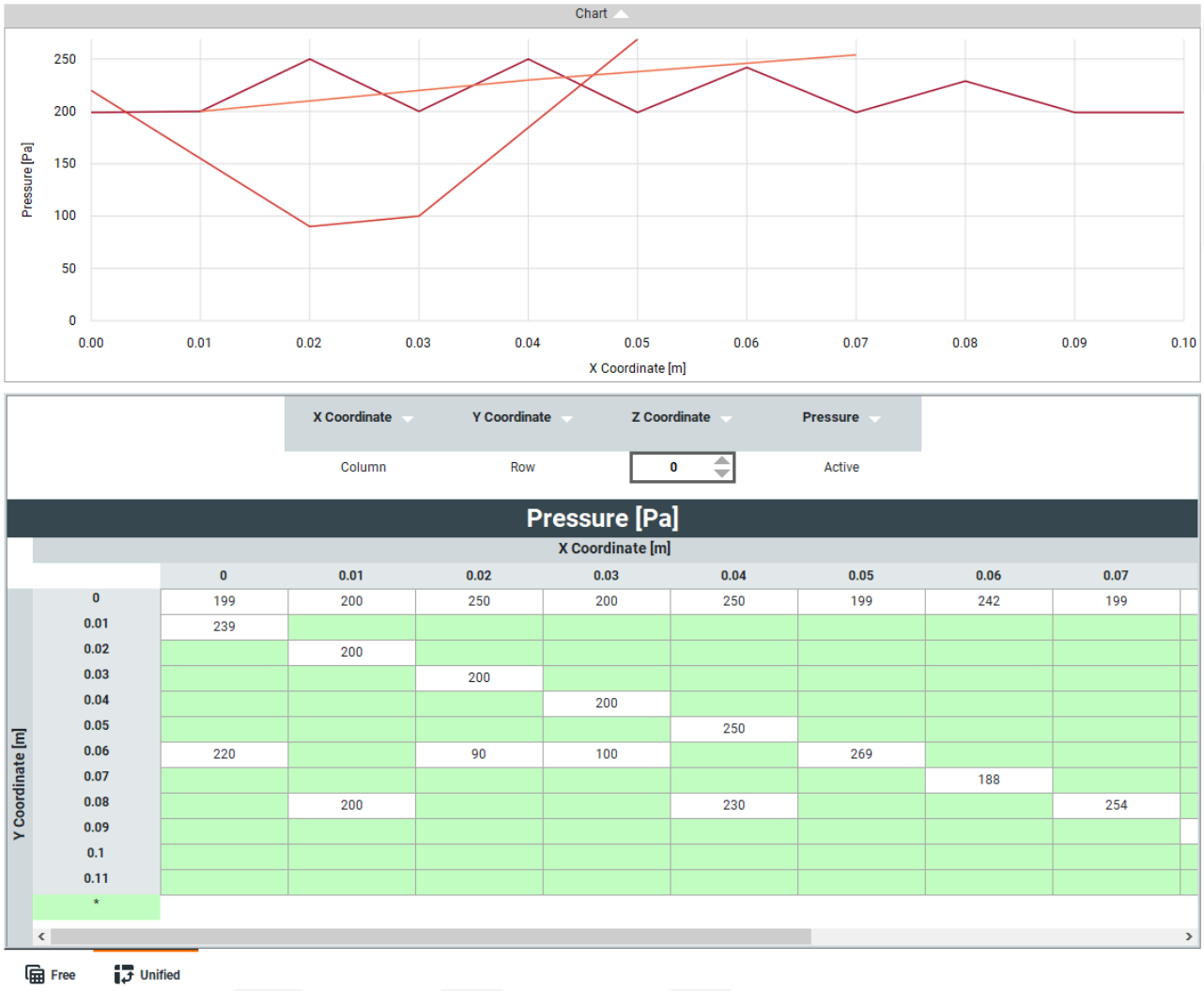
For example, the table below displays the pressure at the **X Coordinate** and **Y Coordinate** where the **Z Coordinate** is equal to zero (0). When you change the value of **Z Coordinate**, the table shows the corresponding pressure values for the **X Coordinate** and **Y Coordinate**.

The application plots the first dependent variable in the table against the independent variable that is selected as a table column. The example below plots pressure versus the x-coordinate, which is identified as the column in unified view.

The application remembers which view you last selected and will load the table in that view the next time you open it.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



You can perform the following operations on a table in unified view:

- Enter and Edit Data in Unified View (p. 1121)
- Plot and Display Variables (p. 1122)
- Select Active Dependent Variable (p. 1122)
- Hide Unified View Plot (p. 1123)

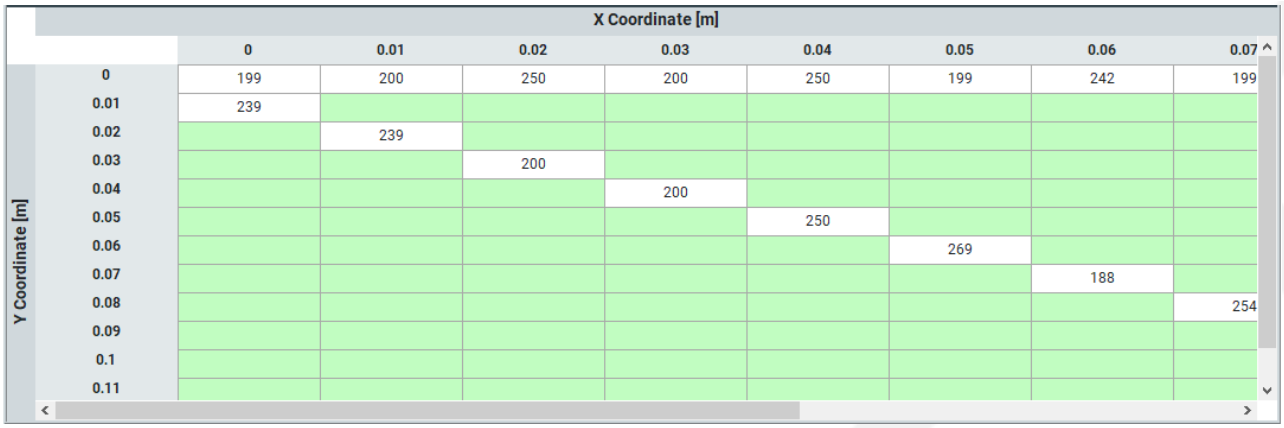
Enter and Edit Data in Unified View

Enter and edit data in unified view in the same way that you enter data in free view. See Enter and Edit Data in a Table (p. 1112) for details.

You can enter dependent variable values into the green cells that correspond to the independent variables that are displayed in the grid. For example, the green cells in the table below identify x and y-coordinates where you can add pressure values to the table. This allows you to easily add values that are missing from your table.

The rows of new data are appended to the end of the table.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Y Coordinate [m] \ X Coordinate [m]	0	0.01	0.02	0.03	0.04	0.05	0.06	0.07
0	199	200	250	200	250	199	242	199
0.01	239							
0.02		239						
0.03			200					
0.04				200				
0.05					250			
0.06						269		
0.07							188	
0.08								254
0.09								
0.1								
0.11								

Note:

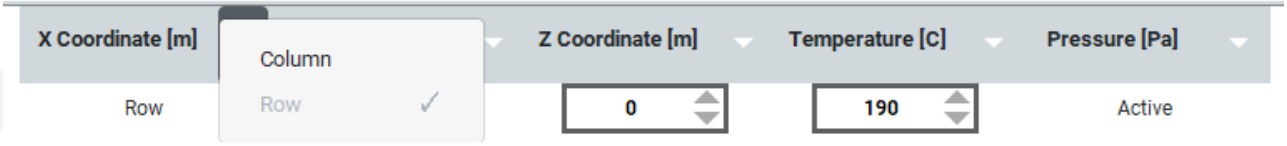
While you can add rows to a table in unified view by entering data in the green cells, you must switch to free view to add columns. See Add a Column to a Table (p. 1118).

Plot and Display Variables

Click the down-arrow next to the name of a variable to change how it is plotted and displayed in unified view.

- Choose **Column** to use the values of that variable as the columns in the unified table grid. The **Column** variable is also plotted on the x-axis of the graph in unified view. If the variable is already selected, **Column** is grayed out and a check mark is displayed next to it.
- Choose **Row** to use the values of that variable as the rows in the unified table grid. If the variable is already selected, **Row** is grayed out and a check mark is displayed next to it

The values of the other independent variables specify which column and row values are plotted and displayed in the table. In the example below, **Z Coordinate** and **Temperature** are not selected as row or column variables. The **X Coordinate** and **Y Coordinate** values at **Z Coordinate** = 0 m and **Temperature** = 190 C are displayed in the table and plotted.



Select Active Dependent Variable

If your table contains more than one dependent variable, you can change which variable is displayed in unified view.

The dependent variable that is currently active – that is, the variable whose values are currently plotted and displayed in the table grid – is listed in the black bar in the middle of the unified display.

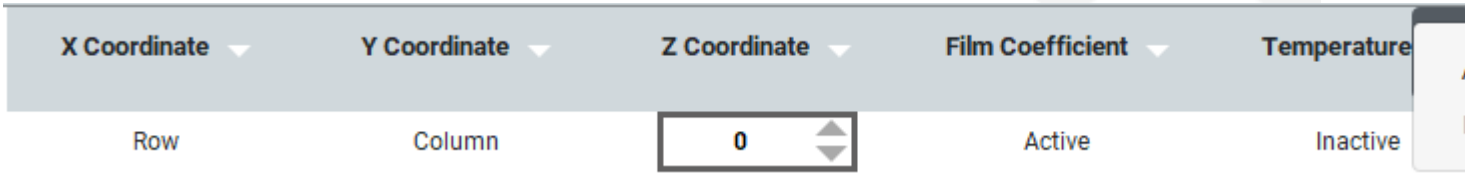
To change the active dependent variable:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

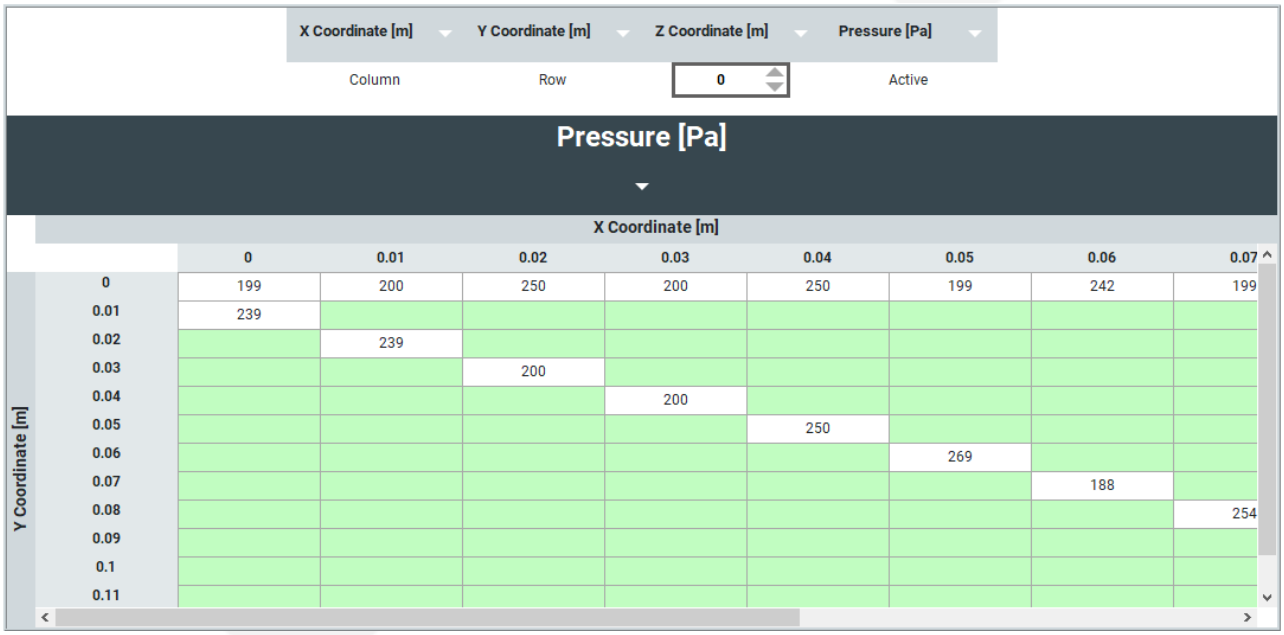
and select a different dependent variable from the Variable menu. Use the search box to find the variable by name. The plot and table now display the values of the newly selected variable.

- Click the down-arrow next to the dependent variable and choose **Active**. The new active dependent variable is shown in the table and plot.
- All other dependent variables are listed as **Inactive**.



Hide Unified View Plot

To hide the plot in unified view, click the **Chart** arrow at the top of the plot. Only table data is displayed.

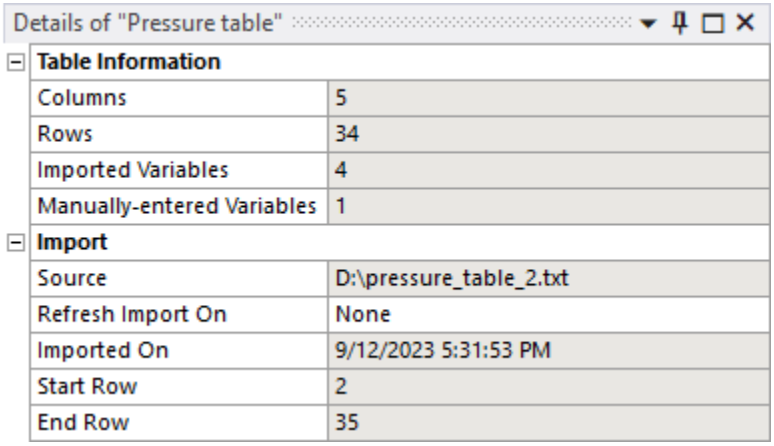


To show the plot, click the **Chart** arrow again.

View Table Details

The application displays the following information about the selected table in the **Details** pane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Details of "Pressure table"

Table Information	
Columns	5
Rows	34
Imported Variables	4
Manually-entered Variables	1
Import	
Source	D:\pressure_table_2.txt
Refresh Import On	None
Imported On	9/12/2023 5:31:53 PM
Start Row	2
End Row	35

Table Information

Columns

Shows the total number of columns in the table.

Rows

Shows the total number of rows in the table.

Imported Variables

If you created the table by importing a data file, this field shows the number of variables that were originally imported in the file. See Import a Table from a Data File (p. 1107).

Manually-entered Variables

Shows the number of variables that you manually added to the table. See Manually Enter a Table (p. 1109) and Add a Column to a Table (p. 1118).

Import

These fields are displayed if you created the table by importing a data file. See Import a Table from a Data File (p. 1107).

Source

Shows the path name of the imported data file.

Refresh Import On

Specifies whether Mechanical reloads the table from the source file before computing a solution. Select one of the following from the drop-down menu:

None

The table is not reloaded (the default). The solver uses the values that currently appear in the table.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Before Solve

Mechanical automatically reloads the table from the source file before the solver starts. This replaces the current numeric values in the table with the numeric values in the source file, which are then used during the solution.

Note:

Be aware of the following when you select **Before Solve**:

- The original import settings for the table apply when it is reloaded from the source file, including variable assignments, units, and skipped header or footer rows.
- The reload function does not keep track of which column corresponds to each variable. Table data is always read according to column order, starting with the first column of data in the source file. If columns are added to, removed from, or moved within the source file, the data that is reloaded into your table could change.
- If there are fewer columns in the source file than there are in the table, the reload fails and the application displays an error message.

Imported On

Shows the date and time when the file was imported.

Start Row

Shows the first row of data that was imported. If you skipped header rows during import, this will not be the first line in the file.

End Row

Shows the last row of data that was imported. If you skipped footer rows during import, this will not be the last line in the file.

Rename a Table

To change the name of a table:

1. Right-click the name of the table in the **Outline** pane.
2. Choose **Rename** from the menu.
3. Type the new name of the table and press the **Enter** key.

Delete a Table

To delete a table:

1. Right-click the table name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. Choose **Delete** from the menu.

3. The application asks whether you want to delete the object. Click **Yes** to delete the table. Click **No** to cancel.

DRAFT

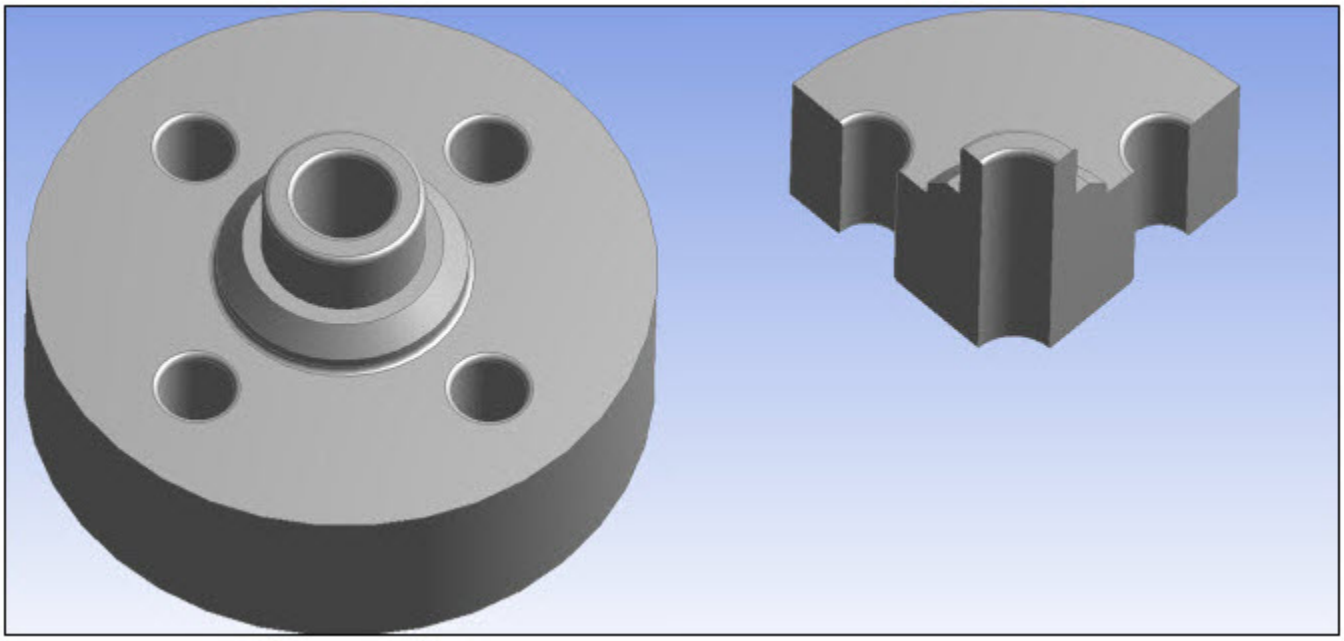
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Defining Symmetry

The **Symmetry** object is a **Model**-level object in the tree hierarchy. You can insert this object by selecting the **Symmetry** option on the Model Context Tab (p. 48).

This feature enables you to use the inherent geometric symmetry of a body to model only a portion of the body for simulation. Using symmetry provides the benefits of faster simulation times and less use of system resources. For example, the model below can be simplified by modeling only ¼ of the geometry by taking advantage of two symmetry planes.



Introduction

Making use of the **Symmetry** feature requires an understanding of the geometry symmetry and the symmetry of loading and boundary conditions. If geometric symmetry exists, and the loading and boundary conditions are suitable, then the model can be simplified to just the symmetry sector of the model.

DesignModeler can be used to simplify a full model into a symmetric model. This is done by identifying symmetry planes in the body. DesignModeler will then slice the full model and retain only the symmetry portion of the model. (See Symmetry in the DesignModeler help). When the Mechanical application attaches to a symmetry model from DesignModeler, a **Symmetry** folder is placed in the tree and each **Symmetry Plane** from DesignModeler is given a **Symmetry Region** object in the tree. In addition, **Named Selection** objects are created for each symmetry edge or face. (See Symmetry Workflow in DesignModeler (p. 1180).)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Once you have inserted a **Symmetry** object into the tree, the following options are available:

Types of Regions
Symmetry Workflow in DesignModeler
Symmetry Workflow in Mechanical
General Axisymmetric Workflow in Mechanical

Types of Regions

The **Symmetry** folder supports the following objects:

- **Symmetry Region** (p. 1128): This option is supported for Static and Transient Structural analyses and Thermal/Thermal-Electric analyses.
- **Linear Periodic** (p. 1130): When inserted, this option displays in the tree as a **Symmetry Region** object only when the **Type** property is set to **Linear Periodic**. As needed, this property can be modified. It is supported for Static Structural analyses and Thermal/Thermal-Electric analyses.
- **Periodic Region** (p. 1131): This option is supported for Magnetostatic analyses.
- **Cyclic Region** (p. 1134): This option is supported for Harmonic Response (Full), Modal, Static Structural, Thermal, Harmonic Acoustics, Modal Acoustics, Static Acoustics, and LS-DYNA analyses.
- **Pre-Meshed Cyclic Region** (p. 1138): This option is supported for Harmonic Response (Full), Modal, Static Structural, Harmonic Acoustics, Modal Acoustics, and Static Acoustics analyses.
- **General Axisymmetric** (p. 1167): This option is supported for 3D Static Structural analyses only.

Note:

Periodic and Cyclic regions:

- Support 3D analyses only.
- Ensure that a mesh is cyclic and suitable for fluids analyses (the mesh is then matched).

For models generated originally as symmetry models, you may create a **Symmetry** folder and manually identify **Symmetry Region** objects or **Periodic/Cyclic Region** objects. (See Symmetry Workflow in Mechanical (p. 1181).)

Symmetry Region Overview

A symmetry region refers to dimensionally reducing the model based on a mirror plane. Symmetry regions are supported for:

- Structural and Thermal Symmetry (p. 1129)
- Structural and Thermal Anti-Symmetry (p. 1129)
- Structural and Thermal Linear Periodic Symmetry (p. 1130)

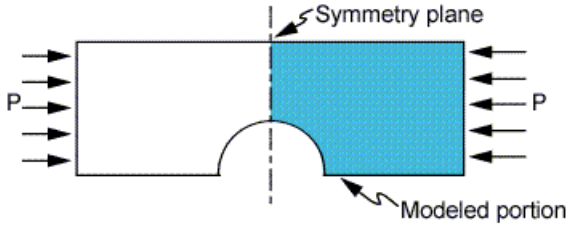
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Electromagnetic Symmetry (p. 1130)
- Electromagnetic Anti-Symmetry (p. 1131)
- Explicit Dynamics Symmetry

Structural and Thermal Symmetry

A symmetric structural boundary condition means that out-of-plane displacements and in-plane rotations are set to zero. The following figure illustrates a symmetric boundary condition. Structural symmetry is applicable to solid and surface bodies.



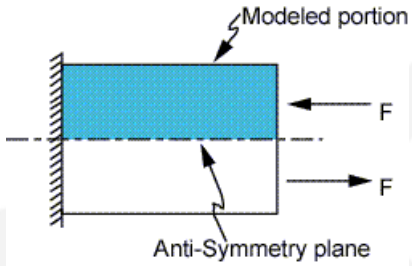
Two-dimensional plate model with symmetry

Note:

Thermal symmetry conditions are naturally satisfied.

Structural and Thermal Anti-Symmetry

An anti-symmetric boundary condition means that the rotation normal to the anti-symmetric face is constrained. The following figure illustrates an anti-symmetric boundary condition. Structural anti-symmetry is applicable to solid and surface bodies.



Two-dimensional plate model with anti-symmetry

Note:

- The **Anti-Symmetric** option does not prevent motion normal to the symmetry face. This is appropriate if all loads on the structure are in-plane with the symmetry plane. If applied loads, or loads resulting from large deflection introduce force components normal to the face, an additional load constraint on the normal displacement may be required.
- Thermal anti-symmetry conditions behave the same way as symmetry conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Structural and Thermal Linear Periodic Symmetry

The **Linear Periodic** boundary condition is used to simulate models with translational symmetry, where the structure is assumed to repeat itself in one particular direction to infinity. This feature supports only a single direction for the entire model (more than one direction is not supported) for structural analyses and thermal/thermal-electric analyses. The application uses the Mechanical APDL command **CE** to apply this boundary condition in a structural environment as compared to the **CP** commands used in the thermal and thermal-electric environments.

For structural modeling with the **Behavior** property set to **Free**, the application creates a pilot node and specifies constraint equations for the high and low regions as follows:

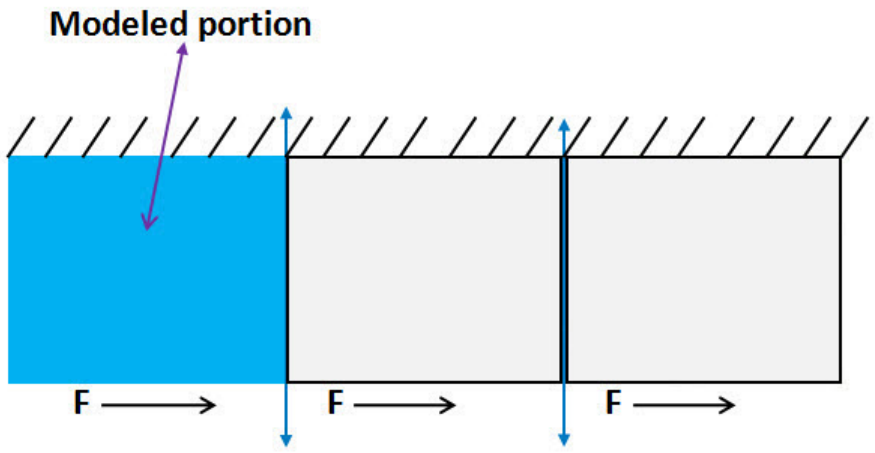
DOF(low) - DOF(high) = DOF(PilotNode)

The Pilot Node is left unconstrained. The Pilot Node is sent to the Mechanical APDL solver using the following parameter (variable): `_pilotNode`

For structural modeling with the **Behavior** property set to **Coupled**, the application directly specifies constraint equations for the high and low regions as follows:

DOF(low) - DOF(high) = 0

This forces DOF(low) to be coupled with DOF(high) in each pair of matched nodes.



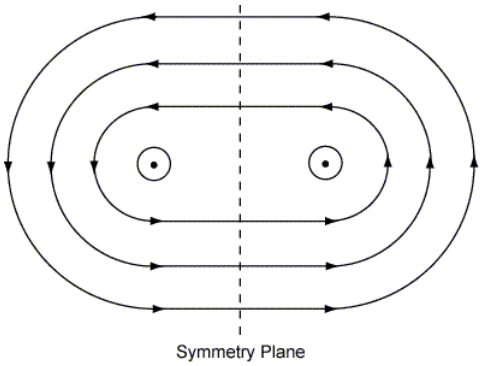
Two-dimensional plate model with translational symmetry

Electromagnetic Symmetry

Symmetry conditions exist for electromagnetic current sources and permanent magnets when the sources on both sides of the symmetry plane are of the same magnitude and in the *same* direction as shown in the following example.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

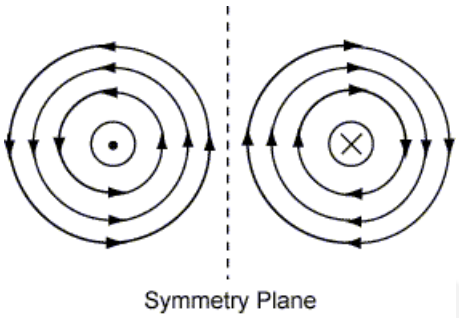
Published: 2023-10-23



Electromagnetic symmetric conditions imply Flux Normal boundary conditions, which are naturally satisfied.

Electromagnetic Anti-Symmetry

Anti-Symmetry conditions exist for electromagnetic current sources and permanent magnets when the sources on both sides of the symmetry plane are of the same magnitude but in the *opposite* direction as shown in the following example.



Electromagnetic anti-symmetric conditions imply Flux Parallel boundary conditions, which you must apply to selected faces.

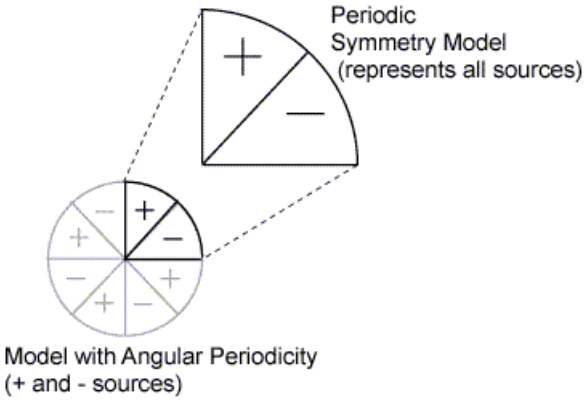
Periodic Region Overview

The **Periodic Region** object is used to define periodic or anti–periodic behavior in a particular model for Electromagnetic analyses (see Electromagnetic Periodic Symmetry (p. 1132) section).

Electromagnetic Periodicity

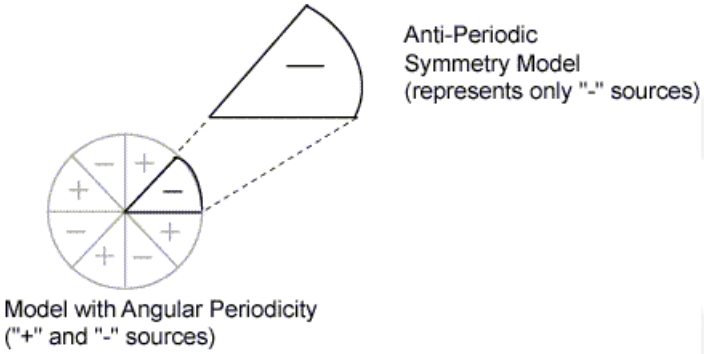
A model exhibits angular periodicity when its geometry and sources occur in a periodic pattern around some point in the geometry, and the repeating portion that you are modeling represents *all* of the sources, as shown below (see the Periodicity Example (p. 1133)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Electromagnetic Anti-Periodicity

A model exhibits angular anti-periodicity when its geometry and sources occur in a periodic pattern around some point in the geometry and the repeating portion that you are modeling represents *a subset of all* of the sources, as shown below.



Electromagnetic Periodic Symmetry

Electric machines and generators, solenoid actuators and cyclotrons are just a few examples of numerous electromagnetic devices that exhibit circular symmetrical periodic type of symmetry (p. 1131).

An automated periodic symmetry analysis conserves time and CPU resources and delivers analysis results that correspond to the entire structure.

The overall procedure in Ansys Workbench for simulating structures that are periodically symmetric is to run a magnetostatic (p. 546) analysis and perform the following specialized steps:

1. Insert a Periodic Region symmetry object in the tree. This step is necessary to enable Ansys Workbench to perform a periodic symmetry analysis.
2. Define the low and high boundaries of the **Periodic Region** by selecting the appropriate faces in the **Low Boundary** and **High Boundary** fields.
3. Define type of symmetry as **Periodic** or **Anti-Periodic** (see Periodicity Example (p. 1133)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. The solver will automatically take into account defined periodicity, and reported results will correspond to the full symmetry model (except volumetric type results as Force Summation, Energy probe, and so on).

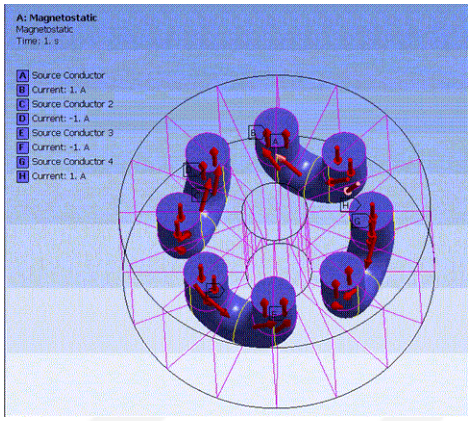
Note:

For a magnetic field simulation with periodic regions, you must be careful when applying flux parallel boundary conditions to adjacent faces. If the adjacent faces of the periodic faces build up a ring and all are subject to flux parallel conditions, that implies a total flux of zero through the periodic face. In some applications that is not a physically correct requirement. One solution is to extend the periodic sector to include the symmetry axis.

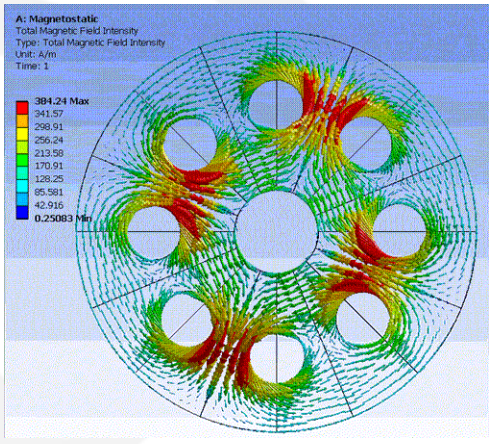
See the Periodicity Example (p. 1133) section for further details.

Periodicity Example

Periodicity is illustrated in the following example. A coil arrangement consists of 4 coils emulated by stranded conductors. A ½ symmetry model of surrounding air is created. The model is conveniently broken into 16 sectors for easy subdivision into periodic sectors and for comparison of results.

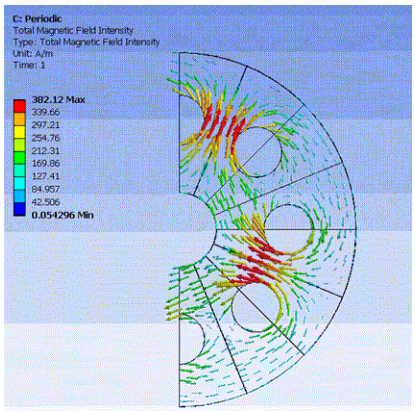


Below is a display of the Magnetic Field Intensity for the ½ symmetry model at the mid-plane. The arrows clearly indicate an opportunity to model the domain for both Periodic or Anti-periodic sectors. Periodic planes are shown to exist at 180 degree intervals. Anti-periodic planes are shown to exist at 90 degree intervals.

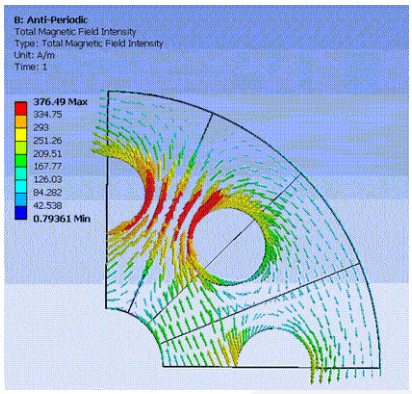


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The model can be cut in half to model Periodic planes. Applying periodic symmetry planes at 90 degrees and 270 degrees leads to the following results.



The model can be cut in half again to model Anti-Periodic planes. Applying anti-periodic symmetry planes at 0 degrees and 90 degrees leads to the following results.



Cyclic Region Overview

Fan wheels, spur gears, and turbine blades are all examples of models that can benefit from cyclic symmetry.

An automated cyclic symmetry analysis conserves time and CPU resources and enables you to view analysis results on the entire structure for a Static Structural, Modal, or Full Harmonic Response analysis.

Note:

Static and transient thermal analyses are also possible using the **Cyclic Region** object in Mechanical even though the cyclic symmetry solution in Mechanical APDL does not support elements with thermal DOFs. However, the results of analyses containing thermal DOFs can only be seen on a single sector.

When you use a Cyclic Region in an LS-DYNA analysis the results can only be seen on a single sector.

Ansys Workbench automates cyclic symmetry analysis by:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Solving for the behavior of a *single* symmetric sector (part of a circular component or assembly). See The Base Sector for more information.
- Using the single-sector solution to construct the response behavior of the full circular component or assembly (as a postprocessing step). Not available in an LS-DYNA analysis.

For example, by analyzing a single 10° sector of a 36-blade turbine wheel assembly, you can obtain the complete 360° model solution via simple postprocessing calculations. Using twice the usual number of degrees of freedom (DOFs) in this case, the single sector represents a 1/36th part of the model.

Restrictions

Note the following when working with cyclic symmetry:

- Always align **Remote Force**, **Moment**, and/or **Nodal Force** loading conditions with the same Z Axis direction used by the **Cyclic Region** object.

  Note:

  Remote Force and Moment are not available in an LS-DYNA analysis. There is no limitation on the **Nodal Force** direction for LS-DYNA.

- The presence of Mesh Connections in a body with cyclic symmetry definition may create conflicts within the mesh during the solution process.
- Cyclic symmetry does not support the use of the Average Across Bodies (p. 1981) option.

  Note:

  For an LS-DYNA analysis, average results (results computed over an element, like stress and strain) can be displayed for a single sector. All display options (unaveraged, averaged, nodal difference, nodal fraction, etc.) are functional when displaying results for LS-DYNA analyses.

- Layered Sections (p. 868) cannot be applied to a model that uses cyclic symmetry.
- Mechanical 2D Analyses do **not** support cyclic results.
- Line bodies are not supported for cyclic symmetry analyses.
- Gasket bodies are not supported for cyclic symmetry analyses.
- When specifying certain Probes (p. 1947) (deformation, stress, strain) during Cyclic analyses, Mechanical does not currently support probes specified when the **Location Method** is set to:
  - **Coordinate System**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– **Geometry Selection** and you define the associated **Geometry** property using the **Hit Point Coordinate** option (Graphics Toolbar (p. 97)).

Warning:

1. Make sure that you do not scope a **Remote Displacement**, **Remote Force**, or a **Moment** to a geometric entity that shares an edge, vertex, or node with either the specified **High Boundary** or **Low Boundary** properties of the **Cyclic Region** because the application could generate incorrect results.
2. Only **Static Structural** analyses support the ability to specify a **Remote Point** that coincides with the cyclic symmetry axis. In addition, for the analysis, you must:
   - Set the Remote Point **Behavior** property to **Deformable** or **Rigid**.
   - Set the **Future Analysis** option (**Analysis Settings** > Analysis Data Management (p. 1484)) to None.
   - Make sure that you divide the load magnitude by the specified number of sectors.
3. A **Remote Point** scoped to a sector is equivalent to having one Remote Point for each sector in the expanded solution.
4. If you specify a node-based load, such as **Direct FE** loads, **Remote Force**, **Moment**, etc., on both the low and high edge nodes, it will be taken into account twice in the solution due to the cyclic constraint equations.

See the Modeling and Loading Limitations section in the *Mechanical APDL Cyclic Symmetry Analysis Guide* for more information.

General Application

The overall procedure in Ansys Workbench for simulating models that are cyclically symmetric is to run a static structural, modal, full harmonic, LS-DYNA, or thermal analysis and perform the following specialized steps:

1. Insert a Cyclic Region or **Pre-Meshed Cyclic Region** object. This step is necessary to enable Mechanical to perform a cyclic symmetry analysis. Multiple **Cyclic Region** objects are permitted but they must refer to the same **Coordinate System** to specify the symmetry axis and sweep the same angle. Only one **Pre-Meshed Cyclic Region** can be added and it cannot be used in combination with any other symmetry objects.

   Note:

   Multistage Cyclic Symmetry Analysis (p. 1145) is an exception. Each stage in a multistage cyclic symmetry analysis can have a different sweep angle and **Pre-Meshed Cyclic Region**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

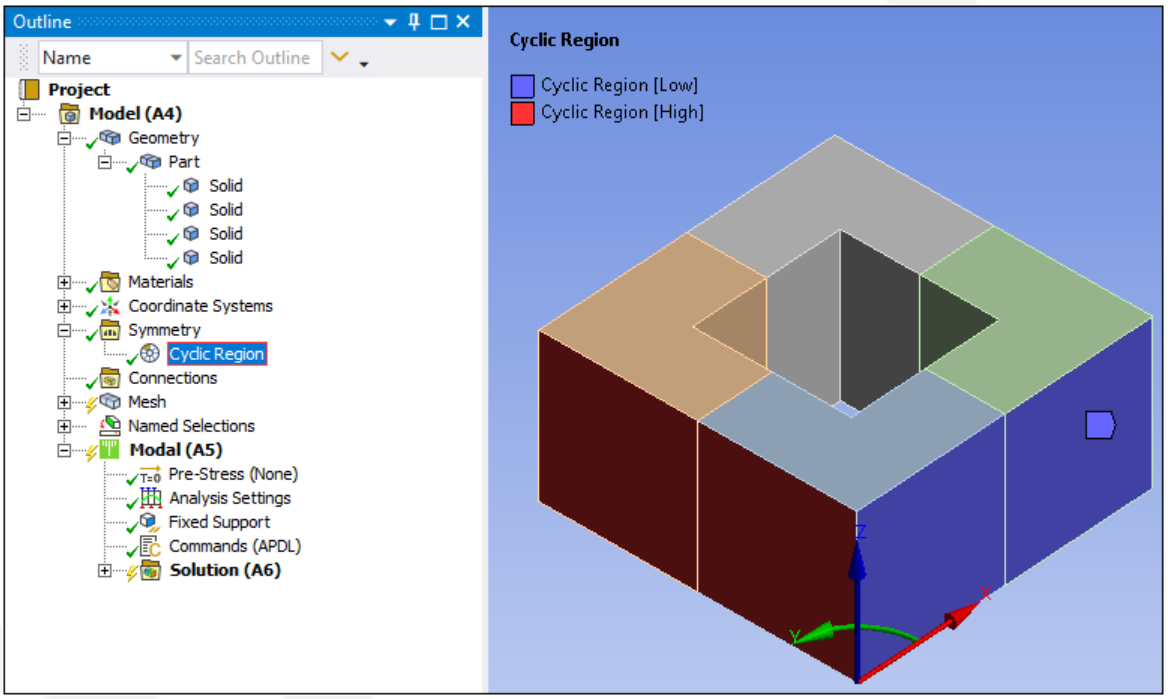
Published: 2023-10-23

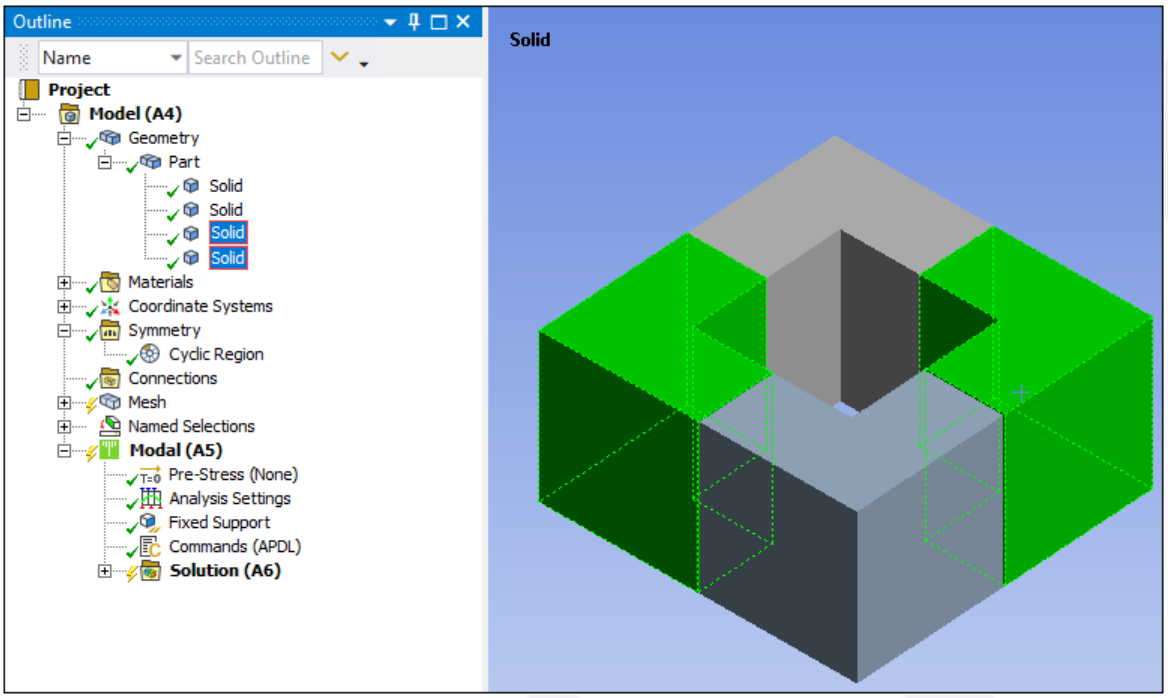
2. Define the low and high boundaries by selecting the appropriate faces in the **Low Boundary** and **High Boundary** fields. Each selection can consist of one or more faces over one or more parts, but they must be paired properly. To be valid, each face in **Low Boundary** must be accompanied by its twin in **High Boundary**. Also, ensure that each face and its twin belong to the same multibody part (p. 847) (although it is not necessary that they belong to the same body), using DesignModeler to adjust your multibody parts as needed.

Note:

- For the **Periodic Region** and the **Cyclic Region** objects, your low/high selections are used to match the mesh of the two boundaries.
- The **Pre-Meshed Cyclic Region** object does not influence the mesh.

The example shown below illustrates two equally valid **Low Boundary** and **High Boundary** twin faces. One twin set of faces, located in the corner body, includes faces that are both included in that same body. Another twin set includes faces that are not on the same body, but are included in the same multibody part, as shown in the second figure.





Note:

High Boundary and **Low Boundary** should be exactly same in shape and size, otherwise Mechanical will not be able to map nodes from **Low Boundary** to **High Boundary** to create full model from a single sector.

3. Continue with the remainder of the analysis. Consult the sections below as applicable to the analysis type.

Refer to the following sections for further details on cyclic symmetry:

Pre-Meshed Cyclic Symmetry

Multistage Cyclic Symmetry Analysis

Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

Cyclic Symmetry in a Thermal Analysis

Pre-Meshed Cyclic Symmetry

The **Pre-Meshed Cyclic Region** is a symmetry option that does process the specified selections for the **Low Boundary** and **High Boundary** properties. Normally, the application uses these selections to specify the nodes to write cyclic constraint equations; however, this object instead uses the cycle angle to determine the nodes used to create cyclic symmetry.

This option is used to define cyclic symmetry for cases where you **cannot** generate a matched mesh using the low and high boundaries. Not being able to match the mesh is often the case when you import mesh from an external source, such as Model Assembly, External Model, or ACP. To

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

match your mesh, the **Pre-Meshed Cyclic Region** enables you to select the exact orientation of nodal coordinate systems on the sector boundaries. And the feature includes the additional property **Relative Distance Tolerance** that enables you to modify the automatic sector boundary detection of low and high boundary.

Note:

- When the solver detects matching node pairs, you can view the matched nodes using the **Cyclic** option of the **Display** property (FE Connection Visibility (p. 1840) category) of the **Solution Information** object (p. 1840).
- This feature instructs the application to find all nodes separated by the sector angle and apply the cyclic boundary condition to them. Nodes separated by the sector angle cannot be excluded from being part of the cyclic symmetry condition.

Requirements

Note the following:

- This symmetry option inherits the limitations and restrictions of the **Cyclic Region**. For consistency with the cyclic symmetry solution method, some restrictions apply when defining the orientations for nodes on the sector boundaries. Refer to the Cyclic Region (p. 1134) section for additional information.
- The **Number of Sectors** property requires an entry (a natural number, N>2).
- Loads and supports are assumed to have the same spatial relation with respect to the cyclic axis in all sectors. Also the loads and supports defined in Mechanical are applied for each and every sector by Mechanical APDL.

Applying Constraints

Unlike the use of the **Cyclic Region** object, the application will not attempt to convert supports to nodal DOF constraints (p. 1789) on either of the sector boundaries (low or high). This means that any support that depends on these conversions should not contain faces, edges, or vertices that touch the sector boundaries. Exempt from this requirement are:

- Fixed Support (p. 1664)
- Nodal Orientation (p. 1719)
- Nodal Displacement (p. 1728)

Nodal Displacements and Nodal Orientations can be used to manually adjust any node-based degree-of-freedom constraints on problematic topologies so they become consistent with both the loading conditions and the cyclic symmetry method. The solver can be used to automatically assign these DOFs (Set the **Boundary DOF Orientation** property to **Chosen by Solver**), but then Nodal Displacements should be used with caution because final DOF directions may not be known prior to solution. You can use Nodal Triads (p. 1935) to verify how Mechanical APDL orients the nodes.

To manually prescribe orientations on the sector boundary, set the **Boundary DOF Orientation** property to **Manual**, and then apply any Nodal Orientations necessary to meet restrictions posed

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

by the cyclic symmetry method. Review the example provided later in this section for additional information.

Nodal Orientation Requirements

By default, all nodes are oriented using the Global Coordinate System. To orient the nodal coordinate systems in a manner consistent with cyclic symmetry, the following conditions are required:

Nodes on Sector Boundaries

Nodes on sector boundaries are free to orient with any direction as long as each node pair meets this condition: Nodal orientations on the low and high sector differ by a rotation of exactly 360°/N about the cyclic axis. The difference must be in phase with the cyclic axis. For example, if the sector angle is 90°, then the axis for a node must be obtained by rotating the coordinate system for the corresponding node on the low boundary by +90° around the cyclic axis.

When **Boundary DOF Orientation** is set to **Chosen by Solver**, Mechanical APDL typically aligns the nodal x axis radially, y tangentially and z axially with respect to the cyclic axis.

Nodes on the Cyclic Axis

If your model contains nodes on the cyclic axis, they require special treatment. The nodes' orientation must produce an axis which is aligned with the cyclic axis.

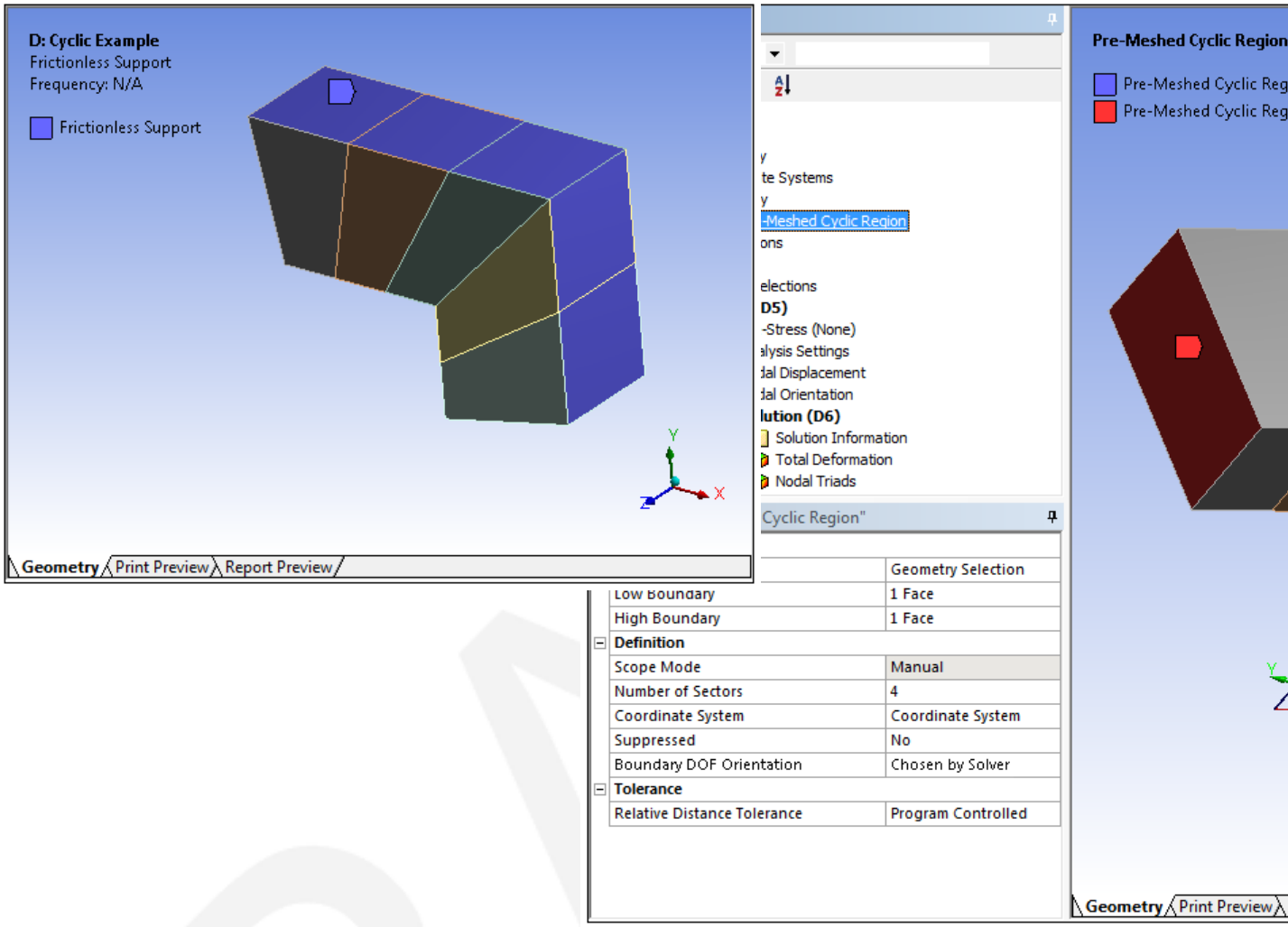
Example: Applying a Frictionless Support Manually via Direct FE

This example demonstrates how to apply the equivalent of a Frictionless Support using Nodal Displacements and Nodal Orientations on a quarter section of a cyclically symmetric model using a **Pre-Meshed Cyclic Region**. The first illustration below depicts a **Frictionless Support** applied to the exterior faces of a symmetry model. Because this boundary condition shares topology with sector boundaries, it will not solve.

The second image illustrates a defined **Pre-Meshed Cyclic Region** object. It selects the same low/high face selections you would use for a **Cyclic Region**. It is a quarter-section so the **Number of Sectors** property is set to **4** and the **Boundary DOF Orientation** property is set to **Manual**.

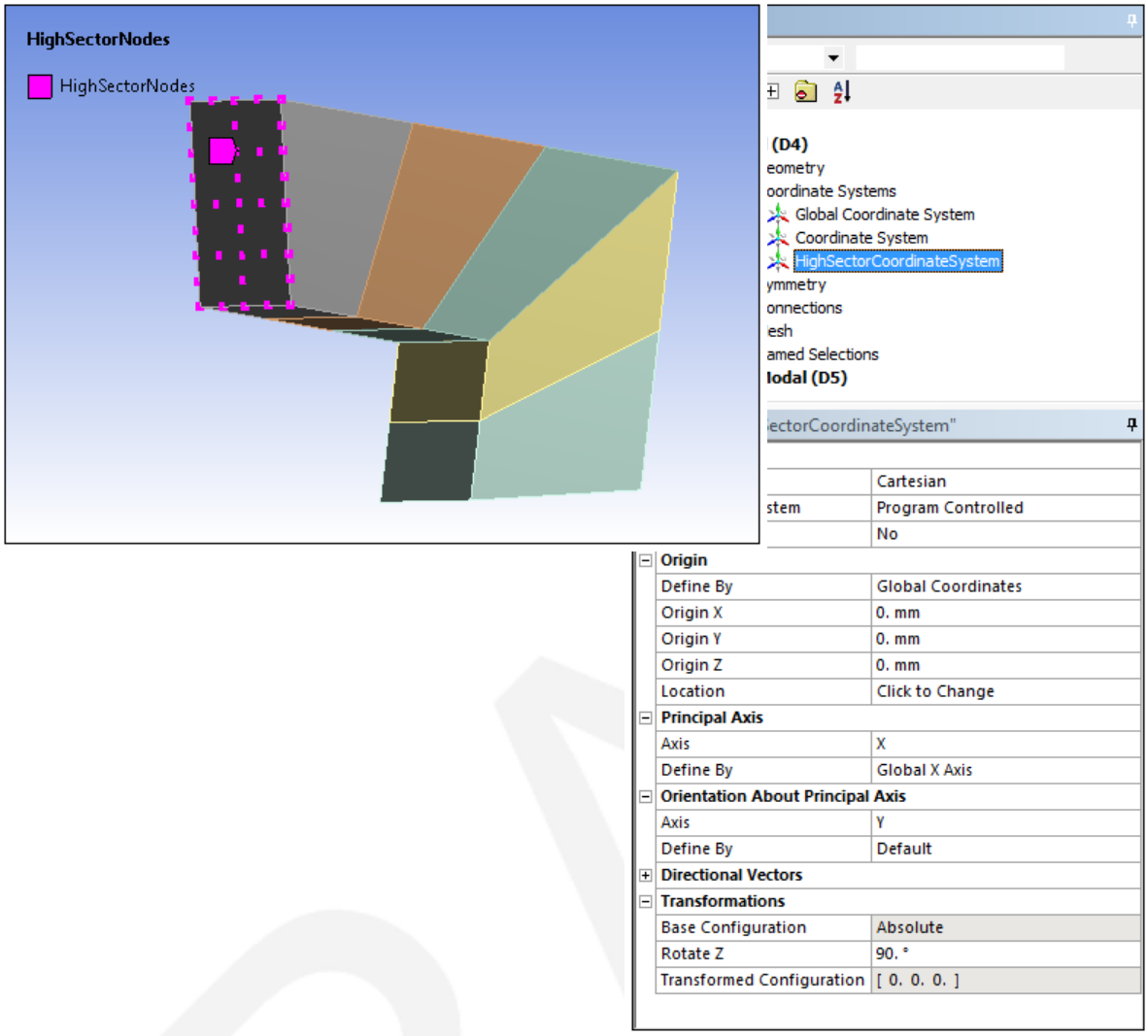
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



To meet the nodal orientation requirements (p. 1140), we need to rotate the high boundary nodes. That means we first need to create a node-based Named Selection of the high sector node ("HighSectorNodes" illustrated below). Then, to achieve the desired orientation, we need to create Cartesian Coordinate System. This coordinate system requires a rotational transformation (p. 1195) about the z axis so that we can rotate the nodes 90° (360°/4 = 90°). Below we have created the necessary coordinate system, "HighSectorCoordinateSystem" that includes the needed rotation (**Rotate Z** property) about the z axis.

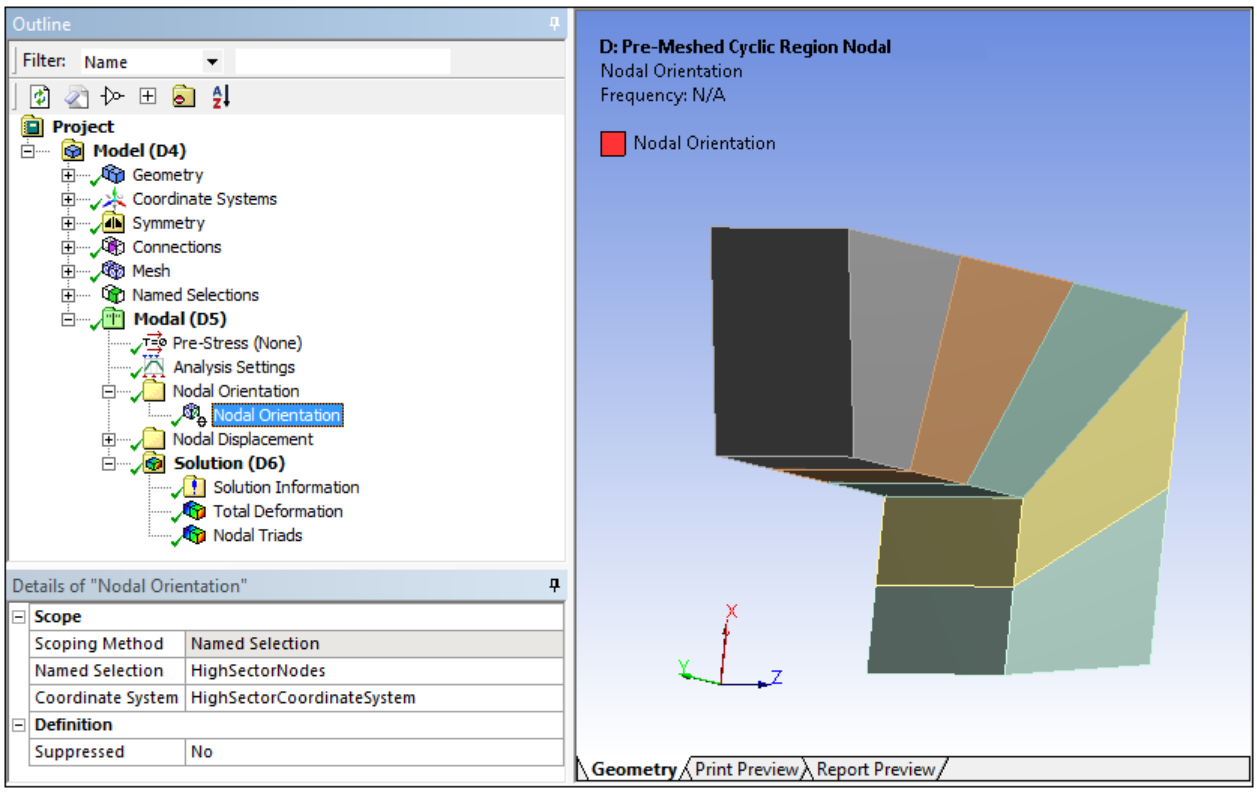
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Now, a **Nodal Orientation** object is inserted using the coordinate system and the node-based Named Selection, to meet the nodal orientation conditions required by the cyclic symmetry method.

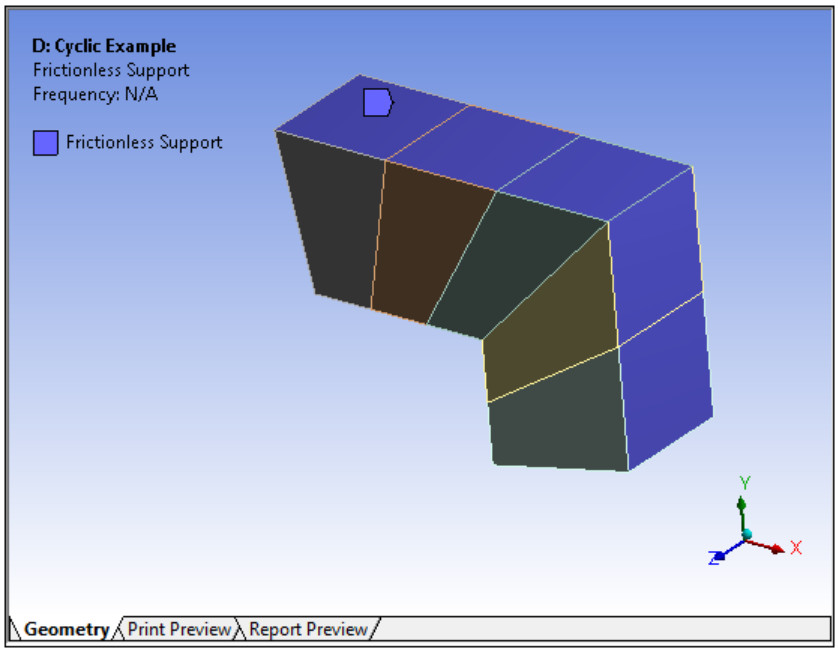
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Understanding Frictionless Support

As with all boundary conditions, a Frictionless Support applies displacements to nodes in their nodal coordinate system. In particular, a Frictionless Support applies displacements which fix the nodes in the direction of the normal of the element faces. As stated above, in this example we are applying the equivalent of a Frictionless Support for several exterior faces. Of these exterior faces, two are normal to the global +X, and three are normal to the global +Y.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

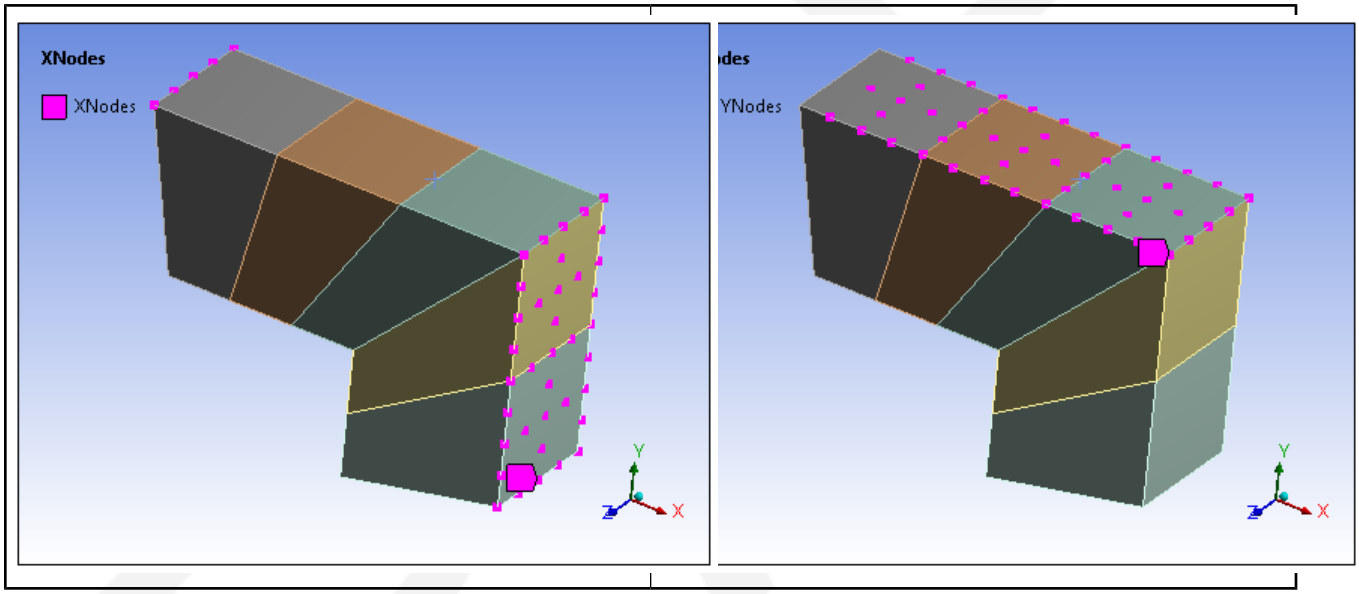
Oriented +X Normal

The intent is to fix these nodes in global X. The nodes on these faces have no **Nodal Orientation** applied on them, so they are oriented with the Global Coordinate System. A Nodal Displacement of X=0 is required for these nodes (Requirement 1 - XNodes).

Oriented +Y Normal

The intent is to fix these nodes in global Y. The majority of the nodes on these faces have no **Nodal Orientation** applied on them. A **Nodal Displacement** of Y=0 is required for these nodes (Requirement 2 - YNodes). The orientation of the remaining nodes on the edge of the high sector have been specified by the **Nodal Orientation** applied above. The Global Y for these nodes is equal to their Nodal X. Therefore, a **Nodal Displacement** of X=0 is required for these nodes (Requirement 3 - XNodes).

The node-based Named Selections needed to meet our requirements are illustrated below.



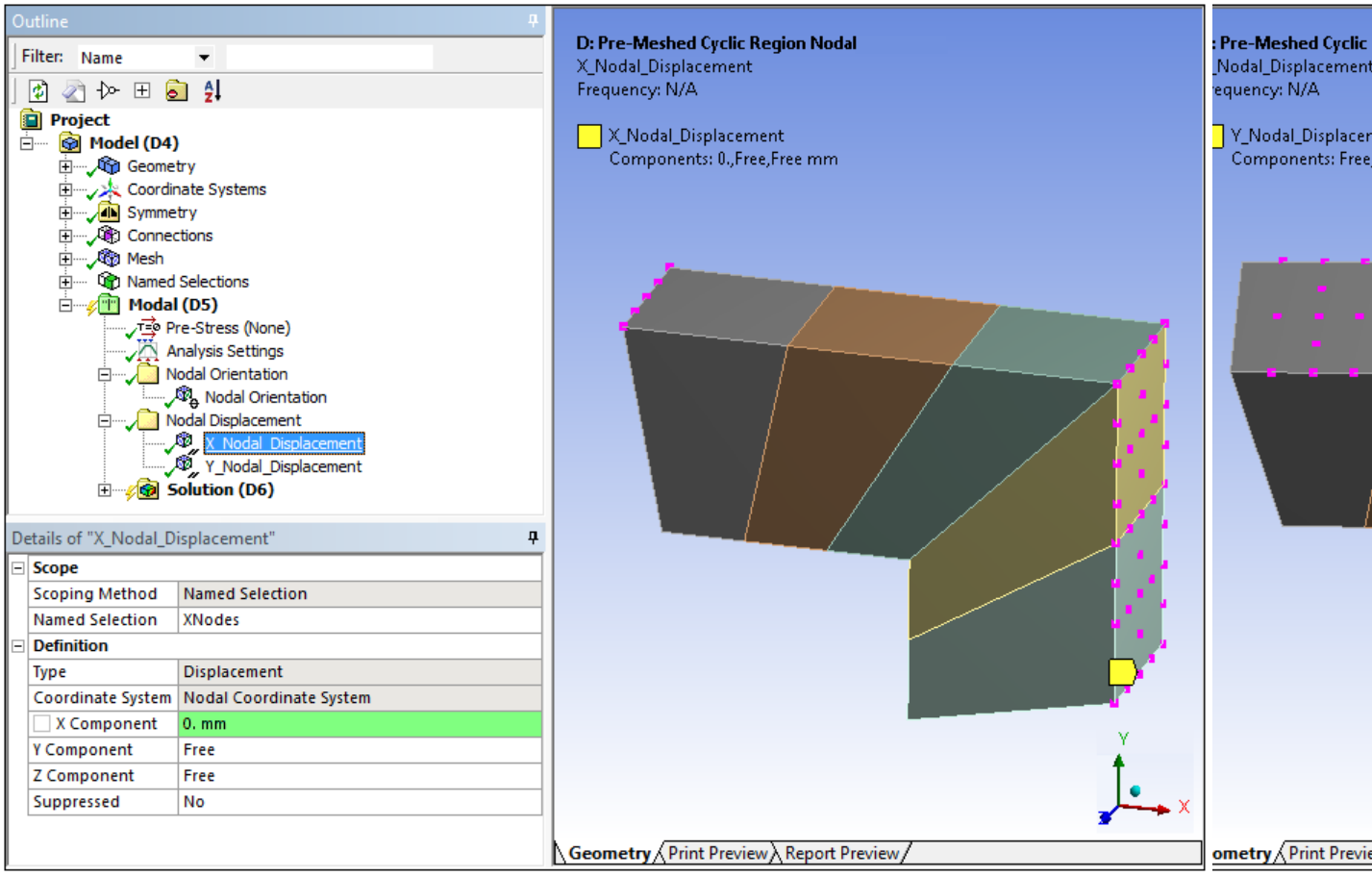
Applying Nodal Displacements

At this point, we have identified three required nodal displacements. However, we find that two of these requirements use the same nodal displacement (X=0). The definitions for the resulting **Nodal Displacement** objects is illustrated below.

X=0 **Y=0**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Using the Coordinate System, Named Selections, Nodal Orientations, and Nodal Displacements, we have manually applied the equivalent of a Frictionless Support that is consistent with cyclic symmetry for use in a **Pre-Meshed Cyclic Region**.

Multistage Cyclic Symmetry Analysis

For complex symmetric models, the application's multistage cyclic symmetry capability enables you to combine two or more independent cyclically symmetric systems with different sector counts. The purpose of symmetry is to reduce calculations times and disk and memory requirements. Multistage uses the existing symmetry capabilities to simplify the calculation process but adds multiple regions of symmetry. In doing so, the multistage feature incorporates:

- **Cyclic Region** (p. 1134) or **Pre-Mesh Cyclic Region** (p. 1138): This object specifies the symmetric region.
- Stage: This object defines each of the bodies corresponding to a given cyclic component.
- Interstage: This object specifies the connections between each **Stage** of the multistage analysis.

Use the procedure below to create a multistage cyclic symmetry analysis. More information and example problems can be found at the following links:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For an overview of the general procedure to reduce model size and computational cost in models with symmetry, see Symmetry Workflow in Mechanical (p. 1181).
- For a detailed discussion and underlying theory, see *Multistage Cyclic Symmetry Analysis Guide*.
- For tutorials that explain in detail the procedure for setting up a multistage cyclic symmetry analysis in the Mechanical Application, see

  Static and Modal Analysis of a Compressor Model with 4 Axial Stages in the *Mechanical Tutorials*

  Static Analysis of a 2-Stage Disk with Pinholes in the *Mechanical Tutorials*
- For more example problems that show the multistage cyclic symmetry work flow solved in Mechanical APDL, see Examples in the *Multistage Cyclic Symmetry Analysis Guide*.
- The interstage displacement is the sum of contributions of many harmonic functions, only some of which significantly affect the response. For an example problem that demonstrates how to select relevant harmonic indices (HIs) to include in a multiharmonic multistage cyclic symmetry analysis for an accurate response, see Static Analysis of a 2-Stage Disk with Pinholes. To automate HI selection, use the multistage harmonic choice wizard (see Multistage Harmonic Choice Wizard in the *Mechanical Add-ons Guide*).

Important:

This type of symmetry analysis does not support boundary conditions imported from an **External Model** system. If your simulation includes these (unsuppressed) boundary conditions, the **Symmetry** object becomes invalid.

Supported Analysis Types

Multistage analyses can be solved for the following analysis types:

- Linear Perturbation Modal
- Modal
- Static Structural
- Steady-State Thermal

Application

This procedure assumes that you have opened an appropriate model in Mechanical to create a multistage cyclic symmetry analysis:

1. Insert and specify the required number of **Cyclic Region** or **Pre-Mesh Cyclic Region** objects.
2. For each symmetric region, specify a **Stage** object:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

a. Using the scoping properties, specify the entire body/bodies of each region. This region is associated with the **Cyclic Region** or **Pre-Mesh Cyclic Region** objects defined in the previous step. Only body selection is supported.

b. In the **Cyclic Region** property, specify the **Cyclic Region** or **Pre-Mesh Cyclic Region** corresponding to your body selections.

3. To define the connection between stages, you must specify an **Interstage** object. Under the **Connections** folder, insert an **Interstage** object and scope it to the two faces (**Source** and **Target** properties) that connect the previously defined stages. Only face selection is supported.

Note:

Ansys recommends that you modify the mesh of the connection so that each side uses quadratic elements that are preferably the same type and size.

---

4. Setup environment and perform a solution.

Note:

This analysis application supports the use of the following imported loads:

- Temperature (p. 1756) and Convection Coefficient (p. 1744) for Steady-State Thermal analyses.
- Body Temperature (p. 1740) and Pressure (p. 1752) for Static Structural analyses.

Note:

If you have a result object selected, certain mesh-based features are not available, including mesh selection filters (nodes (p. 269), elements, and element faces (p. 276)), as well as the ability to display a Node ID in the probe label (p. 206).

Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

When you perform a Static Structural (p. 652) or a Static Acoustics analysis that involves cyclic symmetry (p. 1134), unique features are available for loads/supports and reviewing results. These features are described in the following sections:

Applying Loads and Supports for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

Reviewing Results for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

Applying Loads and Supports for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

The following support limitations and specifications must be observed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The following boundary conditions are **not** supported:
  - Bearing Load
  - Hydrostatic Pressure
  - Fluid Solid Interface (Supported for Static Acoustics but cannot be applied to the Low/High Boundary)
- The following *remote* boundary conditions are **not** supported:
  - Joints
  - Bearing
- Inertial (p. 1500) boundary conditions as well as the **Moment** (p. 1574) boundary condition are restricted to the axial direction. Therefore, you must set the **Define By** property of the specified boundary conditions to **Components**. Only the **Z Component** property can be non-zero and the Z-Axis of the specified **Coordinate System** of the boundary condition must match the coordinate system used by the corresponding **Cyclic Region**.In similar fashion, you must define the **Remote Point** (p. 1090) for **Moment** loads so that its points of application lie along the cyclic axis (see warning below).

Additional restrictions apply while specifying supports for a static structural analysis. For example, Elastic Supports and Compression Only Supports are not available. The loads and supports should not include any face selections (for example, on 3D solids) that already belong to either the low or high boundaries of the cyclic symmetry sector. Loads and supports may include edges (for example, on 3D solids) on those boundaries, however.

Warning:

Only **Static Structural** analyses support the ability to specify a **Remote Point** that coincides with the cyclic symmetry axis. In addition, for the analysis, you must:

- Set the Remote Point **Behavior** property to **Deformable** or **Rigid**.
- Set the **Future Analysis** option (**Analysis Settings** > Analysis Data Management (p. 1484)) to None.
- Make sure that you divide the load magnitude by the specified number of sectors.

Loads and supports are assumed to have the same spatial relation for the cyclic axis in all sectors.

In preparation for solution, the boundary conditions on the geometry are converted into node constraints in the mesh (see Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) for more information). When these boundary conditions involve nodes along the sector boundaries (low, high, and axial boundaries), their constraints are integrated to properly reflect the symmetry. As an example, the low and high edges may feature more node

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

constraints than are applied to each individually, in order to remain consistent with an equivalent full model.

Important:

When analyzing a cyclically symmetric structure that rotates without supports, you may encounter rigid body movement. In this scenario, the recommended action is specifying **Direct** for the **Solver Type** property.

Reviewing Results for Cyclic Symmetry in a Static Structural or Static Acoustics Analysis

When simulating cyclic symmetry in a static structural analysis, the application supports the static structural analysis result types (p. 2017) that involve full symmetry, with the exception of Path (p. 1940) and Fatigue (p. 2165) results.

Even though only one cyclic sector is analyzed, results are valid for the full symmetry model. You can control the post-processing and display of cyclic results using the **Cyclic Solution Display** options on the **Solution** object:

- **Number of Sectors**: This option controls the extent the model is expanded from the raw solution. The value indicates how many sectors should be processed, displayed and animated. Results generate more quickly and consume less memory and file storage when fewer sectors are requested. To set the value as Program Controlled, enter zero; this value reveals the full expansion.
- **Starting at Sector**: Selects the specific sectors to include within the expansion. For example, if Number of Sectors is set to 1, sectors 1 through N are revealed one at a time. To set the value as **Program Controlled**, enter zero; this value reveals the specified number of sectors from sector 1 onwards.

In addition, individual result objects contain a **Starting at Sector** property. For it to display, the **Number of Sectors** property above must be set to a value greater than 0. Furthermore, this property overrides the setting of the above **Starting at Sector** property of the **Solution** object. This property gives you the ability to define different starting sectors for each result object.

Note:

- Extremum values (for example, Minimum, Maximum) correspond only to the portion of the model selected in the Cyclic Solution Display.
- For a Multistage Cyclic Symmetry (p. 1145) analysis, the **Number of Sectors** and **Starting at Sector** properties apply to all Stage objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 20: Unexpanded One Sector Model Display

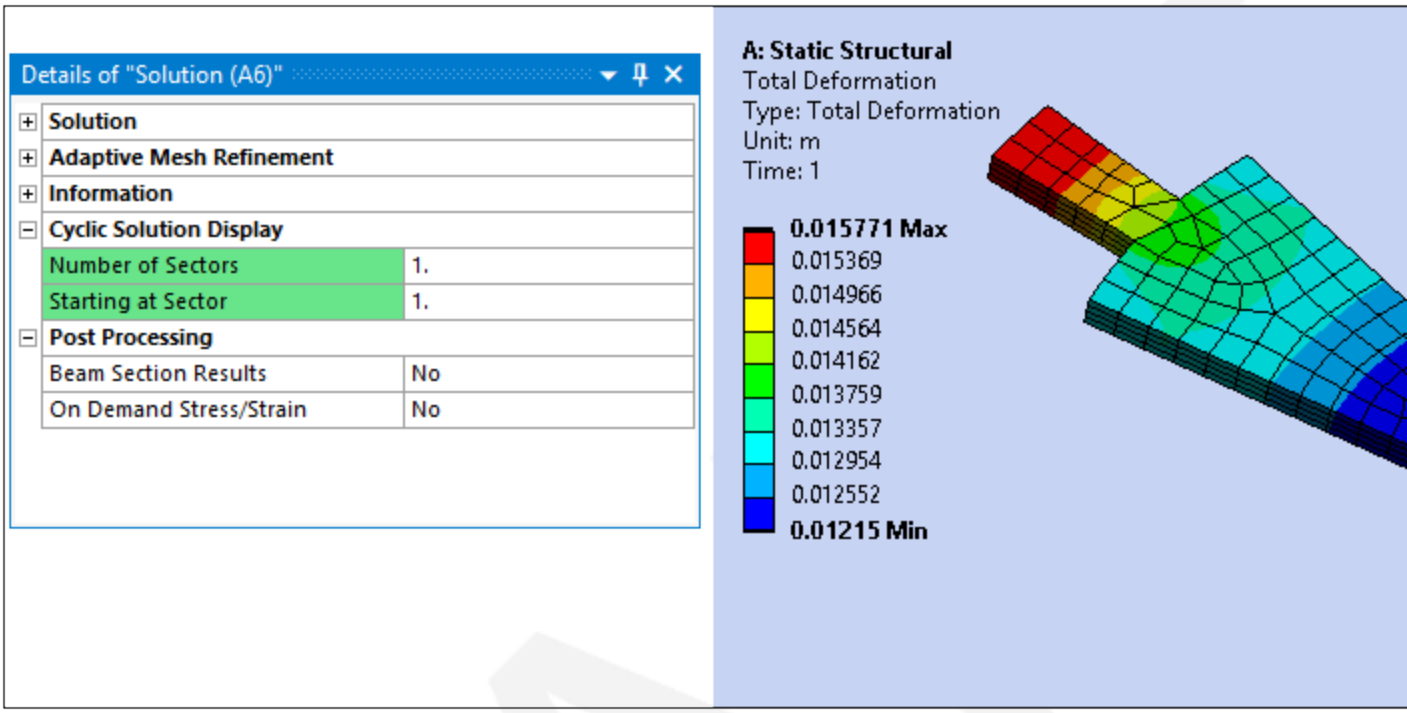
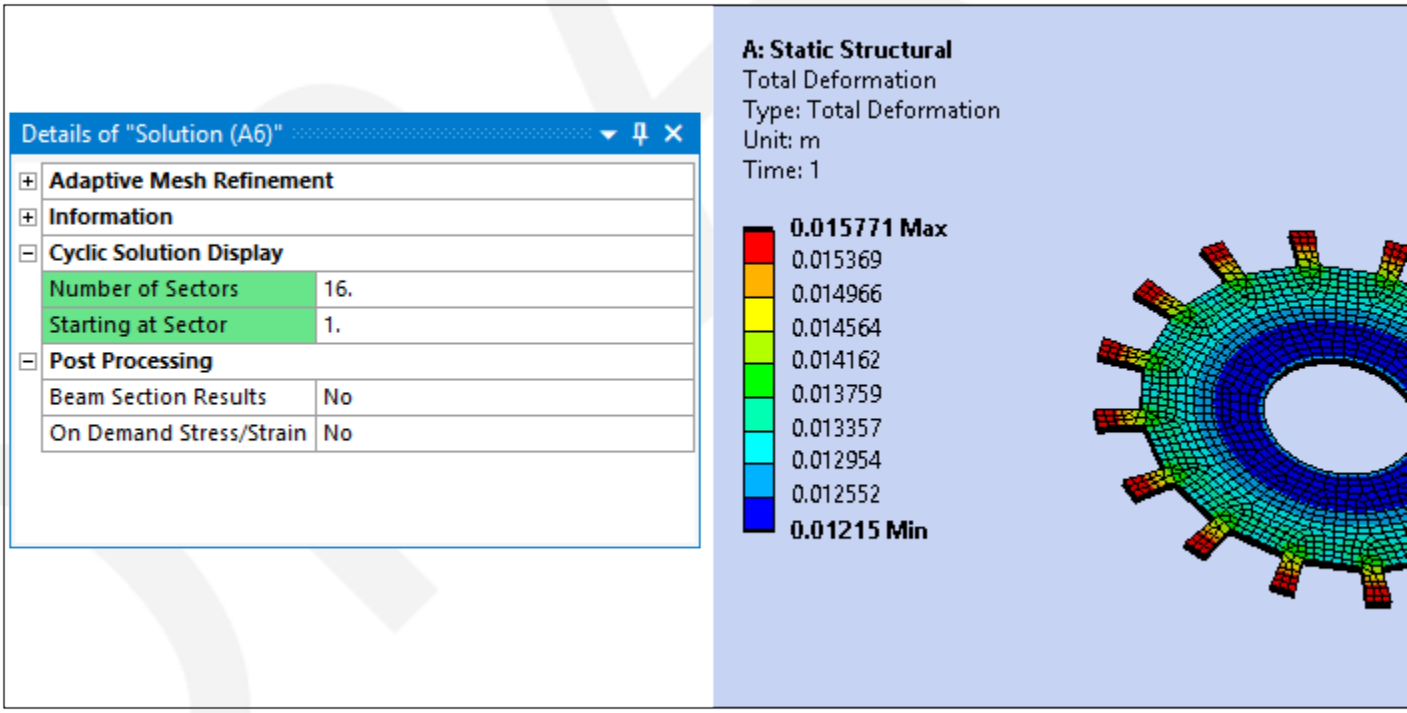


Figure 21: Expanded Full Symmetry Model Display



Note:

- The application does not support the expanded full view of a model for element-based results (one value per element) such as Volume. That is, results with a Data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Type of Elemental. Nodal Triads as well as Crack and Fracture results are also not expanded.

- The results for the **Energy Probe**, **Force Reaction** probe, and **Moment Reaction** probe are calculated for the full symmetry model.
- Unaveraged contact results do not expand to all expanded sectors in a cyclic analysis.
- Expanded result visualization is not available to the Samcef or ABAQUS solver.

Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

Currently, Mechanical supports cyclic symmetry for the **Full** Harmonic Response (p. 496) (including pre-stress effects) and FSI Harmonic Acoustics analyses only. When you perform a Harmonic Response (Full) or FSI Harmonic Acoustics analysis that involves cyclic symmetry, unique features are available for loads and supports as well as for reviewing results. These features are described in the following subsections:

Applying Loads and Supports for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

Reviewing Results for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

Applying Loads and Supports for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

Applying loads and supports for a cyclic Full Harmonic Response analysis (standalone and pre-stressed) follows the same requirements as those outlined in the Harmonic Response Analysis (p. 496) and Harmonic Response (Full) Analysis Using Pre-Stressed Structural System (p. 506) sections of the Mechanical Help except for the requirements and restrictions discussed below.

General Loading Application Requirements

The following support limitations and specifications must be observed for a standalone (Full) Harmonic Response analysis when cyclic symmetry is defined:

- The following boundary conditions are **not** supported:
  - Bearing Load
  - Rotating Force
  - Incident Wave Source
  - Port In Duct
  - Diffuse Sound Field
- The following *remote* boundary conditions are **not** supported:
  - Joints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - Bearing

- Inertial (p. 1500) boundary conditions as well as the **Moment** (p. 1574) boundary condition are restricted to the axial direction. Therefore, you must set the **Define By** property of the specified boundary conditions to **Components**. Only the **Z Component** property can be non-zero and the Z-Axis of the specified **Coordinate System** of the boundary condition must match the coordinate system used by the corresponding **Cyclic Region**. In similar fashion, you must define the Remote Point (p. 1090) for **Moment** loads so that its points of application lie along the cyclic axis.
- Elastic Supports and Compression Only Supports are not available.

Warning:

- Make sure that you do not scope a **Remote Displacement**, **Remote Force**, or a **Moment** to a geometric entity that shares an edge, vertex, or node with either the specified **High Boundary** or **Low Boundary** properties of the **Cyclic Region** because the application could generate incorrect results.
- Specified **Remote Points** cannot coincide with the cyclic symmetry axis, as this could generate incorrect results. However, there are exceptions for a standalone modal or an MSUP modal analysis. If the analysis includes a **Displacement** or a **Moment** (for a subsequent MSUP) that is applied about the cyclic symmetry axis and then to all sectors, then the **Displacement** or **Moment**:
  - Must be scoped to a Remote Point that has its **Behavior** property set to **Deformable**.
  - Must have all DOFs fixed except the DOF about the cyclic symmetry axis, and,
  - The specified **Remote Point** must lie on the cyclic axis of symmetry.

Note:

If you specify a **Fluid Solid Interface** load, you could expose non-physical results if the remote boundary conditions have their **Behavior** property set to **Deformable**.

Additional restrictions apply while specifying supports for a cyclic Harmonic Response (Full) analysis. Also, the loads and supports should not include any face selections (for example, on 3D solids) that already belong to either the low or high boundaries of the cyclic symmetry sector. Loads and supports may include edges (for example, on 3D solids) on those boundaries, however.

By default, loads and supports are assumed to have the same spatial relation for the cyclic axis in all sectors. Also the loads and supports defined in Mechanical are applied for each and every sector by Mechanical APDL.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Non-Cyclic Loading

For Full Harmonic Analysis, you can specify non-cyclic loading to excite a specific harmonic index or indices for the following loads. These loads include the **Non-Cyclic Loading Type** property. This property enables you to specify harmonic indices to be excited by the load. See the Non-Cyclic Loading (p. 1153) section for additional information.

- Pressure (p. 1521)
- Force (p. 1543) (applied to a face, edge, or vertex)
- Moment (p. 1574)
- Remote Force (p. 1552)
- Nodal Force (p. 1721)

Note:

Ansys recommends that you do not define remote boundary conditions along the cyclic axis of symmetry when you have specified non-cyclic loads.

Boundary Condition Conversion to Node Constraints

In preparation for solution, the boundary conditions on the geometry are converted into node constraints in the mesh (see Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) for more information). When these boundary conditions involve nodes along the sector boundaries (low, high, and axial boundaries), their constraints are integrated to properly reflect the symmetry. As an example, the low and high edges may feature more node constraints than are applied to each individually, in order to remain consistent with an equivalent full model.

Non-Cyclic Loading

The application supports non-cyclic loading for Full Harmonic cyclic solutions. A load is non-cyclic when it varies between sectors and involves at least one harmonic index greater than zero.

The application considers the arbitrary forces acting on the full system as the sum of a finite number of spatial Fourier harmonics. The application analyzes the structure for each spatial harmonic index by applying constraint equations between the *basic sector* and duplicate sector. For each spatial Fourier harmonic, the program solves a corresponding equation, then expands and sums the calculated harmonics of the response to give the response for each substructure. Be sure to review the Limitations (p. 1155) at the end of the section.

Tip:

Review the Understanding the Solution Architecture section in the *Mechanical APDL Cyclic Symmetry Analysis Guide* for more information.

In Mechanical, you could specify the non-cyclic loading for the supported loads by setting the **Non-Cyclic Loading Type** property to **Harmonic Index** or **Sector Number**. You can then

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

specify a desired **Harmonic Index** or **Sector Number** value. If you want to apply the load varying across several harmonic indices or sectors, you could specify the load **Magnitude** using **Tabular Data**, which varies based on the variable you specify.

Engine-Order Loading (or Traveling Wave Excitation)

Engine-order loading (traveling wave excitation) is a form of non-cyclic loading supported in Mechanical when you set the **Non-Cyclic Loading Type** property to **Harmonic Index** . This section uses the terms harmonic index and engine order interchangeably.

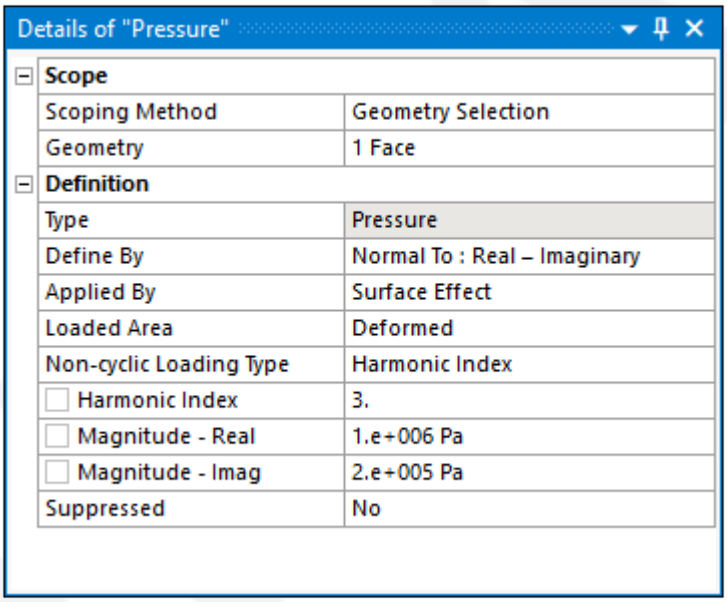
Typically, engine-order loading is simply a count of the number of stators, combustion nozzles, etc., that cause the disturbance. An engine-order excitation typically occurs due to circumferential disturbances in the flow field, for instance from upstream stators or vanes.

Examples Cases

The following examples illustrates how to apply loading based on specified inputs for the supported load types.

Case 1: Constant Load and Constant Harmonic Index

The Pressure load illustrated here has constant loading and a constant Harmonic Index.



When you specify a Harmonic Index (HI), real and imaginary values of pressure (P_{real} and P_{Imag}), the contribution of the load at a sector i is calculated as follows:

$$\left(P_{real}\right)_i = P_{real} \cdot \cos(\alpha) - P_{imag} \cdot \sin(\alpha)$$
$$\left(P_{imag}\right)_i = P_{real} \cdot \sin(\alpha) + P_{imag} \cdot \cos(\alpha) \quad (88)$$

Where:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

$$\alpha = \frac{2 \cdot \pi \cdot (i-1) \cdot HI}{HS}$$
$$i = 1 \text{ to } NS$$
$$NS = Number\ of\ Sectors$$

The values of the Harmonic Index specified should be:

1 to NS/2; if NS is even.

1 to (NS-1)/2; if NS is odd.

Case 2: Tabular Load with Harmonic Index as Independent Variable

You could also specify the loading applied to multiple harmonic indices using tabular data as illustrated here:

Tabular Data

	Harmonic Index	Magnitude - Real [Pa]	Magnitude - Imag [Pa]
1	3.	1.e+006	2.e+005
2	4.	3.e+006	4.e+006
*			

The load is expanded using Fourier transform as specified in **Equation 88** above, for each harmonic index and their contributions are applied for each sector.

Case 3: Tabular Load with Frequency (or X, Y, Z) as Independent Variable

The load specified at each frequency is expanded using Fourier transform as specified in **Equation 88** above, and their contributions are applied for each sector.

Tabular Data

	Frequency [Hz]	Magnitude - Real [Pa]	Magnitude - Imag [Pa]
1	100.	3.e+006	4.e+006
2	200.	1.e+006	2.e+006
*			

Limitations

Review the following limitations for the **Non-Cyclic Loading Type** feature:

General

- When you have a non-cyclic **Tabular Load** and the **Independent Variable** property is set to **Frequency**, please ensure that you define the load such that there is a non-zero magnitude for the load at each frequency point in the tabular data. Otherwise, the application may ignore some of these defined loads.
- A Remote Point scoped to a Sector is equivalent to having one Remote Point for each Sector in the expanded solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you specify a node-based load, such as Direct FE loads, Remote Force, Moments, etc., on both the low and high edge nodes, it will be taken into account twice in the solution due to the cyclic constraint equations.

Engine-Order Loading

- Applying a **Remote Force**, **Moment**, or **Nodal Force** in the direction that **does not** align with the Z direction of the cyclic axis could generate harmonic indices other than the ones requested.
- Defining a non-cyclic loads using the **Function** option is not currently supported.

See the Modeling and Loading Limitations section in the *Mechanical APDL Cyclic Symmetry Analysis Guide* for more information.

Reviewing Results for Cyclic Symmetry in a Harmonic Response or FSI Harmonic Acoustics Analysis

When simulating cyclic symmetry in a Harmonic Response (Full) analysis, the same results are available as results in Harmonic Response analyses (p. 496) that involve full symmetry with the exception of Linearized Stresses and Fatigue results (p. 2165) (which are underdefined). In addition, purely elemental results, such as Volume, Kinetic Energy, and Potential Energy display only in the base sector. To verify a result is elemental, examine the **Worksheet** (from the Solution object) for **Solution Quantities and Result Summary** (p. 1961) > **Available Solution Quantities**.

Even though only one cyclic sector is analyzed, results (excluding elemental results) are valid for the full symmetry model. You can control the post-processing and display of cyclic results using the following options of the **Cyclic Solution Display** category of the **Solution** folder:

- **Number of Sectors**: This option controls the extent the model is expanded from the raw solution. The value indicates how many sectors should be processed, displayed and animated. Results generate more quickly and consume less memory and file storage when fewer sectors are requested. To set the value as Program Controlled, enter zero; this value reveals the full expansion.
- **Starting at Sector**: Selects the specific sectors to include within the expansion. For example, if Number of Sectors is set to 1, sectors 1 through N are revealed one at a time. To set the value as Program Controlled, enter zero; this value reveals the specified number of sectors from sector 1 onwards.

In addition, individual result objects contain a **Starting at Sector** property. For it to display, the **Number of Sectors** property above must be set to a value greater than 0. Furthermore, this property overrides the setting of the above **Starting at Sector** property of the **Solution** object. This property gives you the ability to define different starting sectors for each result object.

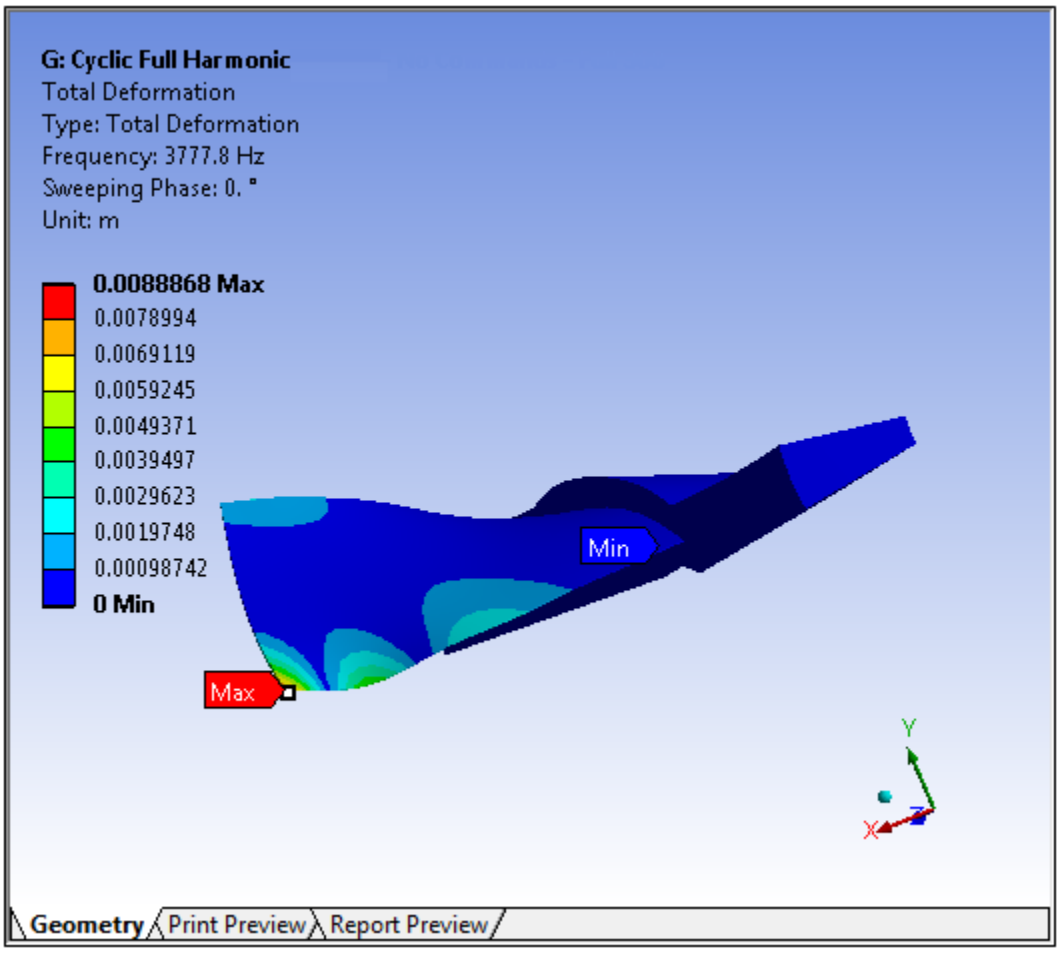
Note:

Extremum values (for example, Minimum, Maximum) correspond only to the portion of the model selected by these properties (under **Cyclic Solution Display**).

Unexpanded One Sector Model Display

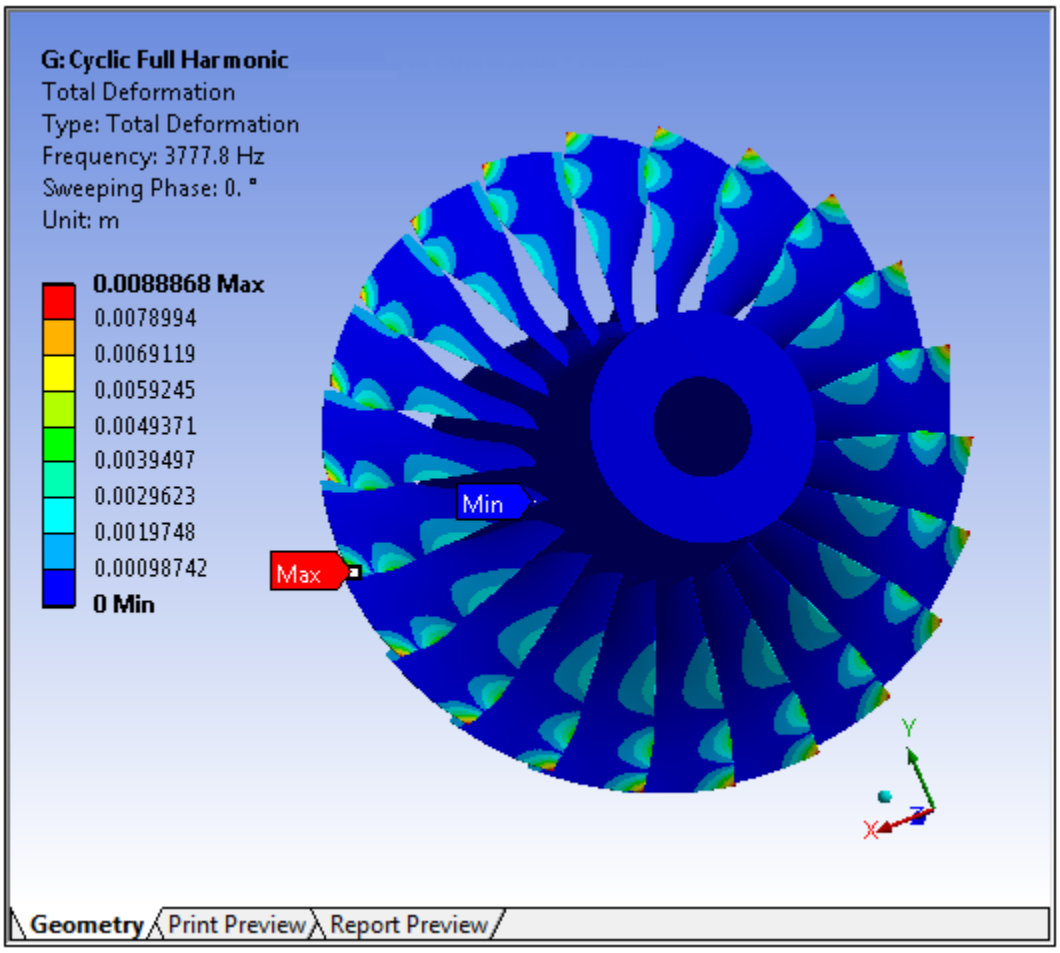
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



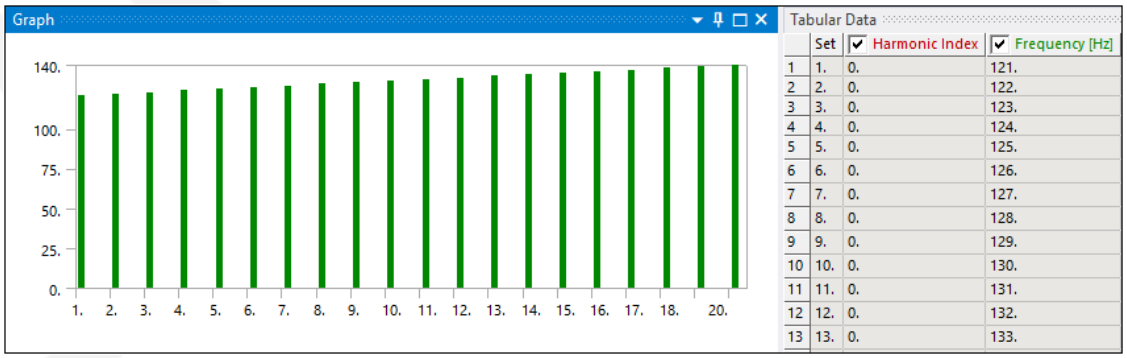
Expanded Full Symmetry Model Display

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

- Probe results are not available for Harmonic Response (Full) analyses when cyclic symmetry is used.
- Selecting the **Solution** object or a result object and then selecting the **Frequency** checkbox in the **Tabular Data** window displays the following bar graph. The **Harmonic Index** values in **Tabular Data** are zero if the loading is cyclic.

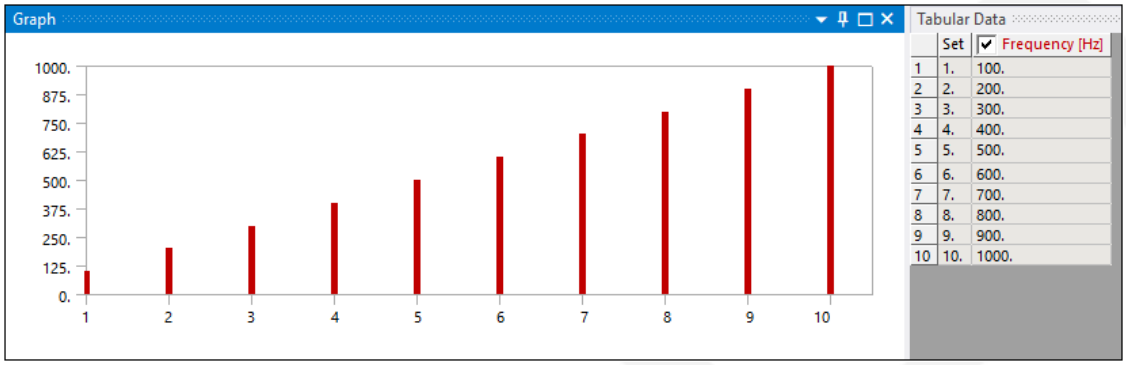


- When Non-Cyclic loads (p. 1153) are present in the model, the **Harmonic Index** is not shown. When you request a result at a frequency, the cyclically expanded result is

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

displayed which accounts for the contribution from all the available harmonic index at the requested frequency. This is similar to the output from the Mechanical APDL **/CYCEXPAND** command.



- When Non-Cyclic loads (p. 1153) are present in the model, the **Sector Number** property is displayed. You can set this property to view the frequency and phase response results on the desired sector.

Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

When you perform a Modal (p. 515) or FSI Modal Acoustics analysis that involves cyclic symmetry (p. 1134), unique features are available for loads/supports, analysis settings, and reviewing results. These features are described in the following sections:

Applying Loads and Supports for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

Analysis Settings for Cyclic Symmetry in a Modal Analysis

Analysis Settings for Cyclic Symmetry in a FSI Modal Acoustics Analysis

Reviewing Results for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

Applying Loads and Supports for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

The following support limitations and specifications must be observed:

- Elastic Supports and Compression Only Supports are **not** permitted.
- Supports should not include any face selections (for example, on 3D solids) that already belong to either the low or high boundaries of the cyclic symmetry sector. Supports may include edges (for example, on 3D solids) on those boundaries, however.
- Only the following remote boundary conditions **are** supported:
  - Remote Displacement
  - Point Mass

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– Spring

Warning:

Make sure that the location of the **Remote Points** does not coincide with the cyclic symmetry axis, as this could generate incorrect results. Two exceptions exist for standalone or MSUP modal analyses: if either a **Displacement** or a **Moment** (for a subsequent MSUP) is applied about the cyclic symmetry axis and then to all sectors, it must:

- be scoped to a **Remote Point** that has its **Behavior** property set to **Deformable**,
- have all DOFs fixed except the DOF about the cyclic symmetry axis, and
- the **Remote Point** must lie on the cyclic axis of symmetry.

In preparation for solution, the boundary conditions on the geometry are converted into node constraints in the mesh (see Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) for more information). When these boundary conditions involve nodes along the sector boundaries (low, high and axial boundaries), their constraints are integrated to properly reflect the symmetry. As an example, the low and high edges may feature more node constraints than are applied to each individually, in order to remain consistent with an equivalent full model.

If the modal analysis is activated as pre-stressed (p. 349), no other modal loads/supports are allowed. On the other hand you can apply all pertinent structural loads/supports (p. 1147) in the previous cyclic static analysis.

When using the Samcef or ABAQUS solver, compatibility of supports with cyclic symmetry is checked internally. If an incompatibility is detected a warning or error message will be displayed, and the solve will be interrupted.

Analysis Settings for Cyclic Symmetry in a Modal Analysis

A Modal analysis involving cyclic symmetry includes a Cyclic Controls (p. 1446) category that enables you to solve the harmonic index for all values, or for a range of values. This category is available if you have defined a **Cyclic Region** in the analysis.

Note:

Currently for Modal Analysis with Cyclic Symmetry:

- The Unsymmetric **Solver Type** (UNSYM) is not supported.
- **Damping** is not supported (Fully Damped, DAMPED, or Reduced Damped, QRDAMP).
- Expansion is only available for harmonic indices > 0 with the Samcef or ABAQUS solver.

For more information about the associated Mechanical APDL command, see the **MODOPT** section of the *Mechanical APDL Command Reference*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Settings for Cyclic Symmetry in a FSI Modal Acoustics Analysis

A modal analysis involving cyclic symmetry includes a **Cyclic Controls** (p. 1446) category in the **Analysis Settings** object that enables you to solve the harmonic index for all values or for a range of values. This category is available if you have defined a **Cyclic Region** object in the analysis.

Note:

Currently for FSI Modal Acoustics Analysis with Cyclic Symmetry:

- The Unsymmetric Solver Type (UNSYM) is supported.
- Damping is not supported (Fully Damped, DAMPED).

See the **MODOPT** section of the *Mechanical APDL Command Reference* for additional information.

Reviewing Results for Cyclic Symmetry in a Modal or FSI Modal Acoustics Analysis

A Modal analysis involving cyclic symmetry includes additional options to help you navigate and interpret the results. In particular, there are features to:

- Review the complete range of modes: you may request the modes to be sorted by their serial number in the results file or by their frequency value in the spectrum.
- Review combinations of degenerate modes through the complete range of phase angles.

When simulating cyclic symmetry in a Modal analysis, the same results are available as for a modal analysis with full symmetry, with the exception of Linearized Stresses (which are undefined). In addition, purely elemental results, such as Volume, Kinetic Energy, and Potential Energy display only in the base sector. To verify a result is elemental, examine the **Worksheet** (from the Solution object) for **Solution Quantities and Result Summary** (p. 1961) > **Available Solution Quantities**.

Although only one cyclic sector is analyzed, results (excluding elemental results) are valid for the full symmetry model. You can control the post-processing and display of cyclic results using the following options of the **Cyclic Solution Display** category of the **Solution** folder:

- **Number of Sectors**: This option controls the extent the model is expanded from the raw solution. The value indicates how many sectors should be processed, displayed and animated. Results generate more quickly and consume less memory and file storage when fewer sectors are requested. To set the value as Program Controlled, enter zero; this value reveals the full expansion.
- **Starting at Sector**: Selects the specific sectors to include within the expansion. For example, if Number of Sectors is set to 1, sectors 1 through N are revealed one at a time. To set the value as Program Controlled, enter zero; this value reveals the specified number of sectors from sector 1 onwards.

In addition, individual result objects contain a **Starting at Sector** property. For it to display, the **Number of Sectors** property above must be set to a value greater than 0. Furthermore, this

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

property overrides the setting of the above **Starting at Sector** property of the **Solution** object. This property gives you the ability to define different starting sectors for each result object.

Note:

- Extremum values (for example, Minimum, Maximum) correspond only to the portion of the model selected by these properties (under **Cyclic Solution Display**).
- To expand a mode shape, Mechanical applies the scale factors as defined in the Complete Mode Shape Derivation section of the *Mechanical APDL Theory Reference*. Normalization to unity is not supported.
- For a Multistage Cyclic Symmetry (p. 1145) analysis, the **Number of Sectors** and **Starting at Sector** properties apply to all Stage objects.

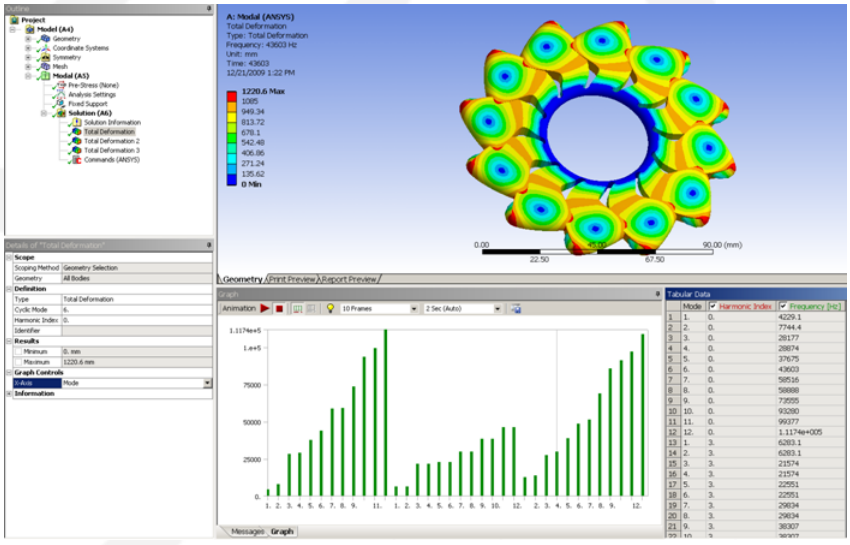
Because these features involve reviewing the mode shapes and contours at individual points within a range, they leverage the charting facilities of the **Graph** and **Tabular Data** windows (p. 193) together with the 3D contour plotting of the **Graphics** view.

Reviewing the Complete Range of Modes

You may request the modes to be sorted in the **Graph** window by their set number in the results file or by their frequency value in the spectrum. You may then interact with the plot (p. 1165) to generate specific mode shapes and contours of interest.

To control how modes are sorted, use the **X-Axis** setting under **Graph Controls** in the Details view of the result and set to either **Mode** or **Frequency**:

- **Mode**: This choice will designate the x-axis in the **Graph** window to indicate the set numbers for each mode (within a harmonic index) in the results file. Each mode will have a vertical bar whose height represents its frequency of vibration. The columns in the **Tabular Data** window are displayed in the order of: **Mode**, **Harmonic Index**, and **Frequency**.

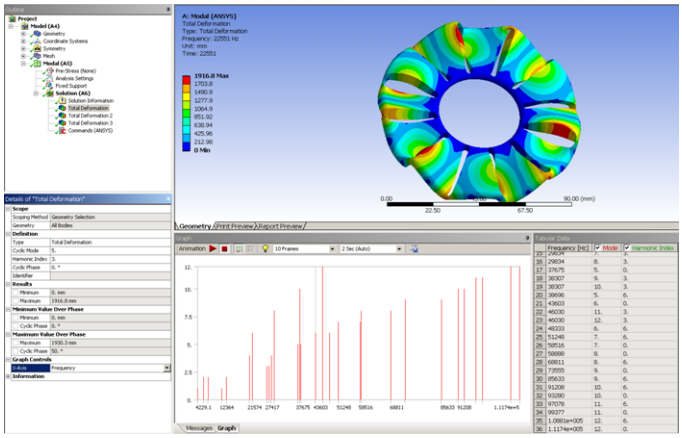


When **X-Axis** is set to **Mode**, the **Definition** category includes settings for **Cyclic Mode** and **Harmonic Index**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Frequency**: This choice will designate the x-axis in the **Graph** window to indicate the mode **Frequency**. Modes are thus sorted by their frequencies of vibration. Each mode will have a vertical bar whose height, for cross-reference, corresponds to the mode number (within its harmonic index). The columns in the **Tabular Data** window are displayed in the order of: **Frequency**, **Mode**, and **Harmonic Index**.



When **X-Axis** is set to **Frequency**, the **Definition** category includes a setting for **Cyclic Phase**. Read-only displays of the **Minimum Value Over Phase** and the **Maximum Value Over Phase** are also available.

- **Phase**: For degenerate modes or couplets, a third option for the **X-Axis** setting under **Graph Controls** is available. This choice will designate the x-axis in the **Graph** window to indicate the phase angle. The graph will show the variation of minimum and maximum value of the result with change in phase angle for the concerned couplet. This setting allows you to analyze the result for a particular mode (for couplets only). The columns in the **Tabular Data** window are displayed in the order of: **Phase**, **Minimum** and **Maximum**. For details on couplets, read the section below.

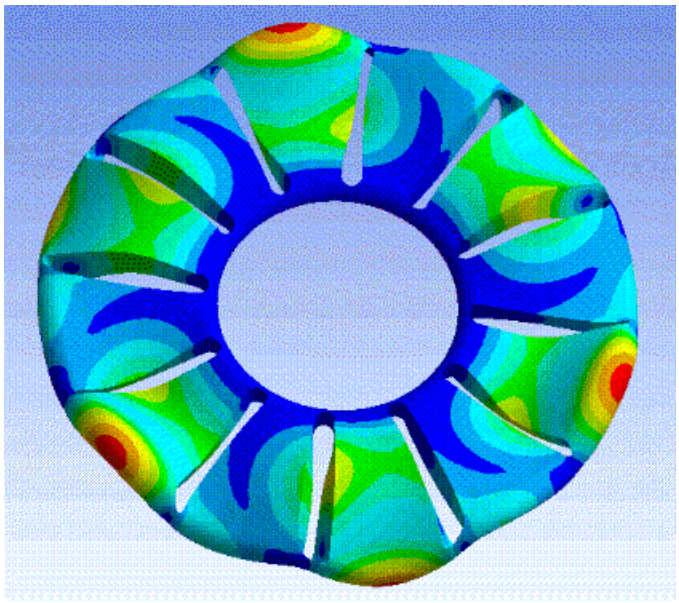
Reviewing results for frequency couplets as a function of cyclic phase angles

An inspection of the results for harmonic indices between 0 and N/2 (that is, 0 < Harmonic Index < N/2) reveals that natural frequencies are reported in pairs by the solver. These pairs of equal value are often termed "couplets." The corresponding mode shapes in each couplet represent two standing waves, one based on a sine and another on a cosine solution of the same spatial frequency, thus having a phase difference of 90°. To appreciate the full range of vibrations possible at a given frequency couplet, it is necessary to review not only the individual mode shapes for sine and cosine (for example, at 0° and 90°) but also their linear combinations which sweep a full cycle of relative phases from 0° to 360°. This sweep is displayed by Mechanical as an animation called a "traveling wave". The following is an example:

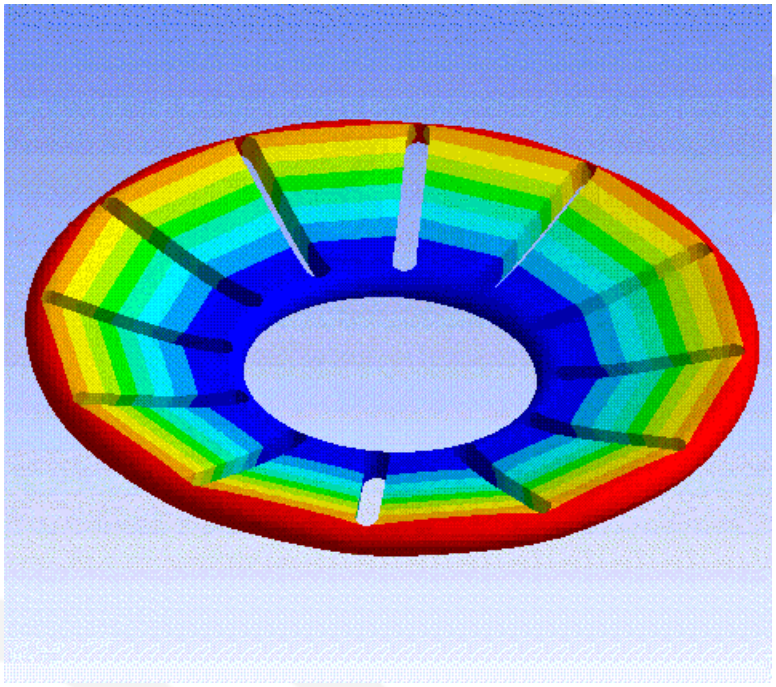
Note:

The following demos are presented in animated GIF format. View online if you are reading the PDF version of the help.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Animations for mode shapes in other harmonic indices, that is, 0 or, for N even, N/2, will yield standing waves. The following animation is an example of a standing wave.



There are options to review the dependence of a result on cyclic phase angle quantitatively. For applicable harmonic indices, results can be defined by:

- Cyclic Phase: Use in combination with the **Cyclic Phase** setting to report the contour at a specific phase. Under this setting, the result will also report the **Minimum Value Over Cyclic Phase** and the **Maximum Value Over Cyclic Phase**.
- Maximum over Cyclic Phase: this contour reveals the peak value of the result as a function of cyclic phase for every node/element.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

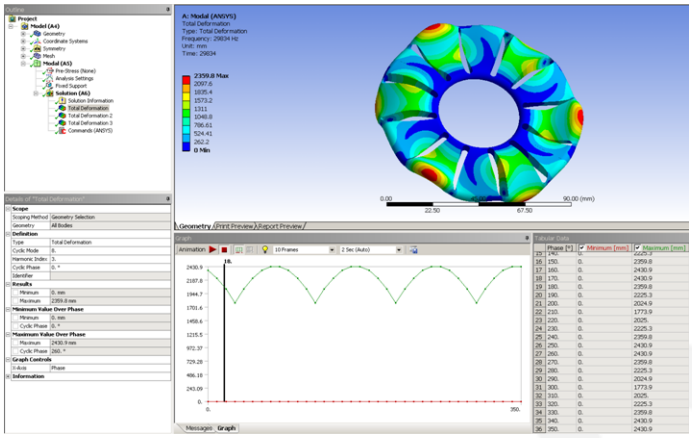
Published: 2023-10-23

- Cyclic Phase of Maximum: this contour reveals the cyclic phase at which the peak value of the result is obtained for every node/element.

When the result is defined by **Cyclic Phase**, it may be convenient to use the interaction options (p. 1165) to pick the value of phase from the **Tabular Data** window as an alternative to direct input in the Details view. To access this feature, set the **X-Axis** to **Phase** under **Graph Controls**.

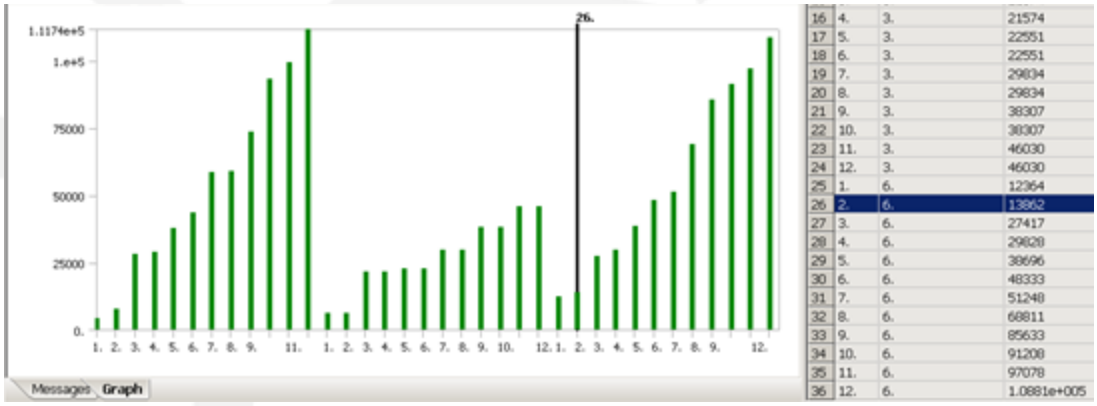
To control the density of the cyclic phase sweep, select **Options** from the **File** tab (p. 37), and then select the **Frequency** (p. 224) category. Modify the **Cyclic Phase Number of Steps** property.

The phase sweep can be disabled individually on a result by setting **Allow Phase Sweep** to **No** in the Details view.



Interaction Options

The **Graph**, **Tabular Data** and the **Graphics** view can be used in concert while reviewing modal cyclic results. For example, if you click in the **Tabular Data** window, a black vertical cursor moves to the corresponding position in the chart. Conversely, if you click a bar (for **Mode** or **Frequency** display) or a node in the chart (for a **Phase** display), the corresponding row is highlighted in the **Tabular Data** window. Multi-selection is also available by dragging the mouse over a range of bars or nodes (in the chart) or rows in the **Tabular Data** window. These are useful in identifying the mode number and harmonic index with specific values of the frequency spectrum.



16	4.	3.	21574
17	5.	3.	22551
18	6.	3.	22551
19	7.	3.	29834
20	8.	3.	29834
21	9.	3.	38307
22	10.	3.	38307
23	11.	3.	46030
24	12.	3.	46030
25	1.	6.	12364
26	2.	6.	13862
27	3.	6.	27417
28	4.	6.	29828
29	5.	6.	38696
30	6.	6.	48333
31	7.	6.	51248
32	8.	6.	68811
33	9.	6.	85633
34	10.	6.	91208
35	11.	6.	97078
36	12.	6.	1.0881e+005

Also, the **Graph** or **Tabular Data** windows can be used to request a specific mode shape at a phase value of interest (if applicable) using context sensitive options. To access these, select an item in the **Graph** or **Tabular Data** windows and click the right mouse button. The following are the most useful options:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Retrieve This Result**: Auto-fills the **Mode** and **Harmonic Index** (for a **Mode** or **Frequency** display) or the **Phase** angle (for a **Phase** display) into the Details view of the result and will force the evaluation of the result with the parameters that were recently changed.
- **Create Mode Shape Results**: processes the selected pairs (**Mode**, **Harmonic Index** defined by dragging in the **Graph** window to produce a light blue rectangle) and inserts results under the **Solution** folder. You must then evaluate these results, since they are not evaluated automatically. This option is not available for **Phase** display.

The following two options are available only if you click the right mouse button in the **Graph** window:

- **Zoom to Range**: Zooms in on a subset of the data in the **Graph** window. Click and hold the left mouse at a step location and drag to another step location. The dragged region will highlight in blue. Next, select **Zoom to Range**. The chart will update with the selected step data filling the entire axis range. This also controls the time range over which animation takes place.
- **Zoom to Fit**: If you have chosen **Zoom to Range** and are working in a zoomed region, choosing **Zoom to Fit** will return the axis to full range covering all steps.

Cyclic Symmetry in a Thermal Analysis

When you perform a steady state thermal analysis (p. 657) or transient thermal analysis (p. 680) that involves cyclic symmetry (p. 1166), unique features are available for loads/supports and reviewing results. These features are described in the following sections:

Applying Loads for Cyclic Symmetry in a Thermal Analysis

Reviewing Results for Cyclic Symmetry in a Thermal Analysis

Applying Loads for Cyclic Symmetry in a Thermal Analysis

For a thermal analysis, in the presence of cyclic symmetry, Coupling loads are not available.

Also, loads should not include any face selections (for example, on 3D solids) that already belong to either the low or high boundaries of the cyclic symmetry sector. Loads may include edges (for example, on 3D solids) on those boundaries, however.

Loads are assumed to have the same spatial relation for the cyclic axis in all sectors.

In preparation for solution, the boundary conditions on the geometry are converted into node constraints in the mesh (see Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) for more information). When these boundary conditions involve nodes along the sector boundaries (low, high and axial boundaries), their constraints are integrated to properly reflect the symmetry. As an example, the low and high edges may feature more node constraints than are applied to each individually, in order to remain consistent with an equivalent full model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Reviewing Results for Cyclic Symmetry in a Thermal Analysis

When simulating cyclic symmetry in a thermal analysis, the same results are available as results in a thermal analysis (p. 2153) that involve full symmetry.

Note:

Radiation Probe results (p. 2155) are calculated for the full symmetry model.

General Axisymmetric Overview

For three-dimensional (3D) Static Structural analyses only, the **General Axisymmetric** feature enables you to create an axisymmetric mesh, in the circumferential direction, on a surface body model that is based on specified nodal planes and an axis. This feature supports edge and vertex scoping only. Additionally, from these surface model edges and vertices, you can generate three-dimensional node-based Named Selections that you can then use as scoping items for other simulation options such as loading conditions and/or results. An axisymmetric model created using this feature greatly reduces the modeling and analysis time compared to that of an equivalent 3D model.

See the General Axisymmetric Workflow in Mechanical (p. 1188) section for the steps to use the feature.

Go to a section topic:

- Element Description (p. 1167)
- General Limitations (p. 1168)
- Connection Limitations (p. 1168)
- Contact Limitations (p. 1168)
- Named Selection Limitations (p. 1168)
- Load and Boundary Condition Limitations (p. 1168)
- Graphical Limitations (p. 1169)
- Result Limitations (p. 1169)
- Graphic Display of Axis and Nodal Planes (p. 1169)
- Mesh Generation and Display (p. 1170)
- Named Selection Display (p. 1172)
- Contact Support (p. 1173)
- Load/Support Scoping Specifications (p. 1174)
- Result Display (p. 1178)

Element Description

This feature uses the Mechanical APDL elements SOLID272 and SOLID273. See the **General Axisymmetric Elements** section in the *Mechanical APDL Element Reference* or review the pertinent sections of the *Mechanical APDL Theory Reference* for additional information.

Limitations

The **Axis** of symmetry you specify should not intersect your surface body and must be in the same plane as the selected surface body. In addition, review the categorized limitations listed below. You cannot use the listed features and capabilities in combination with the General Axisymmetric feature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

General Limitations

- All analysis types other than Static Structural analyses.
- Non-planar surface bodies.
- Element Orientations.
- Point Mass and Distributed Mass.

Connection Limitations

- Beams, Bearings, Joints, and Springs.
- Beam contact formulation type.
- Only bonded contact is supported between General Axisymmetric bodies with one nodal plane.
- When the **Type** property is set to **Bonded** and the **Formulation** property is set to **MPC**, only the **Programmed Controlled** and **Projected, Displacement Only** options for the **Constraint Type** (p. 1248) property are supported. All other options define force distributed constraints that are not supported for General Axisymmetric contacts.
- The **Flip Contact Normals** and the **Flip Target Normals** properties cannot be used to invert the normal direction of edge contacts on 2D surface bodies.

Contact Limitations

- For contact between General Axisymmetric bodies that have one nodal plane, make sure that at least one node is constrained in Z direction for the Target body.
- Contacts imported from an imported base mesh will lose their mesh scoping after the General Axisymmetric mesh is generated. As a result, it is recommended that you create contacts with geometry scoping after importing a base mesh.

Named Selection Limitations

- Named Selections defined on faces that are used to define loads or supports.
- Element-based and Element Face-based Named Selections defined using **Worksheet** criteria.

Load and Boundary Condition Limitations

The following load and boundary conditions are not supported:

- Fluid Solid Interface
- Joint Load and Bearing Load
- Line Pressure or Hydrostatic Pressure
- Radial loading
- Imported Loads other than Imported Body Temperature (p. 1740) and Imported Displacement (p. 1746) (only).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Simply Supported, Fixed Rotation, Frictional, and Cylindrical supports.

Also note the following with regards to loading and boundary conditions:

- Scoping the surface bodies defined using the General Axisymmetric feature to objects such as Symmetry Region, Thickness, Layered Section, Imported Thickness, and Imported Layered Section.
- Deformable Behavior is not supported for objects which create or use remote points like Remote Points, Remote loads, Moments, Point Mass, Beam connections, Bearings, Joints, Springs when scoped to General Axisymmetric bodies.

Graphical Limitations

Note the following scenarios where the load direction arrow (annotation) is not displayed in the **Geometry** window:

- Pressure load and Displacement boundary condition defined by **Components** with a Cylindrical Coordinate System.
- Pressure load defined using the **Normal To** option.

When your analysis includes a Thermal Condition, Imported Body Temperature, Imported Displacement, and/or a Spatially Varying Pressure (Normal To) (p. 1769), load variation contours do not display for any of these loads types unless you have the load scoped to nodes using a node-based Named Selection. Otherwise, the variable loading conditions display in purple and there is an annotation in the **Geometry** window that indicates the load as a "Variable Load."

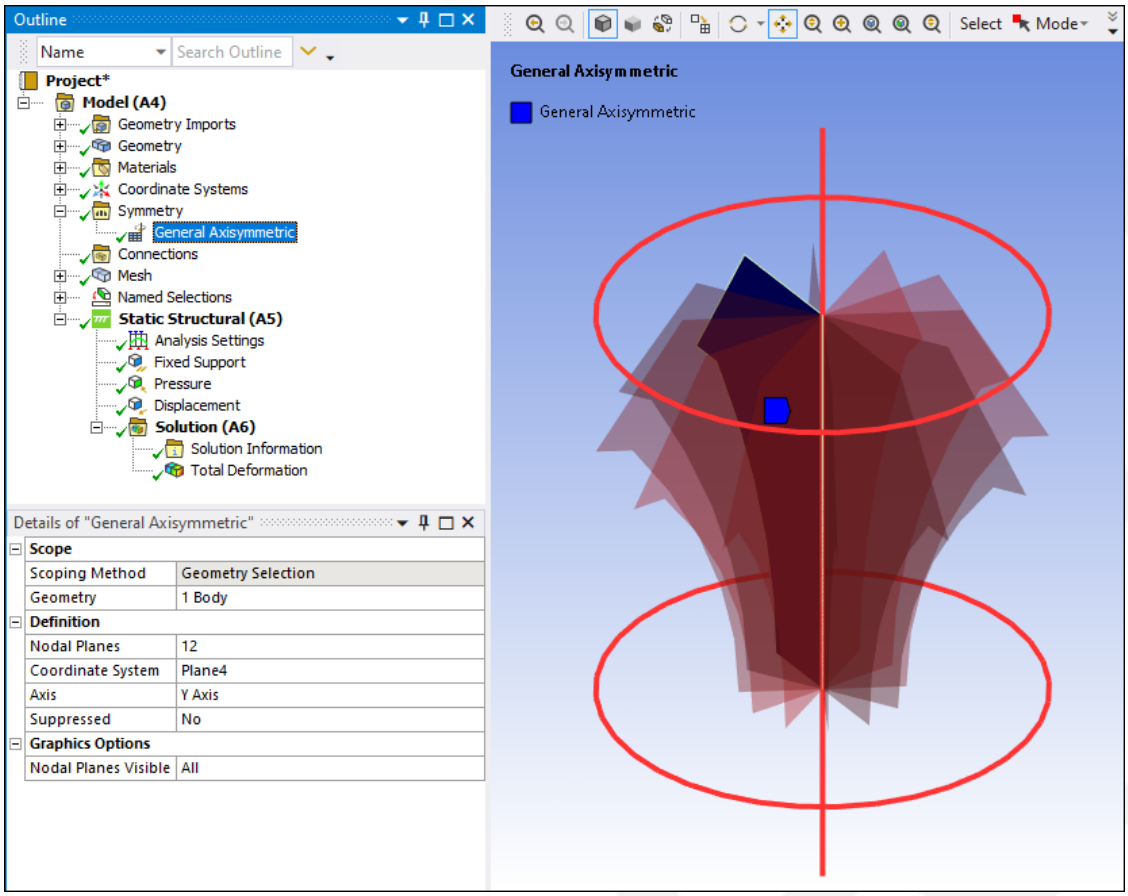
Result Limitations

- Path dependent results such as Linearized stress results.
- Location Probes (**Location Method** property set to **Coordinate System**)
- Path, Surface, and Surface Coating scoping methods of results. That is, results cannot be extracted using Construction Geometries.
- Convergence for results.

Graphic Display

As illustrated here, when you specify a **General Axisymmetric** object, the **Geometry** window automatically displays the axis and the nodal planes around it. The application will generate nodes on these planes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

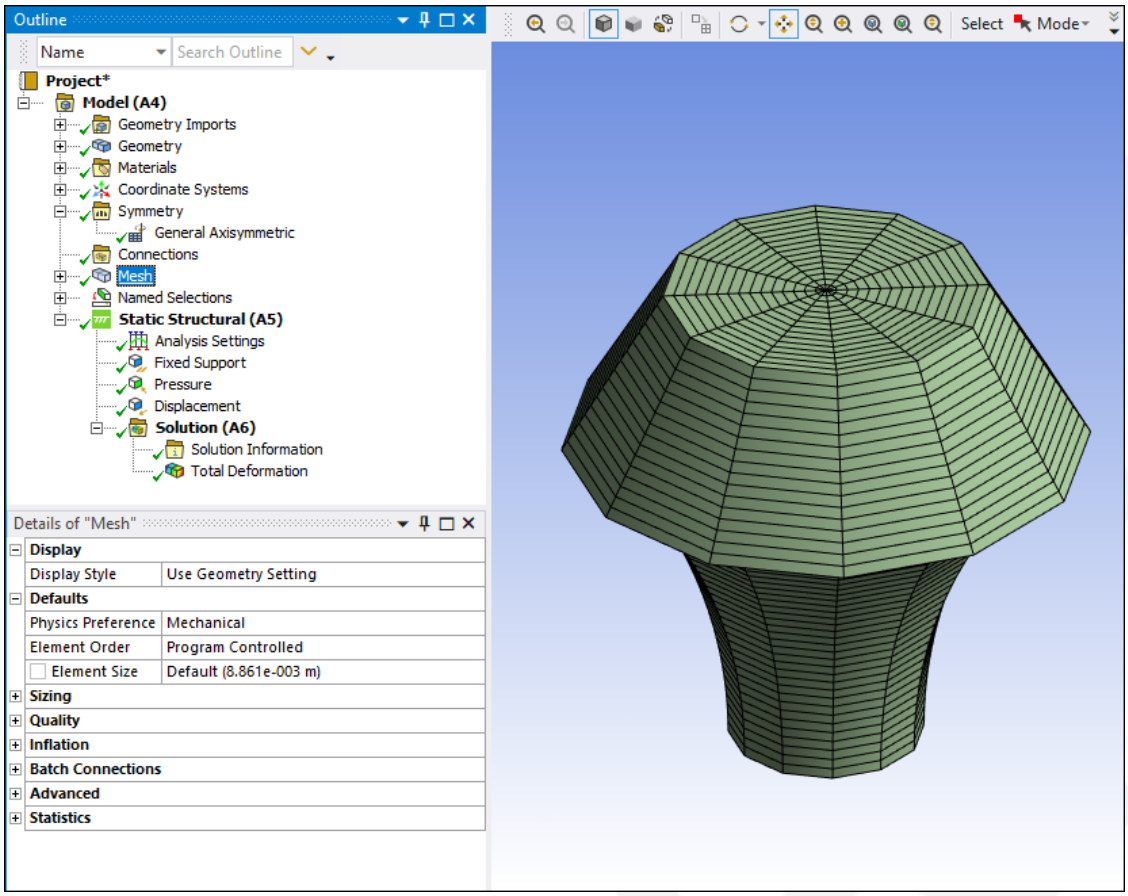


Mesh Generation

Once you fully define the **General Axisymmetric** object, you can generate the General Axisymmetric mesh using the context (right-click) menu option **Generate Mesh**/**Update** on **Mesh** object. The application first creates the base mesh on the surface body and then it generates the General Axisymmetric mesh on all nodal planes in circumferential direction. The example below illustrates a mesh generated for 12 nodal planes.

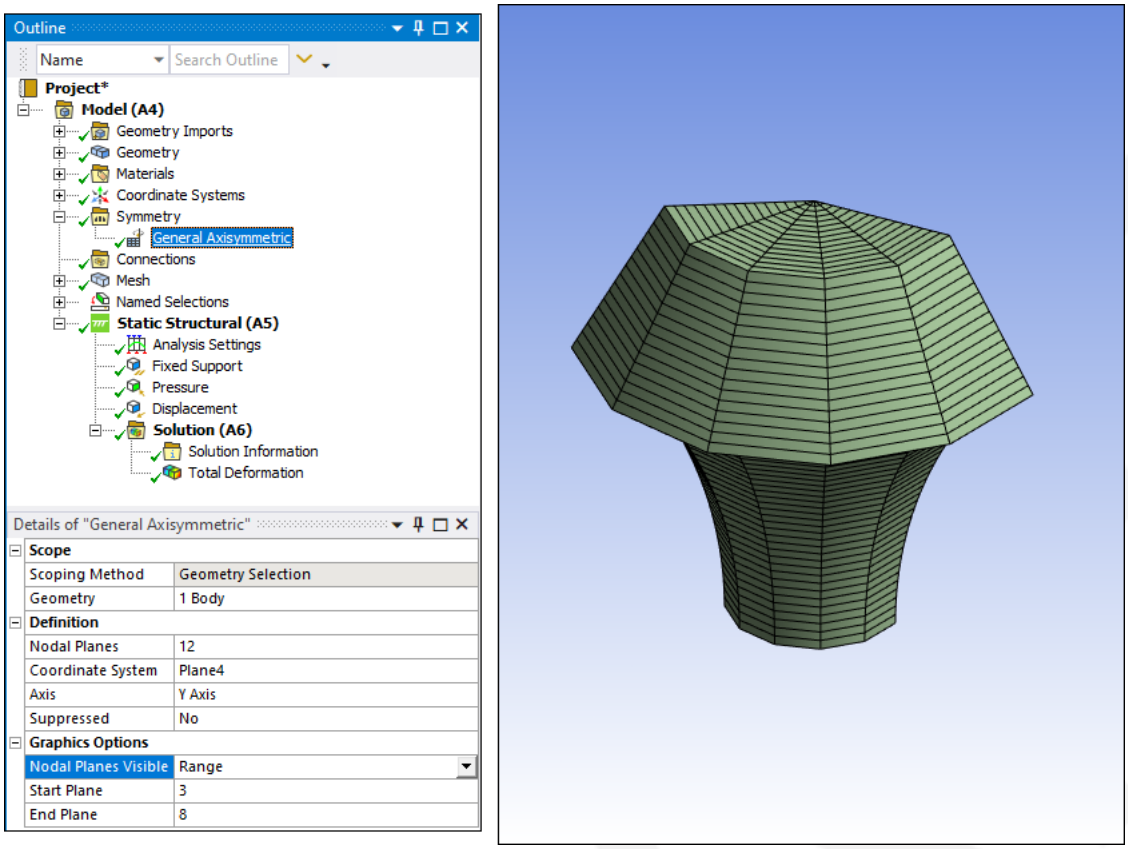
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Additionally, here is an example of the mesh with the **Nodal Planes Visible** property set to **Range**, and the **Start Plane** property set to **3** and **End Plane** property set to **8**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

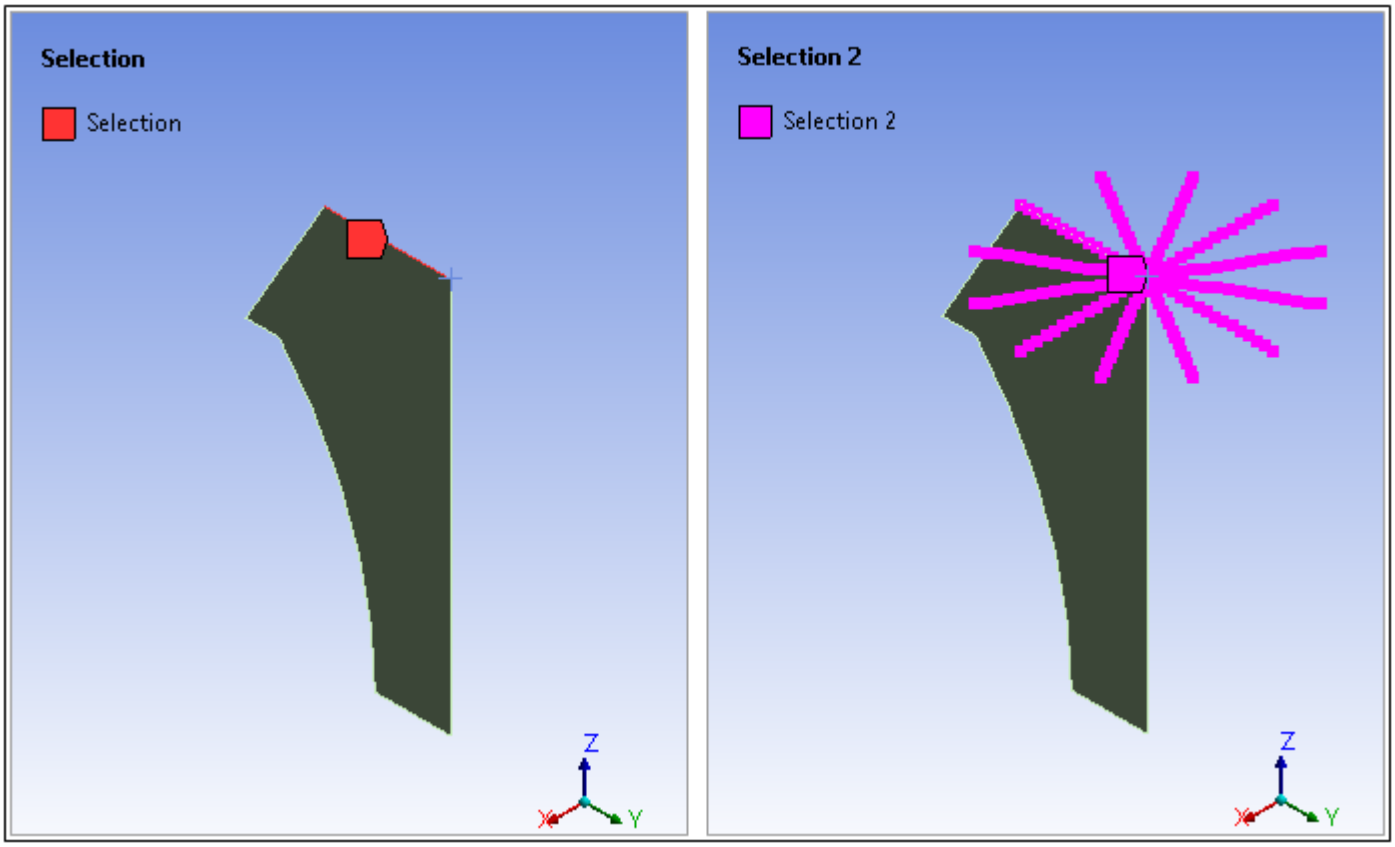


Named Selection Display

As illustrated in the following example, you can create a Named Selection on an edge (or vertex). For this example, you create a node-based Named Selection from the edge-based Named Selection using the context (right-click) menu option Create Nodal Named Selection (p. 1075). This node-based Named Selection represents the circumferential nodes of a full axisymmetric mesh. Node-based Named Selections like this can also be created using the **Worksheet**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

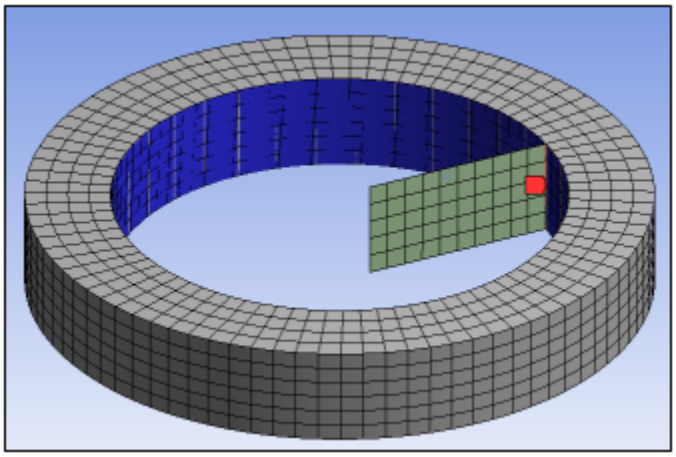


Contact Support

You use this feature on surface bodies. As a result, you need to specify an edge for contact conditions (contact and target selection). Supported contact combinations include:

- The edges of two General Axisymmetric bodies.
- A solid surface as the target side and a General Axisymmetric edge as contact side.
- A 3D shell as the target side and General Axisymmetric edge as the contact side.
- When an edge of a General Axisymmetric body is in contact with a surface or edge of a shell body or with a surface of a solid body, the nodes on the scoped edge in the base nodal plane as well as the equivalent nodes in the circumferential direction form contact. However, if the General Axisymmetric body has only one nodal plane, contact occurs only along the nodes of the edge in the base nodal plane, as illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

For the **Contact Tool** (p. 2079), for node-to-surface contact, **Pressure** displays zeros for results. To display the associated contact force, you must insert a user defined result (p. 2203) called CONTFORC.

The application creates:

- Asymmetric contact pairs.
- Node-to-surface contact pairs (contact elements CONTA175 paired with target elements TARGE170) when the **Nodal Planes** property value specified on the associated **General Axisymmetric** objects equals **3**-**12**.
- 2-D node-to-surface contact pairs (contact elements CONTA175 paired with target elements TARGE169), in the case of a true axisymmetric condition (the **Nodal Planes** property on the associated **General Axisymmetric** objects set to **1**).

Note:

- You cannot create contact pairs between two General Axisymmetric bodies with different nodal planes.
- Only Bonded contact is supported between General Axisymmetric bodies with one nodal plane.
- If the scoped bodies are not in XY plane, contact between General Axisymmetric bodies with only one nodal plane is **not** supported.

Load/Support Application

When specifying loads and supports on your surface body model, you can:

- Apply Direct FE (p. 1719) loading conditions, including: **Nodal Force**, **Nodal Pressure**, and **Nodal Displacement**. You can use these loads to specify non-axisymmetric loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Apply tensile, compressive, and torsional loading using edge or vertex scoping.
- Apply Remote Force, Remote Displacement, and Moments.
- Surface effect elements SURF159 are created to apply a Pressure load when the **Defined By** property is set to **Normal To** and the **Applied By** property is set to **Surface Effect** or when the **Defined By** property set to **Components** or **Vector**.
- An **Imported Body Temperature** load, when applied on General Axisymmetric bodies, uses the **Volumetric** transfer type when the **Mapping Control** property (p. 2389) is set to **Program Controlled** (the recommended data mapping type).
- For imported loads, if you want to map 2D data onto a General Axisymmetric 3D mesh (NP>1), it is recommended that you set the **Dimension** property to **2D** in the Project Schematic when specifying the External Data. In Mechanical, the load is transferred onto the General Axisymmetric mesh using the **2D Projection** property (p. 2389). It is further recommended that you select a cylindrical coordinate system for **2D Projection** property in order to transfer the data in circumferential direction.

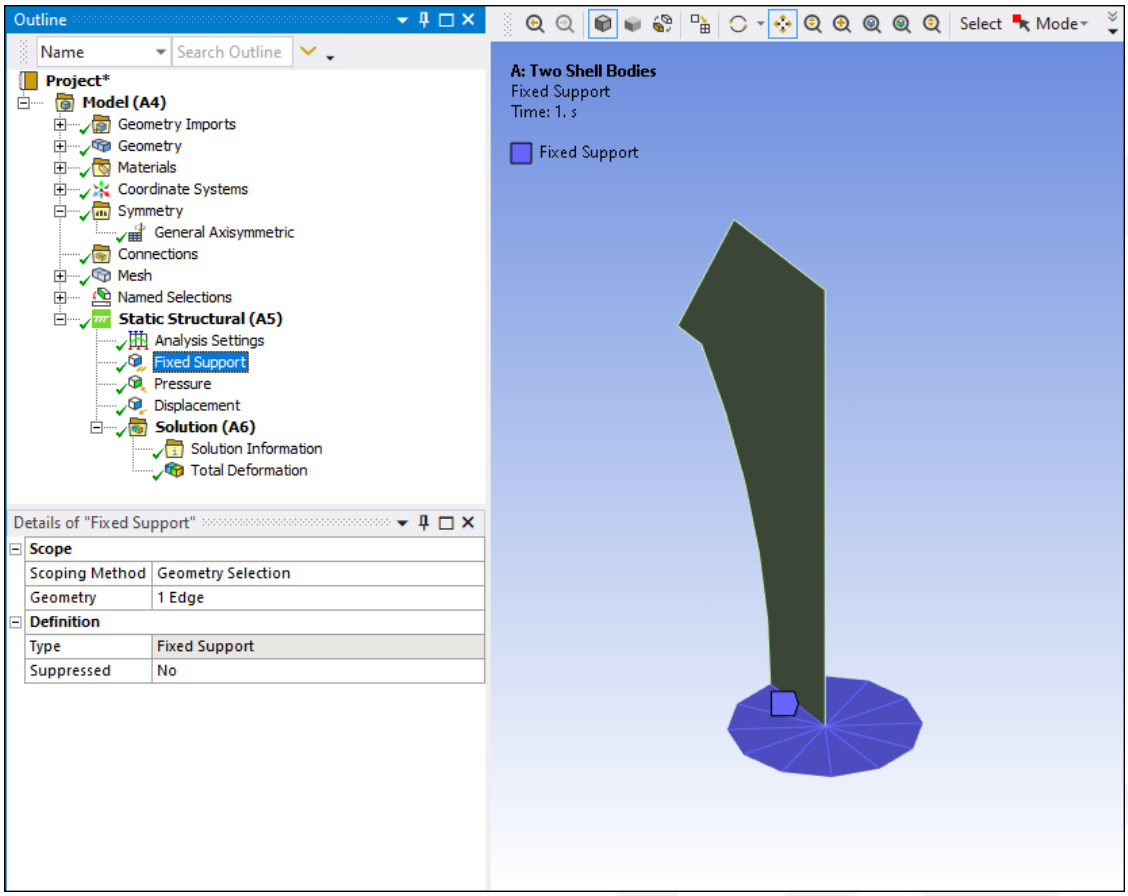
Important:

Loads on General Axisymmetric bodies with one nodal plane may produce torsion. Use constraints along the body plane normal if torsion is not desired.

When you scope loads and/or supports to an edge on surface geometry, the solver transfers the load to the nodes of all the nodal planes. This is essentially the same as selecting a face of a full model, as highlighted below.

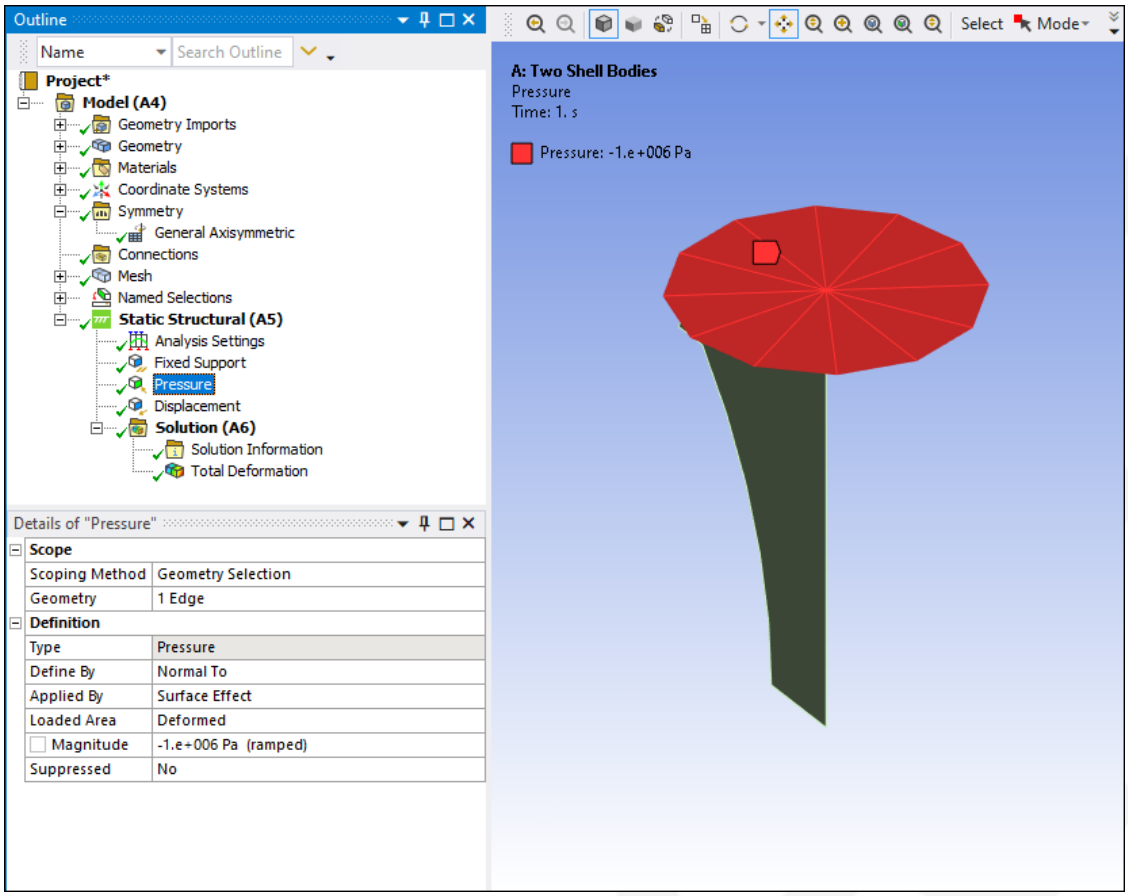
Edge Scoping on a Fixed Support

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Edge Scoping on a Pressure Load

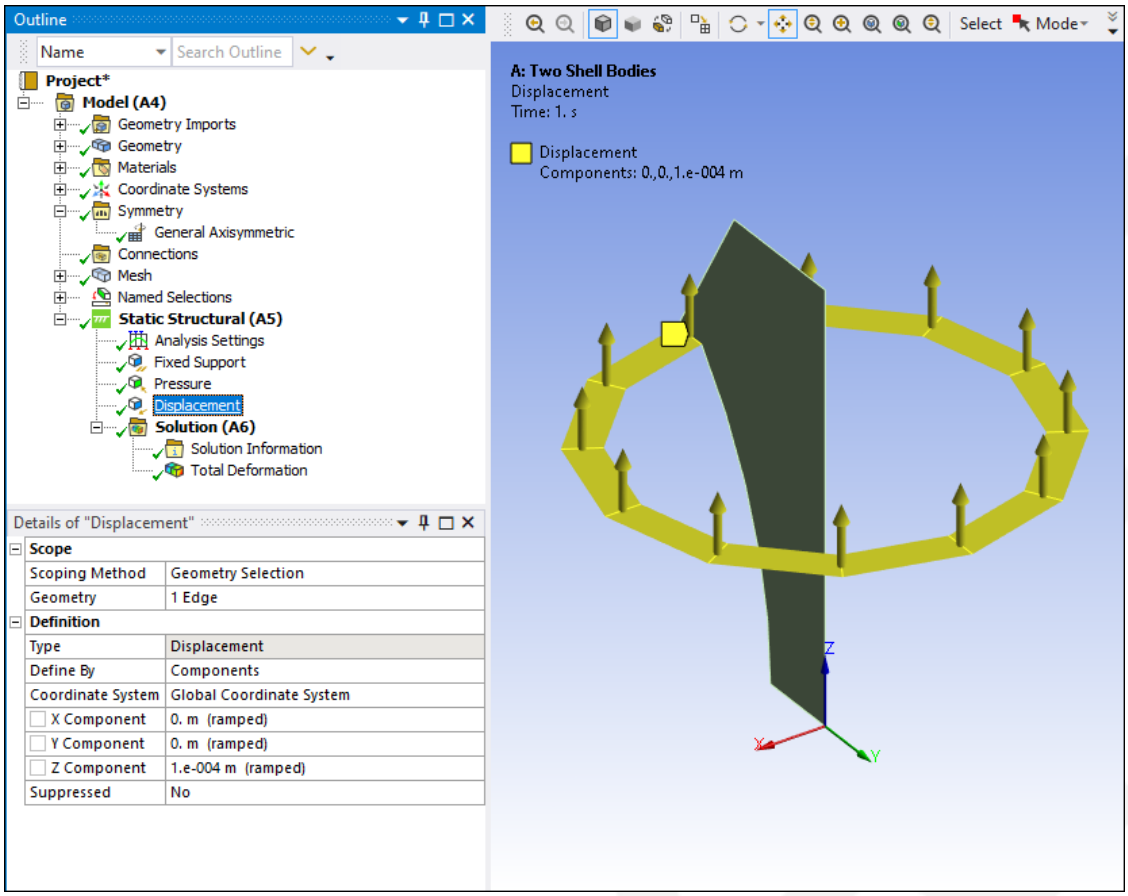
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Similarly, and as illustrated here, when you scope a support or load to a vertex, the solver transfers it onto the corresponding nodes from all nodal planes, which is nothing but an edge of a full model.

Vertex Scoping on a Displacement

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

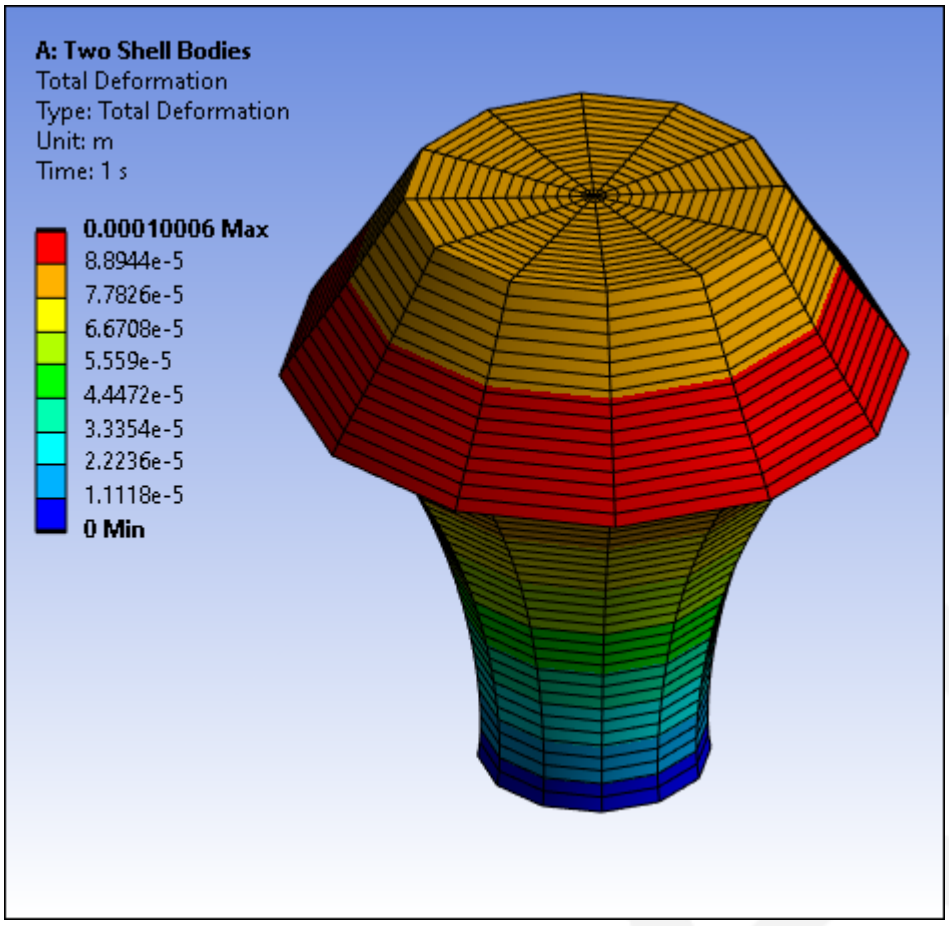


Result Display

Once solved, result contours for this feature display similar to an equivalent 3D solid model.

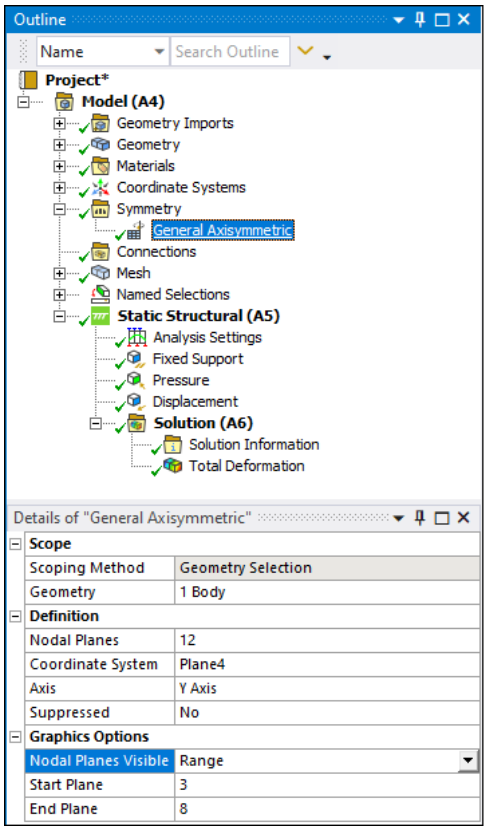
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

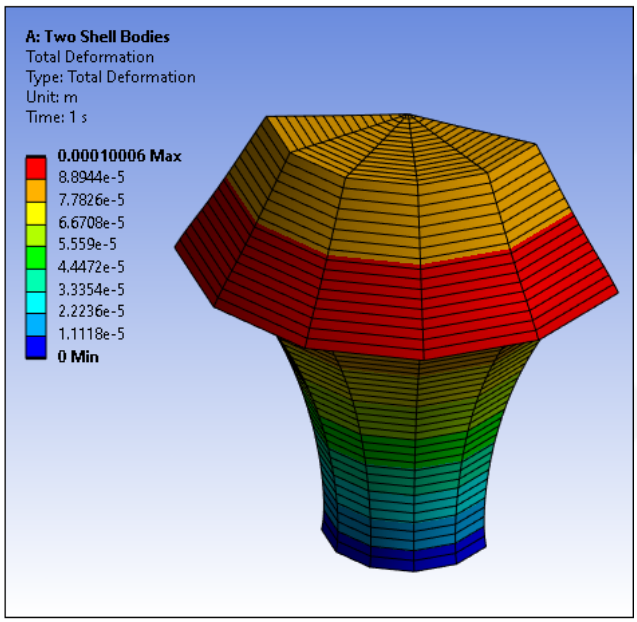
Published: 2023-10-23



This result is an example of the **Nodal Planes Visible** property set to **Range**, and the **Start Plane** property set to **3** and **End Plane** property set to **8**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Symmetry Workflow in DesignModeler

The following procedure describes the steps use to working with Symmetry in DesignModeler.

1. While in DesignModeler, from the **Tools** menu, apply the **Symmetry** feature to the model or define an **Enclosure** .

2. Enter the Mechanical application by double-clicking on the **Model** cell in the **Project Schematic**. The Mechanical application screen appears and includes the following objects in the tree:

   - A Symmetry object.
   - Symmetry Region objects displayed under the **Symmetry** folder. The number of **Symmetry Region** objects corresponds to the number of symmetry planes you defined in DesignModeler.
   - A **Named Selections** folder object. Each child object displayed under this folder replicates the enclosure named selections that were automatically created when you started the Mechanical application.

3. In the Details view of each **Symmetry Region** object, under **Definition**, specify the type of symmetry (p. 1128) by first clicking on the **Type** field, then choosing the type from the drop-down list. Boundary conditions will be applied to the symmetry planes based on both the simulation type and what you specify in the symmetry **Type** field. The **Scope Mode** read-only indication is **Automatic** when you follow this procedure of defining symmetry in DesignModeler. The **Coordinate System** and **Symmetry Normal** fields include data that was "inherited" from DesignModeler. You can change this data if you wish. The **Symmetry Normal** entry must correspond to the **Coordinate System** entry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Symmetry Workflow in Mechanical

The following procedure describes the general steps, for the different types of regions, that you'll use to implement the symmetry feature during an analysis using the Mechanical Application. As needed, refer to Types of Regions (p. 1128) section of the Symmetry documentation for more information about the specific region types.

1. Once you have imported your model into Mechanical, select the **Model** object and select the **Symmetry** option from the **Model** Context Tab (p. 48). Alternatively, you can right-click on the **Model** object or within the **Geometry** window and select **Insert**>**Symmetry** from the context menu.

2. Based on your analysis type, and symmetry needs, insert one of the following Symmetry objects:

   - Symmetry Region (only flexible body scoping is supported)
   - Linear Periodic (this object displays as a **Symmetry Region** object)
   - Periodic Region
   - Cyclic Region
   - Pre-Meshed Cyclic Region

3. Specify the **Scoping Method** property: specify as **Geometry Selection** (default) or **Named Selection**. Based on your selection, one of the following properties is required.

   Geometry/Low Boundary/High Boundary

   These properties are visible when the **Scoping Method** property is set to **Geometry Selection**. You use this property to specify the desired geometric entity. Use the options of the Graphics Toolbar (p. 97) to make your selections.

   For **Periodic Region**, **Cyclic Region**, and **Pre-Meshed Cyclic Region** objects, instead of a **Geometry** property, you are presented with the **Low Boundary** and **High Boundary** properties.

   Each low/high selection can consist of one or more faces over one or more parts, but they must be paired properly. To be valid, each face/edge you specify as the **Low Boundary** must be accompanied by its twin for the **High Boundary**.

   In addition, make sure that each selected face/edge and its twin belong to the same multibody part (although it is not necessary that they belong to the same body). Adjust your multibody parts as needed.

   Note:

   - For the **Periodic Region** and the **Cyclic Region** objects, your low/high selections are used to match the mesh of the two boundaries.
   - For a **Pre-Meshed Cyclic Region** object, it is assumed that the mesh is matched. This object does not influence the mesh.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Named Selection/Low Selection/High Selection

These properties are visible when the **Scoping Method** property is set to **Named Selection**. This field provides a drop-down list of available, and appropriate, user-defined Named Selections.

For **Periodic Region**, **Cyclic Region**, and **Pre-Meshed Cyclic Region** objects, instead of a **Named Selection** property, you are presented with the **Low Selection** and **High Selection** properties.

For a **Periodic Region**, **Cyclic Region**, or **Pre-Meshed Cyclic Region** object, like the **Geometry Selection** option, the **Low Selection** must correspond to the **Low Boundary** component and **High Selection** must correspond to the **High Boundary** component.

The mesh matching requirements are the same as those for a geometry selection.

4. For the **Symmetry Region** and **Periodic Region** objects, and as required, define the **Type** property. It provides a drop-down list of the following options:

   - **Symmetry**
   - **Anti-Symmetry**
   - **Linear Periodic**

   These options essentially apply boundary conditions to the symmetry planes. The **Type** property options support specific analysis types. See the Symmetry Region (p. 1128) section for a more detailed description of use of these options.

5. Specify the **Coordinate System**: Select an appropriate coordinate system from the drop-down list.

   - **Symmetry Region**: Only supports Cartesian coordinate systems.
   - **Periodic Region**, **Cyclic Region**, and **Pre-Meshed Cyclic Region**: Only supports cylindrical coordinate systems.

   As needed, refer to the Coordinate Systems section, Initial Creation and Definition (p. 1192).

6. Based on your symmetry application, you may need to also specify one or more of the following properties:

   - **Symmetry Normal** (**Symmetry Region** only): Specify the normal axis from the drop down list that corresponds to the coordinate system that you chose.
   - **Periodicity Direction**: This option applies a **Symmetry Region** object with the **Type** property set to **Linear Periodic** only. This axis should point into the direction (in user selected Coordinate System) the model should be translated. It might be different from Symmetry Normal property used for other Symmetry Region types.
   - **Behavior**: This property displays during a structural analysis for a **Symmetry Region** object that has the **Type** property set to **Linear Periodic**. This property specifies the coupling behavior between the Low and High Boundaries. Options include:
     - **Free** (default): The application allows the specified **Low Boundary** and **High Boundary** to deform in different directions in order to enable the structure to "breathe."

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Coupled**: The application couples the specified **Low Boundary** and **High Boundary**. This forces the boundaries to experience the same deformation in same direction.

  Refer to the Structural and Thermal Linear Periodic Symmetry (p. 1130) topic for more information.

- **Apply To**: This property displays when piezoelectric physics is present in a Coupled Field (p. 376) analysis that includes a Symmetry Region with the **Type** property set to **Linear Periodic**. This property specifies the degrees of freedom (DOF) applied to the scoped geometry.

  - **Applicable DOF** (default): You can specify Displacement and Voltage DOFs on the geometries of the **Low Boundary** and **High Boundary**.
  - **Displacement**: You can specify Displacement on the geometries of the **Low Boundary** and **High Boundary**.
  - **Voltage**: You can specify Voltage on the geometries of the **Low Boundary** and **High Boundary**.

- **Linear Shift**: This option applies to a **Symmetry Region** object with the **Type** property set to **Linear Periodic** only. This value (positive or negative) represents the increment applied to the node's location in the chosen Periodicity Direction.

  Important:

  If you create more than one **Symmetry Region** object that is specified as **Linear Periodic**, each object must have the same value for the **Linear Shift** property.

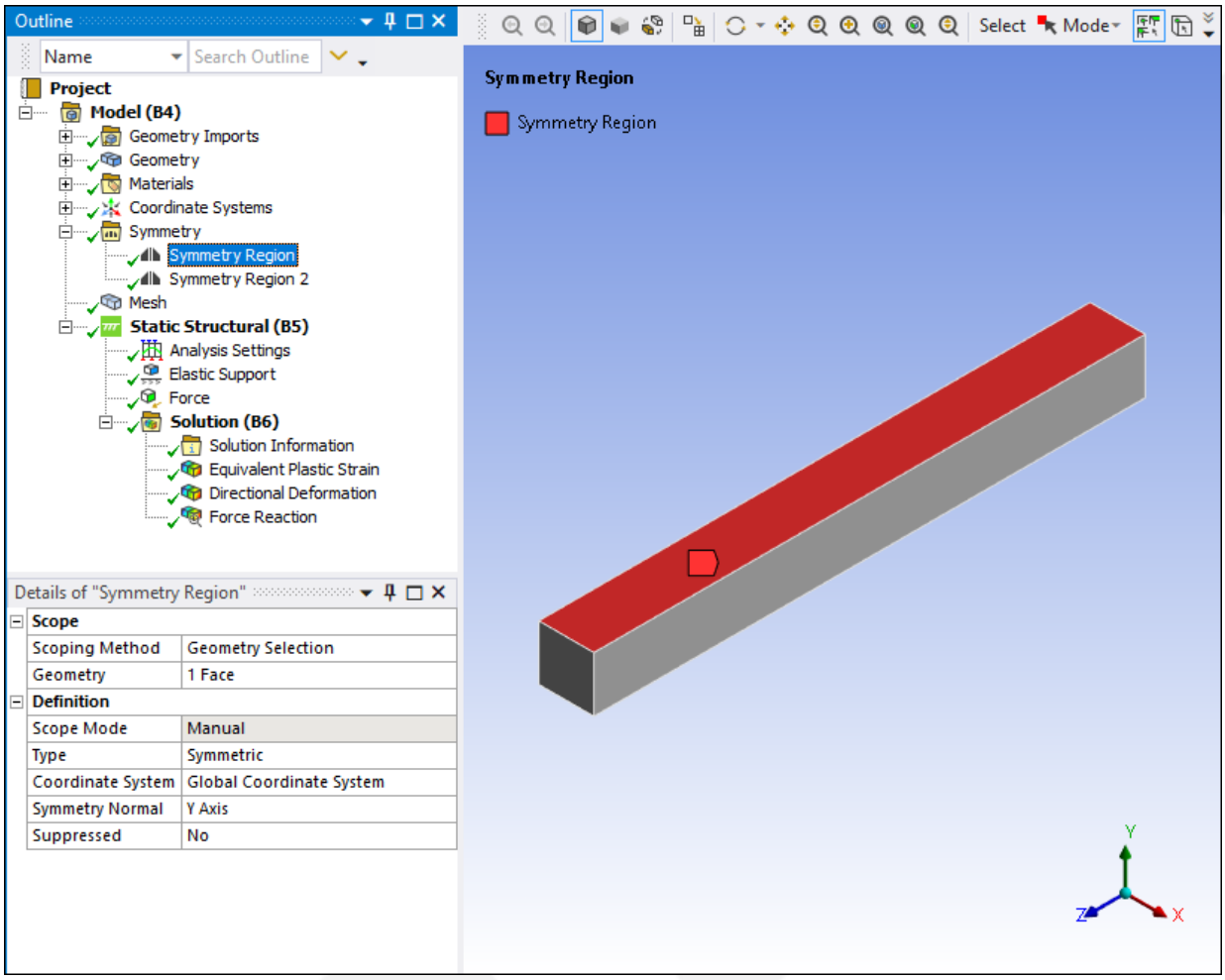
- For a **Pre-Meshed Cyclic Region** only, define the following:

  - **Number Of Sectors**: The entry range is any real number greater than **2**.
  - **Boundary DOF Orientation**:
  - **Relative Distance Tolerance**: The default setting is **Program Controlled** and this setting uses a value of **`-0.0001`**.

Symmetry Examples

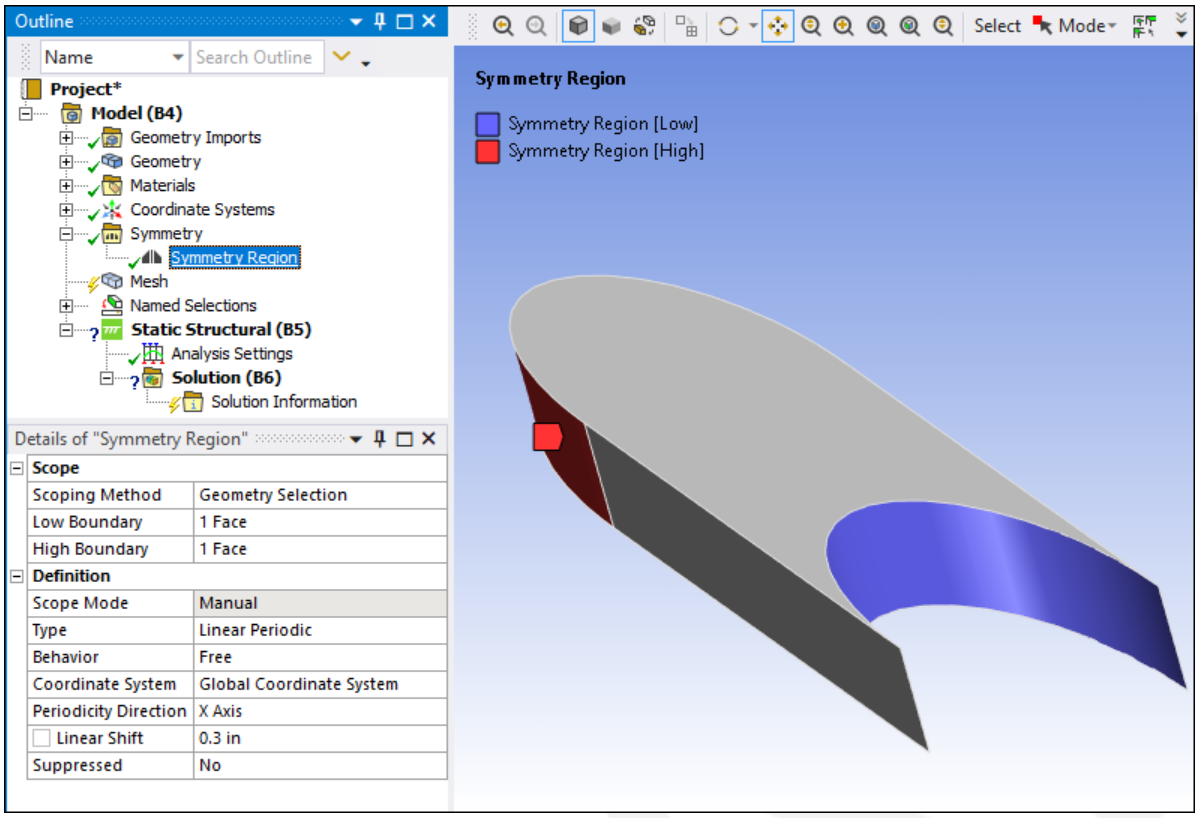
The following example shows a body whose **Symmetry Region** was defined in the Mechanical application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

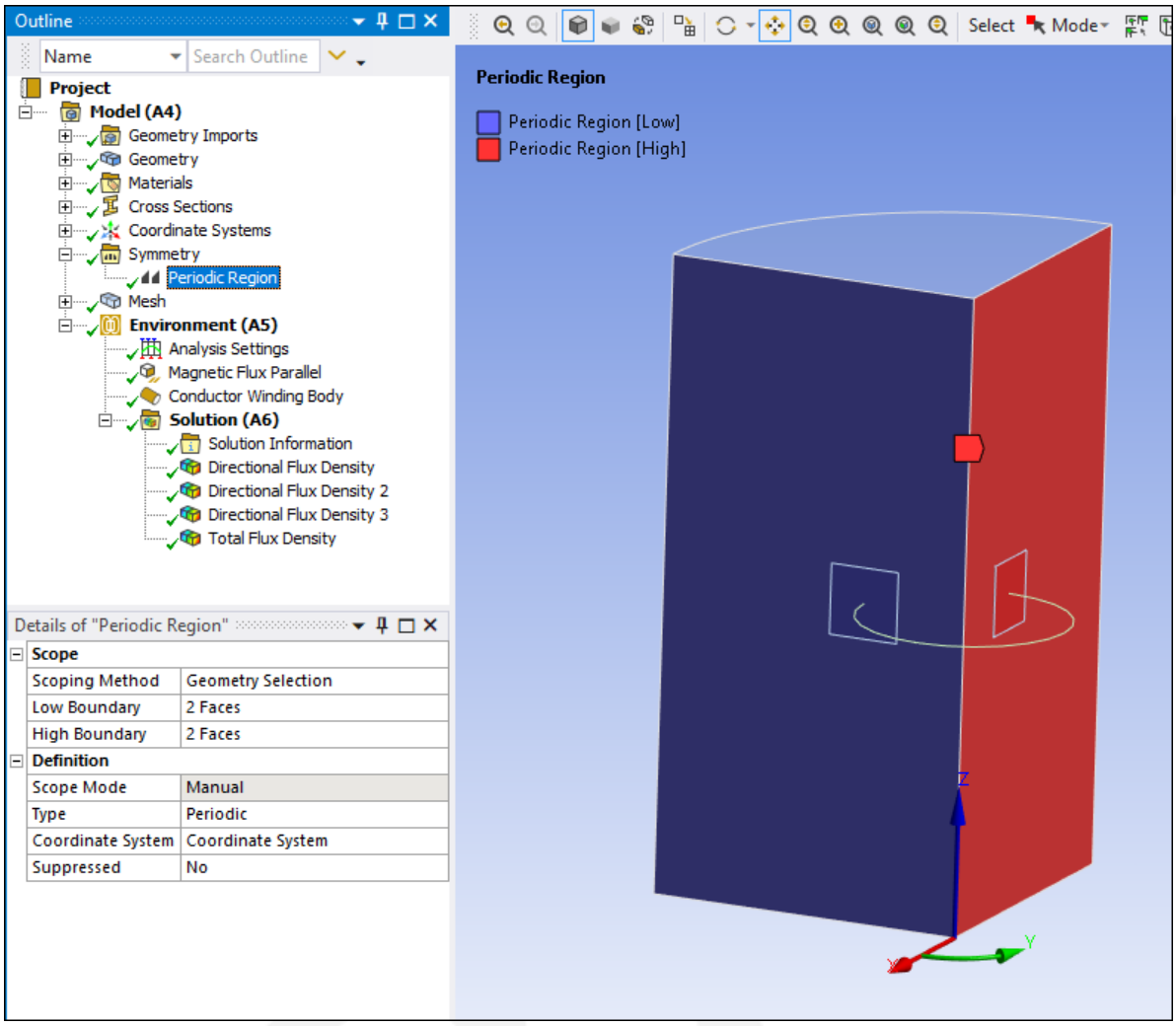


Note:

You can select multiple faces to work with a symmetry region. For Symmetric/Anti-Symmetric Symmetry Regions, all faces selected (or chosen through Named Selection folder) must have only one normal. For Periodic/Cyclic types, you should additionally choose the proper cylindrical coordinate system with the z-axis showing the rotation direction, similar to the Matched Face Mesh meshing option. For a **Symmetry Region** with the **Type** property set to **Linear Periodic**, you should choose the proper Cartesian coordinate system with the **Periodicity Direction** and **Linear Shift** properties showing pertinent values to facilitate conditions similar to the Arbitrary Match Control meshing option. In addition, the **Behavior** property also enables you to specify the coupling behavior between the Low and High Boundaries (**Linear Periodic** only).

The following example shows a body whose **Periodic Region** was defined in the Mechanical application.

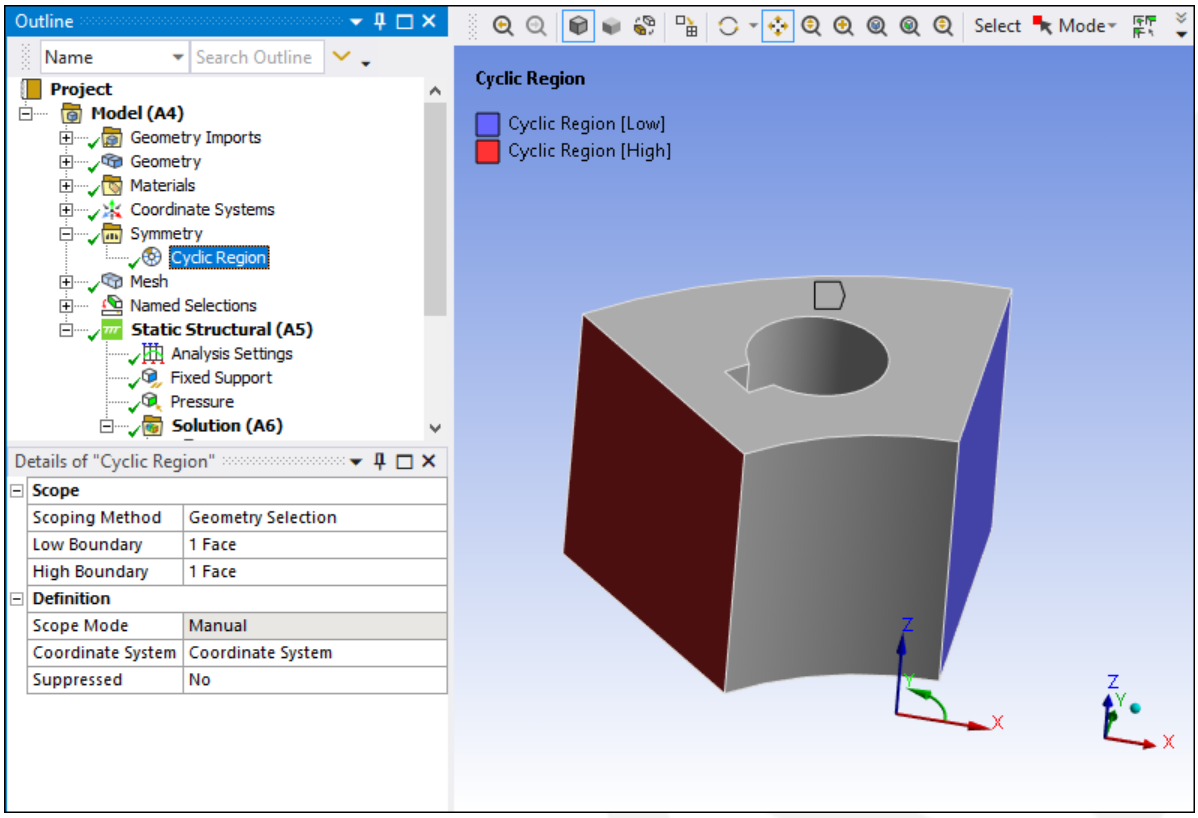
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The following example shows a body whose **Cyclic Region** was defined in the Mechanical application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

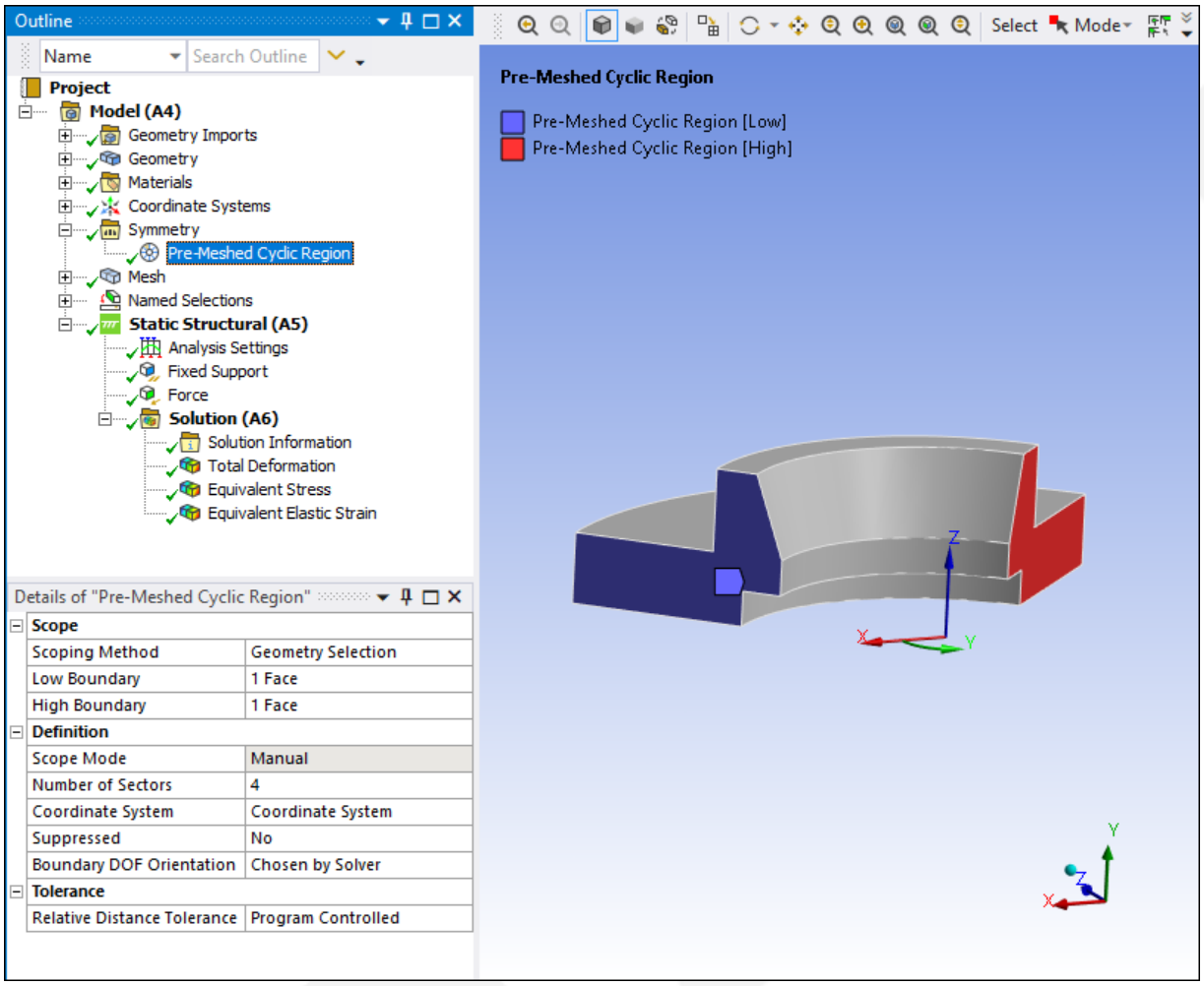
- When using a **Periodic/Cyclic Region** or for a **Symmetry** object whose **Type** is specified as **Linear Periodic**, the mesher automatically sets up match face meshing on the opposite **Low Boundary** and **High Boundary** faces.
- The application displays the following warning if the sector angle does not satisfy the calculation N * angle = 360, with tolerance of 0.5 degree.

  Invalid sector angle : The angle (in degrees) spanned by the base sector should be such that N * angle = 360, where N is an integer and N should be same for all scoped bodies.

A useful feature available is the ability to swap **Low Boundary** and **High Boundary** settings under **Scope** in the Details view. You accomplish this by clicking the right mouse button on the specific symmetry regions (**Ctrl** key or **Shift** key for multiple selections) and choosing **Flip High/Low**.

The following example shows a model defined with the **Pre-Meshed Cyclic Region** object in the Mechanical application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

Except for cyclic symmetry models, symmetry models do not deform for unaveraged results. For example, for an unaveraged stress display, you will see the undeformed shape of the model.

General Axisymmetric Workflow in Mechanical

The following procedure describes the steps to specify a General Axisymmetric object.

1. Once you have imported a surface geometry or a base mesh model into Mechanical, select the **Model** object and then select the **Symmetry** option from the **Model** Context Tab (p. 48). Alternatively, you can right-click on the **Model** object or within the **Geometry** window and select **Insert** > **Symmetry** from the context menu.

2. Select the **Symmetry** folder and then select the **General Axisymmetric** option from the **Symmetry** Context Tab (p. 50). Alternatively, you can right-click on the **Symmetry** folder object or within the **Geometry** window and select **Insert** > **General Axisymmetric** from the context menu.

3. Specify the desired **Geometry**. Only **Body** scoping is supported. You can scope any planar surface body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Using the **Nodal Planes** property, enter your desired number of planes on which nodes should be generated. The entry for this property can be either **1** or **3** through **12**. A **Nodal Planes** entry of **2** is not supported.

5. As needed, modify the **Coordinate System** property. The application uses the **Global Coordinate System** option as the default setting. You can specify a local coordinate system.

6. Specify the **Axis** property. This property specifies the axis about which the mesh nodes are generated. Options include **X Axis**, **Y Axis**, and **Z Axis**.

   Note:

   The specified **Axis** must lie along the selected body, it cannot intersect the body. And, the Axis specified must be on the same plane as the selected surface body.

7. Change the default settings of the **Nodal Planes Visible** property as desired. The default setting displays **All** planes (as specified in the **Nodal Planes** property) or you can set the property to **Range** and select the number of planes you want to display for the mesh and result objects. Enter a consecutive range based on the number of **Nodal Planes**, such as 1-6 or 6-12.

   Note:

   When you change **Graphics Options**:

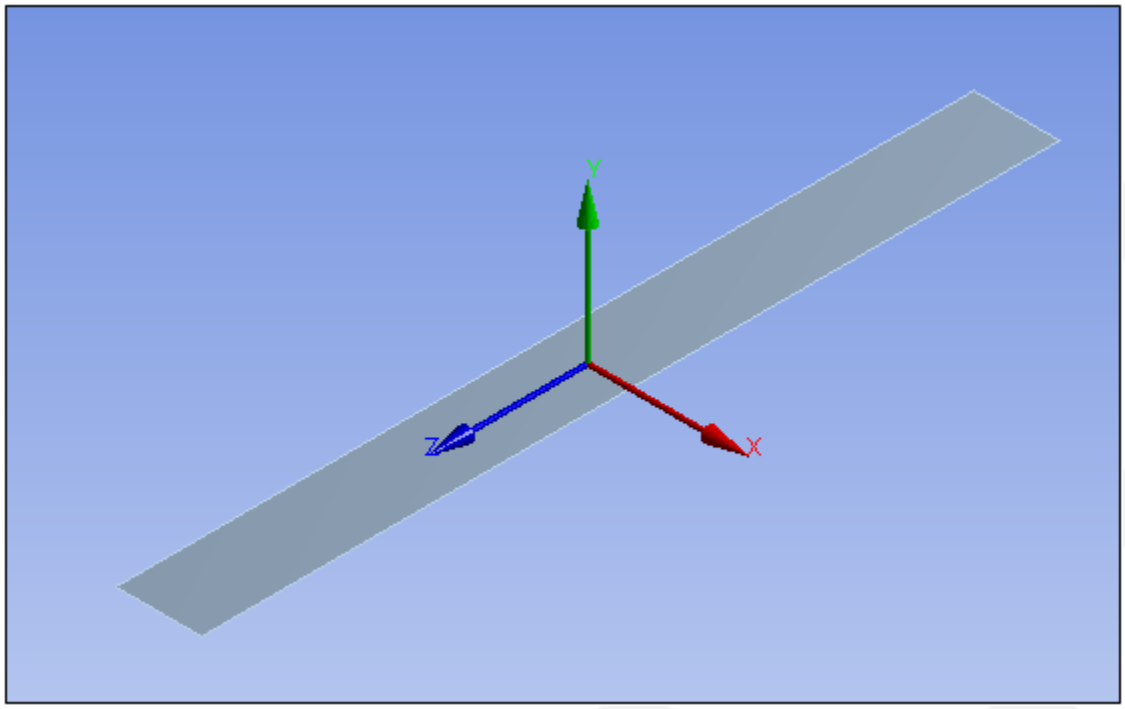
   - The application clears all results. It is necessary to reevaluate the results.
   - For the **Range** option, the display of result **Min** and **Max** values is for the specified range of planes, not the entire model.

Refer to General Axisymmetric Overview (p. 1167) section for additional information, such as load application, limitations, and results options.

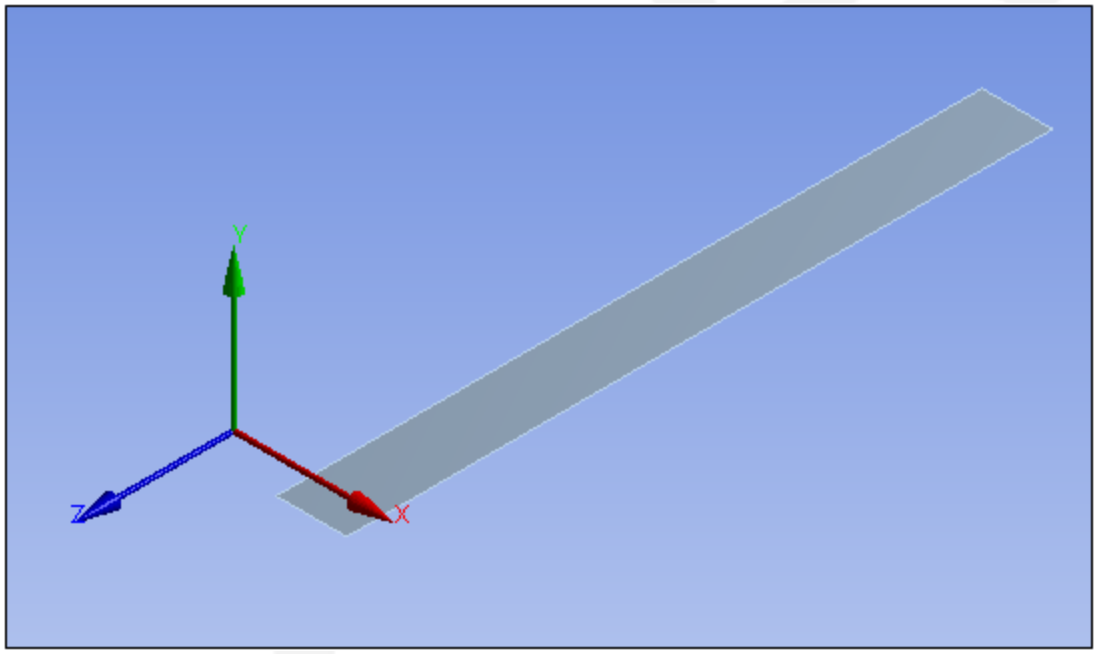
Specification Support Examples

In the example illustrated below, a coordinate system is specified at the centroid of the geometry. As you can see, all three axes of this system cut through the geometry. As a result, you **cannot** specify this coordinate system and any of its axis for a **General Axisymmetric** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



In this example, the X- Axis of the coordinate system cuts through the geometry. As a result, it cannot be specified for the **Axis** property. The Y-Axis is not in the same plane as the surface body so it cannot be specified for the **Axis** property. For this example, you can use only the Z-Axis as the axisymmetric axis.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Setting Up Coordinate Systems

All geometry in the Mechanical application is displayed in the global coordinate system by default. The global coordinate system is the fixed Cartesian (X, Y, Z) coordinate system originally defined for a part.

In addition, you can create unique local coordinate systems to use with springs, joints, various loads, supports, and result probes.

Cartesian coordinates apply to all local coordinate systems. In addition, you can apply cylindrical coordinates to parts, displacements, and forces applied to surface bodies.

Note:

Cylindrical coordinate systems are not supported by the Explicit Dynamics solvers, but may be used for some postprocessing operations.

Annotations are available for coordinate systems. You can toggle the visibility of these annotations in the **Annotation Preferences** dialog box. For more information, see Specifying Annotation Preferences (p. 310).

The following topics are covered in this section:

Creating Coordinate Systems
Importing Coordinate Systems
Applying Coordinate Systems as Reference Locations
Using Coordinate Systems to Specify Joint Locations
Creating Coordinate-Based Section Planes
Transferring Coordinate Systems to the Mechanical APDL Application
Setting Up Coordinate Systems in the Mechanical Configure Tool

Creating Coordinate Systems

The following topics involve the creation of local coordinate systems:

Initial Creation and Definition
Establishing Origin for Associative and Non-Associative Coordinate Systems
Setting Principal Axis and Orientation
Using Transformations
Creating a Coordinate System Based on a Surface Normal
Creating a Coordinate System Based on the Center of Mass
Setting a Coordinate System Origin at the Center of Mass

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

See the Coordinate System Object Reference page of the Help for additional information about the categories and properties of the **Coordinate System** object.

Initial Creation and Definition

To create a new local coordinate system, you first add a Coordinate System object and then specify properties of the **Definition** category.

Create and define a new local coordinate system:

1. Select the **Coordinate Systems** folder in the **Outline** and then select the **Coordinate System** option from the **Insert** group in the **Coordinate Systems** Context tab. This group is also contained on the **Home** tab.

   Tip:

   Alternatively, when the **Coordinate Systems** folder or an existing coordinate system object is selected, right-click the object, or in the **Geometry** window, and select **Insert** > **Coordinate System**.

2. Under the **Definition** category, specify the following properties as needed:

   a. **Type**: Set to **Cartesian** (default) or **Cylindrical**.

   b. **Coordinate System**: Setting include **Program Controlled** (default) or **Manual**. This property assigns the coordinate system reference number (the first argument of the Mechanical APDL **LOCAL** command). The **Program Controlled** option assign the reference number automatically. The **Manual** option enables you to assign a particular reference number in the **Coordinate System ID** field for identification or quick reference of the coordinate system within the input file. You should set the **Coordinate System ID** to a value greater than or equal to 12. If you create more than one local coordinate system, you must ensure that you do not duplicate the **Coordinate System ID**.

   c. **Suppressed**: **Yes** or **No** (default). If you choose to suppress a coordinate system, you remove the object from further treatment, write no related data to the input deck, and cause any objects scoped to the coordinate system to become underdefined (therefore invalidating solutions).

See the next section for continued coordinate system definition.

Establishing Origin for Associative and Non-Associative Coordinate Systems

After creating a local coordinate system, you can further designate it as being associative or non-associative using the properties of the **Origin** category in the Details.

- An *associative* coordinate system is joined to a geometry or mesh scoping throughout preprocessing. Therefore, its position and orientation can be affected by geometry updates, part transformation, as well as through the use of the **Configure** tool (for joints). However, the coordinate system maintains its position throughout the solution process regardless of changes to the geometry as a result of loading.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- A *non-associative* coordinate system is independent of any geometry.

Specify Associative Coordinate System

To establish the origin for an associative coordinate system:

1. Set the **Define By** property to either **Geometry Selection** (default) or **Named Selection**.

   For a **Reference Coordinate System** attached to a joint, use the **Orientation About Principal Axis** category to make the coordinate system associative.

2. If you select:

   - **Geometry Selection**: Select the desired geometry or mesh entities.

     Select the **Click to Change** option in the **Geometry** property.

     Click Apply. A coordinate system symbol displays at the centroid of your selection. The application calculates the centroid of your coordinate system based on a weighted (by volume, area, or length) average for the specified scoping, including:

     – For Solid bodies and solid body elements, the application calculates the weighted average using the volume of the bodies/elements.

     – Surface bodies, faces, surface body elements, and element faces, the application calculates the weighted average using the area of the surfaces/surface elements.

     – Line bodies, edges, and line body elements, the application calculates the weighted average using the length of the line bodies/edges/line elements.

   - **Named Selection**: Select a user-defined Named Selection (p. 1057) from the **Named Selection** property.

   Tip:

   You can automatically specify the **Geometry** of a **Coordinate System** object by first selecting geometry or mesh entities and then inserting a new **Coordinate System** object.

Note:

To create a coordinate system on a **Line Body**, you need to select one or more edges, vertices, nodes, or elements so that the origin is placed at the centroid of the selections.

Specify Non-Associative Coordinate System

To establish the origin for a non-associative coordinate system:

- Set the **Define By** property to **Global Coordinates**. You then define the origin in either of the following ways:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Selecting a point on the exterior of the model using the **Hit Point Coordinate** option:

  1. Set the **Define By** property to **Global Coordinates**.
  2. Select the **Click to Change** field of the **Location** property.
  3. Select the **Hit Point Coordinate** ( ) button on the Graphics Toolbar (p. 97). This option displays coordinates as you move the cursor across the model.
  4. Select the desired origin location on the model. A small cross hair appears at the selected location.
  5. Click **Apply** in the **Location** property field. A coordinate system symbol displays at the origin location. Note that the coordinate values automatically populate the **Origin X**, **Origin Y**, and **Origin Z** properties. You can manually change the location of the coordinate system by changing these values.

- Selecting one or more vertices, nodes, element faces, or elements to define an averaged location of selected geometry or mesh entities:

  1. Set the **Define By** property to **Global Coordinates**.
  2. Select the **Click to Change** field of the **Location** property.
  3. Select the filter (**Vertex**, **Node**, **Element Face**, or **Element**) on the Graphics Toolbar (p. 97).
  4. Select as many vertices, nodes, element faces, or elements as desired and then click the **Apply** option of the **Location** property. The origin coordinate system is specified on the model based on the average location of the selected entities.

- Entering the coordinates directly:

  1. Set **Define By** to **Global Coordinates**.
  2. Enter values in the **Origin X**, **Origin Y**, and **Origin Z** properties.

Setting Principal Axis and Orientation

The definition of the coordinate system involves two vectors, the **Principal Axis** vector and the **Orientation About Principal Axis** vector. The coordinate system respects the plane formed by these two vectors and aligns with the **Principal Axis**.

Principal Axis

Use the **Principal Axis** category to define the **X**, **Y**, or **Z** axes (using **Axis** property) in terms of a:

- **Geometry Selection**: You use this option to specify a geometry or mesh selection to define the direction of the Principal Axis. When selected, an associated **Geometry** property displays. You use the **Geometry** property to specify either one or more faces/element faces, two edges, two vertices, or two or three nodes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Fixed Vector**: Depending upon the **Geometry Selection** in the **Origin** category, this option preserves the current **Geometry Selection** without associativity. When a change occurs to the feature the axis will **not** update automatically to reflect that change.
- **Global X**/**Global Y**/**Global Z** axis: These options force the axis to align to a global X, Y, or Z axis.
- **Hit Point Normal**: This property aligns the axis along a normal vector that represents the normal direction of the local surface curvature of the hit point. You then select a point on the screen to define the **Hit Point Normal** and orient the primary axis. For information on creating a coordinate system aligned with the hit point, see Creating a Coordinate System Based on a Surface Normal (p. 1196).

Orientation About Principal Axis

Use the **Orientation About Principal Axis** Details category to specify the:

- **Axis** property. Options include **X** (default), **Y**, or **Z**.
- **Defined By** property. Options include **Default**, **Geometry Selection**, **Global X Axis**, **Global Y Axis**, **Global Z Axis**, and **Fixed Vector**.

  As above, when using the **Geometry Selection** property, an associated **Geometry** property displays that you use to specify either one or more faces/element faces, two edges, two vertices, or two or three nodes.

Important:

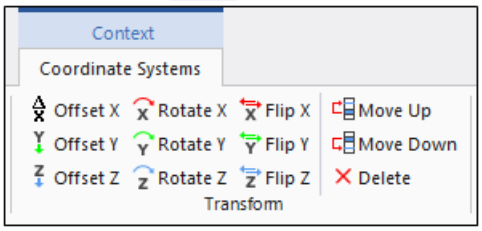
The **Default** setting leaves the orientation of the coordinate system unspecified. Review the values of the properties in the **Directional Vectors** category for the axis data being defined.

Using Transformations

Transformations allow you to "fine tune" the original positioning of the coordinate system. Options are available for offsetting the origin by a translation in each of the x, y and z directions, as well as by rotation about each of the three axes. Flipping of each axis is also available. To exercise transformations, you use buttons on the Coordinate System Context Tab (p. 58) and settings in the **Transformations** category in the Details view.

To transform a coordinate system:

1. Choose a transformation (translation, rotation, or flip) from the **Transform** group.



   Entries appear in the Details as you add transformations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. Enter information in the Details for each transformation.
3. If required:
   - Reorder a transformation by highlighting it in the Details view and using the **Move Up** or **Move Down** options.
   - You can delete a transformation by highlighting it in the Details view and then selecting the **Delete** option of the Transform group on the **Coordinate System** Context Tab.

Creating a Coordinate System Based on a Surface Normal

You can orient a coordinate system based on the surface normal. You have two options. You can orient the principal axis based on the hit point normal of an existing coordinate system, or you can create an aligned coordinate system based on the hit point.

Orienting the Principal Axis by Hit Point Normal

To orient the principal axis based on the hit point normal of an existing coordinate system:

1. Create a coordinate system (p. 1191).
2. Under the **Principal Axis** category, set the **Define By** property to **Hit Point Normal**.
3. Select the **Click to Change** field of the **Hit Point Normal** property.
4. Select the **Hit Point Coordinate** ( ) option on the Graphics Toolbar (p. 97). This option displays coordinates as you move the cursor across the model.
5. In the **Geometry** window, select a desired point.
6. Click the **Apply** option of the **Hit Point Normal** property.

For more information, see Setting Principal Axis and Orientation (p. 1194).

Creating a Coordinate System Aligned with a Hit Point

To create an aligned coordinate system based on the hit point:

1. Enable Hit Point Coordinate mode by toggling the **Hit Point Coordinate** button in the Graphics Toolbar (p. 97).
2. In the **Geometry** window, select a point.
3. Right-click in the **Geometry** window and select **Create Coordinate System Aligned with Hit Point**.

   Mechanical creates a coordinate system on the location of hit point with the primary axis aligning along the hit point normal.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If a hit point is not defined, Mechanical creates a coordinate system on the location of {0,0,0}, with the axis the same as the global coordinate system.

Creating a Coordinate System Based on the Center of Mass

You can create a coordinate system based on the Center of Mass. You have two options. You can set the origin at the Center of Mass, or align the principal axis toward or normal to the Center of Mass.

Create and Define a New Local Coordinate System with the Origin at the Center of Mass

1. Select the **Coordinate Systems** folder in the **Outline**.
2. Select the **Coordinate System at Center of Mass** option from the **Coordinate Systems** group in the **Coordinate Systems** Context tab.

Tip:

Alternatively, when the **Coordinate Systems** folder or an existing coordinate system object is selected, right-click the object, and select **Insert** > **Coordinate System at Center of Mass**.

Selecting this option inserts a Non-Associative Coordinate System with the origin at the Center of Mass of all unsuppressed bodies and Point Mass objects. Distributed Mass objects are not currently considered in the Center of Mass calculation.

Setting the Principal Axis Using the Center of Mass

1. Select the **Coordinate Systems** folder in the **Outline**.
2. Select the **Principal Axis Using Center of Mass** option from the **Coordinate Systems** group in the **Coordinate Systems** Context tab.

This option is only active when the current selection consists 1 or 2 points (vertices or nodes).

When a single point is selected, the principal axis is set to Z and defined by a fixed vector that points from the selected point towards the center of mass of all unsuppressed bodies and point masses.

When two points are selected, the principal axis is set to Z and defined by a fixed vector that points normal to the plane formed by the selected points and the center of mass of all unsuppressed bodies and point masses.

Setting a Coordinate System Origin at the Center of Mass

You can change the location of an existing Coordinate System to the center of mass of your model. This includes all unsuppressed bodies of the model as well as any specified Point Mass and/or Distributed Mass objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Set Origin at Center of Mass

To set a coordinate system's location to the center of the mass:

1. Select an existing coordinate system object.
2. Select the **Set Origin at Center of Mass** option. The coordinate system is automatically set at the center of mass of the model.

> **Note:**
>
> This option does not support reorientation of the **Global Coordinate System**.

Distributed Mass Requirements

In order for a **Distributed Mass** object to be supported, it must be scoped to either edges or faces only and you cannot mix these two scoping options. Furthermore, the calculation of the mass for this object depends upon the setting of the **Mass Type** property. If set to:

- **Mass per Unit Area**, the application calculates the center of mass.
- **Total Mass**, the application calculates the equivalent uniform mass density based on the scoping of the object.

Importing Coordinate Systems

Coordinate systems defined when geometry is imported from DesignModeler, Creo Parametric, or SOLIDWORKS will automatically be created in the Mechanical application. For more information, see the **Attaching Geometry** section under DesignModeler, or see the **Notes** section under Creo Parametric or SOLIDWORKS in the CAD Integration section of the Ansys Workbench help.

If you update the model in the Mechanical application, coordinate systems from these products are refreshed, or newly defined coordinate systems in these products are added to the model.

If a coordinate system was brought in from one of these products but changed in the Mechanical application, the change will not be reflected on an update. Upon an update, a coordinate system that originated from DesignModeler, Creo Parametric, or SOLIDWORKS will be re-inserted into the object tree. The coordinate system that was modified in the Mechanical application will also be in the tree.

Applying Coordinate Systems as Reference Locations

Any local coordinate systems that were created in the Mechanical application, or imported from DesignModeler, Creo Parametric, or SOLIDWORKS, can be applied to a part, or to a Point Mass (p. 901), Spring (p. 1395), Acceleration (p. 1501), Standard Earth Gravity (p. 1509), Rotational Acceleration (p. 1516), Rotational Velocity (p. 1511), Force (p. 1543), Bearing Load (p. 1561), Remote Force (p. 1552), Moment (p. 1574), Displacement (p. 1666), Remote Displacement (p. 1674), or Contact Reaction (p. 2079). This feature is useful because it avoids having to perform a calculation for transforming to the global coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To apply a local coordinate system:

1. Select the tree object that represents one of the applicable items mentioned above.
2. For an Acceleration, Rotational Velocity, Force, Bearing Load, or Moment, in the Details view, set the **Define By** property to **Components**, and then proceed to Step 3. For the other items, proceed directly to step 3.
3. In the Details view, set **Coordinate System** to the name of the local coordinate system that you want to apply. The names in this drop-down list are the same names as those listed in the **Coordinate Systems** branch of the tree outline.

Note:

If you define a load by **Components** in a local coordinate system, changing the **Define By** field to **Vector** will define the load in the global coordinate system. Do not change the **Define By** property to **Vector** if you want the load defined in a local coordinate system.

Using Coordinate Systems to Specify Joint Locations

Whenever you create a joint, an accompanying reference coordinate system is also created. The intent of this coordinate system is for positioning the joint. See the Joint Properties (p. 1316) section for further details.

Creating Coordinate-Based Section Planes

For viewing purposes, you can use the **Section Plane** option from the **Insert** group (p. 42) of the **Home** tab to slice the graphical image of your model based on a predefined coordinate system.

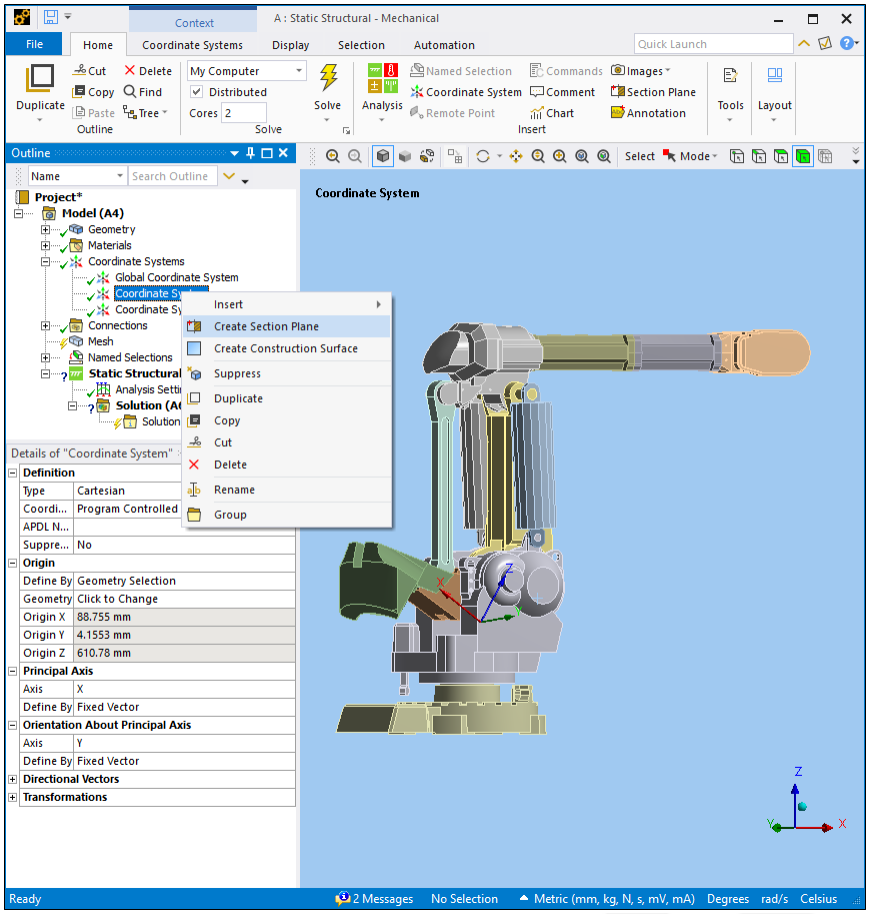
Note:

The Section Plane feature does not support Cylindrical Coordinate Systems.

---

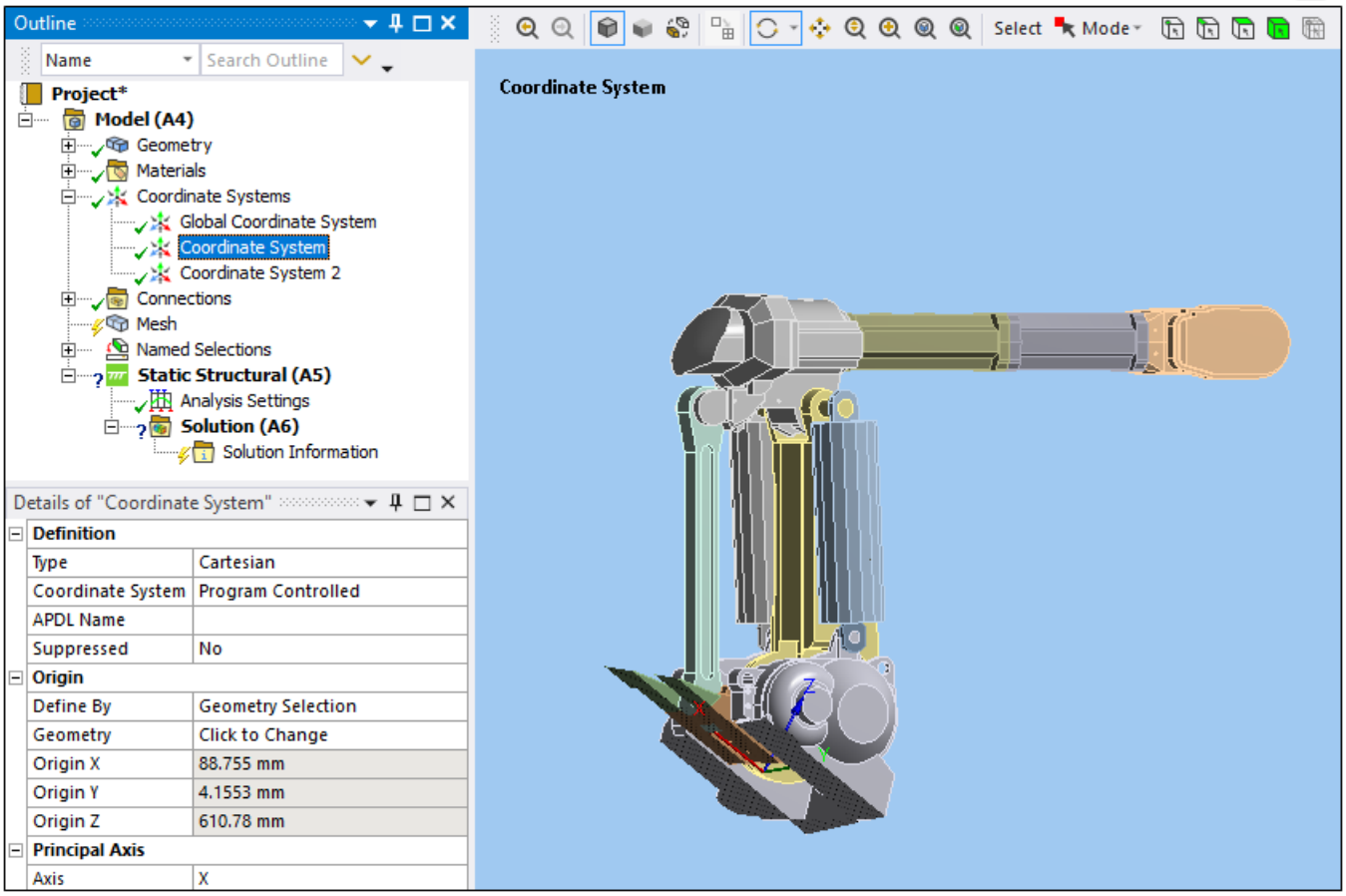
1. Select the desired **Coordinate Systems** object.
2. Right-click the mouse and select **Create Section Plane**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



As illustrated here, the model is sliced based on the Coordinate System.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

This option is also available for Coordinate System objects in the Meshing Application.

Transferring Coordinate Systems to the Mechanical APDL Application

You can transfer coordinate systems to the Mechanical APDL application using any of the following methods:

- Write Input File (p. 1861)
- Load the Mechanical APDL application.
- Commands Objects (p. 2303)

Any coordinate system defined in the Mechanical application and sent to the Mechanical APDL application as part of the finite element model, will be added to the Mechanical APDL application input file as LOCAL commands. For example:

```
/com,********** Send User Defined Coordinate System(s) **********
local,11,0,0.,0.,0.,0.,0.,0.
local,12,1,11.8491750582796,3.03826387968126,-1.5,0.,0.,0.
csys,0
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Setting Up Coordinate Systems in the Mechanical Configure Tool

Prior to a configure operation, when stops are defined in the model, it is important to set the **Initial Position** field to **Override**. This is required to ensure that the initial joint displacements and stops are accurately considered. The reference and mobile coordinate systems must be correctly attached to the corresponding geometry of the joint bodies.

Definition	
Connection Type	Body-Body
Type	Translational
Friction Coefficient	0.
Radius	0. m
Effective Length	0.5 m
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Solid1
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Solid2
Coordinate System	Mobile Coordinate System
Initial Position	Override
Behavior	Rigid
Pinball Region	All
Stops	
X Min Type	None
X Max Type	Stop
X Max	0.1 m

The zero value of the joint's degree of freedom is described in Joint DOF Zero Value Conventions (p. 1296). This joint configuration is established based on the relative position and orientation of the mobile and reference coordinate systems of the joint.

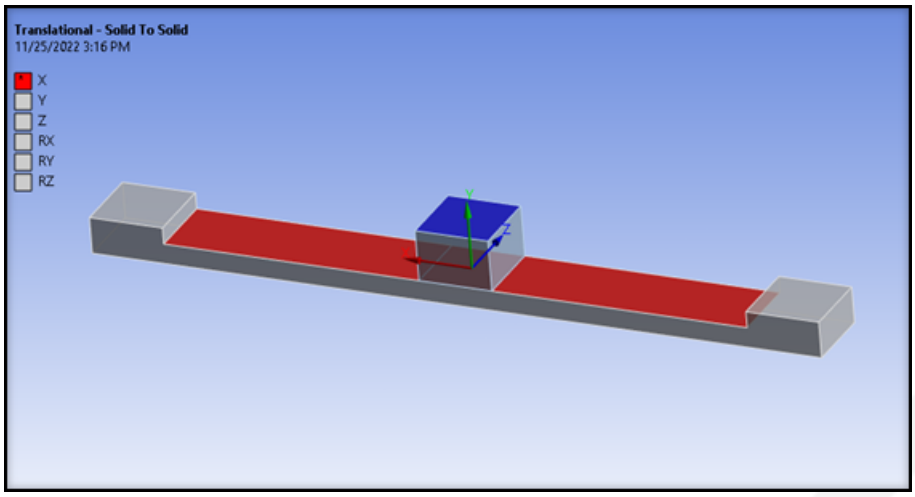
When you configure a joint or body and set the configuration as the new base geometry for the analysis, the position of each joint's mobile and reference coordinate system is updated in correspondence with the definition of these coordinate systems. For example, if the position of the reference coordinate system is defined by a planar surface that moves, the new coordinate system is positioned according to the new position of the surface after the configure operation.

When using the **Unchanged** definition for the Initial Position of the mobile coordinate system, it is re-positioned at the same location as the reference coordinate system. As a result, the new zero value for the joint's degree of freedom is based on the updated configuration, which might be problematic.

To illustrate, this is an example of a translational joint with a minimum and maximum stop:

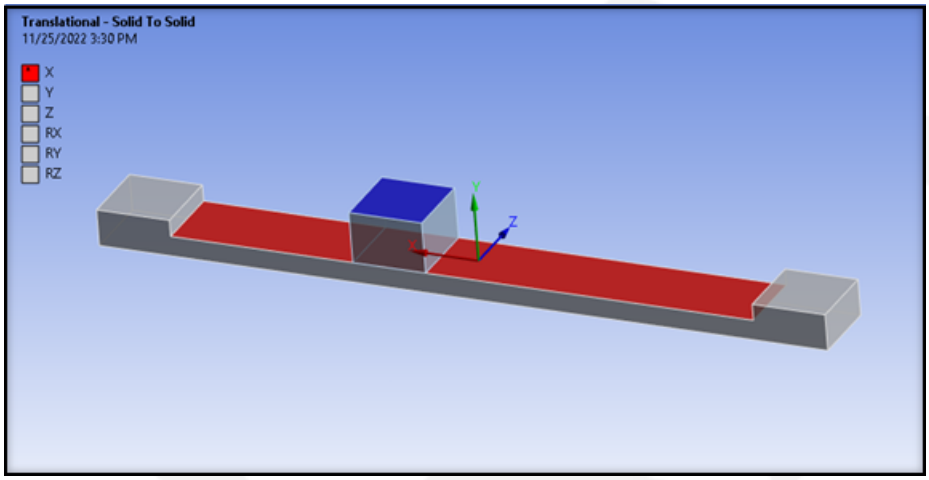
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The stop is initially defined by a lower bound of -35mm and an upper bound of +35mm. The joint coordinate system is defined simply by picking the red reference surface.

Next, configure the translational joint and move the block by 10mm to the right, and set this configuration.



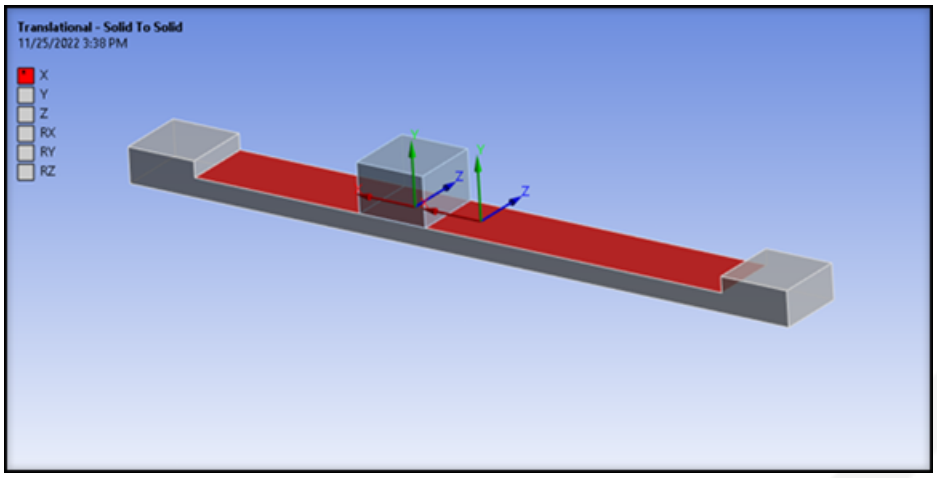
The new reference coordinate system is repositioned by finding the center of the red surface, which has not changed. Therefore, the new reference and mobile coordinate systems still remain. This new configuration is the zero of the joint translation. The stop's bounds (-35, +35) have not changed and are now incorrectly defined.

To obtain a proper definition of the stops, it is important that both reference and mobile coordinate systems are correctly updated. The reference coordinate system must be defined by the red surface, while the mobile coordinate system must be attached to the block.

This can be achieved by setting the Initial Position property of the joint to **Override** and using a face scoping on the block for the reference and mobile coordinate systems.

After this update, each coordinate system is revised based on its definition, and the new configuration is shown below:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



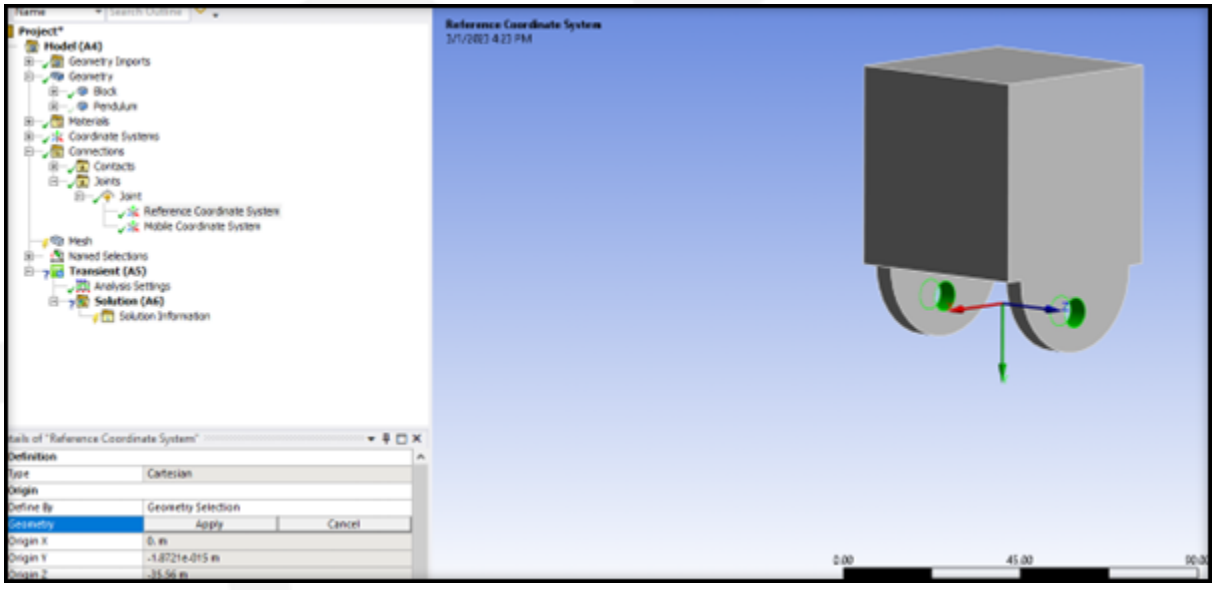
In this configuration, the current value for the joint translation is 10mm, and the definition of the two stops is still valid.

In sum, when the **Override** option is selected, the stop bounds can not be violated with respect to the initial configuration after successive Configure and Set operations.

Consideration for Joints with Rotational Degrees of Freedom

Special attention is required when dealing with revolute joints. More precisely, the orientations of reference and mobile coordinate systems must be defined in accordance with the geometry.

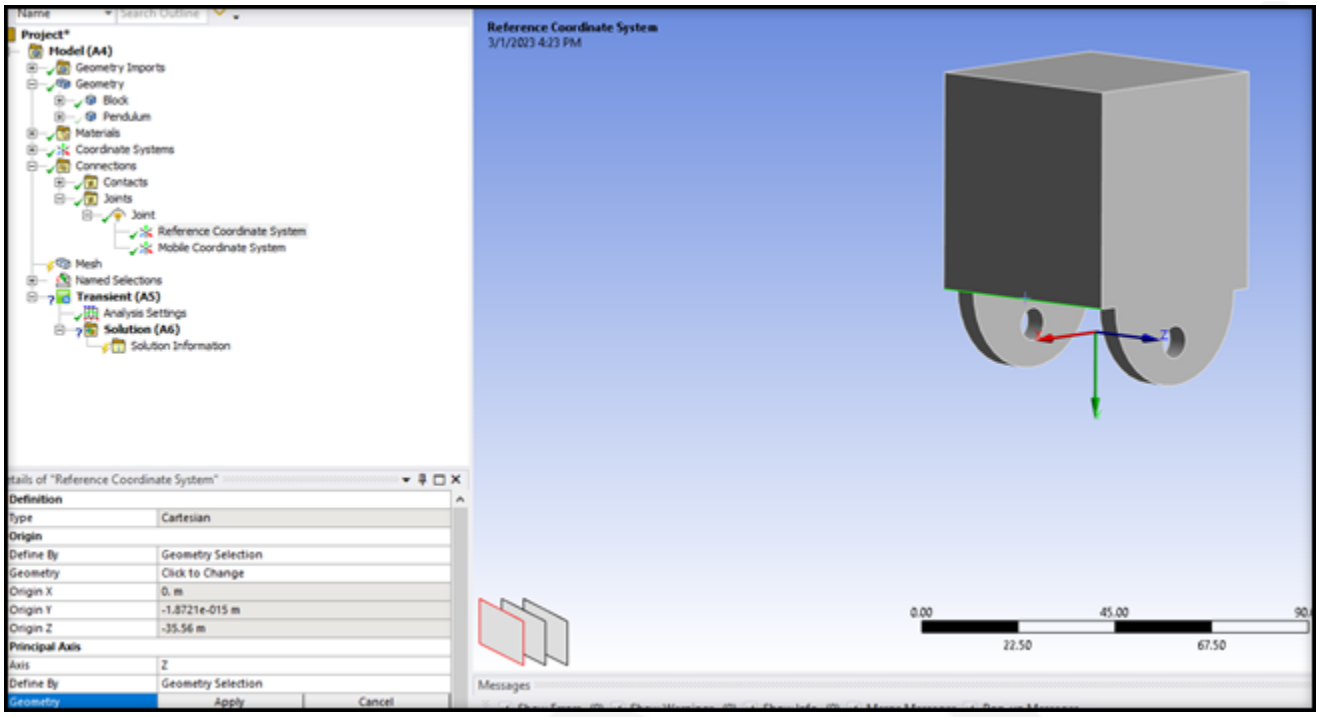
1. The origin of the reference coordinate system must be scoped on an appropriate feature of the reference geometry.



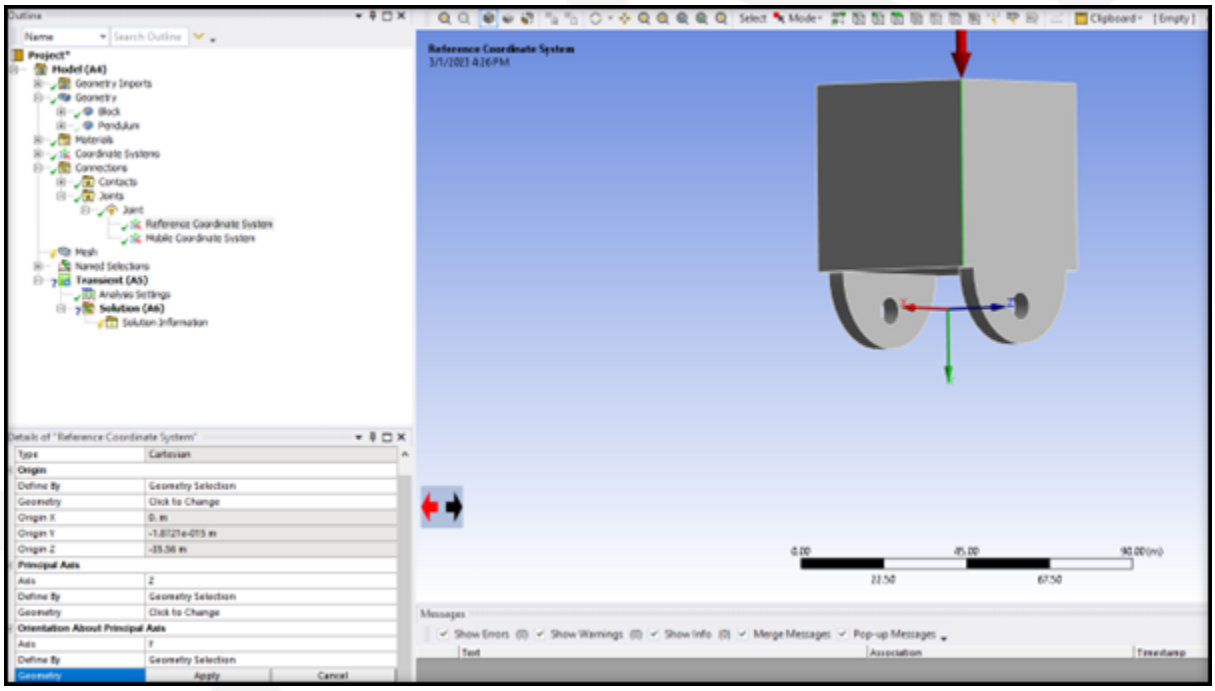
2. The principal axis of the coordinate system must then be defined by a relevant geometric feature of the reference geometry. Here, it is an edge.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

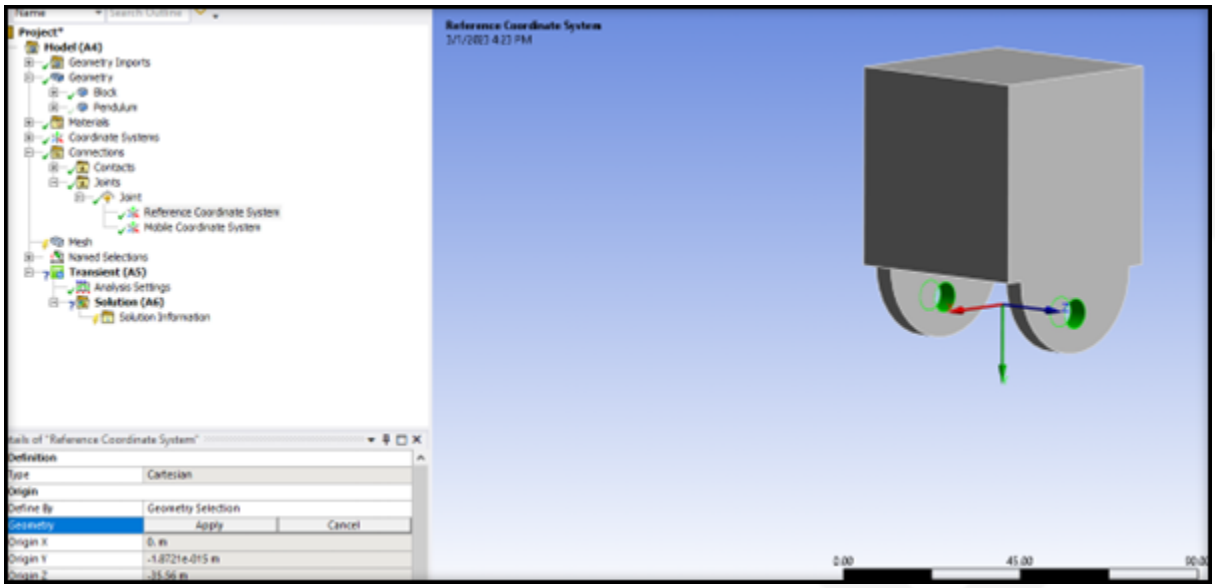


3. The orientation of the coordinate system also needs to be set with respect to the reference body geometry.



4. A similar setting is required for the mobile coordinate system. Once both coordinate systems have the same orientation and are well defined with respect to the geometries, the bounds of stops or locks are properly updated after a configure operation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Setting Connections

Supported connection features consist of Contact, Joint, Spring, Beam Connection, End Release, Spot Weld and Body Interaction (Explicit Dynamics only). Each of these connections can be created manually in the application. Only Contact and Joint can also be generated automatically.

This section describes **Connections** folder, **Connection Group** folder, Automatic Generated Connections, as well as each connection type as outlined below.

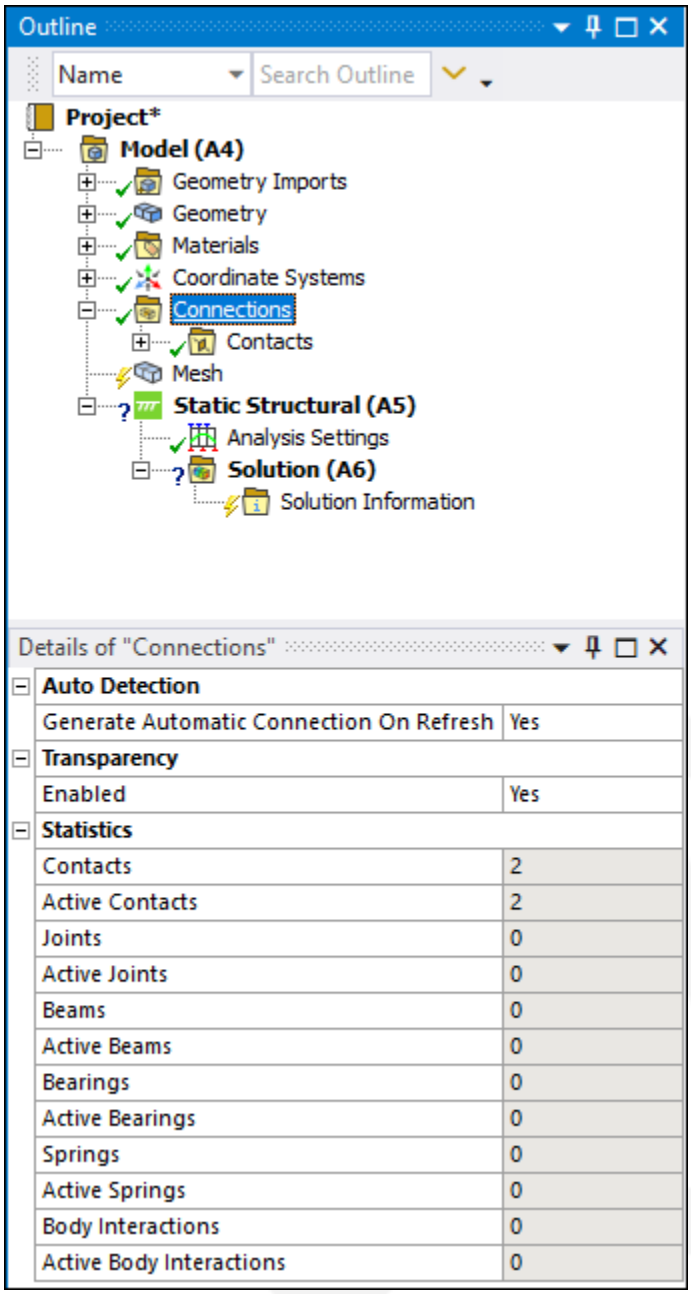
Connections Folder
Connections Worksheet
Connection Group
Connection Features and Operations
Contact
Joints
Springs
Beam Connections
Spot Welds
End Releases
Bearings

Connections Folder

The **Connections** folder is the container for all types of connection objects except for the two types that can be automatically generated (contact and joint). The objects of each of these two types are placed in a sub-folder called the **Connection Group** folder. The Details pane of the **Connections** folder is illustrated below.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The **Connections** folder includes the following categories and properties:

Category	Property/Description
Auto Detection	**Generate Automatic Connection On Refresh**: Options include **Yes** (default) and **No**. This setting turns automatic contact generation on/off when the geometry is refreshed. The process of automatically creating the contact objects is additive. Any existing connection objects of these types that were created manually may be duplicated when the connections

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property/Description
	are automatically regenerated. To avoid duplication, you should first delete any existing contact objects before the geometry is refreshed. **Note:** • Special conditions apply to updating geometry that includes Spot Welds (p. 1407). • The process of automatically creating joint objects is **not** additive. Any existing joint objects are not duplicated when connections are automatically regenerated.
Transparency	**Enabled**: Options include **Yes** (default) and **No**. This setting enables or disables the transparency of the bodies **not** associated with the connection in the graphics display.
Statistics	This category contains read-only properties that display the number of contact conditions and connection types (defined and active) present in project, including: • **Contacts** • **Active Contacts** • **Joints** • **Active Joints** • **Beams** • **Active Beams** • **Bearings** • **Active Bearings** • **Springs** • **Active Springs** • **Body Interactions** • **Active Body Interactions**

Note:

For large deflection analyses that include a Contact Tool (p. 2079) under the **Connections** folder, a beta feature is available that makes sure that the application incorporates large deflection at both the connection and analysis levels of your simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Connections Worksheet

When you are working with the **Connections** object in the tree **Outline**, you can use the Worksheet (p. 192) window to supplement the **Details** view by providing a summary of the contact information, joint information, and the connections between geometry bodies. Select the **Worksheet** option on the Connections Context tab (p. 50) to display the **Worksheet** window. A **Show Preferences** option and a **Generate** option display and enable you to define worksheet data to display and the generate worksheet content.

To toggle on the worksheet:

1. Select the **Worksheet** option from the Context tab.
2. Select the **Show Preferences** option to view the possible data types.
3. Select the check boxes for the data types you want to view.
4. Select the **Generate** option to generate the content. Ansys Mechanical remembers the display preferences you select and will default to those in future sessions.

   Select **Hide Preferences** to hide the preferences and **Refresh** to refresh the worksheet data.

Navigation

The **Worksheet** remains displayed as you select different objects within the **Connections** folder. Selecting an object outside of the folder returns you to the **Graphics** display. Returning to a **Connections** object displays the last active data. Any changes require you to regenerate the **Worksheet** data.

Worksheet Connections Data Types

The data types available in the worksheet are described below. You can turn the displayed properties on and off using the right-click menu.

Contact Information

Displays the properties for each contact.

Joint DOF Checker

Checks the total number of free degrees of freedom and displays the free DOF, based on the number of unsuppressed parts, fixed constraints, and translation joints. If this number is less than 1, the model may be overconstrained, and you should check the model closely and remove any redundant joint constraints. You can use a Redundancy Analysis to detect redundant joint constraints.

Joint Information

Displays the name, type, scope, and status of all joints.

Spring Information

Displays spring connection properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Beam Information

Displays beam connection properties.

Connection Matrix

Displays a matrix that represents the connections between bodies in the geometry. These connections are color-coded by type (as shown in the legend). In the **Preferences**, you can choose the type of data to display, in order to filter out unwanted information. Activate the options by checking the selection box beside the **Connection Matrix** title. The following options can then be selected or deselected as desired.

- **Show Upper Diagonal**
- **Show Diagonal Marker**
- **Show Unconnected Bodies**
- **Show Suppressed Objects**
- **Bundle Connections**

The **Bundle Connections** option is an especially useful tool because it enables you to group **Control Connection Types**. For example, if you have three Spot Welds contained in the same cell of the matrix, activating the **Bundle Connections** option displays the spot welds as "3 Spot Welds" instead of displaying the individual names of all three within the cell.

Note:

- The matrix displays a grounded connection as a connection to itself. For example, if a grounded joint is scoped to body1, then it will be displayed in the cell of column body1 and row body1.
- The **Connection Matrix** is limited to 200 prototypes.

Control Connection Types

The **Control Connection Types** display area provides a list of selectable connection features/types that you can choose to display or to not display within the Connection Matrix. Options include:

- **Contact**
- **Spot Weld**
- **Joint**
- **Spring**
- **Beam**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Context Menu Selection Options

Selecting an item in the table and then right-clicking the mouse provides a menu of the following options, in addition to those associated with the properties of the data types:

- **Go To Selected Items in Tree**: the application displays the associated contact object or objects in the Geometry Window.
- **Reset Columns**: if you change the order of the table content by clicking on a column title, this option resets the order of the table content.
- **Reset Column Widths**: reset the columns to the default width.
- **Edit Column Width**: changes column width (in pixels). You can select multiple columns or rows. A value of zero (default) indicates that the setting is program controlled.
- **Export** (**Connection Matrix** Only - see below)

Exporting the Connection Matrix

You can export a text file version of the **Connection Matrix** from either the **Worksheet** or the **Connections** object in the **Tree Outline**.

To export from the worksheet, right-click the **Connection Matrix** table and select **Export**.

To export from the **Tree Outline**, right-click the **Connections** object and select **Export**.

Connection Group

The role of a **Connection Group** folder is to provide you with the ability to automatically generate Contact or Joint objects for the whole model or for a group of bodies within the model with a tolerance value applied only to this group. Only these two types of connections are provided with the automatic detection capability and only one type of connection objects can be included in a **Connection Group** folder with the exception of Spot Weld (see details in the Spot Weld (p. 1407) section). The generated objects are placed in a **Connection Group** folder which is automatically renamed to "Contacts" or "Joints" depending on the type.

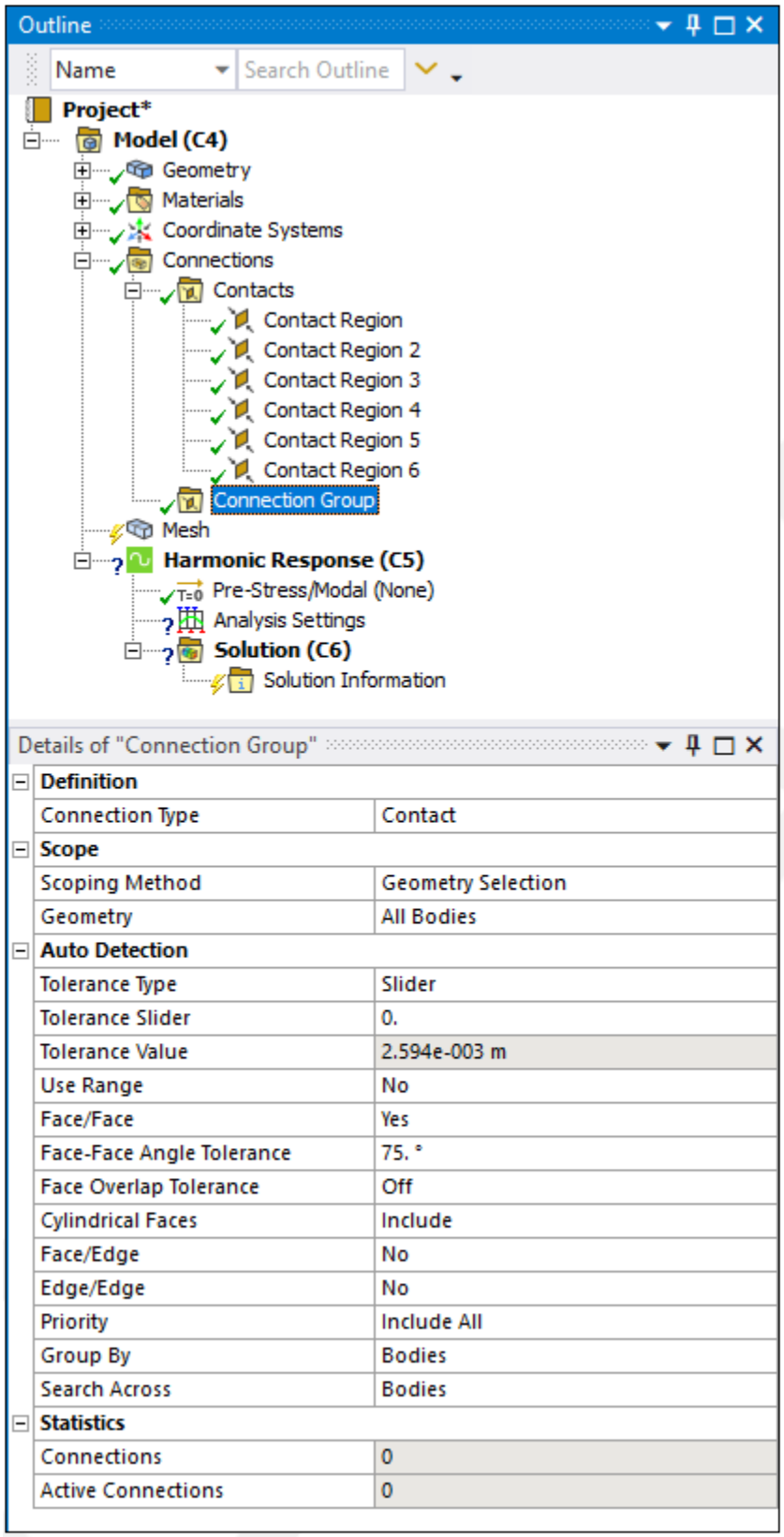
When you import your model into Mechanical, by default, contact detection is performed automatically. This is based on the setting in the Workbench **Options** dialog. On the Workbench Project Page, under **Tools** > **Options** > Mechanical, there is a setting labeled **Auto Detect Contact On Attach**. This setting is selected by default. Furthermore, this automatic contact detection is based on the settings you define in the Options (p. 217) dialog (**Connections** category > **Auto Detection**) of the Mechanical application. To open the Options dialog in Mechanical, select the icon available beside the Help drop-down menu on the title bar. The **File** tab contains this option as well.

Detailed steps for auto/manual generating connection objects are presented in the Connection Features and Operations (p. 1218) section.

The **Connection Group** has the following properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Definition

- **Connection Type**: options include **Contact** and **Joint**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

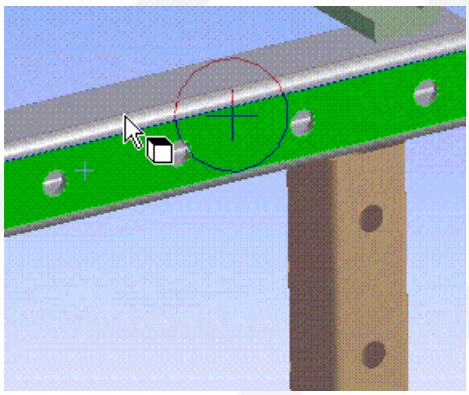
Scope

- **Scoping Method**: options include **Geometry Selection** (default) and **Named Selection**.
  - **Geometry**: used to define geometry selections. Appears when **Scoping Method** is set to **Geometry Selection**.
  - **Named Selection**: drop-down list of available Named Selections. Appears when **Scoping Method** is set to **Named Selection**.

Auto Detection

- **Tolerance Type**: options include **Slider**, **Value**, and **Use Sheet Thickness**. Bodies in an assembly that were created in a CAD system may not have been placed precisely, resulting in small overlaps or gaps along the connections between bodies. You can account for any imprecision by specifying connection detection tolerance. This tolerance can be specified by value when the type is set to **Slider** and **Value**, or sheet thickness of surface bodies when the type is set to **Use Sheet Thickness**. This option is only applicable to Contact and available when the **Group By** property (see below) is set to **None** or **Bodies**.
- **Tolerance Slider**: appears if **Tolerance Type** is set to **Slider**. To tighten the connection detection, move the slider bar closer to +100 and to loosen the connection detection, move the slider bar closer to -100. A tighter tolerance means that the bodies have to be within a smaller region (of either gap or overlap) to be considered in connection; a looser tolerance will have the opposite effect. Be aware that as you adjust the tolerance, the number of connection pairs could increase or decrease.
- **Tolerance Value**: appears if **Tolerance Type** is set to **Slider** or **Value**. This field will be read-only if the **Tolerance Type** is set to **Slider** showing the actual tolerance value based on the slider setting. When the **Tolerance Type** is set to **Value**, you will be able to provide an exact distance for the detection tolerance.

  After you provide a greater than zero value for the **Tolerance Value**, a circle appears around the current cursor location as shown below.

  The radius of the circle is a graphical indication of the current **Tolerance Value**. The circle moves with the cursor, and its radius will change when you change the **Tolerance Value** or the **Tolerance Slider**. The circle appropriately adjusts when the model is zoomed in or out.
- **Use Range**: appears when the **Tolerance Type** property is set to **Slider** or **Value**. Options include **Yes** and **No** (default). If set to **Yes**, you will have the connection detection searches within a range from **Tolerance Value** to **Min Distance Value** inclusive.

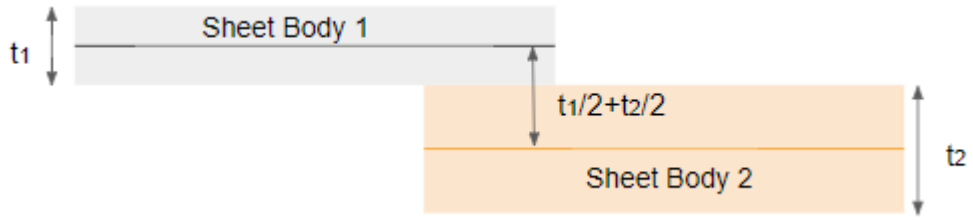
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - **Min Distance Percentage**: appears if **Use Range** is set to **Yes**. This is the percentage of the Tolerance Value to determine the Min Distance Value. The default is 10 percent. You can move the slider to adjust the percentage between 1 and 100.

  - **Min Distance Value**: appears if **Use Range** is set to **Yes**. This is a read-only field that displays the value derived from: **Min Distance Value** = **Min Distance Percentage** * **Tolerance Value**/100.

- **Thickness Scale Factor**: appears if **Tolerance Type** is set to **Use Sheet Thickness**. The default value is **1**.



  Use Sheet Thickness will use sheet bodies' thickness values (t_1 and t_2 above) to find contact between bodies. For example, if the sheets are within $t_1/2 + t_2/2$ * **Thickness Scale Factor**, then Sheet Body 1 and Sheet Body 2 would be found to be in contact.

  Note:

  If sheet bodies and/or shell thicknesses are not precise, the default value could result in some missing contact areas. Increasing the **Thickness Scale Factor** to a larger value, say 1.1, may help in such cases.

  For **Edge/Edge** pairing (see below), the largest thickness among the surface bodies involved is used; however, if the pairing is **Face/Edge**, the thickness of the surface body with the face geometry is used.

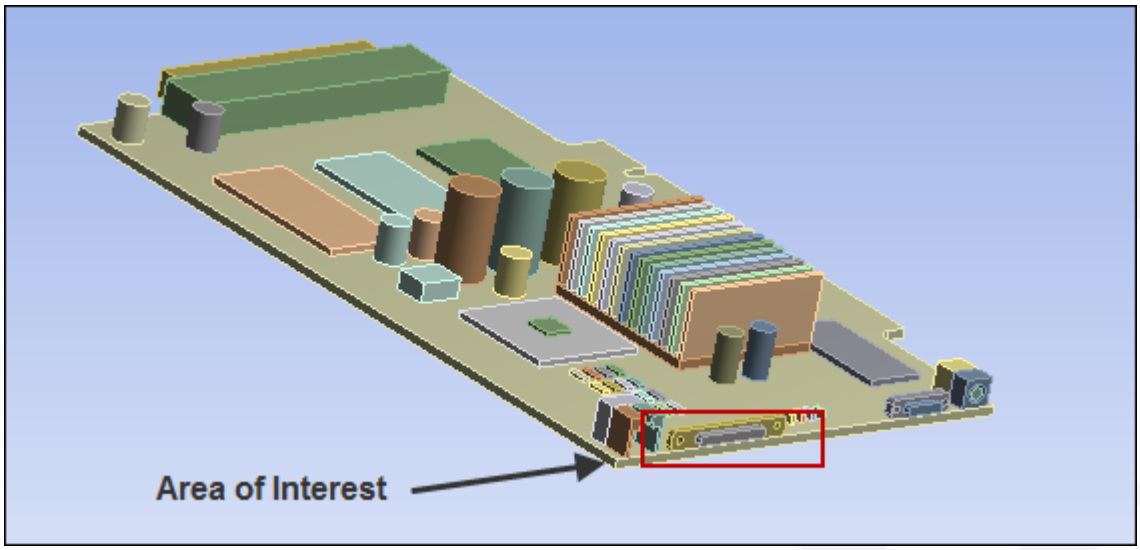
- **Face/Face**: (**Contacts** only) options include **Yes** (default) and **No**. Detects connection between the faces of different bodies. The maximum allowable difference in the normals for which contact is detected is 15 degrees. For Joints, **Face/Face** is the only detection type allowed. That is why the property does not appear in the Details view when the **Connection Type** is **Joint**.

- **Face/Face Angle Tolerance**: This property is visible when the **Face/Face** property is set to **Yes**. When working with **Face/Face** automatic contact detection, this property enables you to define the minimum angle between two face normals. This minimum angle is the threshold below which the application will ignore the faces from proximity detection. The default value is 75°, the minimum value is 0°, and the maximum value is 90° (perpendicular).

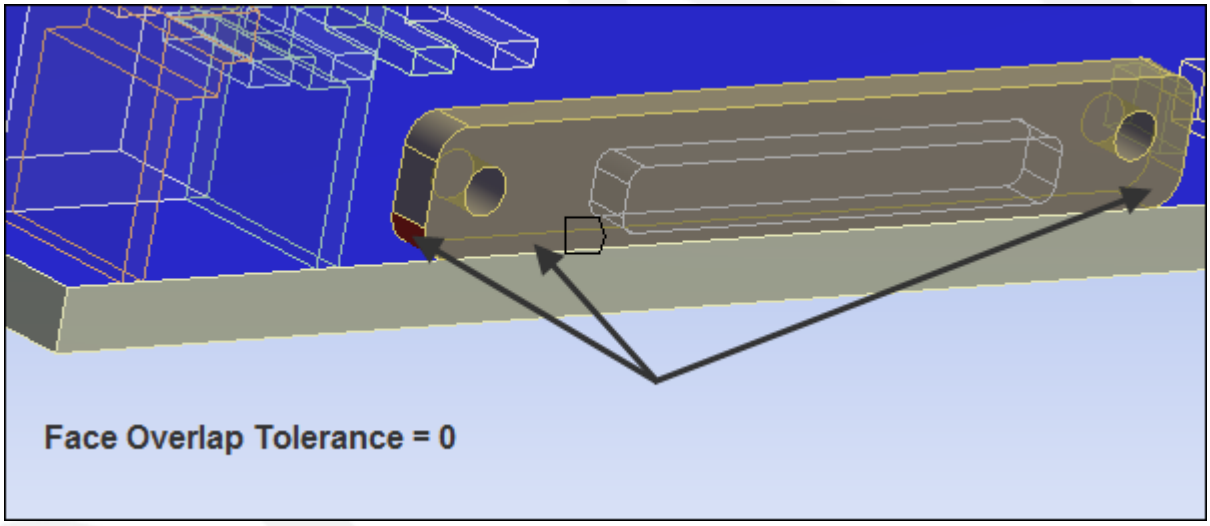
- **Face Overlap Tolerance** (**Contacts** only): This property is visible only when the **Face/Face** property is set to **Yes**. It sets the tolerance for overlap of faces in contact; that is, the minimum percentage of overlap at which a contact pair is created for two overlapping faces. For example, if **Face Overlap Tolerance** is set to 25, a contact pair is created for each pair of faces for which at least 25% of one face overlaps the other. You can set a value from 0 to 100 or retain the default. Setting the tolerance to 0 turns off the overlap checks.

  The following images illustrate the use of **Face Overlap Tolerance**. For the image of the circuit board below, SpaceClaim's **Imprint** tool was used to get common boundaries between parts, and then the model was loaded into Mechanical. Notice the area of interest.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



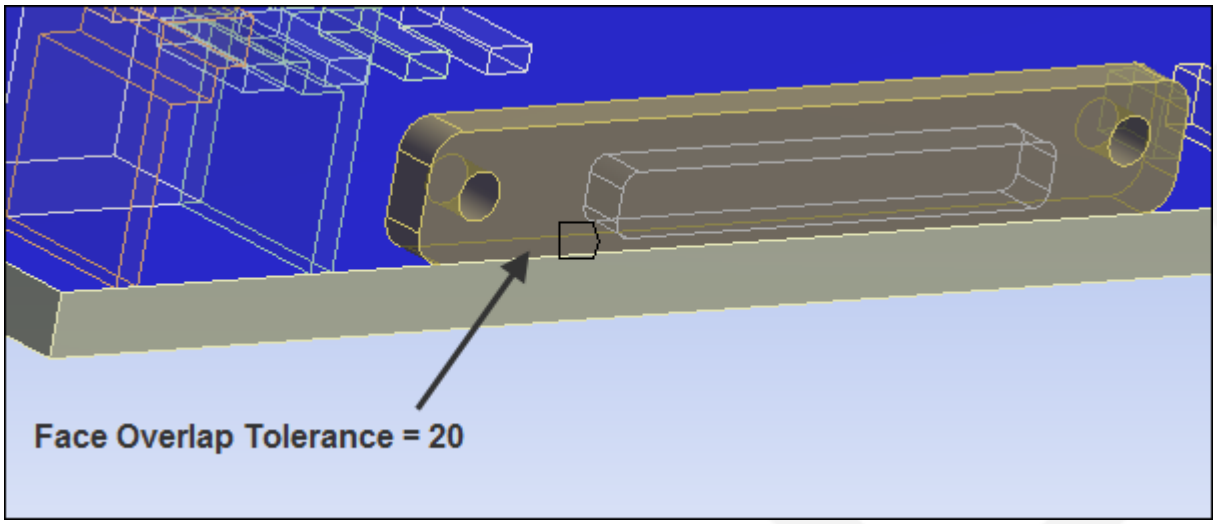
In general, the smaller the face overlap tolerance, the greater the chance that contact will result in extra pairs. The image below shows an enlarged view of the area of interest when a single **Contact Region** was selected in the tree. With **Face Overlap Tolerance** set to 0, the 3 faces identified by the arrows were scoped automatically to the **Contact** property of the **Contact Region**, and 1 face (the large blue face) was scoped automatically to the **Target** property of the **Contact Region**.



To get more precise contact pairs, you can increase the value of **Face Overlap Tolerance**. In the image below, the same **Contact Region** was selected in the tree but **Face Overlap Tolerance** was set to 20. In this case, the 2 small fillets were not found to be in contact with the large blue face, so only the 1 face identified by the arrow was scoped to the **Contact** property of the **Contact Region**, and 1 face (the large blue face) was scoped to the **Target** property of the **Contact Region**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- **Cylindrical Faces** (*3D Only*): This property is only visible when the **Connection Type** property is set to **Contact** and the **Face/Face** property is set to **Yes**. Available options are **Include** (default), **Exclude**, and **Only**. This property determines how the application handles cylindrical faces during automatic contact generation either upon geometry attach or manually on a Connection Group. For example, given a simulation that contains bolted joints, where the bolt shank should have frictionless contact applied and the bolt head should have bonded contact applied. Setting this property appropriately creates contacts during automatic generation that define cylindrical contact (**Only** setting for the bolt shank) and the flat contact (**Exclude** for the bolt head).

- **Face/Edge**: options include **Yes**, **No** (default), **Only Solid Body Edges**, **Only Surface Body Edges**, and **Only Beam Body Edges**. Detects connection between faces and edges of different bodies. Faces are designated as targets and edges are designated as contacts. To determine connection with all faces, for **Only Solid Body Edges**, face to edge connection uses the edges of solid bodies; for **Only Surface Body Edges**, it uses only edges of surface bodies; and for **Only Beam Body Edges**, it uses only edges of beam bodies.

- **Edge Overlap Tolerance** (**Contacts** only): This property is visible only when the **Face/Edge** property is set to **Yes**, **Only Solid Body Edges**, or **Only Surface Body Edges**. It sets the tolerance for overlap of an edge and a face in contact; that is, the minimum percentage of overlap at which a contact pair is created for an edge and a face that overlap. For example, if **Edge Overlap Tolerance** is set to 25, a contact pair is created for an edge and a face when at least 25% of the edge overlaps the face. You can set a value from 0 to 100 or retain the default. Setting the tolerance to 0 turns off the overlap checks.

- **Edge/Edge**: options include **Yes** and **No**. Detects connection between edges of different bodies.

- **Priority**: options include **Include All**, **Face Overrides** and **Edge Overrides**. For very large models the number of connection objects can sometimes become overwhelming and redundant, especially when multiple detection types are chosen. Selecting some type of priority other than **Include All** will lessen the number of connection objects generated during **Create Automatic Connections** by giving designated connection types precedence over other types. **Face Overrides** gives **Face/Face** option precedence over both **Face/Edge** and **Edge/Edge** options. It also gives **Face/Edge** option precedence over **Edge/Edge** option. In general, when **Face Overrides** priority is set with **Face/Edge** and **Edge/Edge** options, no **Edge/Edge** connection pairs will be detected. **Edge Overrides** gives **Edge/Edge** option precedence over both **Face/Edge** and **Face/Face** options, no **Face/Face** connections pairs will be detected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Group By**: options include **None**, **Bodies** (default), **Parts**, and **Faces**. This property enables you to group the automatically generated connections (p. 1218) objects. Setting **Group By** to **Bodies** (default) or to **Parts** means that connection faces and edges that lie on the same bodies or same parts will be included into a single connection object.

  Setting **Group By** to **None** means that the grouping of geometries that lie on the same bodies or same parts will *not* occur. Any connection objects generated will have only one entity scoped to each side (that is, one face or one edge). Applications for choosing **None** in the case of contact are:

  – If there are a large number of source/target faces in a single region. Choosing **None** avoids excessive contact search times in the Mechanical APDL solver.

  – If you want to define different contact behaviors on separate regions with contact of two parts. For example, for a bolt/bracket contact case, you may want to have bonded contact (p. 1235) between the bolt threads/bracket and frictionless contact (p. 1235) between the bolt head/bracket.

- **Search Across**: This property enables automatic connection detection through the following options:

  – **Bodies** (default): Between bodies.

  – **Parts**: Between bodies of different parts, that is, not between bodies within the same multibody part.

  – **Assemblies**: Between bodies from different sub-assemblies (sources) in an Assembled Model (p. 989).

  – **Anywhere**: Detects any connections regardless of where the geometry lies, including different parts. However, if the connections are within the *same body*, this option finds only Face/Face connections, even if the **Face/Edge** setting is turned **On**.

  – **Files**: Between bodies from different External Model source files and between copies of an External Model source file.

- **Fixed Joints**: (**Joint** only) options include **Yes** and **No**. This property determines if **Fixed Joints** are to be automatically generated. See the Automatic Joint Creation (p. 1359) section for details.

- **Revolute Joints**: (**Joint** only) options include **Yes** and **No**. This property determines if **Revolute Joints** are to be automatically generated. See the Automatic Joint Creation (p. 1359) section for details.

Connection Features and Operations

This section examines several common actions you can take when working with connections. Go to a desired topic:

- Automatically Generate Connections (p. 1219)
- Manually Inserting Connection Objects (p. 1220)
- Checking for Overlapping Contact Regions (p. 1220)
- Setting Default APDL Names (p. 1220)
- Searching for Duplicate Pairs (p. 1220)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Moving and Copying Connection Objects (p. 1221)
- Creating Contact Sizing Controls for Contact Regions Automatically (p. 1221)
- Treatment of Legacy Databases (p. 1222)

Automatically Generate Connections

You can automatically generate supported connections for a group of bodies in a model and use a separate tolerance value for that group. The supported connection types are Contact Region (p. 1226) and Joint (p. 1292).

To automatically generate connections for a group of bodies:

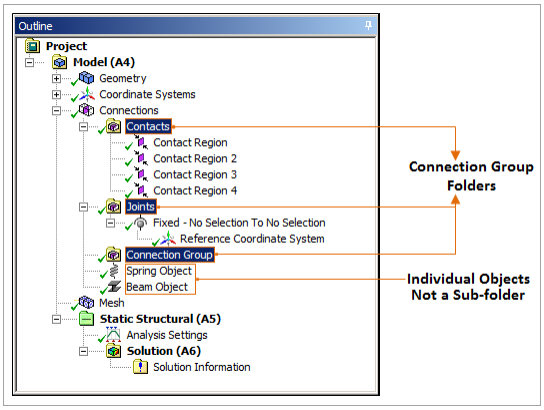
1. Insert a **Connection Group** group folder under the **Connections** folder by selecting the **Connection Group** option on the Connections (p. 50) Context tab or by selecting **Insert** > **Connection Group** from the context menu (right mouse click) for this folder.
2. From the Details view of the **Connection Group** object, select the desired **Connection Type**. The default is **Contact**.
3. Select some bodies in the model based on the **Scoping Method**. The default is **Geometry Selection** scoped to **All Bodies**.
4. If applicable, set the **Auto Detection** properties. Note that these properties will be applied only to scoped geometries for this connection group.
5. Choose **Create Automatic Connections** from the context menu (right mouse click) for the **Connection Group**.

Note:

For small models, the auto contact detection process runs so fast that the **Contact Detection Status** (progress bar) dialog box does not get displayed. However, for large models with many possible contact pairs, the progress bar dialog box is displayed showing the contact detection progress. If you click the **Cancel** button on the dialog box while contact detection is processing, the detection process stops. Any contact pairs found by that moment are discarded and no new contacts are added to the tree.

The resulting connection objects will be placed under this folder and the folder name will be changed from its default name **Connection Group** to a name based on the connection type. The folder name for contacts will be **Contacts** and for joints it will be **Joints**. Once the **Connection Group** folder contains a child object, the **Connection Type** property cannot be changed. Each **Connection Group** folder will hold objects of the same type and will include a worksheet that displays only content pertaining to that folder. When two or more **Connection Group** folders are selected and you choose **Create Automatic Connections**, auto detection for the selected **Connection Group** folders will be performed. The **Create Automatic Connections** option is also available from the context menu (right mouse click) for the **Connections** folder provided there is at least one **Connection Group** folder present. When you choose this command from the **Connections** folder, auto detection will be performed for all connection groups under this folder.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Manually Inserting Connection Objects

You can insert any supported connection objects manually using Context tab options or using the context (right-click) menu on the **Connections** or **Connection Group** folder. When inserting a connection object from the **Connections** folder, a **Connection Group** object will automatically be created in addition to the connection object itself. When inserting a connection object from a **Connection Group** folder, if it is an empty folder, any supported type of object can be inserted. However, if the folder already contains at least one object, only objects of the same type can be inserted.

Checking for Overlapping Contact Regions

Overlapping contact conditions arise when a face or edge is contained in more than one **Contact Region**. Mechanical supports this situation for the Mechanical APDL solver; however, if your analysis is incorporated into a fluid analysis, issues can arise.

Mechanical provides the context menu option Check Overlapping Contact Regions to identify any overlapping contact regions in your model so that you can address them appropriately.

Setting Default APDL Names

The context (right-click) menu option for the Contact Region object, **Set Default APDL Names**, enables you generate system default names for the **Contact**/**Target APDL Name** properties. These properties are optional and only available for the Mechanical APDL solver. See the Setting Default APDL Names (p. 1273) section for more information.

Searching for Duplicate Pairs

Generating connections (Contacts (p. 1222) or Joints (p. 1292)) and Mesh Connections, either automatically or manually, may result in the same geometry pair being scoped by more than one connection object. This may over constrain the model which may lead to convergence difficulty problems in the solver. If this situation occurs, use the context (right-click) menu option **Search Connections for Duplicate Pairs**,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

to check connection objects for duplicate pairs. You can then modify or the geometry scoping of the duplicated pairs or delete the duplicate connection objects.

Note:

The application considers connection objects whose scoping has been flipped when searching for duplicate pairs. For example, if you have two Contact Regions that are scoped to the same geometry pair but have reversed Contact and Target settings, these regions are considered a duplicate pair.

When one or more duplicate pairs are found in the existing connection objects (which includes **Mesh Connections**), the application displays the following warning message in the **Messages** window for a connection object that shares the same geometry pair.

> "This connection object shares the same geometries with one or more connection objects. This may over-constrain the model. Consider eliminating some connection objects."

To search for connection objects that share the same geometry pair with more than one connection object, select multiple connection objects before selecting **Search Connections for Duplicate Pairs**. Or you may issue the search from a **Connections**/**Connection Group** or Mesh Connections/Mesh Connection Group folder, the application searches all connection objects under this folders for duplicate pairs.

Once the application detects duplicates, right-click the message in the **Messages** window and select **Go To Object** from the context menu. The application highlights the associated connection object in the tree. To find other connection objects that share the same geometry pair, right-click the highlighted object and select **Go To Connections for Duplicate Pairs** from the context menu; all connection objects that share the same geometry pair will be highlighted in the tree.

Moving and Copying Connection Objects

To move a connection object to another folder of the same connection type, drag the object and drop it on that folder. For example, to move a contact region object, drag the object from its current **Contacts** folder and drop it on another folder whose **Connection Type** is **Contact** (possibly named **Contacts 2**).

To copy a connection object to another folder of the same connection type, hold the **Ctrl** key while performing the move procedure described above.

Creating Contact Sizing Controls for Contact Regions Automatically

You can create contact sizing controls automatically by:

- Dragging and dropping the **Contacts** folder onto the **Mesh** object to create a Contact Sizing control for each contact region in the folder.
- Selecting the **Contacts** folder or an individual **Contact Region** in the Tree and using the RMB option **Create** > **Contact Sizing** to create Contact Sizing controls for the selected contact regions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Treatment of Legacy Databases

Supported connection objects from databases of previous versions of Ansys Workbench will be grouped based on their types and migrated into **Connection Group** folders.

Contacts from legacy databases are resumed with the **Protected** property set to **No**.

Contact

The following topics are covered in this section:

Contact Overview
Contact Formulation Theory
Contact Settings
Supported Contact Types
Setting Contact Conditions Manually
Contact Ease of Use Features
Contact in Rigid Dynamics
Best Practices for Specifying Contact Conditions

Contact Overview

Contact conditions are created when an assembly is imported into the application and it detects that two separate bodies (solid, surface, and line bodies) touch one another (they are mutually tangent). Bodies/surfaces in contact:

- Do not 'interpenetrate'.
- Can transmit compressive normal forces and tangential friction forces.
- Can be bonded together (Linear)
- Able to separate and collide (Nonlinear)

Surfaces that are free to separate and move away from one another are said to have changing-status nonlinearity. That is, the stiffness of the system depends on the contact status, whether parts are touching or separated.

Use the Contact Tool (p. 2079) to help you coordinate contact conditions before loading and as part of the final solution.

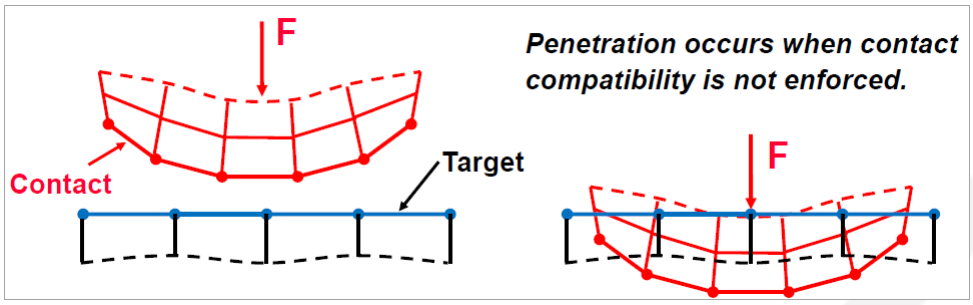
Contact Formulation Theory

Contact solutions are often very complicated. It is recommended that, whenever possible, you employ the **Program Controlled** setting. However, in order to better understand your selections, this section examines the specifics of Formulations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Because contacting bodies do not interpenetrate, the application must establish a relationship between the two surfaces to prevent them from passing through each other in the analysis. When the application prevents interpenetration, it is said to enforce "contact compatibility".



In order to enforce compatibility at the contact interface, the application offers several different contact Formulations. These Formulations define the solution method used. The options for the **Formulation** property are listed below. For additional information, see Formulation (p. 1241) property topic of the **Advanced Settings** section.

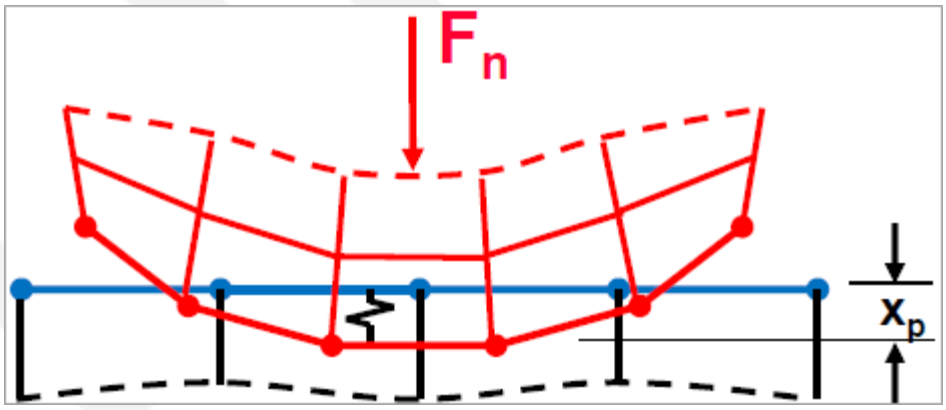
- **Program Controlled** (default)
- Pure Penalty (p. 1223)
- Augmented Lagrange (p. 1223)
- Normal Lagrange (p. 1224)
- MPC (p. 1225)
- Beam (p. 1225)

Pure Penalty and Augmented Lagrange Contact Formulation

For nonlinear solid body contact of faces, **Pure Penalty** or **Augmented Lagrange** formulations can be used. Both of these are penalty-based contact formulations:

$$\mathbf{F_{Normal} = k_{Normal} x_{Penetration}}$$

The finite contact Force, F_n, is a concept of contact stiffness, k_{Normal}. The higher the contact stiffness, the lower the penetration, x_p, as illustrated here.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Ideally, for an infinite k_{Normal}, one would get zero penetration. This is not numerically possible with penalty-based methods, but as long as x_p is small or negligible, the solution results are accurate.

The main difference between Pure Penalty and Augmented Lagrange methods is that Augmented Lagrange augments the contact force (pressure) calculations:

Pure Penalty: $\mathbf{F_{Normal} = k_{Normal} x_{Penetration}}$

Augmented Lagrange: $\mathbf{F_{Normal} = k_{Normal} x_{Penetration}} + \lambda$

Because of the extra term λ, the Augmented Lagrange method is less sensitive to the magnitude of the contact stiffness k_{Normal}.

Normal Lagrange Contact Formulation

Another available option is **Normal Lagrange**. This formulation adds an extra degree of freedom (contact pressure) to satisfy contact compatibility. Consequently, instead of resolving contact force as contact stiffness and penetration, contact force (contact pressure) is solved for explicitly as an extra DOF.

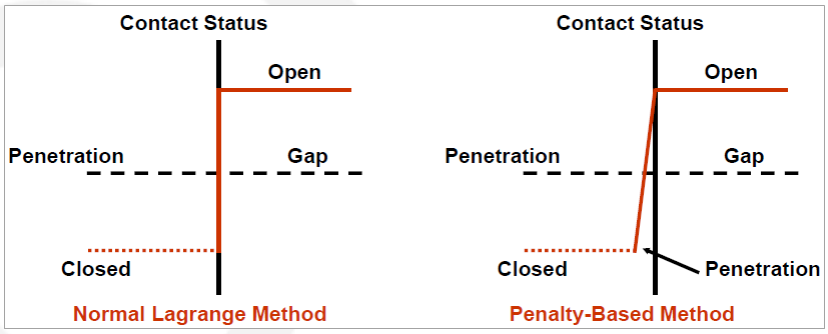
F_{Normal} = DOF

Specifications:

- Enforces zero/nearly zero penetration with pressure DOF.
- Does *not* require a normal contact stiffness (zero elastic slip)
- Requires **Direct Solver**, which can increase computation requirements.

Normal Lagrange Chattering

Chattering is an issue which often occurs with Normal Lagrange method. If no penetration is allowed (left), then the contact status is either open or closed (a step function). This can sometimes make convergence more difficult because contact points may oscillate between an open and closed status. This is called "chattering". If some slight penetration is allowed (right), it can make it easier to converge since contact is no longer a step change.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Multi-Point Constraint (MPC) Contact Formulation

For the specific case of **Bonded** and **No Separation** **Types** (p. 1234) of contact between two faces, a Multi-Point Constraint (**MPC**) formulation is available. MPC internally adds constraint equations to "tie" the displacements between contacting surfaces. This approach is not penalty-based or Lagrange multiplier-based. It is a direct, efficient way of relating surfaces of contact regions which are bonded. Large-deformation effects are supported with MPC-based Bonded contact.

Beam Contact Formulation

Beam Formulation is available for the **Bonded** Type (p. 1234) only. This formulation works by "stitching" the contact topologies together using **massless** linear Beam Elements.

Comparison of Formulations

Some of the primary aspects of contact formulations are compared below.

Table 3: Comparing Contact Formulations

<table>
<tr><th>Pure Penalty</th><th>Augmented Lagrange</th><th>Normal Lagrange</th><th>MPC</th><th>Beam</th></tr>
<tr><td>Good convergence behavior (few equilibrium iterations).</td><td>Additional equilibrium iterations needed if penetration is too large.</td><td>Additional equilibrium iterations needed if chattering is present.</td><td colspan="2">Excellent convergence behavior (one equilibrium iteration).</td></tr>
<tr><td>Sensitive to selection of normal contact stiffness.</td><td>Less sensitive to selection of normal contact stiffness.</td><td colspan="2">No normal contact stiffness is required.</td><td>NA</td></tr>
<tr><td>Contact penetration is present and uncontrolled.</td><td>Contact penetration is present but controlled to some degree.</td><td>Usually, penetration is near-zero.</td><td>No Penetration.</td><td>Penetration is minimal with a stiff enough material definition.</td></tr>
<tr><td colspan="3">Useful for any type of contact behavior.</td><td>Only Bonded & No Separation behaviors.</td><td>Bonded Only</td></tr>
<tr><td colspan="2">Iterative or Direct Solvers can be used.</td><td>Only Direct Solver can be Used.</td><td colspan="2">Iterative or Direct Solvers can be used.</td></tr>
<tr><td colspan="2">Symmetric or Asymmetric contact available.</td><td colspan="2">Asymmetric contact Only.</td><td>NA</td></tr>
<tr><td colspan="2">Contact detection at integration points.</td><td colspan="2">Contact Detection at Nodes.</td><td>NA</td></tr>
</table>

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Contact Settings

When a model is imported into Workbench Mechanical, the default setting of the application is to automatically detect instances where two bodies are in contact and to generate the corresponding Contact Region objects.

When a **Contact Region** is selected in the Outline, as illustrated here, contact settings are available in the **Details** pane, and are included in the following categories:

- **Scope** (p. 1227): settings for displaying, selecting, or listing contact and target geometries.
- **Definition** (p. 1234): commonly used contact settings.
- **Advanced** (p. 1240): advanced controls that are primarily program controlled.
- **Display** (p. 1256): display of element normal direction.
- **Geometric Modification** (p. 1257): settings for further defining contact interface behaviors.

Details of "Frictionless - V1Mid To HNear"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	V1Mid
Target Bodies	HNear
Protected	No
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	1.1979 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	1.e-003 mm
Contact Geometry Correction	Bolt Thread
Orientation	Revolute Axis
Starting Point	Coordinate System
Ending Point	Coordinate System 2
Mean Pitch Diameter	2.5 mm
Pitch Distance	1.5 mm
Thread Angle	60. °
Thread Type	Single-Thread
Handedness	Right-Handed

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Scope Settings

The properties for the **Scope** category are described in the following table. Also make sure you review the Stiffness Behavior Support Specifications (p. 1234) topic at the end of the section.

Property	*Description/Selections*
Scoping Method	Specifies whether the **Contact Region** is applied to a **Geometry Selection** (default), a Named Selection (p. 89), or to a **Pre-Generated Interface** for fracture mechanics (Interface Delamination) when you are using the Ansys Composite PrepPost (ACP) application.
Interface	This property displays when you select **Pre-Generated Interface** as the **Scoping Method**. It provides a drop-down list of the available interface layers that were imported from ACP.
Contact	Displays/selects which geometries (bodies, faces, edges, or vertices) or mesh entities (elements [3D shell only], element faces or node set) are considered in contact. The geometries can be manually selected or automatically generated. **Important:** Note the following requirements when scoping with mesh nodes. • Can only be performed using manual scoping (no automatic detection). • Is not supported in combination with the **Shell Thickness Effect** property (see below). For **Face/Edge** or **Body/Edge** contact, the edge must be designated as **Contact**. **Body** scoping always includes all pertinent faces in the solution process. A contact pair can have a flexible-rigid scoping, but the flexible side of the pair must always be the **Contact** side. If the Contact side of the contact pair is scoped to multiple bodies, **all** of the bodies must have the same Stiffness Behavior (p. 852), either **Rigid** or **Flexible**. **Note:** When the Rigid Body Mesh Behavior is set to **Dimensionally Reduced**, the **Contact** scoping can not be scoped to a **Named Selection of Mesh Entities**.
Target	Displays which body element (body, face, or edge) or mesh entities (elements [3D shell only] or element faces) is considered

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	*Description/Selections*
	Target (versus Contact). This element can be manually set or automatically generated. For **Face/Edge** or **Body/Edge** contact, the face or body must be designated as **Target**. **Body** scoping includes all pertinent faces in the solution process. If the **Contact** side of the contact pair has a flexible Stiffness Behavior (p. 852) then the **Target** side can be rigid. Multiple *rigid* bodies cannot be selected for the **Target** side scoping of the contact pair. The selection of multiple rigid bodies for the **Target** invalidates the Contact Region object and an error message is generated following the solution process. **Note:** If you click this field, the bodies are highlighted. **Note:** When the Rigid Body Mesh Behavior is set to **Dimensionally Reduced**, the **Target** scoping can not be scoped to a **Named Selection of Mesh Entities**.
Shared Target Body	This property displays when the **Target** is scoped to either edges or faces that are shared by more than one body and: • The **Contact** or **Target** property is scoped to edges or faces that are shared by the scoping of a General Axisymmetric object. **Or…** • When the **Contact** and **Target** properties are scoped to the edges and the **Edge Contact Type** property is set to **Nodes on Edge**. When displayed, this property provides a drop-down list that includes the bodies that share the edge or face. Scoping the property to one of these body selections specifies that the target elements are generated on top of elements of the selected body and their normal is outward to the boundary of the body.
Reverse Shared Contact Normal	**Reverse Shared Contact Normal**: This property displays when the **Contact** property scoping of the contact condition includes the following: 1. One or more 2D edges, shared by more than one body, and that are not specified by a **General Axisymmetric**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	*Description/Selections*
	object. The **Target** scoping cannot be specified by a **General Axisymmetric** object, either. 2. One or more 3D faces that are shared by more than one body. When displayed, you select subset of shared contact edges or faces to reverse the direction of the normal for the generated contact elements. **Important:** If a shared face is shared by both shell and solid bodies, the **Reverse Shared Contact Normal** property only displays when **Contact Shell Face** property is set to **Top** or **Bottom**. **Note:** You can set the **Element Normals** property of the **Display** category to **Yes** to display the normal directions of the elements in contact.
Reverse Shared Target Normal	**Reverse Shared Target Normal**: This property displays when the **Target** property scoping of the contact condition includes the following: 1. One or more 2D edges, shared by more than one body, and that are not specified by a **General Axisymmetric** object. The **Contact** scoping cannot be specified by a **General Axisymmetric** object, either. 2. One or more 3D faces that are shared by more than one body. When displayed, you select subset of shared target edges or faces to reverse the direction of the normal for the generated target elements. **Important:** If a shared face is shared by both shell and solid bodies, the **Reverse Shared Target Normal** property

Property	*Description/Selections*
	only displays when **Target Shell Face** property is set to **Top** or **Bottom**. **Note:** You can set the **Element Normals** property of the **Display** category to **Yes** to display the normal directions of the elements in contact.
Contact Bodies	This read-only property displays the name of the parts included in the **Contact** side of the **Contact Region**.
Target Bodies	This read-only property displays the name of the parts included in the **Target** side of the **Contact Region**.
Edge Contact Type	This property is visible when the **Contact** geometry is an Edge (of a line, shell, or solid body) and the **Target** geometry is an Edge, Face, or Element Face (of a shell or solid body). Property options include: • **Program Controlled** (default): For 3D Structural analyses, the application internally uses the **Line Segments** (CONTA177 element) option. For all other cases, the application uses the **Nodes on Edge** (CONTA175 element) option as the internal setting. **Exception**: If you are performing a substructure analysis, or if your analysis is linked to a downstream **Structural Optimization** analysis (**Future Analysis** property set to **Structural Optimization**), then the application uses the CONTA175 element. • **Nodes On Edge**: When selected, the application uses the CONTA175 element for the **Contact Region**. • **Line Segments**: This option is only supported for structural physics-based analyses. When selected, the application uses the CONTA177 element for the **Contact Region**. **Note:** This property is not supported for contact between the edges of two line bodies.
Line-Line Detection	The application displays this property when you: • Specifying beam edges as the scoping for both the **Contact** and **Target** properties. *Or...*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	*Description/Selections*
	• Specify the **Contact** property as an edge (of a line, shell, or solid body) and the **Target** property an edge, face, or element face (of a shell or solid body) and the **Edge Contact Type** property set to **Line Segments** and the **Formulation** property is set to **Augmented Lagrange** or **Pure Penalty**. Property options include: • **External - Only 1 Segment** • **External - Up to 4 Segments** (default) • **External - Up to 8 Segments** • **Internal Pipe Contact** (only visible for beam-to-beam edge contact) **Note:** • The external options enable you to specify the maximum number of target segments interacting with each contact detection point simultaneously. • The **Internal Pipe Contact** can be used to model a beam (or pipe) sliding inside another hollow beam (or pipe). However, if the inner beam is much stiffer than the outer beam, the inner beam can be the target surface. This setting specifies a maximum of four target segments for the **Contact** edge (CONTA177). • See the CONTA177 and 3D Beam-To-Beam Contact sections for additional technical details about this property.
Beam-Beam Model	The application displays this property when it detects beam-to-beam contact. It enables you to specify the contact traction-based model the application uses for Beam-to-Beam contact. Options include: • **Exclude Crossing Beams**: For this option, the application excludes the contact for any beams that cross one another. The Mechanical APDL Reference for this option is KEYOPT(3) = 1. • **Only Crossing Beams**: For this option, the application includes only the contact for beams that cross one

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	*Description/Selections*
	another. The Mechanical APDL Reference for this option is KEYOPT(3) = 3. • **All** (default): This is the default setting. The application includes all beam contact scenarios, which are: beam/edge to surface contact, parallel beam-to-beam contact, and crossing beam-to-beam contact. The Mechanical APDL Reference for this option is KEYOPT(3) = 2. For additional Mechanical APDL specific information, see KEYOPT(3) in the *Mechanical APDL Contact Technology Guide*.
Protected	Specifies if the contact entities (faces, edges, and vertices) are protected topology. Set the property to **Yes** to respect the geometry features the **Contact** is scoped to and ensure proper association between the geometry and mesh. Set the property to **No** to indicate that the topology may not be protected.
Contact Shell Face	Specifies whether the **Contact** should be applied on a surface body's top face or bottom face. When scoped to an element face, the contact is applied to the scoped side (top/bottom) of the shell face. If you set **Contact Shell Face** to the default option, **Program Controlled**, then the **Target Shell Face** option must also be set to **Program Controlled**. The **Program Controlled** default option is not valid for nonlinear contact types. This option displays only when you scope a surface body to **Contact Bodies**.
Target Shell Face	Specifies whether the **Target** should be applied on a surface body's top face or bottom face. When scoped to an element face, the target is applied to the scoped side (top/bottom) of the shell face. If you set **Target Shell Face** to the default option, **Program Controlled**, then the **Contact Shell Face** option must also be set to **Program Controlled**. The **Program Controlled** default option is not valid for nonlinear contact types. This option displays only when you scope a surface body to **Target Bodies**.
Shell Thickness Effect (See notes below as well as Using KEYOPT(11))	This property appears when the scoping of the contact or target includes a Surface Body. Options include: • **Yes**: Include the property. • **No** (default): Exclude the property. When set to **Yes**, the contact object becomes under-defined if the Offset Type of any scoped surface body is set to a value other than **Middle**. In this situation, the following error message will be displayed: "The shell thickness effect of a contact pair is turned on; however, the offset type of a shell body in contact is set to other than Middle. Set its offset type to Middle." In the presence of a Thickness, Imported Thickness, Layered Section, or an Imported Layered Section object, the following

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	*Description/Selections*
	warning message will be issued if a solve is requested: "The shell thickness effect of a contact pair is turned on. Make sure that the offset type of the thickness, imported thickness, layered sections and imported layered sections objects associated with the shell bodies in contact are set to Middle."

Shell Thickness Effect

The Shell Thickness Effect allows users to automatically include the thickness of the surface body during contact calculations. Instead of contact being detected on the face of the surface body, contact will be detected a distance of half the thickness away from the face.

If the surface body undergoes large strains and changes thickness, the updated (current) thickness is also used in the contact calculations. However, to be able to take advantage of this feature, the Offset Type (p. 862) must be set to **Middle**.

For cases where Offset Type (p. 862) is set to **Top** or **Bottom**, you can do the following:

- For a given contact region, if contact is occurring on the same face (**Top** or **Bottom**) as the offset, no special settings are required. The location of the nodes and elements of the surface body represent the actual position of that face.
- For **Rough**, **Frictionless**, or **Frictional** contact types, if contact is occurring on the opposite face as the offset, specify a contact Offset equal to the shell thickness for the Interface Treatment (p. 1257). Note that changes in shell thickness for large strain analyses will not be considered.

Note:

If the Shell Thickness Effect is activated, and you have specified a contact Offset for the Interface Treatment (p. 1257), the total offset will be half the thickness of the surface body plus the defined contact offset.

Postprocessing surface bodies using the **Shell Thickness Effect** has the following special considerations:

- Because contact is detected half of the thickness from the middle of the surface body, viewing surface body results without **Thick Shell and Beam** (See the Style group (p. 80) of the **Display** tab) effects turned on will show an apparent gap between contact bodies. This is normal since contact is being detected away from the location of the nodes and elements.
- When using the Contact Tool (p. 2079) to postprocess penetration or gaps, these values are measured from the middle of the surface bodies (location of the nodes and elements), regardless of whether or not the shell thickness effect is active.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Stiffness Behavior Support Specifications

Note:

- All geometric entities selected for a contact condition, on either the **Contact** or the **Target** side of the contact pair, must have the same setting for the **Stiffness Behavior** property.
- If the **Stiffness Behavior** property of a geometry is set to **Rigid**, you must set the **Definition** property to **Asymmetric**.
- You cannot scope the **Target** side in a contact pair to more than one rigid body.
- If you have both rigid and flexible bodies in your contact pair, you must scope the rigid body as a **Target**.
- For the Mechanical APDL solver, you cannot scope the **Target** side in a contact pair to the edge of a rigid body.

Definition Settings

The differences in the contact settings determine how the contacting bodies can move relative to one another. This category provides the following properties.

- Type (p. 1234)
- Scope Mode (p. 1236)
- Behavior (p. 1236)
- Trim Contact (p. 1237)
- Contact/Target APDL Name (p. 1240)
- Suppressed (p. 1240)

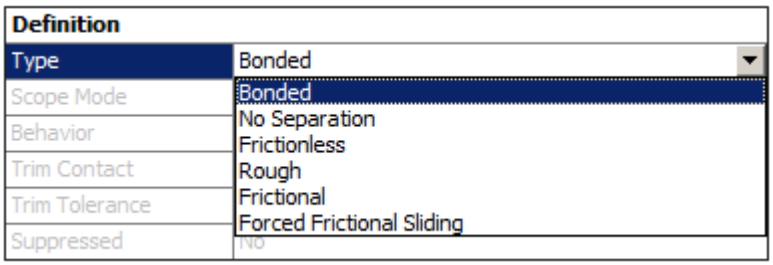
Type

Choosing the appropriate contact type depends on the type of problem you are trying to solve. If modeling the ability of bodies to separate or open slightly is important and/or obtaining the stresses very near a contact interface is important, consider using one of the nonlinear contact types (**Frictionless**, **Rough**, **Frictional**), which can model gaps and more accurately model the true area of contact. However, using these contact types usually results in longer solution times and can have possible convergence problems due to the contact nonlinearity. If convergence problems arise or if determining the exact area of contact is critical, consider using a finer mesh (using the Sizing control) on the contact faces or edges.

The available contact types are listed below. Most of the types apply to Contact Regions made up of faces only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- **Bonded**: This is the default configuration and applies to all contact regions (surfaces, solids, lines, faces, edges). If contact regions are bonded, then no sliding or separation between faces or edges is allowed. Think of the region as *glued*. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. If contact is determined on the mathematical model, any gaps will be closed and any initial penetration will be ignored. [Not supported for Rigid Dynamics. Fixed joint can be used instead.]
- **No Separation**: This contact setting is similar to the **Bonded** case. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). Separation of the geometries in contact is not allowed.
- **Frictionless**: This setting models standard unilateral contact; that is, normal pressure equals zero if separation occurs. Therefore, gaps can form in the model between bodies depending on the loading. This solution is nonlinear because the area of contact may change as the load is applied. A zero coefficient of friction is assumed and allows free sliding. The model should be well constrained when using this contact setting. Weak springs are added to the assembly to help stabilize the model in order to achieve a reasonable solution.
- **Rough**: Similar to the frictionless setting, this setting models perfectly rough frictional contact where there is no sliding. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). By default, no automatic closing of gaps is performed. This case corresponds to an infinite friction coefficient between the contacting bodies. [Not supported for Explicit Dynamics analyses.]
- **Frictional**: In this setting, the two contacting geometries can carry shear stresses up to a certain magnitude across their interface before they start sliding relative to each other. This state is known as "sticking." The model defines an equivalent shear stress at which sliding on the geometry begins as a fraction of the contact pressure. Once the shear stress is exceeded, the two geometries will slide relative to each other. The coefficient of friction can be any non-negative value. [Not supported for Rigid Dynamics. **Forced Frictional Sliding** should be used instead.]
- **Forced Frictional Sliding**: In this setting, a tangent resisting force is applied at each contact point. The tangent force is proportional to the normal contact force. This setting is similar to **Frictional** except that there is no "sticking" state. [Supported only for Rigid Dynamics]

  By default the friction is not applied during collision. Collisions are treated as if the contact is frictionless regardless the friction coefficient. The following commands override this behavior and include friction in shock resolution (see Rigid Dynamics Command Objects Library in the *Mechanical User's Guide* (p. 563) for more information).

  ```
  options=CS_SolverOptions()

  options.FrictionForShock=1
  ```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Note that shock resolution assumes permanent sliding during shock, which may lead to unrealistic results when the friction coefficient is greater than 0.5.

- **Friction Coefficient**: Enables you to enter a friction coefficient. Displayed only for frictional contact applications.

Note:

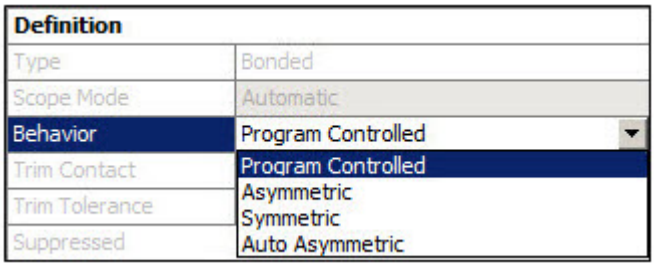
- For the **Bonded** and **No Separation** contact **Type**, you can simulate the separation of a **Contact Region** as it reaches some predefined opening criteria using the Contact Debonding (p. 476) feature.
- Refer to KEYOPT(12) in the *Mechanical APDL Contact Technology Guide* for more information about modelling different contact surface behaviors.

Scope Mode

This is a *read-only* property that displays how the selected **Contact Region** was generated. Either automatically generated by the application (**Automatic**) or constructed or modified by the user (Manual (p. 1266)). Note that this property is not supported for Rigid Body Dynamics analyses.

Behavior

This property will appear only for 3D Face/Face or 2D Edge/Edge contacts. For 3D Edge/Edge or Face/Edge contacts, internally the program will set the contact behavior to **Asymmetric** (see below). Note that this property is not supported for Rigid Body Dynamics analyses.



Sets contact pair to one of the following:

- **Program Controlled** (*Default* for the Mechanical APDL solver): internally the contact behavior is set to the following options based on the stated condition:
  - **Auto Asymmetric** (see below): for Flexible-Flexible bodies.
  - **Asymmetric** (see below): for Flexible-Rigid bodies.
  - **Symmetric** (see below): for Flexible-Flexible bodies that are scoped to a Nonlinear Adaptive Region (p. 1707).

    For Rigid-Rigid contacts, the Behavior property is under-defined for the **Program Controlled** setting. The validation check is performed at the Contact object level when all environment branches are using the Mechanical APDL solver. If the solver target for one of the environments

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

is other than Mechanical APDL, then this validation check will be carried out at the environment level; the environment branch will become under-defined.

- **Asymmetric**: Contact will be asymmetric for the solve. All face/edge and edge/edge contacts will be asymmetric. [In Explicit Dynamics analyses this is supported for Bonded connections.]

  Asymmetric contact has one face as **Contact** and one face as **Target** (as defined under Scope (p. 1227) settings), creating a single contact pair. This is sometimes called "one-pass contact," and is usually the most efficient way to model face-to-face contact for solid bodies.

  The **Behavior** property setting must be **Asymmetric** if the scoping includes a body specified with rigid Stiffness Behavior (p. 852).

- **Symmetric**: Contact will be symmetric for the solve. The symmetric pairs will have the same contact characteristics (using KEYOPT(8)=1) except when the Nonlinear Adaptive Region (p. 1707) object is present.

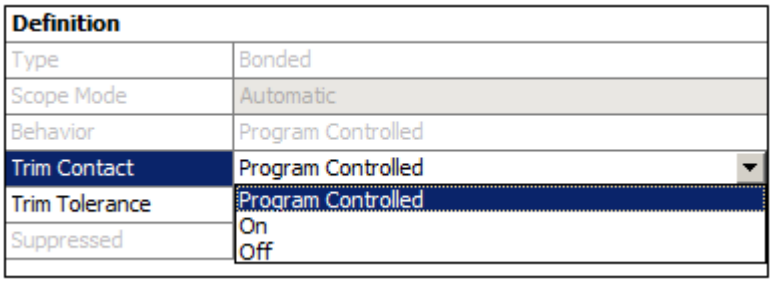
- **Auto Asymmetric**: Automatically creates an asymmetric (p. 1237) contact pair, if possible. This can significantly improve performance in some instances. When you choose this setting, during the solution phase the solver will automatically choose the more appropriate contact face designation. You can also designate the roles of each face in the contact pair manually. [In Explicit Dynamics analyses this option is available for Bonded connections; see Bonded Type.]

Note:

Refer to KEYOPT(8) in the *Mechanical APDL Contact Technology Guide* for more information about asymmetric and symmetric contact selection.

Trim Contact

The **Trim Contact** feature can speed up the solution time by reducing the number of contact elements sent to the solver for consideration. For a Contact Region interface of a Condensed Part, this property enables you to reduce the number of master DOFs the application sends to the solver. Note that this feature is not supported for Rigid Body Dynamics analyses.



Trim Contact options include:

- **Program Controlled**: This is the default setting. The application chooses the appropriate setting. Typically, the application sets **Trim Contact** to **On**. However, if there are manually created contact conditions, no trimming is performed. By default, for Condensed Part generation, no trimming of the master DOFs is performed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **On**: During the process of creating the solver input file, checking is performed to determine the proximity between source and target elements. Elements from the source and target sides which are not in close proximity (determined by a tolerance) are not written to the file and therefore ignored in the analysis.
- **Off**: No contact trimming is performed.

The checking process is performed to identify if there is overlap between the bounding boxes of the elements involved. If the bounding box of an element does not overlap the bounding box of an opposing face or element set, that element is excluded from the solution. Before the elements are checked, the bounding boxes are expanded using the **Trim Tolerance** property (explained below) so that overlapping can be detected.

Trim Tolerance

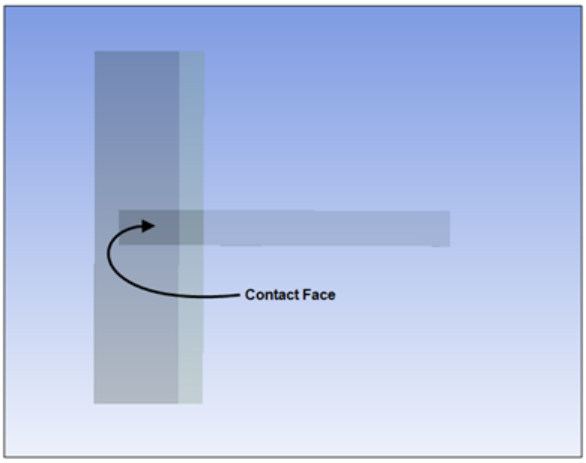
This property provides the ability to define the tolerance value that is used to expand the bounding boxes of the elements before the trimming process is performed.

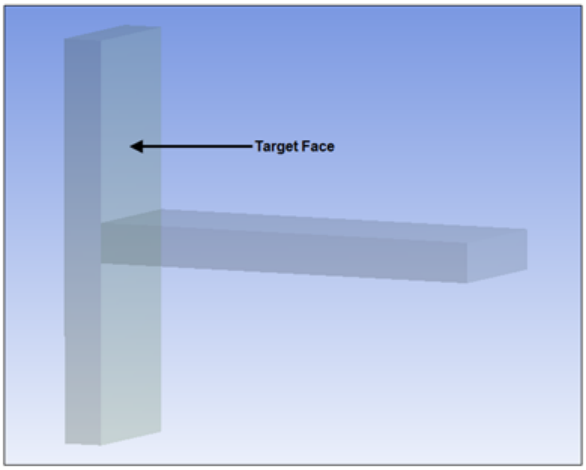
This property is available for both automatic and manual contacts when the **Trim Contact** is set to **On**. It is only available for automatic contacts when the **Trim Contact** is set to **Program Controlled** since no trimming is performed for manual contacts. For automatic contacts, this property displays the value that was used for contact detection and it is a read-only field. For manual contacts, enter a value greater than zero.

Note that a doubling expansion effect can result from the bounding box expansion since the bounding box of both the source and target elements are expanded. An example of the double expansion effect is illustrated below where the Trim Tolerance is defined as 10 mm. For simplicity sake, the size of the elements is specified as 5mm. Therefore, the bounding boxes for the contact/target elements will extend 10mm (two elements) in each direction as represented by the orange boxes, solid and dashed. For each face, Contact and Target, the number of elements that will be used are illustrated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

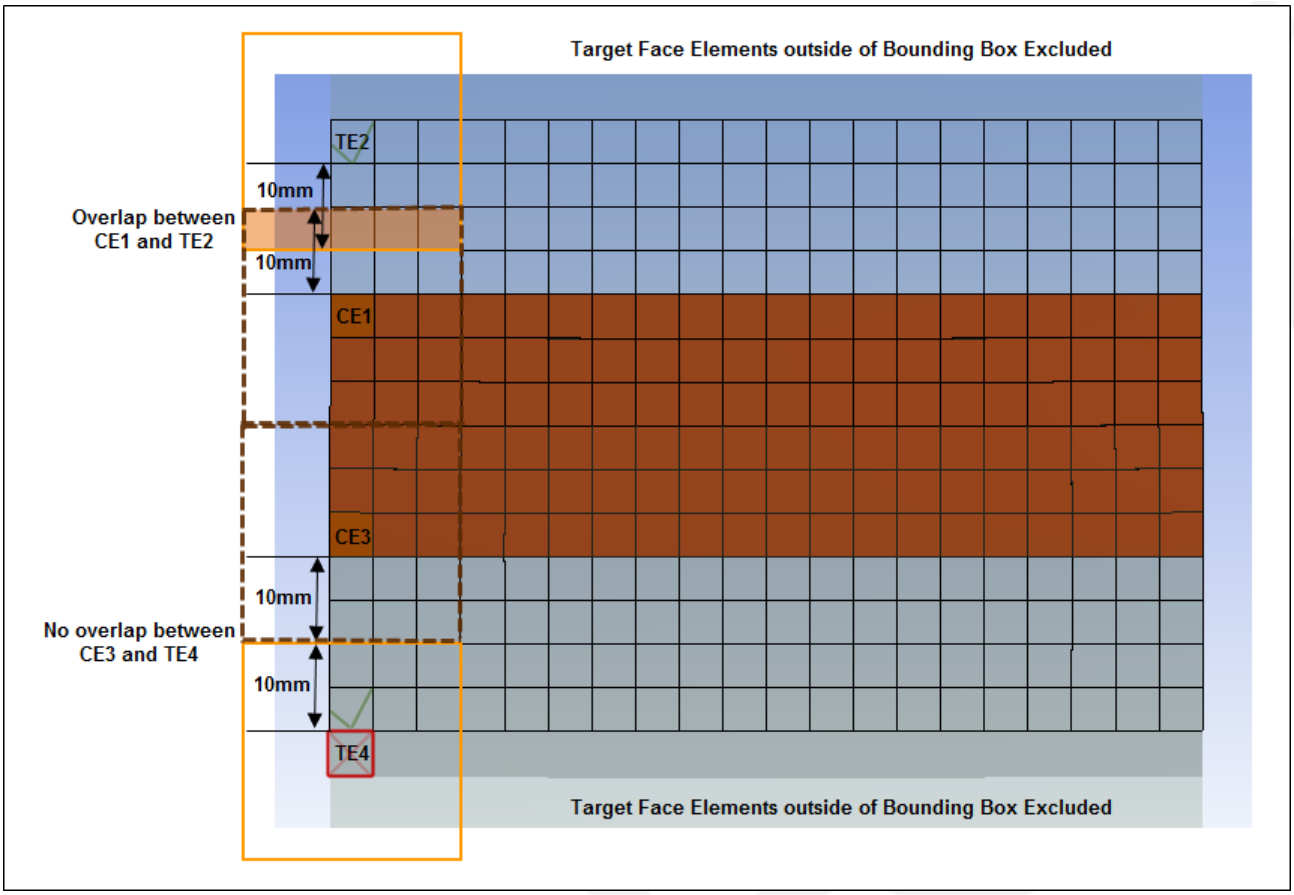
Published: 2023-10-23





The brown area illustrated below represents the elements from the contact face. On the corresponding target side exist potential elements from the entire target face. The elements of the target face that will be kept are drawn in black. On the target Face, each element bounding box is expanded by 10mm and an overlap is sought against each element from the contact side. Referring to the image below, the bounding boxes between Contact Element 1 (**CE1**) and Target Element 2 (**TE2**) overlap and therefore **TE2** is included in the analysis. Meanwhile, **CE3** and **TE4** do not overlap and as a result, **TE4** is not included in the analysis. This results in a reduced number of elements in the analysis and, typically, a faster solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Contact/Target APDL Name

These properties are optional and only available for the Mechanical APDL solver. They enable you to specify an APDL parameter name for the contact and the target element types. Using these names, you can easily identify the contact and target element type and their corresponding real constant set that you can reference and use in a Commands (APDL) object.

Note:

Because element key options for all contact and target element types can be modified or set using the **KEYOPT** command with the *ITYPE* argument set **CONT** or **TARG**, Ansys recommends that you do not specify the **Contact/Target APDL Name** property using a variable that begins with "Cont" or "Targ," such as "Contact_1" or "Target _1."

If the **Contact/Target APDL Name** entry modifies an element key option (KEYOPT), the application will automatically modify **all** contact and target element types.

Suppressed

Specifies whether or not the Contact Region is included in the solution.

Advanced Settings

The **Advanced** category provides the following properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

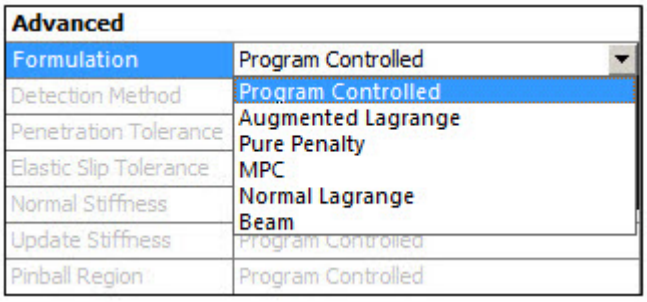
Published: 2023-10-23

- Formulation (p. 1241)
- Small Sliding (p. 1244)
- Detection Method (p. 1244)
- Penetration Tolerance (p. 1246)
- Elastic Slip Tolerance (p. 1247)
- Normal Stiffness (p. 1249)
- Constraint Type (p. 1248)
- Update Stiffness (p. 1249)
- Electric Capacitance (p. 1250)
- Pressure at Zero Penetration (p. 1251)
- Initial Clearance (p. 1251)
- Stabilization Damping Factor (p. 1252)
- Thermal Conductance (p. 1253)
- Pinball Region (p. 1253)
- Pinball Radius (p. 1254)
- Time Step Controls (p. 1255)
- Electric Conductance (p. 1254)
- Restitution Factor—Rigid Dynamics Solver Only (p. 1255)
- Continuous Distance Computation—Rigid Dynamics Solver Only (p. 1255)

Formulation

Formulation options allow you to specify which algorithm the software uses for a particular Contact pair computation.

Property options include:



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option	*Description*	**Mechanical APDL Reference**
Program Controlled (default)	For this setting, the application internally selects the: • **Pure Penalty** option for contact between two rigid bodies. • **MPC** option when the **Inverse Option** property to **Yes** (**Analysis Settings** > Advanced (p. 1461)). • **MPC** option for bonded contact defined between solid-solid, solid-shell, shell-shell, and shell-beam bodies in 3D structural analyses. **Exclusions** The application uses the **Augmented Lagrange** option internally when the analysis: – Includes Additive Manufacturing processes. – If the **Physics Type** property is set to an option other **Structural**, such as **Thermal** or **Acoustics**. This property is generally populated based on the analysis type you select, but could vary for a Coupled Field analysis, as shown in the **Physics Definition** category of the environment. – Includes a **Cyclic Symmetry** object. – Is a **Substructure Generation** analysis. – Is linked to a downstream **Structural Optimization** analysis. – Has a contact region scoped to a body that is also scoped to a **General Axisymmetric** object. – If the bonded contact has an edge or a face overlapping with certain boundary conditions, such as **Fixed Support**, **Simple Support**, **Frictionless Support**, **Displacement** (as support - all entries as zero), **Normal Displacement** (as support - all entries as zero), **Cylindrical Displacement** (as support - all entries as zero), and loads such as **Remote Force** and **Remote Displacement**. • **Augmented Lagrange** option for all other contact situations.	-
Pure Penalty	Basic contact formulation based on Penalty method.	KEYOPT(2) = 1
Augmented Lagrange	Also a penalty-based method. Compared to the **Pure Penalty** method, this method usually leads to better conditioning and is less sensitive to the magnitude of the contact stiffness coefficient. However, in some analyses, the Augmented Lagrange method may require additional iterations, especially if the deformed mesh becomes too distorted.	KEYOPT(2) = 0

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

MPC	Available for **Bonded** and for **No Separation** contact **Types**. Multipoint Constraint equations are created internally during the Mechanical APDL solution to tie the bodies together. This can be helpful if truly linear contact is desired or to handle the nonzero mode issue for free vibration that can occur if a penalty function is used. Note that contact based results (such as pressure) will be zero. **Note:** When modeling Shell-Solid assemblies with the **MPC** contact Formulation, the contact surface/edge must be on the shell side and the target surface must be on the solid side. However, you can override this requirement to support certain special cases, such as acoustics. See the Modeling a Shell-Solid Assembly section of the *Mechanical APDL Contact Technology Guide* for additional information.	KEYOPT(2) = 2
Normal Lagrange	Enforces zero penetration when contact is closed making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. Normal Stiffness is not applicable for this setting. Normal Lagrange adds contact traction to the model as additional degrees of freedom and requires additional iterations to stabilize contact conditions. It often increases the computational cost compared to the Augmented Lagrange setting. The Iterative setting (under Solver Type) cannot be used with this method.	KEYOPT(2) = 3
Beam	Available for **Bonded** (p. 1235) Type only. This formulation works by "stitching" the contact topologies together using **massless** linear Beam Elements.	-

For additional Mechanical APDL specific information, see KEYOPT(2) in the *Mechanical APDL Contact Technology Guide*.

Note:

Cases involving large gaps and faces bonded together can result in fictitious moments being transmitted across a boundary.

Fluid Structure Interface Contact Conditions

Note the following for contact conditions defined at a Fluid Structure Interface:

- The Contact side must be on the acoustic body and the Target must be on the structural body.
- The Bonded contact type setting and the Pure Penalty formulation is supported in addition to MPC formulation.
- Pure Penalty formulation is not supported for contact conditions between two acoustic bodies.
- The Nodal-Dual Shape Function Projection (keyo,cid,4,4) option, of the Detection Method (p. 1244) property, is used by default for FSI contact defined using the Pure Penalty formulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The Combined option (keyo,cid,4,5), of the Detection Method (p. 1244) property, is not supported for the MPC formulation type

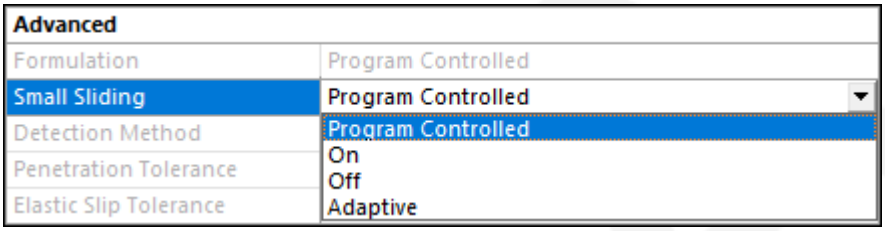
Small Sliding

The **Small Sliding** property displays and activates an assumption of relatively-small sliding (less than 20% of the sum of the average length of each contact element for the contact pair). If small sliding is known to occur, this feature can make your solution more efficient and robust. The **Program Controlled** setting for the **Small Sliding** property automatically sets the property to **On** in most situations if the **Large Deflection** property is set to **Off** or the **Formulation** property is set to **Bonded** contact. The default setting for this property can be changed using the **Small Sliding** option in the **Connections** (p. 219) category of the **Options** preference.

Note:

When you specify a Coupled Field (p. 376) analysis using Electrostatic physics only (standalone), the application automatically disables this property.

Property options include:



Option	Description
Program Controlled	This is the default setting. The application determines whether to select small or finite sliding.
Yes	Enable small sliding.
No	Disable small sliding and set finite sliding.
Adaptive	Activates adaptive small sliding.

For additional information, see the Selecting a Sliding Behavior topic in the *Mechanical APDL Contact Technology Guide*.

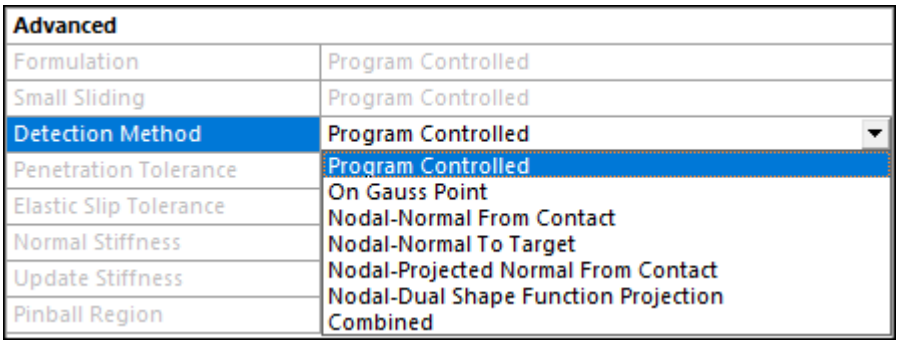
Detection Method

Detection Method enables you to choose the location of contact detection used in the analysis in order to obtain a good convergence. It is applicable to 3D face-face contacts and 2D edge-edge contacts.

Property options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Option	Description
Program Controlled	The application uses Gauss integration points (**On Gauss Point**) when the **Formulation** property is set to **Pure Penalty** or **Augmented Lagrange**. It uses nodal point (**Nodal-Normal to Target**) when the **Formulation** property is set to **MPC** or **Normal Lagrange**. When working with Gasket bodies, the application automatically selects the option, **Nodal-Projected Normal From Contact**.
On Gauss Point	The contact detection location is at the Gauss integration points. This option is not applicable to contacts with MPC or Normal Lagrange formulation.
Nodal - Normal From Contact	The contact detection location is on a nodal point where the contact normal is perpendicular to the contact surface.
Nodal - Normal To Target	The contact detection location is on a nodal point where the contact normal is perpendicular to the target surface.
Nodal - Projected Normal From Contact	The contact detection location is at contact nodal points in an overlapping region of the contact and target surfaces (projection-based method). **Note:** Additive Manufacturing Process simulations do not support projection-based contact.
Nodal - Dual Shape Function Projection	The contact detection location is at contact nodal points in an overlapping region of the Contact and Target surfaces (Dual Shape function projection-based method). This option is efficient in terms of solution performance and memory usage and generally remedies potential over constraint due to MPC equations.
Combined	The Mechanical APDL solver will use the optimized detection approach based on the bodies in contact.

For additional Mechanical APDL specific information, see Selecting Location of Contact Detection (specifically, **KEYOPT(4)** related information) in the *Mechanical APDL Contact Technology Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

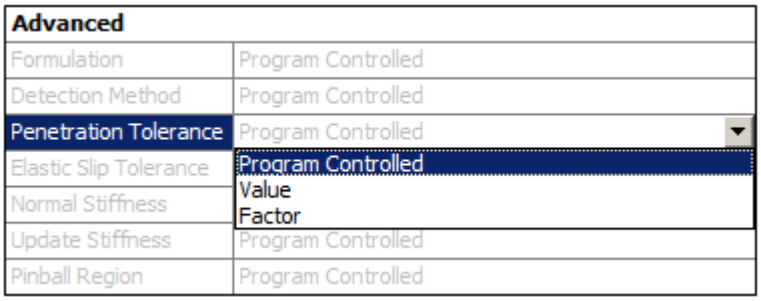
Penetration Tolerance

The **Penetration Tolerance** property enables you to specify the **Penetration Tolerance Value** or the **Penetration Tolerance Factor** for a contact when the **Formulation** property is set to **Program Controlled**, **Pure Penalty**, or **Augmented Lagrange**.

Note:

The **Update Stiffness** property must be set to either **Program Controlled**, **Each Iteration**, or **Each Iteration, Aggressive** for the **Penetration Tolerance** property to be displayed when **Formulation** is set to **Pure Penalty**.

Property options include:



Option	*Description*
Program Controlled	This is the default setting. The **Penetration Tolerance** is calculated by the program.
Value	Enter the **Penetration Tolerance Value** directly. This entry is a length measurement (foot, meter, etc.). Only non-zero positive values are valid.
Factor	Enter the **Penetration Tolerance Factor** directly. This entry must be equal to or greater than zero but must also be less than 1.0. This entry has no unit.

Penetration Tolerance Value

The **Penetration Tolerance Value** property displays when **Penetration Tolerance** is set to **Value**. You enter a Value.

Penetration Tolerance Factor

The **Penetration Tolerance Factor** property displays when **Penetration Tolerance** is set to **Factor**. You enter a Factor.

Note:

When viewing the Connections Worksheet (p. 1210), a **Value** displays as a negative number and a **Factor** displays as a positive number.

For additional information, see the Determining Contact Stiffness and Allowable Penetration, specifically Using FKN and FTOLN, section of the *Mechanical APDL Contact Technology Guide* (Surface-to-Surface Contact).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

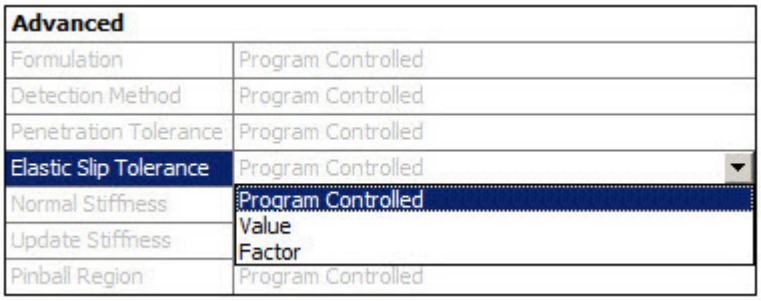
Elastic Slip Tolerance

The **Elastic Slip Tolerance** property enables you to set the allowable elastic slip value for a contact when the Formulation is set to Normal Lagrange or when the contact stiffness is set to update each iteration (**Update Stiffness** is set to **Each Iteration** or **Each Iteration, Aggressive**).

Note:

Elastic Slip Tolerance is not applicable when the contact Type is set to Frictionless or No Separation.

Property options include:



Option	*Description*
Program Controlled	This is the default setting. The **Elastic Slip Tolerance Value** is calculated by the application.
Value	Enter the **Elastic Slip Tolerance Value** directly. This entry is a length measurement (foot, meter, etc.). Only non-zero positive values are valid.
Factor	Enter the **Elastic Slip Tolerance Factor** directly. This entry must be equal to or greater than zero but must also be less than 1.0. This entry has no unit.

Elastic Slip Tolerance Value

The **Elastic Slip Tolerance Value** property displays when **Elastic Slip Tolerance** is set to **Value**. You enter a Value.

Elastic Slip Tolerance Factor

The **Elastic Slip Tolerance Factor** property displays when **Elastic Slip Tolerance** is set to **Factor**. You enter a Factor.

Note:

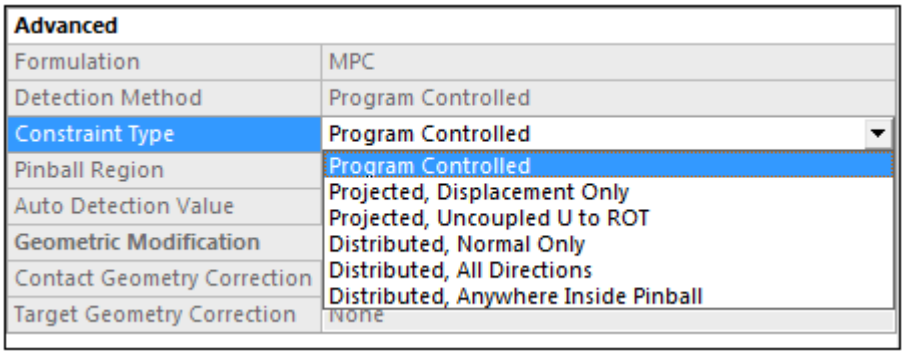
When viewing the Connections Worksheet (p. 1210), a **Value** displays as a negative number and a **Factor** displays as a positive number.

For additional information, see the Determining Contact Stiffness and Allowable Penetration, specifically Using FKT and SLTO, section of the *Mechanical APDL Contact Technology Guide* (Surface-to-Surface Contact).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Constraint Type

The **Constraint Type** property controls the type of MPC constraint to be created for bonded contact. This displays only if **Formulation** property is set to **MPC**. The property includes the following options:



Option	*Description*
Program Controlled (default)	Internally, this setting corresponds to the: • **Projected, Displacement Only** option for General Axisymmetric (p. 1167) contact and for contact between rigid and deformable bodies. • **Distributed, All Directions** option for all other contact types.
Projected, Displacement Only	Constraints are constructed to couple the translational DOFs only. Projected constraint if an intersection is found from the contact normal to the target surface.
Projected, Uncoupled U to ROT	The rotational and displacement constraints will not be coupled together. This option can model situations where the surface body edges line up well and a moment is not created from the physical surface body positions. Therefore, it is most accurate for the constraints to leave the displacements/rotations uncoupled. This provides an answer which is closer to a matching mesh solution. Using a coupled constraint causes artificial constraints to be added causing an inaccurate solution.
Distributed, Normal Only	Both translational DOFs and rotational DOFs of contact nodes and translational DOFs of target nodes are included in the constraint set in a coupled manner.
Distributed, All Directions	Represents the most common type of surface body contact. Constraints are constructed to couple the translational and rotational DOFs. In most types of surface body contact, an offset will exist. Due to this offset there will be a moment created. To get the correct moment, the rotation/displacement DOF's must be coupled together. If the program cannot detect any contact in the target normal direction, it will then search anywhere inside the pinball for contact.
Distributed, Anywhere Inside Pinball	Constraints are coupled and created anywhere to be found inside the pinball region. Therefore, the pinball size is important as a larger pinball will result in a larger constraint set. This option is useful when you wish to fully constrain one contact side completely to another.

For additional information, see the **Controlling Degrees of Freedom Used in the MPC Constraint** topic in the Modeling Solid-Solid and Shell-Shell Assemblies section of the *Mechanical APDL Contact*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Technology Guide. Also note that the Mechanical APDL entry for the Constraint Type is KEYOPT(5) for element TARGE170.

Normal Stiffness

Defines a contact **Normal Stiffness** factor. Property options include:

Option	*Description*
Program Controlled	This is the default setting. The Normal Stiffness Factor is calculated by the program. If only **Bonded** or **No Separation** contact exists, the value is set to 10. If any other type of contact exists, all of the program controlled regions (including **Bonded** or **No Separation**) will use the Mechanical APDL default (Real Constant FKN).
Factor	You enter the **Normal Stiffness Factor** (see below). This is a unit-less entry.
Absolute Value	You enter the **Normal Stiffness Value** (see below).

Normal Stiffness Factor

This property appears when the Normal Stiffness property is set to **Factor**. It enables you to specify the **Normal Stiffness Factor**. Only non-zero positive values are supported. The usual factor range is from **0.01** - **10**The default value is selected by the application. A smaller value provides for easier convergence but with more penetration. The default value is appropriate for bulk deformation. If bending deformation dominates, use a smaller value (**0.01** - **0.1**).

For additional information specific to Mechanical APDL, see the following sections:

- Determining Contact Stiffness and Allowable Penetration section of the *Mechanical APDL Contact Technology Guide* (Surface-to-Surface Contact).
- Using FKN and FTOLN section of the *Mechanical APDL Contact Technology Guide* (Surface-to-Surface Contact).

Normal Stiffness Value

This property appears when the **Normal Stiffness** property is set to **Absolute Value**. It enables you to specify the **Normal Stiffness Value**. The application supports positive values only. The Units for this value are based on the types of contact involved. For a traction based model, the application uses Force/Volume (for example, N/m^3) and for a force-based model, the application uses Force/Length (for example, N/m). A force-based model is used for face-to-edge contacts and edge-to-edge (not including line bodies).

Update Stiffness

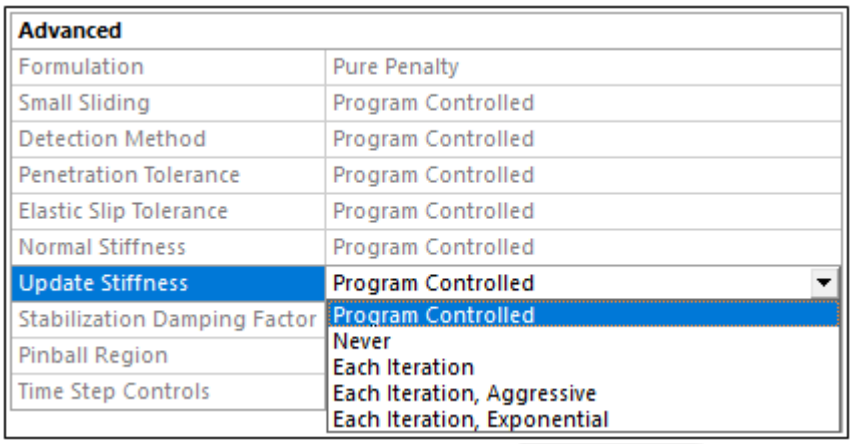
This property enables you to specify if the program should update (change) the contact stiffness during the solution. If you choose any of these stiffness update settings, the application modifies the stiffness (raise/lower/leave unchanged) based on the physics of the model (that is, the underlying element stress and penetration). To use the options of this property, you need to set the **Formulation** property to either **Augmented Lagrange** or **Pure Penalty**, the two formulations where contact stiffness is applicable. For the **Each Iteration, Exponential** option, the **Formulation** property must be set to **Pure Penalty**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

An advantage of specifying an Update Stiffness setting is that stiffness is automatically determined that allows both convergence and minimal penetration. Also, if this setting is used, problems may converge in a Newton-Raphson sense, that would not otherwise.

You can use a **Result Tracker** (p. 1844) to monitor a changing contact stiffness throughout the solution.

Property options are illustrated and described below.



Option	*Description*
Program Controlled	This is the default setting. The application sets the property to **Never** for contacts between two rigid bodies and to **Each Iteration** for all other cases. You can change the default using the Options (p. 217) dialog.
Never	This is the default setting. Turns off the program's automatic Update Stiffness feature.
Each Iteration	Sets the program to update stiffness at the end of each equilibrium iteration. This choice is recommended if you are unsure of a **Normal Stiffness Factor** to use in order to obtain good results.
Each Iteration, Aggressive	Sets the program to update stiffness at the end of each equilibrium iteration, but compared to the **Each Iteration**, this option allows for a more aggressive changing of the value range.
Each Iteration, Exponential	This option requires the **Type** property to be set to either **Frictional** or **Frictionless** and the **Formulation** property to **Pure Penalty**. When selected, the **Pressure at Zero Penetration** and the **Initial Clearance** properties display. This option updates stiffness using an exponential pressure-penetration relationship. For detailed information about this option, see the Exponential Pressure-Penetration Relationship (KEYOPT(6) = 3) topic (of the *Set the Real Constants and Element KEYOPTS* section) in the Mechanical APDL Contact Technology Guide.

Electric Capacitance

This property controls the electric contact capacitance value used in an electric contact simulation. Property options are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

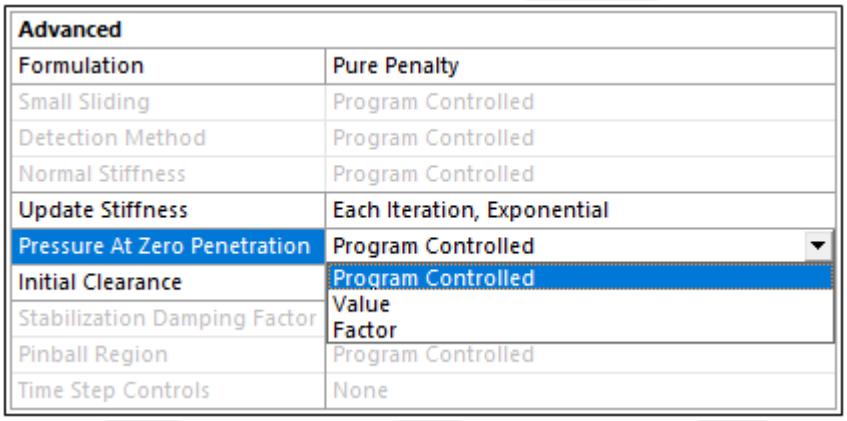
Option	Description
Program Controlled	This is the default setting. Using this setting, the application calculates the value for the electric contact capacitance. The value is calculated based on the maximum value of the average of permittivity and the size of the model. For orthotropic materials, the application only considers Relative Permittivity in the X direction.
Manual	You use this setting to specify an electric capacitance value.

Electric Capacitance Value

This property displays when you set the **Electric Capacitance** property to **Manual**. You use this property to specify a **Electric Capacitance Value** (positive values only).

Pressure at Zero Penetration

This property corresponds to the PZER real constant used in the Mechanical APDL application. It defines the pressure when there is zero penetration between Contact and Target geometries. See the Exponential Pressure-Penetration Relationship (KEYOPT(6) = 3) topic (of the *Set the Real Constants and Element KEYOPTS* section) in the Mechanical APDL Contact Technology Guide for a detailed description of this property and its function. Property options are illustrated and described below.



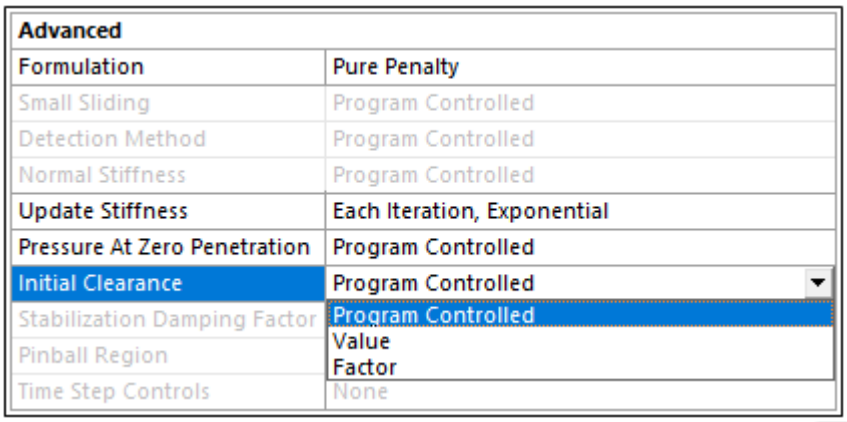
Option	*Description*	Mechanical APDL Reference
Program Controlled	This is the default setting. The application automatically calculates and selects default pressure values.	KEYOPT(6) = 3
Value	Using this option, you can manually specify a pressure value. This entry has a unit of measure for pressure (Pa, etc.). Only non-zero positive values are valid.	
Factor	Using this option, you can manually specify a factor of the default pressure value. This entry must be equal to or greater than zero but must also be less than 1.0. This entry has no unit.	

Initial Clearance

This property corresponds to the CZER real constant used in the Mechanical APDL application. It defines the initial clearance or gap at which the contact pressure begins to act on the Contact and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Target geometries. See the Exponential Pressure-Penetration Relationship (KEYOPT(6) = 3) topic (of the *Set the Real Constants and Element KEYOPTS* section) in the Mechanical APDL Contact Technology Guide for a detailed description of this property and its function. Property options are illustrated and described below.



Option	*Description*	*Mechanical APDL Reference*
Program Controlled	This is the default setting. The application automatically calculates and selects default clearance values.	KEYOPT(6) = 3
Value	Using this option, you can manually specify a clearance value. This entry is a length measurement (meter, etc.). Only non-zero positive values are valid.	
Factor	Using this option, you can manually specify a factor of the default clearance value. This entry must be equal to or greater than zero but must also be less than 1.0. This entry has no unit.	

Stabilization Damping Factor

A contact you define may initially have a near open status due to small gaps between the element meshes or between the integration points of the contact and target elements. The contact will not get detected during the analysis and can cause a rigid body motion of the bodies defined in the contact. The stabilization damping factor provides a certain resistance to damp the relative motion between the contacting surfaces and prevents rigid body motion. This contact damping factor is applied in the contact normal direction and it is valid only for the **Frictionless**, **Rough**, and **Frictional** options of the **Type** property. The damping is applied to each load step where the contact status is open. The value of the stabilization damping factor should be large enough to prevent rigid body motion but small enough to ensure a solution. A value of **1** is usually appropriate. Property options are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option	*Description*	**Mechanical APDL Reference**
Stabilization Damping Factor	If the value provided for the damping factor is: • **0** (default value), damping is activated for the first load step only. The solver uses a default value of **1.0** as the **Stabilization Damping Factor**. • Non-zero, then damping is always activated regardless of the contact status of the previous substeps. The solver uses (KEYOPT(15) = 3) to apply stabilization damping.	Real Constant FDMN

Thermal Conductance

Controls the thermal contact conductance value used in a thermal contact simulation. Property options are described below.

Option	*Description*
Program Controlled	This is the default setting. The program will calculate the value for the thermal contact conductance. The value will be set to a sufficiently high enough value (based on the thermal conductivities and the model size) to model perfect contact with minimal thermal resistance. Note that the **Program Controlled** option is not valid for composite materials. For orthotropic materials, the application only considers Thermal Conductivity in the X direction.
Manual	The **Thermal Conductance Value** is input directly by the user.

Thermal Conductance Value

This property displays when the **Thermal Conductance** property is set to **Manual**. Enables entry of the Thermal Conductance Value. The property only supports positive values and can be specified as a parameter. The Units for this value are based on the types of contact involved. For 3D faces and 2D edges, the units are HEAT/(TIME * TEMPERATURE* AREA). For contact between 3D edges and vertices, the units are HEAT/(TIME * TEMPERATURE) with the value applied to every node in the contact side. For more information about the units used for thermal contact conductance coefficient, see Table 78 (p. 1920) and Table 79 (p. 1920) in the Solving Units (p. 1874) section.

For additional Mechanical APDL specific information, see the Modeling Thermal Contact, specifically Modeling Conduction>Using TCC, section of the *Mechanical APDL Contact Technology Guide* (Multiphysics Contact).

Pinball Region

This option enables you to specify the contact search size, commonly referred to as the **Pinball Region**. Setting a pinball region can be useful in cases where initially, bodies are far enough away from one another that, by default, the application does not detect that they are in contact. In this case, you can increase the pinball region. For example, you could have a surface body that was generated by offsetting a face of a solid body. Depending upon the thickness, this could leave a large gap. Or, for a large deflection problem, a considerable pinball region to compensate for the possibility of large amounts of over penetration. In general, if you want two regions to be bonded

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

together that may be far apart, you should specify a pinball region that is large enough to ensure that contact occurs.

However, for the **Bonded** and **No Separation** contact types (p. 1234), you must be careful when specifying a large **Pinball Region**. For these types of contact, the application considers any region found within the pinball region to be in contact. For the other contact types, this consideration is not as critical because the application performs additional calculations to determine if the two bodies are truly in contact. The pinball region defines the searching range where these calculations will occur. In addition, a large gap can transmit fictitious moments across the boundary.

Property options include:

Option	*Description*
Program Controlled	This is the default setting. The application calculates the **Pinball Region** value.
Auto Detection Value	This option is only available for automatically generated contacts. The pinball region will be equal to the tolerance value used in generating the contacts. The value is displayed as read-only in the **Auto Detection Value** field. **Auto Detection Value** is the recommended option for cases where the automatic contact detection region is larger than a **Program Controlled** region. In such cases, some contact pairs that were detected automatically may not be considered in contact for a solution.
Radius	When selected, the **Pinball Radius** property displays. You use this property to manually specify a radius value.

Rigid Dynamics Solver

For the Rigid Dynamics solver, the **Pinball Region** property is used to control the touching tolerance. By default, the Rigid Dynamics solver automatically computes the touching tolerance using the sizes of the surfaces in the contact region. These default values are sufficient in most of cases, but inadequate touching tolerance may arise in cases where contact surfaces are especially large or small (small fillet for instance). In such cases, the value of the touching tolerance can be directly specified using the following properties:

Option	*Description*
Program Controlled (default)	The touching tolerance is automatically computed by the Rigid Body Dynamics solver from the sizes of the contact surfaces.
Radius	The value of the touching tolerance is directly given by user.

Pinball Radius

The numerical value for the **Pinball Radius**. This property is displayed only if **Pinball Region** is set to **Radius**.

Electric Conductance

Controls the electric contact conductance value used in an electric contact simulation. Property options are described below.

Option	*Description*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Program Controlled	This is the default setting. The program will calculate the value for the electric contact conductance. The value will be set to a sufficiently high enough value (based on the electric conductivities and the model size) to model perfect contact with minimal electric resistance.
Manual	The Electric Conductance value is input directly by the user.

Note:

The Electric Analysis result, Joule Heat (p. 2162), when generated by nonzero contact resistance is not supported.

Electric Conductance Value

Allows input of the Electric Conductance value (in units of electric conductance per area). Only positive values are allowed. This choice is displayed only if **Manual** is specified for **Electric Conductance**.

Time Step Controls

Allows you to specify if changes in contact behavior should control automatic time stepping. This choice is displayed only for nonlinear contact (**Type** is set to **Frictionless**, **Rough**, or **Frictional**). Property options are described below.

Option	*Description*
None	This is the default setting. Contact behavior does not control automatic time stepping. This option is appropriate for most analyses when automatic time stepping is activated and a small time step size is allowed.
Automatic Bisection	Contact behavior is reviewed at the end of each substep to determine whether excessive penetration or drastic changes in contact status have occurred. If so, the substep is reevaluated using a time increment that is bisected (reduced by half).
Predict for Impact	Performs same bisection on the basis of contact as the **Automatic Bisection** option and also predicts the minimal time increment needed to detect changes in contact behavior. This option is recommended if you anticipate impact in the analysis.
Use Impact Constraints	Activates impact constraints with automatic adjustment of the time increment. This option includes constraints on penetration and relative velocity to more accurately predict the duration of impact and the rebound velocities after separation.

Restitution Factor—Rigid Dynamics Solver Only

For the Ansys Rigid Dynamics solver, **Restitution Value** is a property in the **Advanced** group. This value represents the energy lost during shock. It is defined as the ratio between relative velocity prior to the shock and the velocity after the shock. This value can be between 0 and 1. A **Restitution Factor** equal to 1 indicates that no energy is lost during the shock. That is, the rebounding velocity equals the impact velocity (a perfectly elastic collision). The default value is **1**.

Continuous Distance Computation—Rigid Dynamics Solver Only

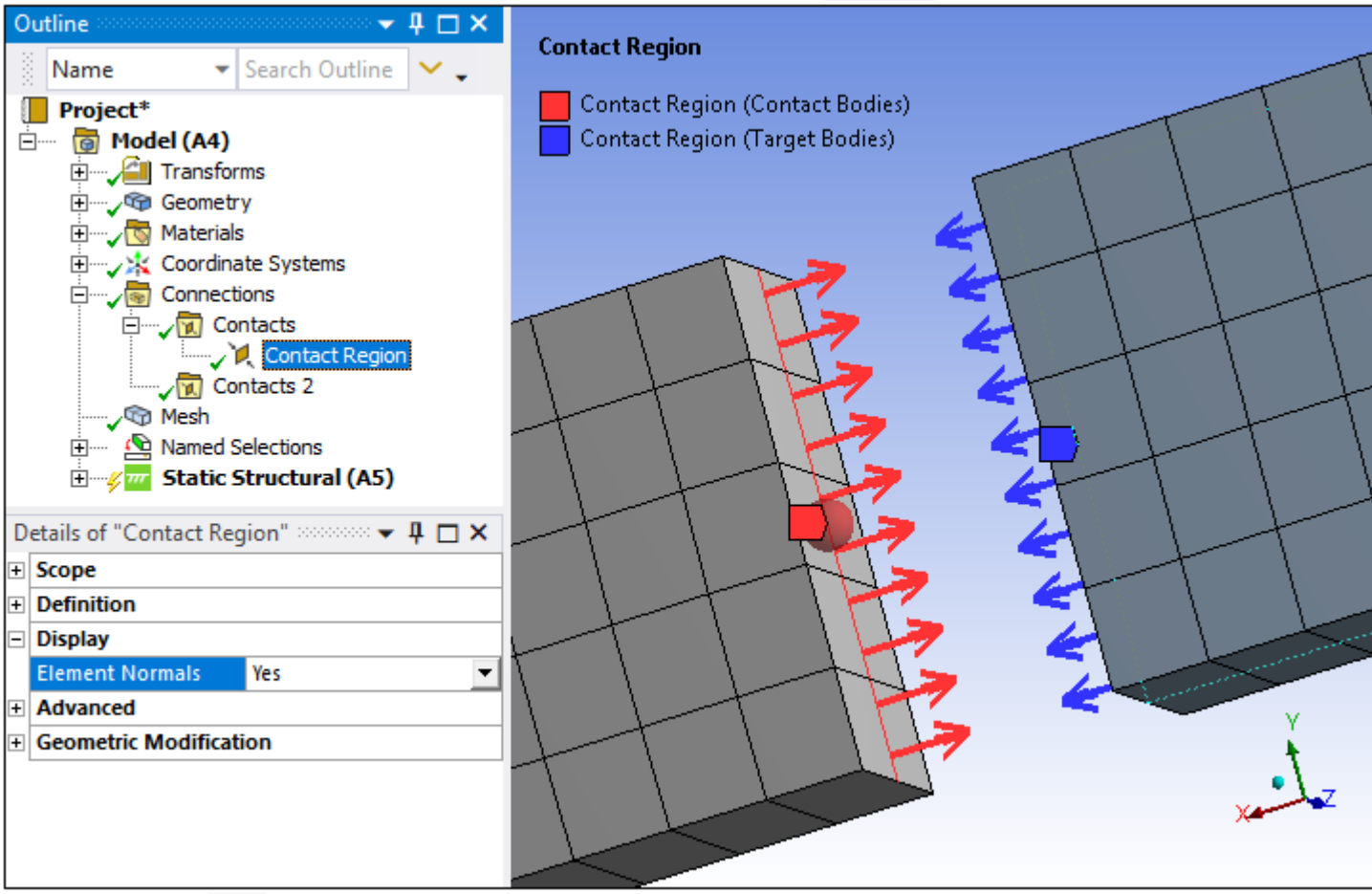
For the Ansys Rigid Dynamics solver, **Continuous Distance Computation** is an additional property in the **Advanced** group. This property continuously computes the contact distance during the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

simulation. By default, the distance is only computed when the contact bodies are significantly close in contact. However, it can force distance computation, even when contact bodies are distant. Enabling **Continuous Distance Computation** is mandatory in order to monitor the distance between contact bodies (see Contact Distance Probes (p. 2126) for further information). The **Continuous Distance Computation** property can be defined as **Program Controlled**. Select either **Yes** to enable distance computation or **No** to disable it. The default value is **Program Controlled**, which disables the distance computation.

Display

The **Display** category is available for edge contacts on 2D surface bodies and face contacts on 3D bodies. This category contains the property **Element Normals**. The options for this property are **No** (default) and **Yes**. Setting the feature to **Yes** enables you to see the normal direction of each element in the **Contact Region**. As needed, you can invert the normal direction using the Flip Contact Normals and/or the Flip Target Normals (p. 1263) properties in the **Geometric Modification** category for 2D edges, or the Reverse Shared Contact Normal/Reverse Shared Target Normal (p. 1228) properties in the **Scope** category for shared 2D edges and 3D faces.



Important:

- If your analysis includes 2D surface bodies in contact and the edge normal directions do not adhere to the right-hand rule, that is, the normals appear to be in the wrong

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

direction, you can use the **Flip Contact Normals** and/or the **Flip Target Normals** properties to reorient the edge directions.

- If a **Contact Region** is defined by nodal scoping, element normals may not be visible for contact elements.
- When a **Contact Region** is scoped to shell faces and the **Contact Shell Face** or the **Target Shell Face** properties are set to **Program Controlled**, the application does not draw the arrows for the element normals (even if the **Element Normals** property is set to **Yes**).

Geometric Modification

The **Geometric Modification** category provides the features described below. As described, this category only displays when certain contact conditions are detected by the application and/or certain property definitions are specified.

- Interface Treatment (p. 1257)
- Contact/Target Geometry Correction (p. 1260)
- Flip Contact/Target Normals (p. 1263)

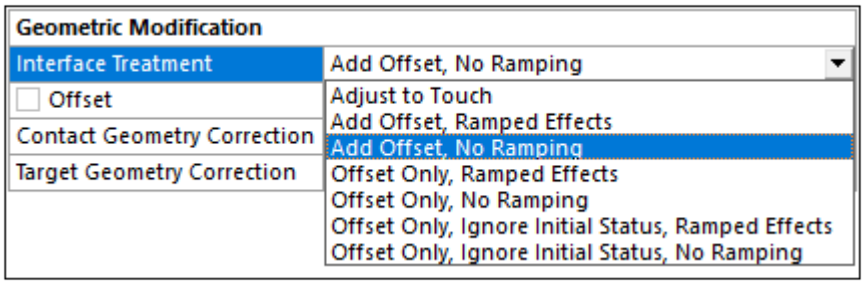
Interface Treatment

The **Interface Treatment** property defines how the contact interface of a contact pair is treated. The property becomes visible when you set the contact **Type** property to either **Frictionless**, **Rough** or **Frictional** (nonlinear contact). For a given contact interface (**Contact Region**) that includes either gaps or penetrations, either locally or uniformly along the overall surface, contact adjustments, such as closing the gaps, removing the penetrations, or shifting the contact surface by an Offset may produce a better setup for initial contact.

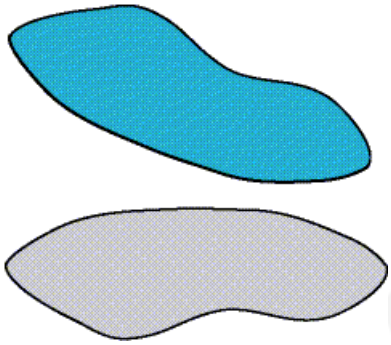
To see the initial contact, you can insert a Contact Tool (p. 2079) under the **Connections** folder. An Initial Information (p. 2084) object is automatically inserted under the **Contact Tool**. Using the context (right-click) menu this object, select the **Generate Initial Contact Results** option. This option produces a table of information for all Contact Regions. The **Status** column indicates the contact interface as **Near Open**, **Closed**, or **Far Open**. Using this information, your selection for the **Interface Treatment** property can improve the contact status from **Far Open** or **Near Open** to **Closed**. Having a **Contact Region** in **Closed** status at the beginning of the analysis may improve the convergence of the model.

When active, the **Interface Treatment** property provides the options illustrated here. Each option is described below.

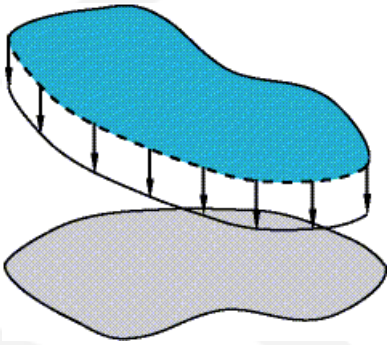
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



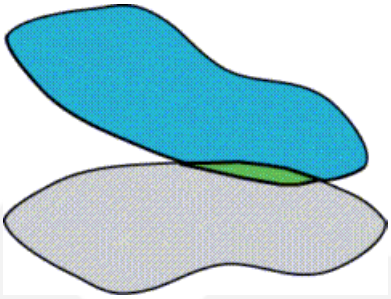
- **Adjust to Touch**: Any initial gaps are closed and any initial penetration is ignored creating an initial stress free state. Contact pairs are "just touching" as illustrated.



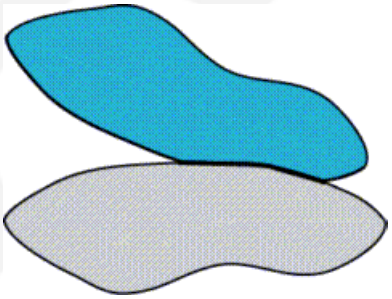
Contact pair *before* any **Interface Treatment**. Gap exists.



Contact pair after **Adjust to Touch** treatment. Gap is closed automatically. Pair is "just touching".



Contact pair before any **Interface Treatment**. Penetration exists.



Contact pair after **Adjust to Touch** treatment. Pair touches at interface.

This setting is useful to make sure initial contact occurs even if any gaps are present (as long as they are within the pinball region). Without using this setting, the bodies may fly apart if any initial gaps exist. Although any initial gaps are ignored, gaps can still form during loading for the nonlinear contact types. For nonlinear contact types (**Frictionless**, **Rough**, and **Frictional**), **Interface Treatment** is displayed where the choices are **Adjust to Touch**, **Add Offset**, **Ramped Effects**, and **Add Offset**, **No Ramping**.

Note:

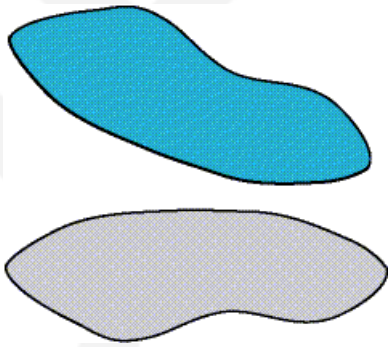
– The **Adjust to Touch** option does not lead to uniform contact between concentric cylinders that contain a small initial gap. For this situation, manually specify the gap as an offset in with the **Add Offset, No Ramping** option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

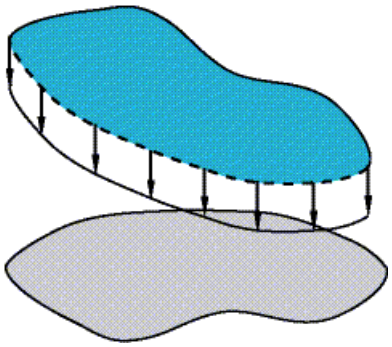
Published: 2023-10-23

– Gaps may still be present if the contact pair has regions of differing gaps. When gaps exist, this setting makes sure that initial contact occurs by closing the smallest gap.

- **Add Offset, Ramped Effects**: Models the true contact gap/penetration plus adds in any user defined offset values. This setting is the closest to the default contact setting used in the Mechanical APDL except that the loading is ramped. Using this setting will not close gaps. Even a slight gap may cause bodies to fly apart. Should this occur, use a small contact offset to bring the bodies into initial contact. Note that this setting is displayed only for nonlinear contact and the ramping occurs over the first load step.
- **Add Offset, No Ramping**: This is the default setting. This option is the same as **Add Offset, Ramped Effects** but loading is not ramped.
- **Offset Only, Ramped Effects**: Using this option, the application ignores any initial geometric penetrations if the initial status is in contact. If the initial status is near field, the application ignores the calculated initial penetration (**Offset** plus geometric gap), and applies only the remaining **Offset** value (remaining from **Offset** plus geometric gap calculation). All loading for this option is ramped.
- **Offset Only, No Ramping**: This option performs the same actions as the **Offset Only, Ramped Effects** options except that loading is step-applied.
- **Offset Only, Ignore Initial Status, Ramped Effects**: This option is the same as the **Offset Only, Ramped Effects** option except that this option ignores any initial geometric penetration or gap when calculating contact regardless of the initial contact status, before applying a uniform offset.
- **Offset Only, Ignore Initial Status, No Ramping**: This option performs the same actions as the **Offset Only, Ignore Initial Status, Ramped Effects** options except that loading is step-applied.
- **Offset**: This property displays for all of the options of the **Interface Treatment** property except **Adjust to Touch**. This property defines the contact offset value. A positive value moves the contacts closer together (increase penetration/reduce gap) and a negative value moves the contacts further apart.



Contact pair *before* any **Interface Treatment**. Gap exists.



Contact pair after **Add Offset** treatment (either option). Gap is closed "manually" based on value entered for **Offset** (positive value shown that includes some penetration).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Contact/Target Geometry Correction

The **Contact Geometry Correction** and the **Target Geometry Correction** properties both have the default setting **None** as well as the following two options for geometry correction.

- Smoothing (p. 1260)
- Bolt Thread (p. 1261)

Smoothing

For Contact and/or Target geometries, the **Smoothing** option enables you to improve the accuracy of circular edges (2D) and spherical or revolute surfaces (3D) by evaluating the contact detection based on the exact geometry instead of the mesh. This feature enables curved geometries to be analyzed more effectively using meshes with dropped midside nodes. For additional technical information about this feature, see the Surface Smoothing section of the *Mechanical APDL Contact Technology Guide*.

Note:

- The Behavior (p. 1236) properties **Symmetric** and **Auto-Asymmetric** are not supported.
- To avoid having the application incorrectly modify the geometry of a contact surface that has drastically deformed, do not use the **Smoothing** feature during an analysis that has specified Large Deflection (p. 1435).

The following properties become visible when the **Contact Geometry Correction** property and/or the **Target Geometry Correction** property are set to **Smoothing**.

Orientation

Property options include:

- The **Program Controlled** option is only valid when the contact scoping is on a sphere or a single, simple cylinder. The option becomes invalid when the specified scoping:
  - Includes more than one face.
  - Is on a single cylinder face with two edges, but the CAD package does not identify it as a cylinder. In this case, you can confirm your model's geometric information using the Selection Information window (p. 180).

  If the **Program Controlled** option is not valid, you can use the **Revolute Axis** option to manually identify the cylinder.

  Note:

  The **Program Controlled** option includes cylindrical endpoint approximation if the selected cylinder face includes more than two edges. These approximated cylinder endpoints are sent with the **SECDATA** command. See the Geometry Cor-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

rection for Contact and Target Surfaces section in the *Mechanical APDL Contact Technology Guide* for more information.

Note:

Make sure the cylinder face only contains edges at the ends of the cylinder. The application issues a warning if the cylinder edges are out of the calculated tolerance limit. In such cases, you can use the **Revolute Axis** option to manually identify the cylinder endpoints.

- **Sphere Center Point** (3D) or **Circle Center Point** (2D): When one of these properties is selected, the **Center Point** property also displays. The **Center Point** property provides a drop-down list of available coordinate systems, Global as well as user-defined. As needed, create a local coordinate system (p. 1191) at the center of your sphere.
- **Revolute Axis**: You use this option for 3D revolute surfaces, such as a cone or a cylinder. The definition of the revolute surface is computed using an internal axis definition. When **Revolute Axis** is selected, the following additional properties display. These properties define the coordinate systems that are used to define the axis around which the contact smoothing is oriented.
  - **Starting Point**
  - **Ending Point**

Bolt Thread

This geometry correction option enables you to model bolt threads. For a 2D axisymmetric model, only edge-to-edge scoping is supported. For a 3D model, only face-to-face scoping is supported. For additional technical information about this feature, see the Simplified Bolt Thread Modeling section of the *Mechanical APDL Contact Technology Guide*.

Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	Bolt Thread
--Orientation	Program Controlled
--Mean Pitch Diameter	5. mm
--Pitch Distance	2. mm
--Thread Angle	60. °
--Thread Type	Single-Thread
--Handedness	Right-Handed

Tip:

When you specify the **Bolt Thread** option, it is strongly recommended that you have a refined mesh. See the Sizing Group (Category) sections of the *Meshing User's Guide* for additional information about mesh refinement.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Support Requirements

In order to use the **Bolt Thread** option, note the following.

- For 3D models, when the **Orientation** property is set to **Program Controlled**, the edges specified for the **Contact** and **Target** properties of the selected cylinders must be circular edges.
- The **Contact Geometry Correction** property is available for all contact **Type** settings *except* for **Bonded**.
- The Behavior (p. 1236) properties **Symmetric** and **Auto-Asymmetric** are not supported.
- It is recommended that you do **not** set the Detection Method (p. 1244) to either **Nodal-Normal To Target** or **On Gauss Point**.
- The diameter of the solid model (bolt and nut bodies) should be based on the major diameter (d).

Property Descriptions

The following properties are visible when **Contact/Target Geometry Correction** is set to **Bolt Thread**.

Orientation

Property options include:

- **Program Controlled** (default): A contact condition with **Contact/Target Geometry Correction** property defined as **Bolt Thread**, is fully defined only when cylindrical contact conditions are detected by the application, otherwise, manual specifications are required.
- **Revolute Axis**: when **Revolute Axis** is selected, the following additional properties display. These properties define the coordinate systems that are used to generate the axis around which the bolt is oriented. They do not correspond to the starting and ending point of the bolt threads.
  - **Starting Point**
  - **Ending Point**

Mean Pitch Diameter

This property defines the average diameter of the threaded bolt.

Pitch Distance

This property defines the length of the thread pitch.

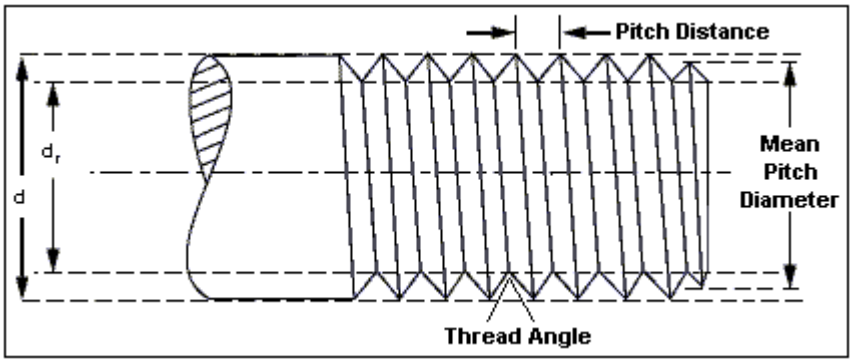
Thread Angle

This property defines the angle of the thread's inclination.

The following diagram illustrates the **Mean Pitch Diameter**, **Pitch Distance**, and **Thread Angle**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Thread Type

This property defines the number of threads on the bolt. Property options include:

- **Single-Thread**
- **Double-Thread**
- **Triple-Thread**

Handedness

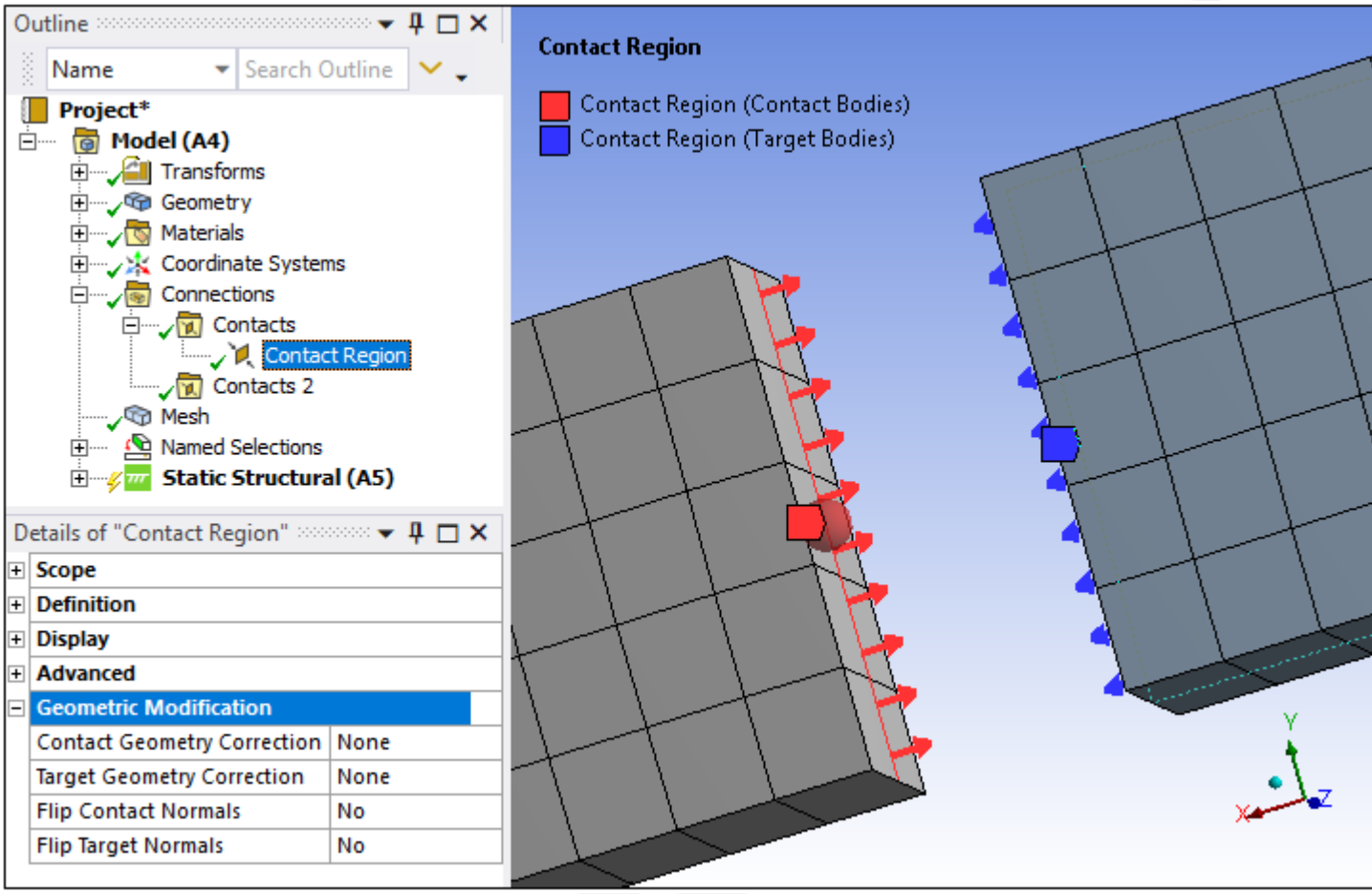
This property defines the bolt as either right or left handed. Property options include:

- **Right-Handed**
- **Left-Handed**

Flip Contact/Target Normals

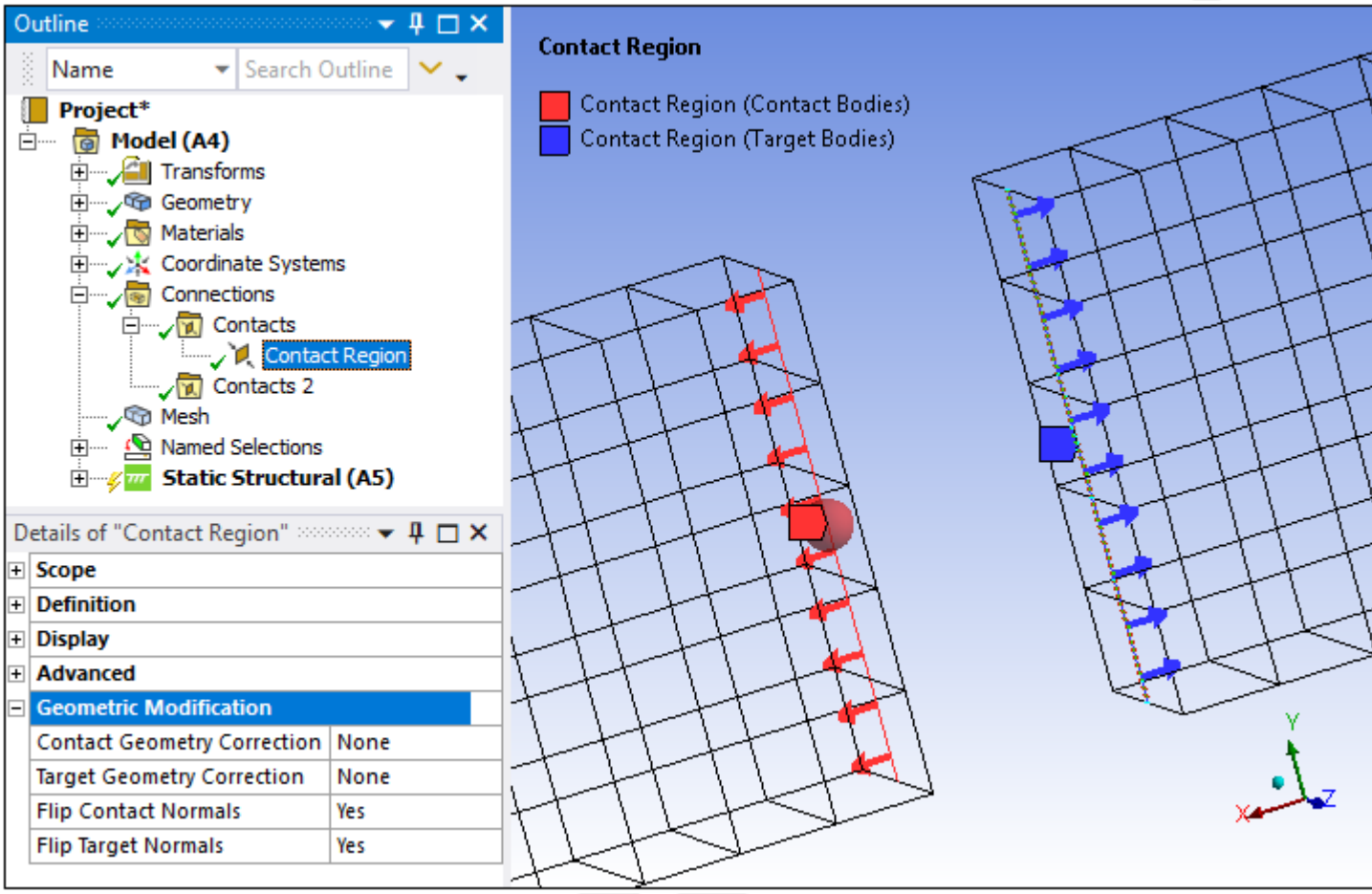
The **Flip Contact Normals** and the **Flip Target Normals** properties enable you to invert the normal direction of edge contacts on 2D surface bodies. You use the Element Normals (p. 1256) property of the **Display** category in combination with these properties to display the normal directions of the edge elements in contact and, as needed, invert the direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Supported Contact Types

The following table identifies the supported formulations and whether symmetry is respected for the various contact geometries.

Contact Geometry	**Face** (**Scope** = **Contact**)	**Edge** (**Scope** = **Contact**)	**Vertex** (Line Bodies Only) (**Scope** = **Contact**)
Face (**Scope** = **Target**)	Symmetry Respected: Yes Contact Element: CONTA171	Symmetry Respected: No Contact Element: CONTA175	Symmetry Respected: No Contact Element: CONTA175
Edge (see notes below) (**Scope** = **Target**)	Not Supported for solving. Contact Element: CONTA175	Symmetry Respected: No Contact Element: CONTA175 (3D) and CONTA172 (2D)	Symmetry Respected: No Contact Element: CONTA175 **Note:** You can scope an edge of a solid or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

			surface body to the vertex of a line body. However, you cannot scope the edge of a line body to the vertex of another line body.
Vertex (**Scope** = **Target**)	Not Supported for solving.	Not Supported for solving.	Not Supported for solving.

Note:

In 2D analyses, only edge-edge contact is supported (the equivalent of 3D face-face contact).

Beam-to-Beam (Edge-Edge) Contact Requirements and Restrictions

Note the following requirements for beam-to-beam (edge-edge) contact:

- Beam-to-Beam (Edge-Edge) contact is supported by structural analyses only (static, transient, modal, harmonic, etc.).
- The application uses contact element CONTA177.
- During contact detection, the contact and target surface are assumed to be the surface of a cylinder. The application automatically calculates the equivalent circular radius based on the associated geometry of the underlying elements.
- Any defined beam offsets on the line bodies are not taken into account for the contact calculations.

Setting Contact Conditions Manually

Manual contact regions represent contact over the entire extent of the contact scope, for example, faces of the contact region.

Automatic contact regions represent contact only to the extent of the scope where the corresponding bodies initially are close to one another. For automatic contact, the contact elements are "trimmed" before solution. The trimming is based on the detection tolerance (p. 1218). The tighter the tolerance, the less number of generated contact elements. Note that if you set **Large Deflection** effects to **On** in the Details view of a Solution object, no trimming will be done due to the possibility of large sliding.

Valid reasons to manually change or add/delete contact regions include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Modeling "large sliding" contact. Contact regions created through auto-detection assume "assembly contact," placing contact faces very near to one another. Manual contact encompasses the entire scope so sliding is better captured. In this case, you may need to add additional contact faces.
- Auto-detection creates more contact pairs than are necessary. In this case, you can delete the unnecessary contact regions.
- Auto-detection may not create contact regions necessary for your analysis. In this case, you must add additional contact regions.

You can set contact conditions manually, rather than (or in addition to) letting the application automatically detect contact regions.

Within a source or target region, the underlying geometry must be of the same geometry type (for example, all surface body faces, all solid body faces). The source and target can be of different geometry types, but within itself, a source must be of the same geometry type, and a target must be of the same geometry type.

To set contact regions manually:

1. Select the **Connections** object in the Outline (p. 105).
2. Right-click and select **Insert** > **Manual Contact Region**. You can also select the **Contact** option on the Context tab.
3. A **Contact Region** item appears in the Outline. Click that item, and under the Details Pane (p. 119), specify the **Contact** and **Target** regions (faces or edges) and the contact type. See the Contact and Target topics in the Scope Settings (p. 1227) section for additional Contact Region scoping restrictions.

Contact Ease of Use Features

The following features are intended to assist you in performing simulations involving contact:

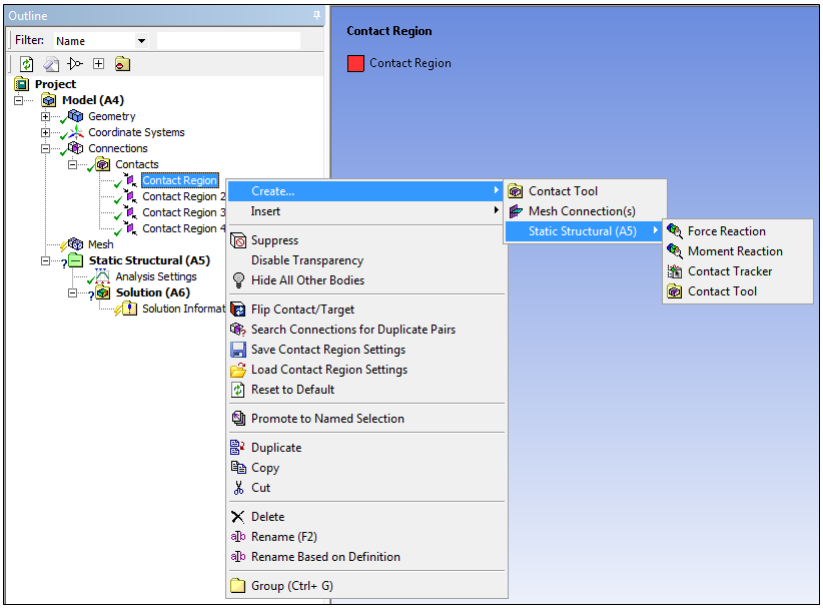
Automatically Generate Objects Scoped to Contact Regions
Controlling Transparency for Contact Regions
Displaying Contact Bodies with Different Colors
Displaying Contact Bodies in Separate Windows
Hiding Bodies Not Scoped to a Contact Region
Renaming Contact Regions Based on Geometry Names
Identifying Contact Regions for a Body
Create Contact Debonding
Flipping Contact and Target Scope Settings
Setting Default APDL Names
Merging Contact Regions That Share Geometry
Saving or Loading Contact Region Settings
Resetting Contact Regions to Default Settings
Locating Bodies Without Contact

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Locating Parts Without Contact

Automatically Generate Objects Scoped to Contact Regions

When you select a Contact Region object (or multiple Contact Region objects) in the tree, the context menu contains the option **Create**, as illustrated below. This option enables you to generate specific contact objects, results, and automatically scope the objects to the selected **Contact Region** or regions.



The **Create** context menu options include:

Contact Tool

This option enables you to create a **Contact Tool** under the **Connections** folder. The application automatically scopes the tool to the selected Contact Region(s) only.

Mesh Connection

This option enables you to create **Mesh Connection** objects from **Contact Regions**. The application scopes the new **Mesh Connection** objects to the geometries of the Contact Region(s) and sets the tolerance to be equal to the trim tolerance of the contact region. The Mesh connections are added into a new **Mesh Connections Group** folder.

Environment Submenu

The **Create** option provides a submenu of options to create specific result objects for environments that include the options shown below. The application inserts the corresponding object under the **Solution** folder for the selected environment and automatically scopes the object to the **Contact Region** (or Contact Regions).

- Force Reaction (p. 2107)
- Moment Reaction (p. 2107)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

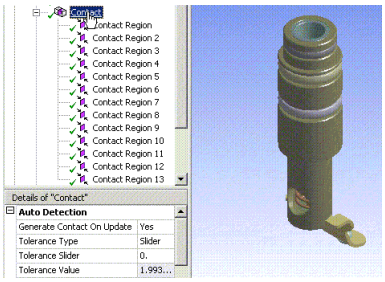
Published: 2023-10-23

- Contact Tracker (p. 1848)
- Contact Tool (p. 2079)

Controlling Transparency for Contact Regions

As shown below, you can graphically highlight an individual contact region.

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.

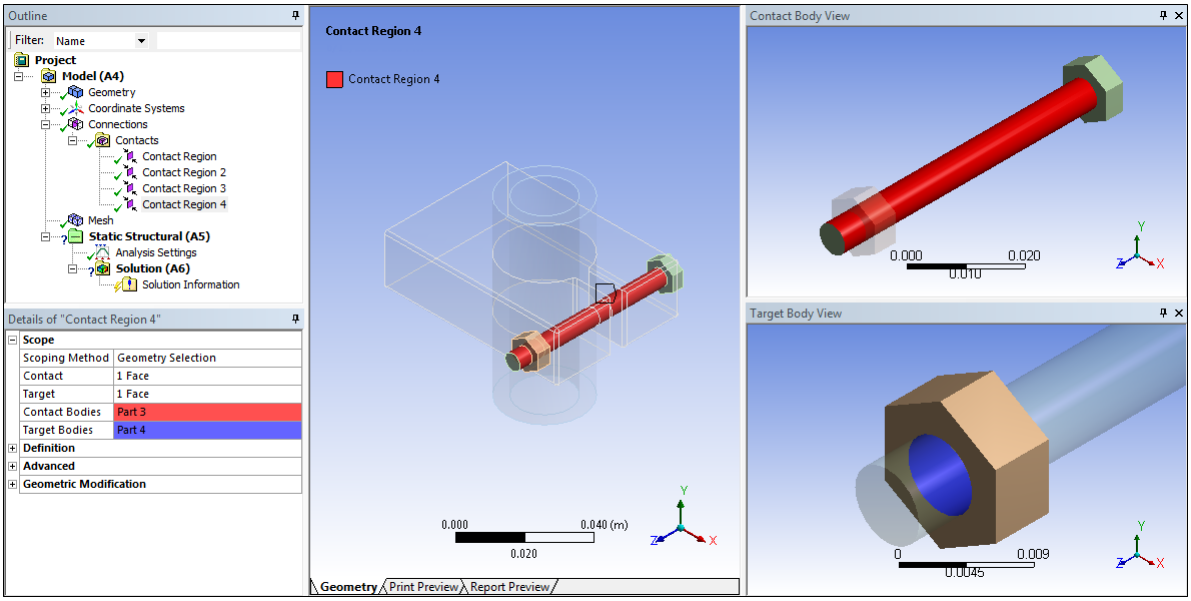


- Select a contact region to highlight the bodies in that region.
- Highlighting is due to internal transparency settings:
  - Transparency is set to 0.8 for bodies in *selected* contact region.
  - Transparency is set to 0.1 for bodies not in selected contact region(s).
  - You can change the default transparency values in using the Connections (p. 219) settings of the **Options** dialog box.
- You can disable the contact region highlighting feature in either the Details view of a contact group branch, or by accessing the context menu (right mouse click) on a contact region or contact group branch of the tree, and choosing **Disable Transparency**.

Displaying Contact Bodies with Different Colors

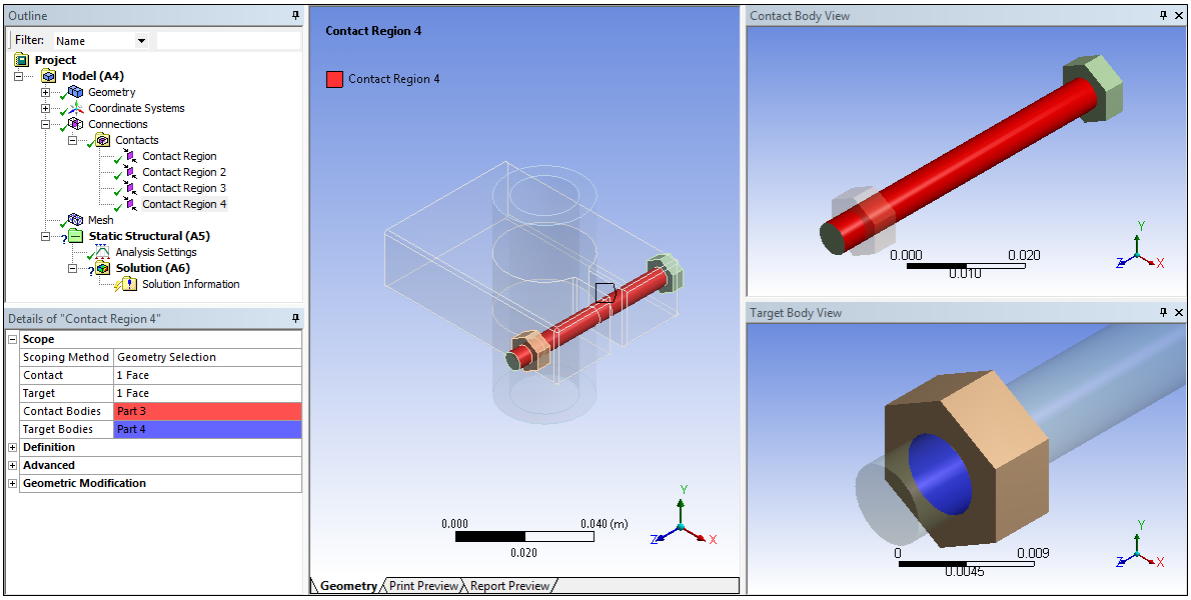
Contact and Target bodies are displayed using default colors (red and blue). You can change the display color using the **Random** option in the Annotations (p. 79) group to display each contact using a color chosen at random each redraw.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Displaying Contact Bodies in Separate Windows

By default, the **Body Views** option on the Connections Context Tab (p. 50), is active to display parts and connections in separate auxiliary windows. As illustrated below, the different contact bodies (**Contact** and **Target**) have colors codes associated with them, in the **Details** view as well as the graphic windows. The Body Views feature is available for joints, contacts, springs, and beam connections.



Note:

The Body Views feature does not support the use of hotkeys and hotkey combinations (p. 314).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Hiding Bodies Not Scoped to a Contact Region

You can hide all bodies except those that are scoped to a specific contact region.

To Hide All Bodies Not Scoped to a Contact Region:

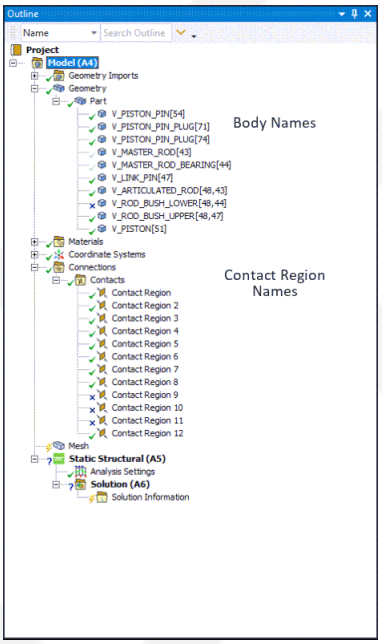
1. Select the Contact Region object whose bodies you do not want to hide.
2. Right-click to display the context menu.
3. Select **Hide All Other Bodies** in the menu. All bodies are hidden except those that are part of the selected contact region.

Renaming Contact Regions Based on Geometry Names

You can change the name of any contact region using the following choices available in the context menu that appears when you click the right mouse button on a particular contact region:

- **Rename**: Enables you to change the contact region name to a name that you type (similar to renaming a file in Windows Explorer).
- **Rename Based on Definition**: Enables you to change the contact region name to include the corresponding names of the items in the **Geometry** branch of the tree that make up the contact region. The items are separated by the word "To" in the new contact region name. You can change all the contact region names at once by clicking the right mouse button on the **Connections** branch, then choosing **Rename Based on Definition** from that context menu. A demonstration of this feature follows.

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When you change the names of contact regions that involve multiple bodies, the region names change to include the word **Multiple** instead of the long list of names associated with multiple bodies. An example is **Bonded – Multiple To Multiple**.

Identifying Contact Regions for a Body

See the description for **Contacts for Selected Bodies** in the Correlating Tree Outline Objects with Model Characteristics (p. 110) section.

Create Contact Debonding

To automatically generate a Contact Debonding object, select a Contact Region and drag and drop it onto the Fracture folder.

Flipping Contact and Target Scope Settings

A valuable feature available when using asymmetric contact is the ability to swap Contact and Target face or edge **Scope** specifications. To flip the scoping, right-click the desired connection and select **Flip Contact/Target**. This is illustrated below for a single region. Note that you can select multiple connection objects to flip using the **Ctrl** or **Shift** keys.

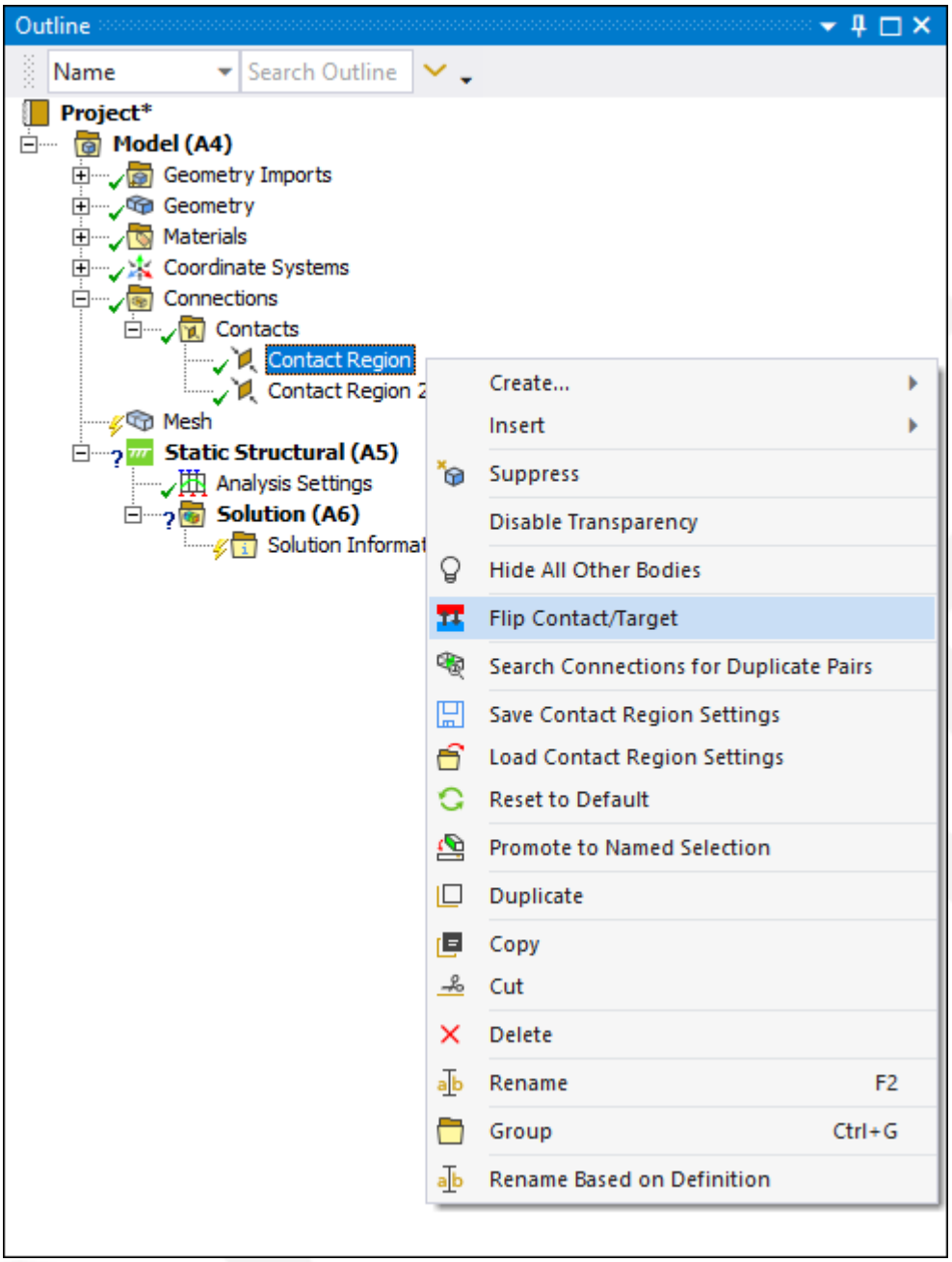
Note:

This feature is not applicable to **Face/Edge** contact where faces are always designated as targets and edges are always designated as contacts.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Setting Default APDL Names

The context (right-click) menu option for the Contact Region object, **Set Default APDL Names**, enables you generate system default names for the **Contact/Target APDL Name** properties. These properties are optional and only available for the Mechanical APDL solver. You use them to specify an APDL parameter name for the contact and the target element types. Using these names, you can easily identify the contact and target element type and their corresponding real constant set that you can reference and use in a Commands (APDL) object.

The application creates the names based on object identifiers and prepends that value with either "_con" or "_tgt," as shown in the following examples:

- **Contact APDL Name**: _con<object id>, such as **_con66**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Target APDL Name**: _tgt<object id>, such as **`_tgt66`**

Merging Contact Regions That Share Geometry

You can merge two or more contact regions into one contact region, provided they share the same type of geometry (edges or faces).

To Merge Contact Regions That Share Geometry:

1. Select two or more contact regions in the tree that share the same type of geometry (edges or faces). Use the **Shift** or **Ctrl** key for multiple selections.
2. Right-click to display the context menu.
3. Select **Merge Selected Contact Regions** in the menu. This option only appears if the regions share the same geometry types. After selecting the option, a new contact region is appended to the list in the tree. The new region represents the merged regions. The individual contact regions that you selected to form the merged region are no longer represented in the list.

Saving or Loading Contact Region Settings

You can save the configuration settings of a contact region to an XML file. You can also load settings from an XML file to configure other contact regions.

To Save Configuration Settings of a Contact Region:

1. Select the contact region whose settings you want to save.
2. Right-click to display the context menu.
3. Select **Save Contact Region Settings** in the menu. This option does not appear if you selected more than one contact region.
4. Specify the name and destination of the file. An XML file is created that contains the configuration settings of the contact region.

Note:

The XML file contains properties that are universally applied to contact regions. For this reason, source and target geometries are not included in the file.

To Load Configuration Settings to Contact Regions:

1. Select the contact regions whose settings you want to assign. Use the **Shift** or **Ctrl** key for multiple selections.
2. Right-click to display the context menu.
3. Select **Load Contact Region Settings** in the menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Specify the name and location of the XML file that contains the configuration settings of a contact region. Those settings are applied to the selected contact regions and will appear in the Details view of these regions.

Resetting Contact Regions to Default Settings

You can reset the default configuration settings of selected contact regions.

To Reset Default Configuration Settings of Contact Regions:

1. Select the contact regions whose settings you want to reset to default values. Use the **Shift** or **Ctrl** key for multiple selections.
2. Right-click to display the context menu.
3. Select **Reset to Default** in the menu. Default settings are applied to the selected contact regions and will appear in the Details view of these regions.

Locating Bodies Without Contact

See the description for **Bodies Without Contacts in Tree** in the Correlating Tree Outline Objects with Model Characteristics (p. 110) section.

Locating Parts Without Contact

See the description for **Parts Without Contacts in Tree** in the Correlating Tree Outline Objects with Model Characteristics (p. 110) section.

Contact in Rigid Dynamics

Contact conditions are formed where rigid bodies meet. While the default contact settings and automatic detection capabilities are often sufficient for structural analyses, the default contact definition must be extended to adjacent surfaces in some cases. This is because the nature of rigid dynamics usually implies very large displacements and rotations.

In rigid dynamics, supported contact types depend on where the contact is defined and which time integration type (p. 1436) is used:

- Frictionless and forced frictional contact are available for all time integration schemes.
- Rough contact and no separation contact are supported by the Runge-Kutta (p. 638), Generalized-Alpha (p. 639), and Stabilized Generalized-Alpha (p. 641) time integration schemes.
- Bonded contact is only supported within condensed parts.

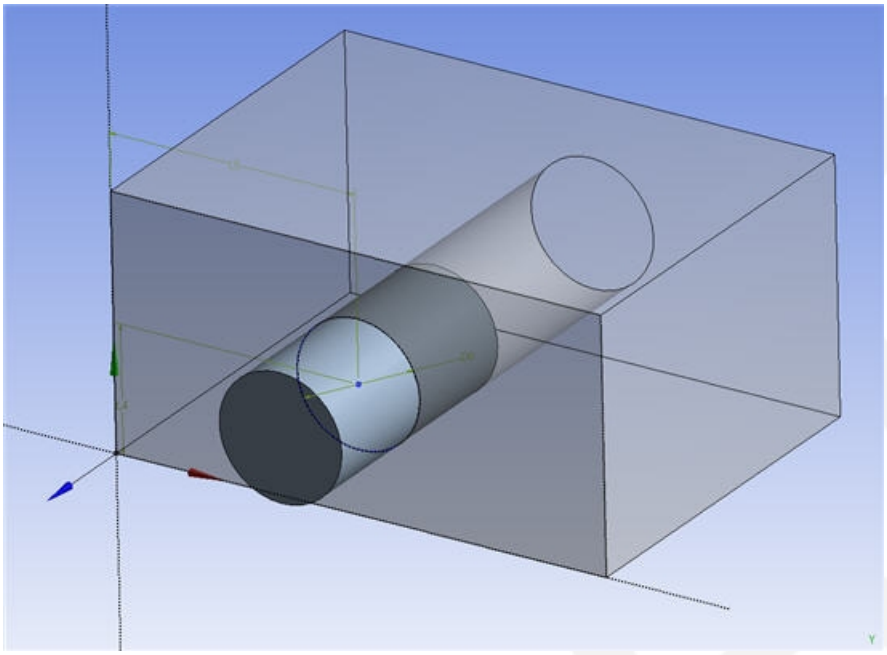
The contact is always based on **Pure Lagrange** formulation. Contact constraint equations are updated at each time step, and added to the system matrix through additional forces of degrees of freedom called **Lagrange Multipliers**. In this formulation, there is no contact stiffness. Contact constraints are satisfied when the bodies are touching, and they are nonexistent when bodies are separated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Contact and Rigid Bodies

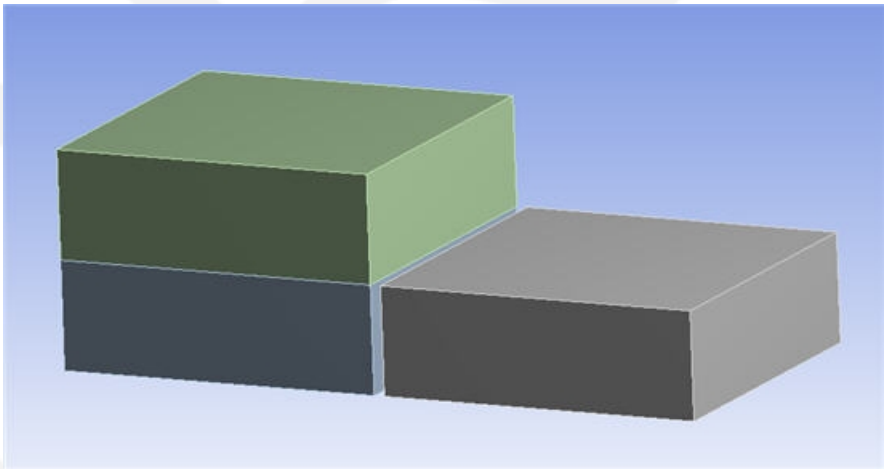
Contact is formulated between rigid bodies. Hence, there is no possibility of deforming the bodies to satisfy the contact constraint equations. If the contact equations cannot eventually be satisfied, the solution will not proceed. To illustrate this, two examples are considered:

Example 4: Cylindrical Shaft in a Block



- If the diameter of the cylindrical shaft is smaller than that of the hole, motion is possible.
- If the diameter of the cylindrical shaft is larger than that of the hole, the simulation is not possible.
- If the two diameters are exactly equal, then the analysis might fail.

Example 5: Block Sliding on Two Blocks



- If the green block slides horizontally from left to right and the height of the right block is less than that of the left block, motion is possible.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- If the height of the two bottom blocks is identical and a vertical contact surface is defined between the two bottom blocks, the block might hit the vertical surface, and the solution will not proceed.
- If the height of the right block is greater than the height of the left block, the green block will move back to the left.

Note:

Avoid ambiguous configurations whenever possible. Consider creating fillets on sharp edges as a workaround.

Contact Mesh

You can scope the contact objects to rigid bodies using 3-D faces in solid bodies, or pick the entire body. When you create this type of contact, the surfaces and edges in the contact region are meshed.

The mesh is used differently depending on the **RBD Contact Detection** setting:

When **RBD Contact Detection** is set to **Geometry-Based**, the mesh helps to speed up the solution by providing an initial position to the contact points that are calculated, and it helps to drive the number of contact points used between the bodies when in contact. As each body has up to 6 degrees of freedom, a contact between two rigid bodies will restrain up to 6 relative degrees of freedom. This means that a reasonably coarse mesh is generally sufficient to define the contact surface. The contact solver will use this mesh to initiate the contact geometry calculation, but will then project back the contact points to CAD geometry. Refining the mesh can increase the solution time without always increasing the quality of the solution. Conversely, refining the mesh can be useful if the geometry is concave and the solver reports a high amount of shocks for the pair involving the concave surfaces.

When **RBD Contact Detection** is set to **Mesh-Based**, contact points are on the mesh, while the contact normal is still evaluated on the geometry. In conjunction with the Moreau-Jean (p. 642) time integration type, mesh-based contact detection can handle more refined meshes that capture smaller geometry details.

Contact and Time Step

The rigid solver uses event-based time integration. Over each time step, the solver evaluates the trajectory of the bodies, and checks when these trajectories interfere. When interference is found (as with stops on joints), a shock will be analyzed, leading to a new velocity distribution. The physics of the velocity redistribution during the shock is based on the conservation of momentum and energy. The amount of energy lost during the shock is quantified by the coefficient of restitution. For details, see Joint Stops and Locks (p. 1360). The trajectory detection of interferences allows the use of rather large time steps without missing the contacts; however, transitions between adjacent contact surfaces in certain situations (such as sliding situations) often require smaller time steps.

In contrast to Penalty based simulation that introduces an artificial deformation of the bodies and therefore, high frequencies in the simulation, the pure Lagrange formulation used in the rigid dynamics formulation does not change the frequency content of the simulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

A solution that includes contact requires an increased amount of geometrical calculation, resulting in a significantly higher overall simulation time than a solution without contact. As such, it is recommended that joints stops are used in place of contacts whenever possible.

Limitations

For models with sliding contacts (such as cams, guiding grooves, and so on), small bounces due to nonzero restitution factors can cause an increase in simulation time and instabilities. Using a restitution factor of zero will significantly speed up the simulation.

The Rigid Dynamics solver unifies contact regions defined between the same pairs of parts/bodies. Consequently, defining more than one contact region between the same pairs of bodies may lead to unpredictable results. The following guidelines are strongly recommended:

- All contact regions defined between the same pairs of parts/bodies must have the same type. Mixing different types (for example, frictionless and rough) may lead to incorrect results.
- All contact regions defined between the same pairs of parts/bodies must follow the same order. A body defined as a target body in one contact region must not be defined as contact body in another contact region between the same pairs of parts/bodies.

Best Practices for Contact in Rigid Body Analyses

The following guidelines are considered best practices for optimizing performance with contact when simulating complex models with large numbers of contact elements, long duration, or high speed.

Time Integration Type

- Use the Moreau-Jean time stepping method (p. 1436) when non-smooth and for:
  - Models with large number of contact elements
  - Models with many locks/stops
- Use a small **Initial Time Step**.
- Use a small **Max Time Step**.
- Use output controls (p. 1481) to limit the number of steps saved.

Mesh

- Use low order triangle element shape.
- Use mapped mesh when possible.
- Make sure the mesh is capturing the fine details of your model.

Contact Pairs

Contact can be defined on surfaces or on entire bodies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- When defining contact on a surface, one surface/surface pair is defined per surface. If your contact has less than a dozen surfaces on each side, surface to surface contact should be used. When dealing with a larger number of surfaces, use body to body contact. For example, when modeling contact between the teeth of two gears, body to body contact is best.
- When using body to body contact, mesh-based contact is the only option.
- Setting the coefficient of restitution to 0 introduces some dissipation into the model and adds robustness.

Pinball

Pinball radius is automatically calculated based on the size of the contact surface pairs. Most of the time, this value is correct. In cases where the pinball radius is very small, increasing it slightly should correct the issue (don't go over 10 times the default value). The actual value of the pinball radius can be found in the solver log file:

```
Forced Frictional Sliding – Body1 to Body2 is open. Contact radius = 7.828508e-004
```

Special Cases

When a gap is small and surface exact geometry matters, you should use geometry-based contact. In cases like this, Moreau-Jean time-stepping can still be used.

Redundancies

The rigid body solver solves for forces between bodies. When the model is redundant, these forces are harder to determine. The solver has multiple algorithms that are used in sequence to determine forces between bodies. At time 0, the solver often has no information about forces, so adding an initial step to bring all the parts of the model into proper touching configuration can help. When applicable, use the 2-D option (only generates a few contact points per pair).

When redundancies can't be avoided, you can attempt to force convergence by inserting the **`ContactUseRobustPSOR(itermax)`** command in a snippet with **`itermax`** being a large number (5000 - 10000 is typical). This will increase solution time, and you should carefully review the joint and contact forces.

Monitoring Contact

The contact log file shows the status of contacts in the simulation. In the following example, one pair goes from separated to touching at time 0.151928:

```
Region: Solid - Solid - Separated...
Number Of Points 7

Region: Solid - Solid - Separated...
Number Of Points 7

Region: Solid - Solid - Separated...
Number Of Points 7
_____________________________

Region: Solid - Solid - ...
Shock 1 time prediction is = 0.151928 alpha 0.638561
_____________________________
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Region: Solid - Solid - Touching...
Number Of Points 7
```

Checking the contact log file can help you troubleshoot contact in your model.

- Shocks are not reported for Moreau-Jean time-stepping (p. 1436), but the status does change.
- Look for contact status **`Penetrated`** if the solution fails at time = 0. You can configure joints to avoid this initial penetration.
- Check the number of points for a given contact pair. Avoid large and small numbers. If necessary, change the mesh density.

Monitoring the Solver

The solver output monitors the solver progress. It lists the status of contacts and stops.

```
Contact "Frictionless - Solid To Solid" not active (0 points received)
_____________________________________________________________
Contact "Frictionless - Solid To Solid" not active (0 points received)
_____________________________________________________________
Current Time: 0.12 Current time step 0.05
Frictionless - Solid To Solid is open.
_____________________________________________________________
Contact "Frictionless - Solid To Solid" not active (0 points received)
_____________________________________________________________
Contact "Frictionless - Solid To Solid" Received 7 contact points
Stopped Frictionless - Solid To Solid at 0.151928

Time 0.1519 Solving Shock 1

Stopped Frictionless - Solid To Solid at 0.151928
Time 0.151928 Processing Event "Contact"
Frictionless - Solid To Solid is open.
_____________________________________________________________
Contact "Frictionless - Solid To Solid" not active (7 points received)
```

The solver log file will also report redundancies. When detected, the solver will mention **LCP** errors:

```
*** Error:
w[4]=1.097266e-005 > threshold2=7.859939e-007, z[4]=6.639837e-001, > threshold=8.865630e-009
0 C_BlockMatrix::Block::CheckLCPSolution(), C_BlockMatrix.cpp line 2174

Murty by Factorization algorithm failed
Trying PSOR
PSOR algorithm reached maximum number of iterations: 680
Initial residual 4.455025e+000 Last 3 residual values are 2.194531e-005 2.194531e-005 2.194531e-005
PSOR algorithm failed
Trying Lemke
```

`threshold` is the force in the contact that the solver needs to reach for convergence. The value depends on global forces in the model.

A model with touching contact with no forces is always difficult for the solver. Adding small forces to stabilize the solution usually helps with convergence. In many cases, gravity is sufficient.

Friction

Frictionless and Forced frictional contact (p. 1234) are the only supported contact types for Moreau-Jean time-stepping (p. 1436).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Best Practices for Specifying Contact Conditions

For typical structural problems, the presence of nonlinear contact is often the reason for increased solution times. This can be exacerbated by poorly defined contact conditions, which may also create further instability and cause convergence issues and require additional processing time. Given properly defined contact conditions, contact results converge more efficiently and results tend to be smoother.

The following topics are examined in this section to help you achieve better results through proper contact definition:

Contact Setup and Verification
Solver Preparation
Addressing Non-Convergence

Contact Setup and Verification

This section describes some of the tools that you should try to use when generating and managing contact.

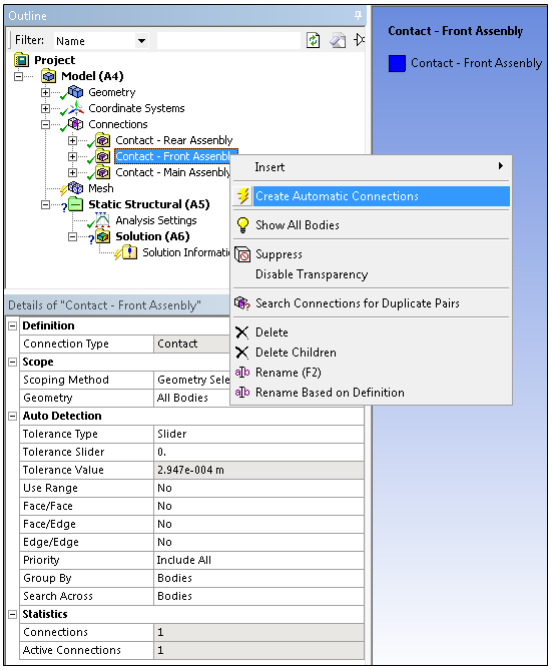
- Automatic Contact Generation (p. 1281)
- Manual Contact Pairs (p. 1282)
- Multiple Contact Folders (p. 1282)
- Contact Search and Select (p. 1283)
- Worksheet Options (p. 1283)
- Body Views (p. 1284)
- Shell Contact Normal Directions (p. 1284)
- Initial Contact Tool (p. 1285)

Automatic Contact Generation

On geometry attach, or when requested, Contact pairs can be automatically detected and generated in Mechanical. Search options also can be set on the Contact folder, and include:

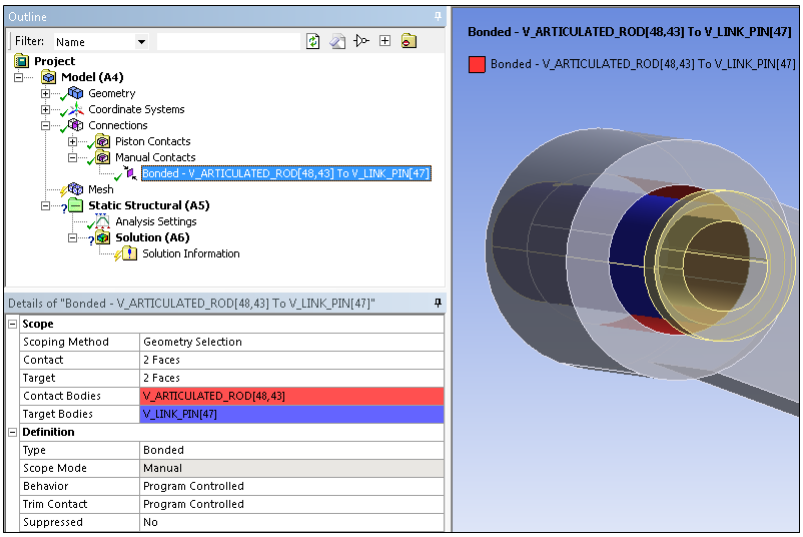
- Scoping (search entire assembly or only selected bodies)
- Auto Detection Tolerance (see Connection Group Folder (p. 1212) Help section)
- Topology Types (Face/Face)
- Grouping Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Manual Contact Pairs

There are times when you need to create contact conditions manually (p. 1266). This could be necessary when a required contact is not detected automatically, such as when a gap exists that is outside of the tolerance, as shown in the example below.



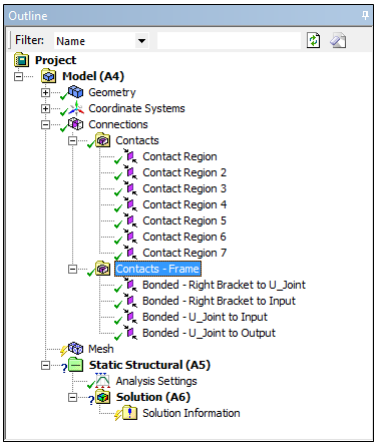
Multiple Contact Folders

Creating multiple contact folders can be beneficial. Multiple folders enable you to:

- Create different automatic detection settings on different bodies.
- Organize a large numbers of contact regions more efficiently.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Contact Search and Select

To find connections associated with a geometric selection, the context menu (right-click) **Go To** options (p. 110).

Tip:

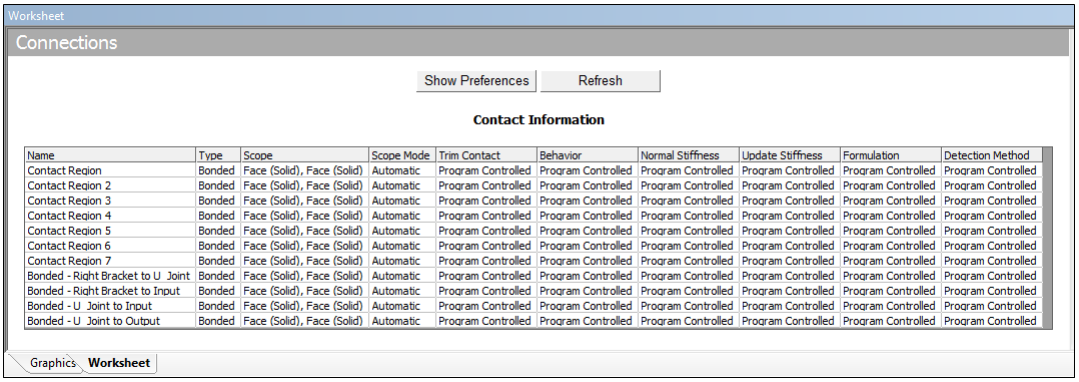
Use tags (p. 2349) to keep track of contacts pairs of interest.

Worksheet Options

As illustrated below, the connections Worksheet (p. 1210) view provides a detailed layout for reviewing all of your contact settings, and includes the capability to:

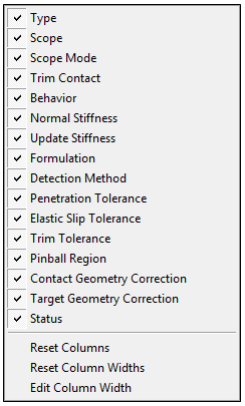
- Sort rows by clicking the column header.
- Control the visibility of each column/property using the selections available on the right-click (context) menu.

Contact Worksheet



Name	Type	Scope	Scope Mode	Trim Contact	Behavior	Normal Stiffness	Update Stiffness	Formulation	Detection Method
Contact Region	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 2	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 3	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 4	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 5	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 6	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Contact Region 7	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Bonded - Right Bracket to U_Joint	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Bonded - Right Bracket to Input	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Bonded - U_Joint to Input	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Bonded - U_Joint to Output	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlled

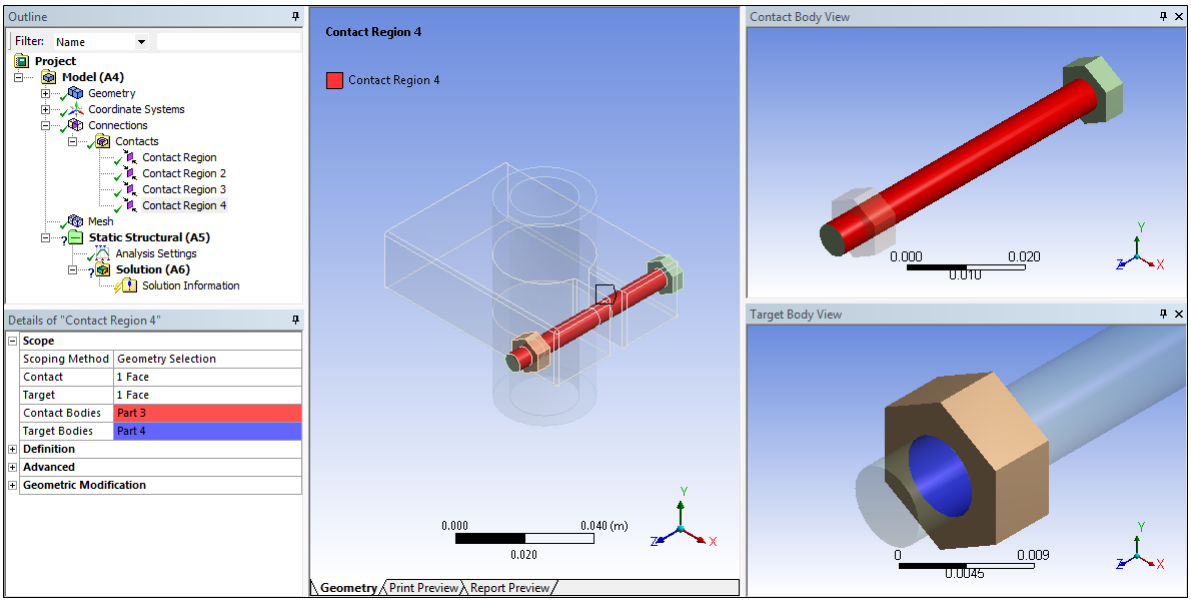
RMB Menu



Body Views

A helpful way to spatially identify a contact, especially when the contact occurs on the interior of your model, is to use the **Body Views** feature on the Connections Context Tab (p. 50) to display parts in separate auxiliary windows (p. 1270).

This is the default display option when you select a Contact object, and as illustrated below, the different contact bodies (**Contact** and **Target**) have colors codes associated with them in the Details view as well as the graphic windows.



Note:

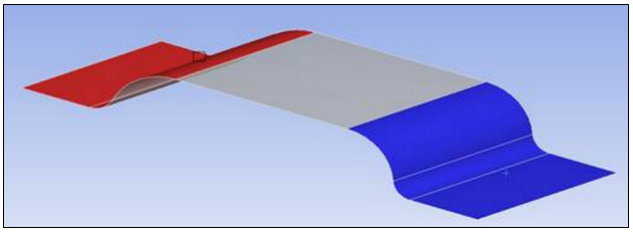
The Body Views feature does not support the use of hotkeys and hotkey combinations (p. 314).

Shell Contact Normal Directions

Contact on shell bodies can only be detected if the direction of the contact is appropriately defined. For example, to model the self-contact when pinching the following shell:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Undesirable Normals

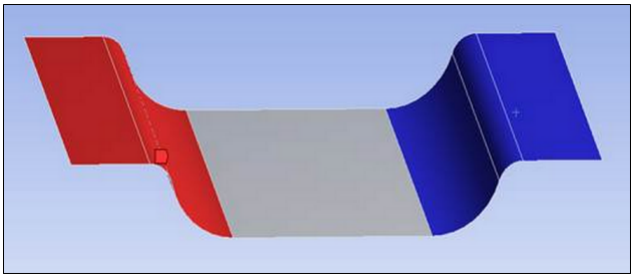


Details View

Details of "Contact Region"

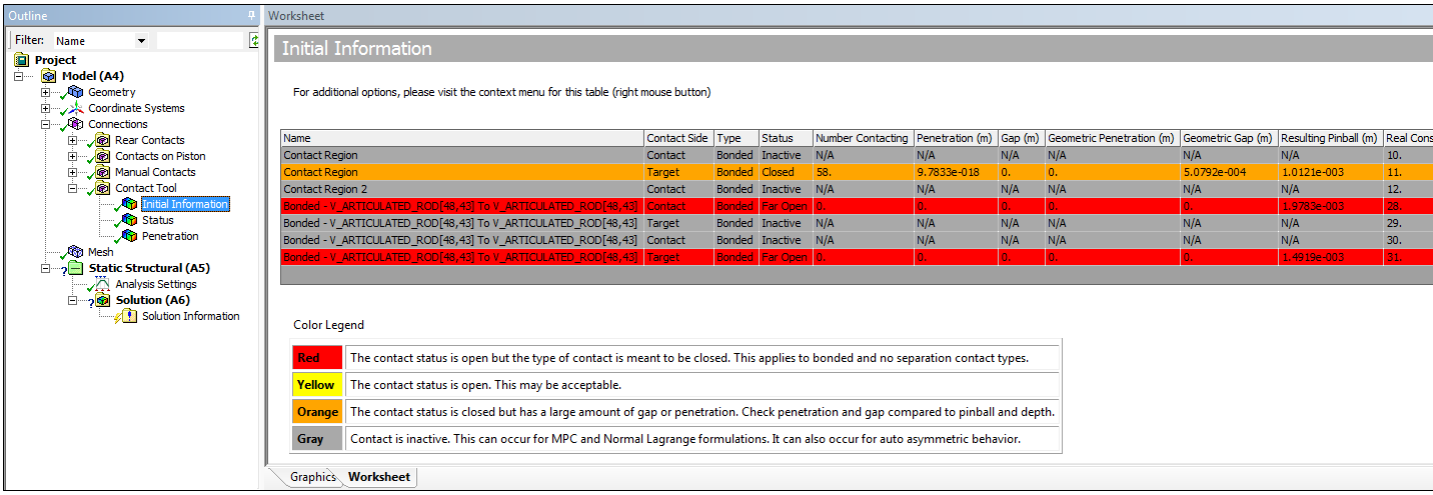
Scope	
Scoping Method	Geometry Selection
Contact	3 Faces
Target	3 Faces
Contact Bodies	Surface Body
Target Bodies	Surface Body
Contact Shell Face	Bottom
Target Shell Face	Bottom
Shell Thickness Effect	No
Definition	
Advanced	
Geometric Modification	

Desired Normals



Contact Tool

You can use the Contact Tool (p. 2079) to quickly learn about the contact status as seen by the solver (*before* you solve). **Initial Information** is illustrated below.



Name	Contact Side	Type	Status	Number Contacting	Penetration (m)	Gap (m)	Geometric Penetration (m)	Geometric Gap (m)	Resulting Pinball (m)	Real Cons
Contact Region	Contact	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	10.
Contact Region	Target	Bonded	Closed	58.	9.7833e-018	0.	0.	5.0792e-004	1.0121e-003	11.
Contact Region 2	Contact	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	12.
Bonded - V_ARTICULATED_ROD[48,43] To V_ARTICULATED_ROD[48,43]	Contact	Bonded	Far Open	0.	0.	0.	0.	0.	1.9783e-003	28.
Bonded - V_ARTICULATED_ROD[48,43] To V_ARTICULATED_ROD[48,43]	Target	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	29.
Bonded - V_ARTICULATED_ROD[48,43] To V_ARTICULATED_ROD[48,43]	Contact	Bonded	Inactive	N/A	N/A	N/A	N/A	N/A	N/A	30.
Bonded - V_ARTICULATED_ROD[48,43] To V_ARTICULATED_ROD[48,43]	Target	Bonded	Far Open	0.	0.	0.	0.	0.	1.4919e-003	31.

The color coding alerts you to possible issues. And you can calculate contours for results such as **Status** and **Penetration**. In addition, you can see:

- Contact pairs that are open/closed.
- The extent of initial penetration.
- Active/Inactive pairs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solver Preparation

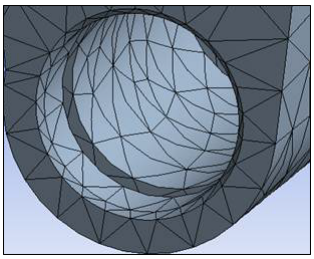
This section describes useful techniques for making sure that you establish robust contact settings and conditions.

- Mesh Quality (p. 1286)
- Mesh Sizing (p. 1287)
- Understanding Contact Formulation (p. 1287)
- Advantages of MPC Contact (p. 1288)
- Contact Stiffness (p. 1288)
- Overlapping Contact and Boundary Conditions (p. 1289)
- Initial Gaps and Rigid Body Motion (p. 1289)

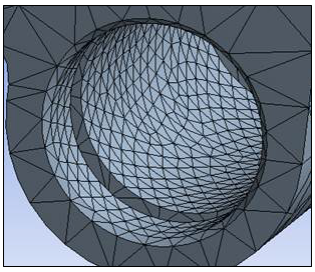
Mesh Quality

Poor mesh quality can cause convergence problems, especially when nonlinear contact is applied on it. Examples of mesh quality are illustrated below. Understanding the use of Local Mesh Controls can help you refine the mesh on your contact conditions.

The contact surface shown here has a mesh that could be improved.



This contact surface shown here has a very good mesh quality.



Tip:

For nonlinear contact models, you can either set **Physics Preference** to **Mechanical** and **Error Limits** to **Aggressive Mechanical**, or set **Physics Preference** to **Nonlinear Mechanical**. See Ansys Workbench and Mechanical APDL Application Meshing Differences for details.

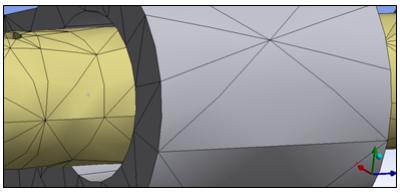
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

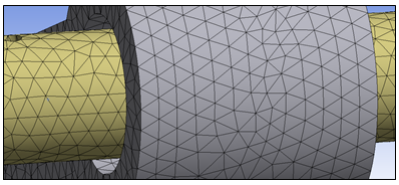
Mesh Sizing

In order to create smooth results, you need to make sure that the elements closely follow the curvature and that your model has sufficient contact elements on curved surfaces, including straight surfaces that may become curved as the analysis proceeds. This is especially true for nonlinear contact conditions. For the best results, use similar mesh densities on both sides of the contact pair. To improve element size and density, modify the mesh using Sizing Controls.

This example shown below illustrates a curve with not enough elements.



The parts are now shown below with improved element sizes and density for each contact side.



Understanding Contact Formulation

Formulation options (p. 1241) allow you to specify which algorithm the software uses for a particular Contact pair computation. Formulations define the solution method used. See the Contact Formulation Theory (p. 1222) section for additional information.

- **Augmented Lagrange** (Default): this setting is suitable for most problems.
- **Pure Penalty**: this setting is suited to contact occurring only on an edge or vertex.
- **MPC** (Multi-Point Constraint): this setting is ideal for all linear contacts when there is no over-constraint.
- **Normal Lagrange**: this **Formulation** option:
  - provides the highest accuracy.
  - works well with material nonlinearities.
  - works well with shells or thin layers.
  - enables interference fit
  - allows large sliding
- **Beam**: this setting is ideal for linear contact when there may be over-constraint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Advantages of MPC Contact

The **MPC** Formulation option (p. 1241) does not provide stiffness settings. It is a purely linear way to connect contacting bodies. This setting is ideal for shell-solid, shell-shell, and beam-shell contacts. MPC prevents artificial stiffness when gaps exist between curved surfaces (another alternative is to use fixed Joints).

One caveat is that MPC is the most sensitive contact type to overconstraint so it should be avoided when there are other contacts or boundary conditions that overlap.

Following a completed solution, you can view MPC equations and other FE Connections (p. 1840) graphically.

Contact Stiffness

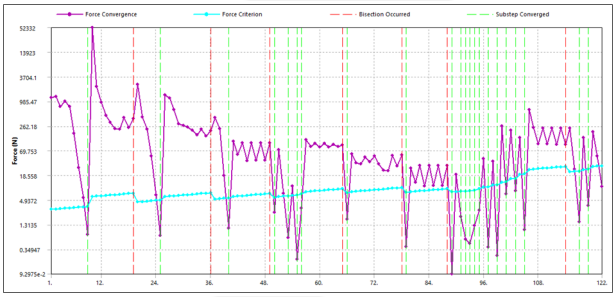
Using the Normal Stiffness (p. 1249) property, you can manually increase the stiffness associated with a contact pair. A high stiffness setting can lead to reduced penetration and an increase in accuracy. However, it can also lead to ill-conditioning and divergence.

For contacts that lead to convergence difficulty, you should try lowering the stiffness.

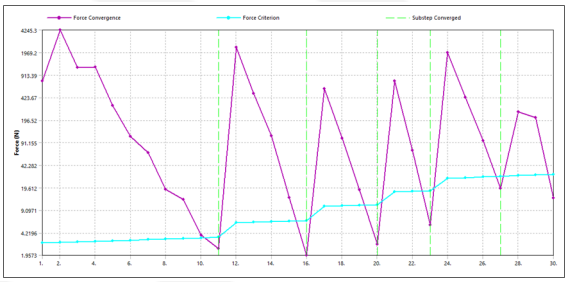
For pretension problems, increase the stiffness, because penetration can strongly influence the pretension forces.

And, when there is difficulty converging due to high penetration, you may need to increase the stiffness.

Here is an example of poor convergence. 122 iterations were required and there are too many bisection instances.



This example shows very good convergence as a result of the stiffness being reduced. No bisections have occurred and only 30 iterations were required.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

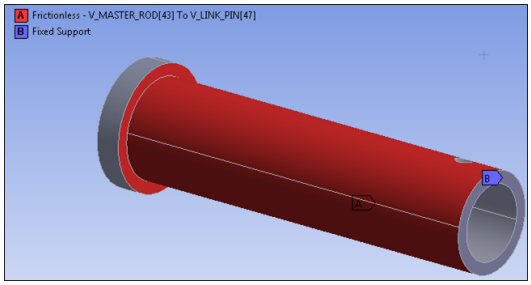
Published: 2023-10-23

Overlapping Contact and Boundary Conditions

Care should be taken when a contacts and constraint type boundary conditions have overlapping topology. In this case, you can use modeling techniques to eliminate the overconstraint, including:

- Modifying the Contact Trimming (p. 1237).
- Modifying the **Pinball Radius** property on the remote boundary conditions. This property specifies a radius (length unit) that the boundary condition is applied to the elements that are within the region.
- Implementing node-based (p. 269) scoping on the boundary conditions.

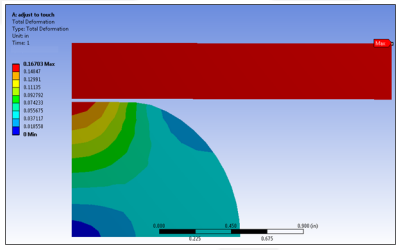
If you cannot eliminate an overcontraint situation, the use of the MPC Formulation should be avoided. The **Beam** Formulation option (p. 1241) can be a good alternative in this case.



Initial Gaps and Rigid Body Motion

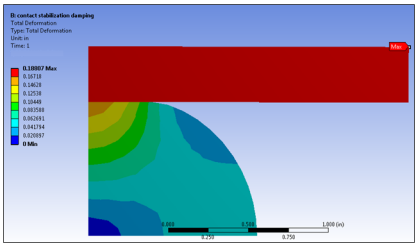
There are times when parts (that have exterior constraints placed on them) are dependent upon contact to prevent rigid body motion. In addition, when small gaps exist (especially for curved contact conditions), rigid body motion can occur at the beginning of the solution before contact engages.

When nonlinear contact is present, small gaps that are initially open can lead to rigid body motion (as shown below). Changing the Interface Treatment (p. 1257) property to **Adjust to Touch** can be an effective means to resolve this gap. Mathematical adjustment to close the gap causes an open region to be visible when post-processing. This gap is simply offset by the contact elements. Note that for concentric cylinders, **Adjust to Touch** is not recommended. Instead the offset should be manually entered.



An alternative is to employ the **Stabilization Damping Factor** (p. 1252) property. Rather than ignoring the gap by offsetting the contact elements, damping is applied and makes the approach of the bodies towards each other less difficult for the solver to handle, leading eventually to closed contact. An example is shown below. Note that you need to use care and verify that the damping does not negatively affect the accuracy of your analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



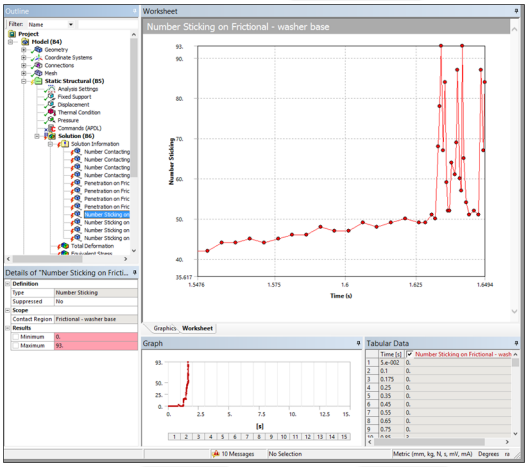
Addressing Non-Convergence

This section describes techniques to troubleshoot convergence problems.

- Diagnostic Tool: Contact Result Tracker (p. 1290)
- Making Use of Partial Solutions (p. 1290)
- Contact Results Tool: Penetration (p. 1291)
- Diagnostic Tools: NR Residuals (p. 1291)
- Checklist for Overcoming Convergence Difficulties (p. 1292)

Diagnostic Tool: Contact Results Tracker

The structural result tracker (p. 1847) for Contact enables you to view contact information "live" as the solution processes. This enables you to observe trends that can help you diagnose problems. For example, a decreasing number of contact points indicate a trend towards a loss of contact.

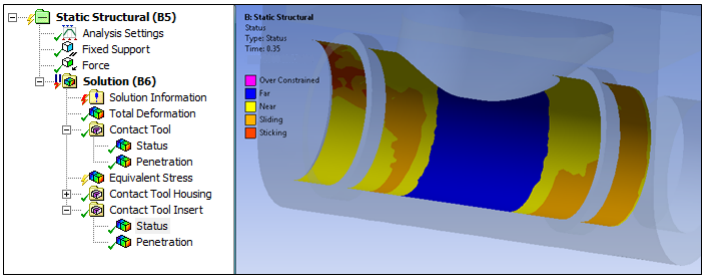


Making Use of Partial Solutions

When a Solution fails to converge, you can review the results at the converged substeps, which can be invaluable for diagnosing problems. Checking the contact result Status (p. 2043) is a useful first step. It can help you understand the global contact behavior of the model and enable you to find problem areas. An example is shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

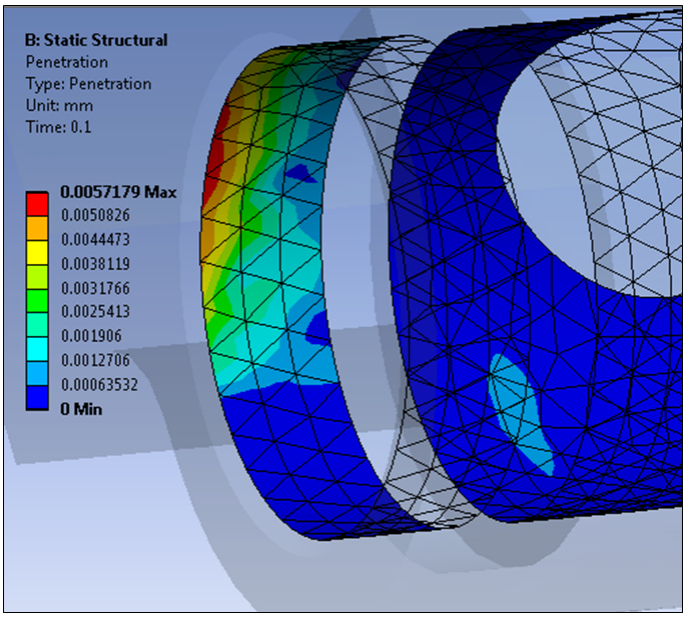
Published: 2023-10-23



Contact Results Tool: Penetration

It is important to ensure that the contact penetration is small relative to local displacements. The reason is that any contact penetration is technically impossible, since bodies do not move into each other. A good check is to add the contact result (p. 2043) **Penetration** to the local displacement and compute a back-of-the-envelope strain calculation. This helps determine if the penetration is significantly affecting the stress calculations. An example is shown below. Ways to reduce penetration include:

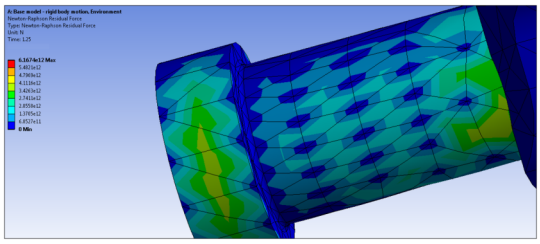
- Increasing the Stiffness (p. 1249).
- Reducing the Penetration Tolerance (p. 1246).
- Setting the Formulation (p. 1241) property to **Normal Lagrange** .



Diagnostic Tool: NR Residuals

For a failed solution, requesting Newton-Raphson residuals helps to identify areas where attention is needed. It is especially useful for models with many nonlinear contacts, and the pair which caused the convergence difficulty is not known. An example is shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Checklist for Overcoming Convergence Difficulties

Here are a few steps to follow when convergence difficulties are encountered.

1. Identify the problematic contact region(s) using:
   - Contact Tracking (p. 1847).
   - Contact Results (p. 2043) (especially **Status** and **Penetration**).
   - Force Convergence plots (See the Solution Information (p. 1833) object Help section).
   - NR residuals.
2. Once identified, possible remedies:
   - Check Mesh Quality (using Mesh Controls)
   - Make sure the model's units of measure are on an appropriate scale
   - Check for proper Initial Conditions (p. 349)/pinball
   - Adjust the contact Stiffness (p. 1249).
   - Use "nodal detection" if the problem is at a corner.
   - Reduce the time step (p. 1418) size before and during the onset of the divergence.
   - If a large friction coefficient is defined (>**`0.25`**) consider using an unsymmetric solver.
   - If the problem manifests immediately based on poor initial contact conditions (not well engaged or rigid body motion) try using aggressive contact stiffness update.
   - **Last resort**: Add stabilization (Contact (p. 1252) or global (p. 2036)).

Joints

The following topics are covered in this section:

Joint Characteristics
Joint Types
Joint Properties
Modifying Joint Coordinate Systems
Joint Stiffness

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Joint Friction
Manual Joint Creation
Automatic Joint Creation
Joint Stops and Locks
Ease of Use Features
Detecting Overconstrained Conditions
Example: Assembling Joints
Example: Configuring Joints
Example: Configuring Bodies

Joint Characteristics

A joint typically serves as a junction where bodies are joined together. Joint types are characterized as fixed or free depending on their rotational and translational degrees of freedom. If you specify a **Joint** as a **Remote Attachment** it is classified as a *remote* boundary condition. Refer to the Remote Boundary Conditions (p. 1735) section for a listing of all remote boundary conditions and their characteristics.

The section includes the following joint characteristics:

- Supported Analysis Types (p. 1293)
- Degrees of Freedom (DOF) (p. 1295)
- Abstraction (p. 1295)
- Initial Conditions (p. 1296)
- DOF Zero Value Conventions (p. 1296)
- Joint Condition Considerations (p. 1297)

Supported Analysis Types

Joints are supported in the following structural analyses:

- Explicit Dynamics (p. 406)
- Harmonic Response (p. 496)
- Modal (p. 515)
- Random Vibration (p. 524)
- Response Spectrum (p. 533)
- Rigid Dynamics (p. 550)
- Static Structural (p. 652)

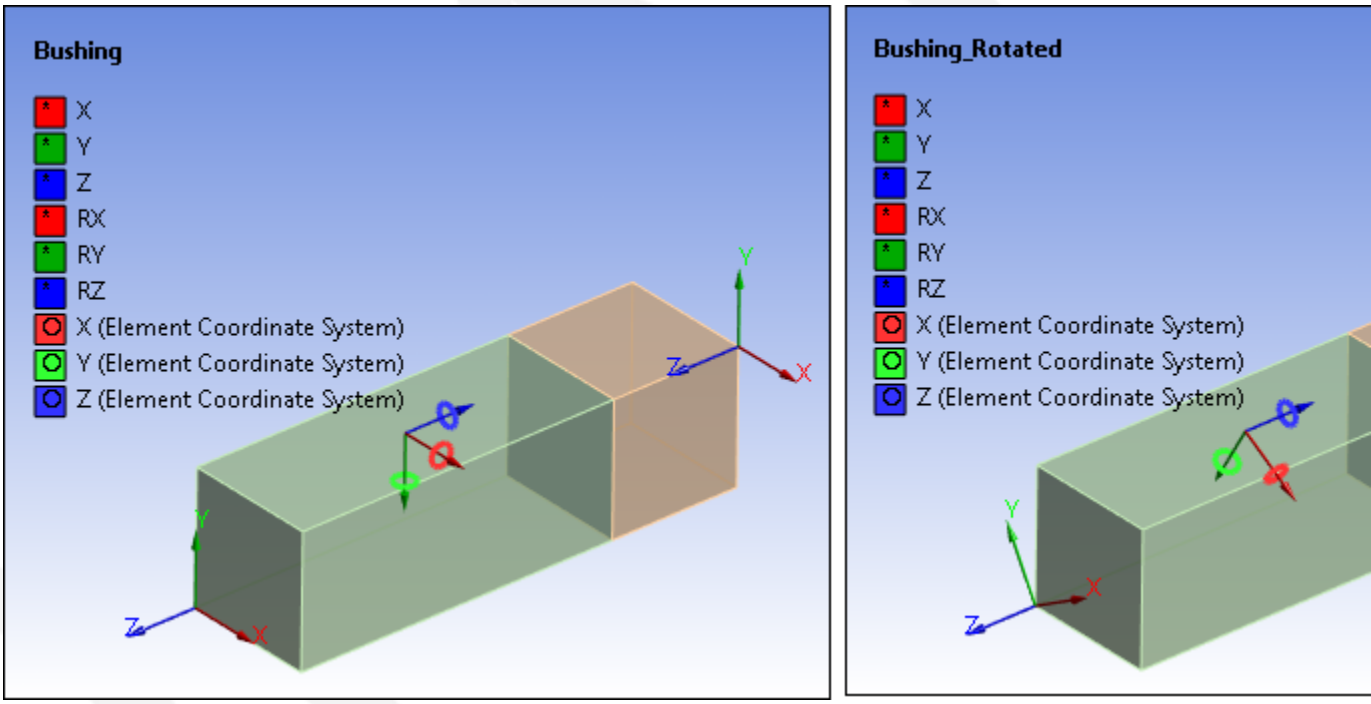
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Transient Structural (p. 665)

Note:

- A joint cannot be applied to a vertex scoped to an end release (p. 1409).
- When you set the **Formulation** property to **Bushing**, the application uses the element COMBI250. As a result:
  - The **Element Coordinate System** is the first coordinate system used to apply rotation.
  - This is followed by nodal rotations per promoted Remote Points (**Reference** or **Mobile**), such as the **NROTAT** command that rotates nodal coordinate systems into the active system. In addition, rotations from these remote points modify the element matrix (see the **Input Data** topic of the COMBI250 element reference).
  - This is then followed by the normal processing sequence for rotations.

  Here is an example of Element Coordinate System annotations ("rings" about each axis), prior to and after rotation.



- Each multibody part made of rigid bodies is treated as a single, rigid part. Consequently, joints must not be created within a multibody part.

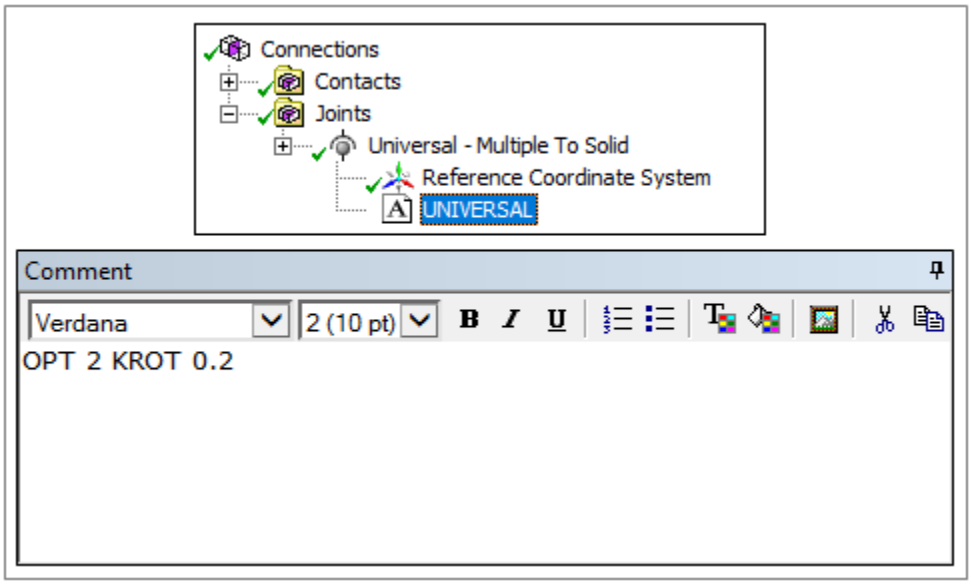
  Similarly, functionalities such as configure, assemble, model topology, or redundancy analysis always consider each multibody part as a single part, regardless stiffness behavior. Joints must be defined between independent parts or multibody parts to use these functionalities.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The Samcef Solver interface supports all joint types except for the fixed joint, slot joint, and the imperfect joints. Only supported joint types are active in the Mechanical interface.

  To maintain consistency with the characteristics of Samcef joints, you must insert a Comment object with the correct name under the joint object. The comment permits you to capture joint properties which are not available in the **Properties** view in Mechanical. The comment functions similarly to a commands object: the content of the comment is appended to the description of the joint in the solver input file. The necessary name for the comment object is listed for each joint type.



Nature of Joint Degrees of Freedom

- For all joints that have both translational degrees of freedom and rotational degrees of freedom, the kinematics of the joint are defined such that the moving coordinate system translates in the reference coordinate system. For example, if a joint is a slot (p. 1301), the translation along X is expressed in the reference coordinate system.

  Once the translation has been applied, the center of the rotation is the location of the moving coordinate system.

- For the Mechanical APDL solver, the relative angular positions for the spherical (p. 1302), general (p. 1310), and bushing (p. 1303) joints are characterized by the Cardan (or Bryant) angles. This requires that the rotations about the local Y axis be restricted between $-\pi/2$ to $+\pi/2$. Therefore, the local Y axis should not be used to simulate the axis of rotation if the expected rotation is large.

Joint Abstraction

Joints are considered as point-to-point in the solution though the user interface shows the actual geometry. Due to this abstraction to a point-to-point joint, geometry interference and overlap between the two parts linked by the joint can be seen during an animation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When using the Ansys Explicit Dynamics solver the contact algorithm will be active for the mesh by default. This means that free DOF's may be restrained by contact forces.

Joint Initial Conditions

The degrees of freedom are determined depending on the selected solver. For the Ansys Rigid Dynamics solver, the degrees of freedom are the relative motion between the parts. For the Ansys Mechanical solver and the Ansys Explicit Dynamics solver, the degrees of freedom are the location and orientation of the center of mass of the bodies.

When initial conditions are applied, there are two means for the Ansys Rigid Dynamics solver to initialize the velocities:

1. A pure kinematic method, only based on the kinematic constraints. It minimizes the position and velocity increments.

2. A method using the inertia matrix. The position increment, scaled by the inertia matrix, is minimized; while the velocity increment is calculated in order to minimize the kinetic energy.

Unless otherwise specified using joint conditions, the Mechanical APDL solver, Ansys Rigid Dynamics solver, and Ansys Explicit Dynamics solver start with initial velocities equal to zero. This has different implications for the solvers. For the Mechanical APDL solver and the Ansys Explicit Dynamics solver, this means that the bodies will be at rest. For the Ansys Rigid Dynamics solver, this means that the relative velocities will be at rest.

Consider, for example, an in-plane double pendulum, with a constant velocity specified for the first grounded link. The two solvers will treat this scenario as follows:

- Using the Ansys Rigid Dynamics solver:

  – If the first method is used, the second link has the same rotational velocity as the first, because the relative velocity is initially equal to zero.

  – If the second method is used, the second link does not start with the same initial velocity as the first link.

- Using the Mechanical APDL solver or the Ansys Explicit Dynamics solver, the second link starts at rest.

Joint DOF Zero Value Conventions

Joints can be defined using one or two coordinate systems: the **Reference Coordinate System** and the **Mobile Coordinate System**.

The use of two coordinate systems can be beneficial in certain situations, such as when a CAD model is not imported in an assembled configuration. Using two coordinate systems also enables you to employ the **Configure** and **Set** features (see Manual Joint Creation (p. 1357)), and it gives you the ability to update a model following a CAD update.

For the Ansys Rigid Dynamics solver, the zero value of the degrees of freedom corresponds to the matching reference coordinate system and moving coordinate system.

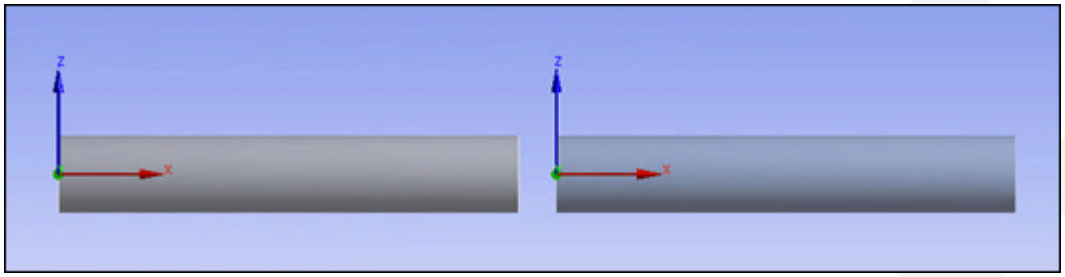
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

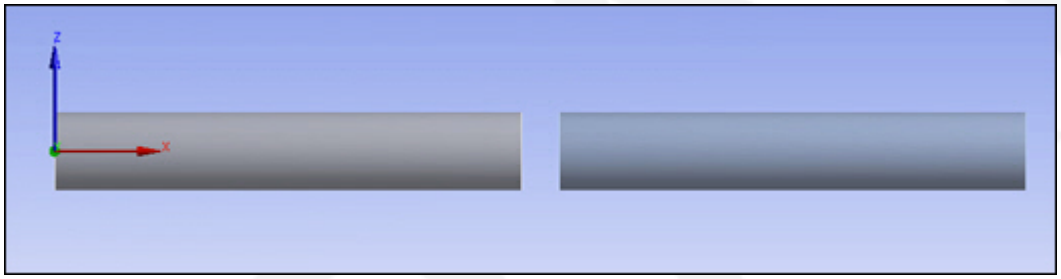
If a joint definition includes only the location of the **Reference Coordinate System** (see Modifying Joint Coordinate Systems (p. 1326)), then the DOF of this joint are initially equal to zero for the geometrical configuration where the joints have been built.

If the **Mobile Coordinate System** is defined using the **Override** option, then the initial value of the degrees of freedom can be a nonzero value.

Consider the example illustrated below. If a Translational (p. 1300) joint is defined between the two parts using two coordinate systems, then the distance along the X axis between the two origins is the joint initial DOF value. For this example, assume the joint initial DOF value is 65 mm.



Conversely, if the joint is defined using a single coordinate as shown below, then the same geometrical configuration has a joint degree of freedom that is equal to zero.



For the Mechanical APDL solver and the Ansys Explicit Dynamics solver, having one or two coordinate systems has no affect. The initial configuration corresponds to the zero value of the degrees of freedom.

Joint Condition Considerations

When applying a **Joint Condition**, behavior varies depending on the solver selected. The following tables demonstrate variations in solver behavior, using the right part of the translational joint illustrated above moving 100 mm towards the other part over a 1 second period. (The distance along the X axis is 65 mm.)

Solver	Displacement Joint Condition	
	Time	Displacement
Ansys Rigid Dynamics – Two Coordinate Systems	0	65
	1	165
Ansys Rigid Dynamics – One Coordinate System	0	0
	1	100
Mechanical APDL, Ansys Explicit Dynamics – Two Coordinate Systems	0	0
	1	100

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mechanical APDL, Ansys Explicit Dynamics – One Coordinate System	0	0
	1	100

You can unify the joint condition input by using a Velocity Joint Condition.

Solver	Velocity Joint Condition	
	Time	Velocity
Ansys Rigid Dynamics – Two Coordinate Systems	0	100
	1	100
Ansys Rigid Dynamics – One Coordinate System	0	100
	1	100
Mechanical APDL, Ansys Explicit Dynamics – Two Coordinate Systems	0	100
	1	100
Mechanical APDL, Ansys Explicit Dynamics – One Coordinate System	0	100
	1	100

Joint Types

You can create the following types of joints in Mechanical:

- Fixed Joint
- Revolute Joint
- Cylindrical Joint
- Translational Joint
- Slot Joint
- Universal Joint
- Spherical Joint
- Planar Joint
- Bushing Joint
- Screw Joint
- Constant Velocity Joint (Homokinetic Joint)
- Distance Joint
- General Joint
- Point on Curve Joint
- Imperfect Joint Types

The above sections include animated joint representations.

Fixed Joint

Constrained degrees of freedom:

All

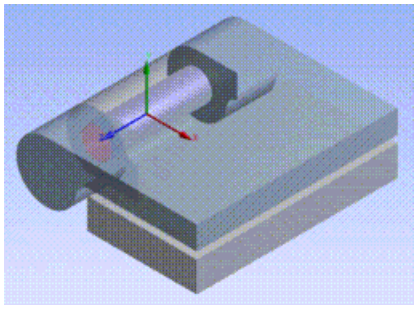
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

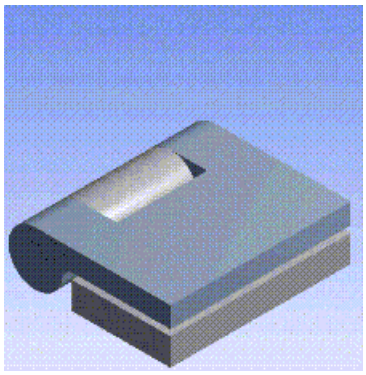
Revolute Joint

Constrained degrees of freedom:

UX, UY, UZ, ROTX, ROTY



Example:



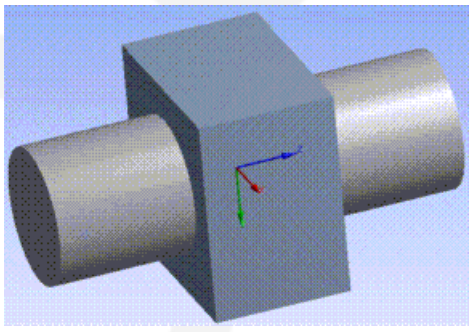
Samcef Solver Comment Object (p. 1295):

REVOLUTE

Cylindrical Joint

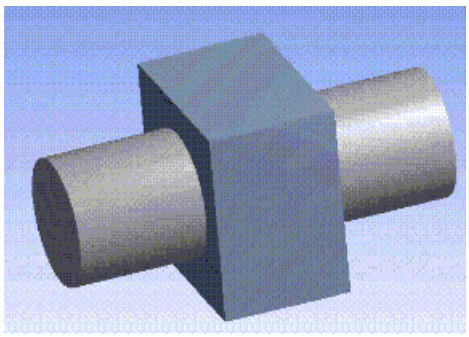
Constrained degrees of freedom:

UX, UY, ROTX, ROTY



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example:



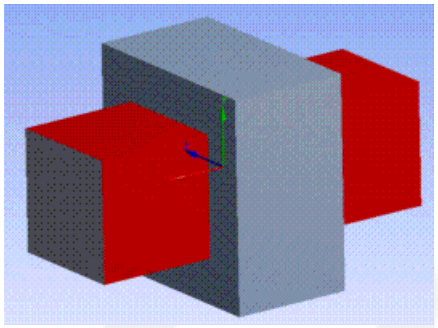
Samcef Solver Comment Object (p. 1295):

```
CYLINDRICAL
```

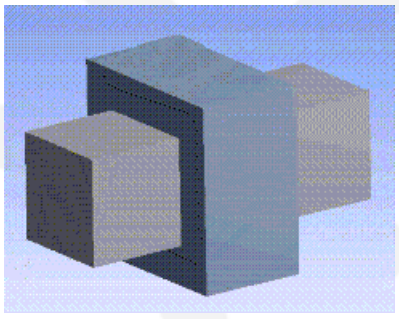
Translational Joint

Constrained degrees of freedom:

UY, UZ, ROTX, ROTY, ROTZ



Example:



Samcef Solver Comment Object (p. 1295):

```
TRANSLATIONAL
```

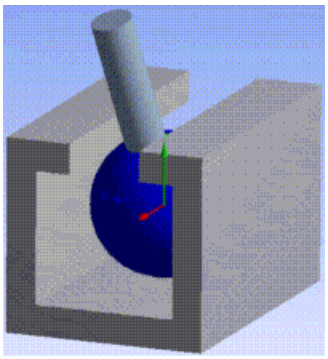
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

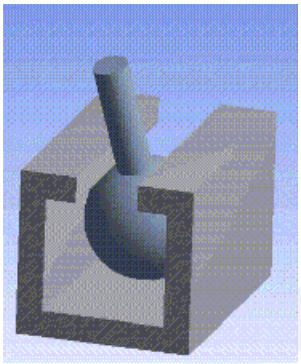
Slot Joint

Constrained degrees of freedom:

UY, UZ



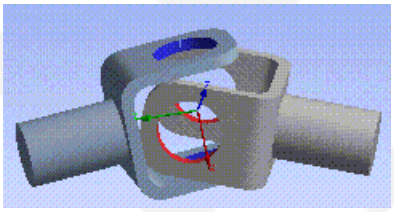
Example:



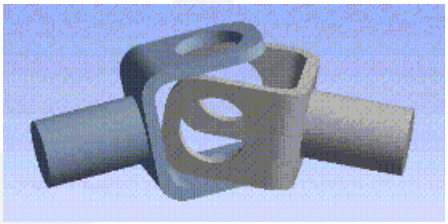
Universal Joint

Constrained degrees of freedom:

UX, UY, UZ, ROTY



Example:



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Samcef Solver Comment Object (p. 1295):

UNIVERSAL

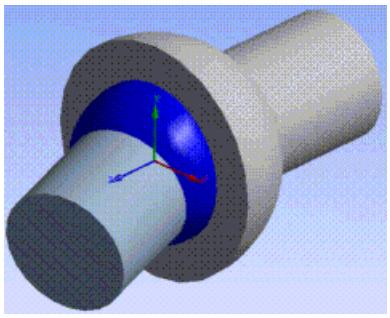
Note:

The Ansys Explicit Dynamics solver does not support Universal Joints.

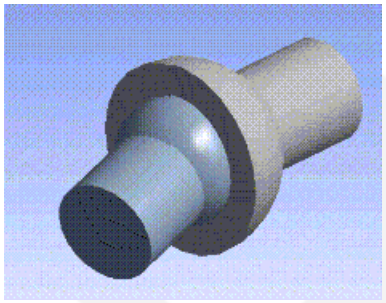
Spherical Joint

Constrained degrees of freedom:

UX, UY, UZ



Example:



Samcef Solver Comment Object (p. 1295):

SPHERICAL or **SPHERICAL FRICTION** (depending on the Samcef joint type you wish to use)

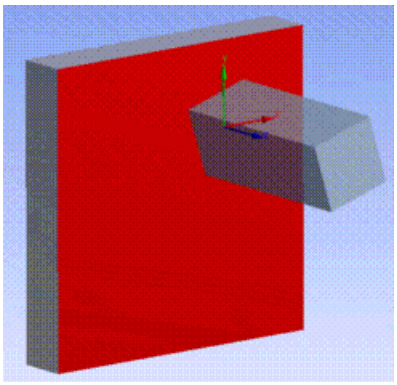
Planar Joint

Constrained degrees of freedom:

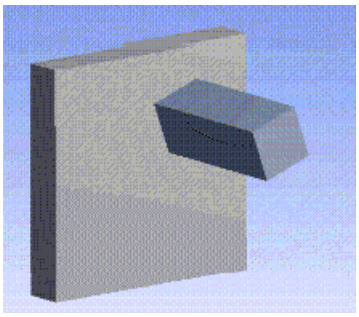
UZ, ROTX, ROTY

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Example:



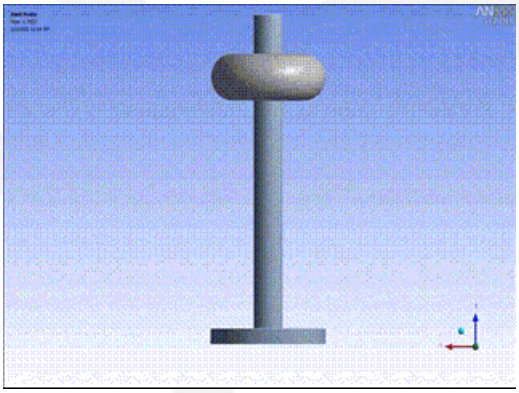
Samcef Solver Comment Object (p. 1295):

PLANAR

Bushing Joint

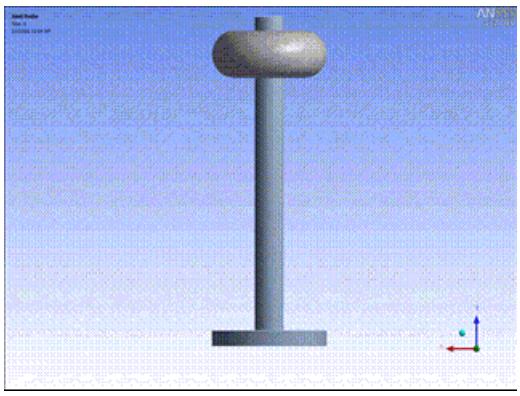
Constrained degrees of freedom:

None



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example:



A **Bushing** has six degrees of freedom, three translations and three rotations, all of which can potentially be characterized by their rotational and translational degrees of freedom as being free or constrained by stiffness.

For a Bushing, the rotational degrees of freedom are defined as follows:

- The first is a rotation around the reference coordinate system X Axis.
- The second is a rotation around the Y Axis after the first rotation is applied.
- The third is a rotation around the Z Axis after the first and second rotations are applied.

The three translations and the three rotations form a set of six degrees of freedom. In addition, the bushing behaves, by design, as an imperfect joint, that is, some forces developed in the joint oppose the motion.

The three translational degrees of freedom expressed in the reference coordinate system and the three rotations are expressed as: Ux, Uy, Uz, and Ψ, Θ, ϕ. The relative velocities in the reference coordinate system are expressed as: Vx, Vy, and Vz. The three components of the relative rotational velocity are expressed as: Ωx, Ωy, and Ωz. Note that these values are not the time derivatives of [Ψ, Θ, ϕ]. They are a linear combination.

The Bushing joint includes a **Formulation** property that enables you to specify a desired element type. The **Formulation** property options include **MPC** (Multi-Point Constraint) and **Bushing**. The **MPC** option uses the MPC184 element and the **Bushing** option uses the COMBI250 element.

Important:

The **Bushing** option is only supported for the following analysis types:

- All Substructure (p. 781) analyses when the **Joint** as included in the **Condensed Part**.
- Modal and Harmonic Response.
- Static Structural when you active Beta Options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For the **MPC Formulation**, the degrees of freedom are expressed in the reference coordinate system as discussed above. The same holds true for the **Bushing Formulation**, however, it uses an element coordinate system as specified by an associated property. This **Element Coordinate System** property is set to the **Global Coordinate System** by default. As desired, you can define an element coordinate system manually.

The forces developed in the Bushing are expressed as:

$$\left\{\frac{F}{T}\right\} = [K]\begin{Bmatrix} U_x \\ U_y \\ U_z \\ \psi \\ \theta \\ \varphi \end{Bmatrix} + [C]\begin{Bmatrix} V_x \\ V_y \\ V_z \\ \Omega_x \\ \Omega_y \\ \Omega_z \end{Bmatrix}$$

Where:

[F] is force and **[T]** is Torque, and **[K]** and **[C]** are 6x6 matrices (defined using Stiffness Coefficients and Dampening Coefficients options). Off diagonal terms in the matrix are coupling terms between the DOFs.

You can use these joints to introduce flexibility to an over-constrained mechanism. Note that very high stiffness terms introduce high frequencies into the system and may penalize the solution time when using the Ansys Rigid Dynamics solver. If you want to suppress motion in one direction entirely, it is more efficient to use Joint DOF Zero Value Conventions (p. 1296) instead of a very high stiffness.

Application

To add a bushing:

1. Select the **Connections** object in the tree and select Bushing from either the **Body-Ground**or **Body-Body** drop-down menus on the Connections Context Tab (p. 50). By default, the **Worksheet** displays.

   Matrix data for the **Stiffness Coefficients** and **Dampening Coefficients** is entered in the **Worksheet**.

   Note:

   - For an **MPC Formulation**, entries are based on a Full Symmetric matrix. For a **Bushing Formulation**, only diagonal entries are available to define.
   - You can change the default display setting of the **Worksheet** using the **Bushing Joint Worksheet View** property in the **Connections** (p. 219) category of the **Options** dialog. Set the property to **No**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

2. As needed, specify a desired element type using the **Formulation** property.

   When you specify the **Bushing** option, the **Element Coordinate System** property also displays. As needed, specify a user-defined coordinate system. The default setting for this property is **Global Coordinate System**.

3. Select the **Graphics** tab. Based on whether your bushing is **Body-Ground** or **Body-Body**, scope the bushing to geometry.

   You can scope a bushing to single or multiple faces, single or multiple edges, or to a single vertex. **Body-Body** scoping requires **Reference** and **Mobile** scoping. The scoping for the **Reference** and **Mobile** sides of the joint cannot be the same.

   Body-Ground assumes that the **Reference** is grounded (fixed). Only the **Mobile** side requires scoping.

4. Specified the various additional properties as needed for your analysis, that may include:

   - **Coordinate System**
   - **Behavior**
   - **Pinball Region**

   See the Joint Properties (p. 1316) section for specific property information.

Non-Linear Force-Deflection Curve

You can use a nonlinear force-deflection curve to simulate a multi-rate bushing with nonlinear stiffness. In Mechanical, you can accomplish this using a linear piecewise curve. You can use this calculation when defining Condensed Parts and using the **MPC Formulation** (only).

Note:

The Ansys Explicit Dynamics solver does not support Tabular Data entry.

To define a nonlinear Force-Deflection curve:

1. In the **Worksheet**, select the cell in which you want to define a non-linear force-deflection curve.
2. Right-click the cell and then select **Tabular**.
3. Enter **Displacement** and **Force** (or **Angle** and **Moment**) values (minimum of two rows of data) in the **Tabular Data** window. The application plots your entries in the **Graph** window.

   Note:

   If tabular entries exist in the stiffness matrix, the Mechanical APDL solver does not account for constant terms and non-diagonal (coupled) terms.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Samcef Solver Comment Object (p. 1295)

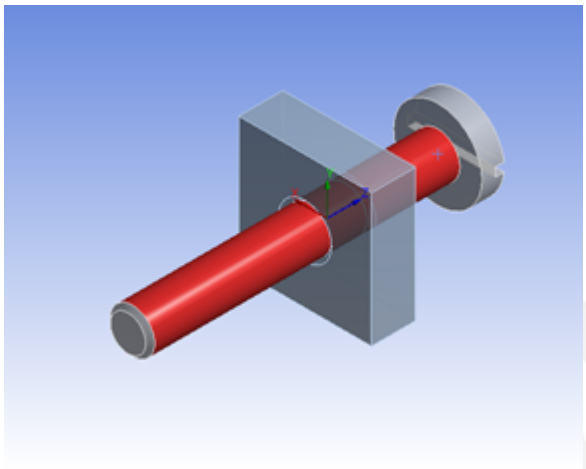
BUSHING

Screw Joint

This joint type is only supported by the Rigid Dynamics solver.

Constrained degrees of freedom:

UX, UY, ROTX, ROTY



UZ=pitch*ROTZ / 2*π (radians) = pitch*ROTZ/360 (degrees)

Specific property:

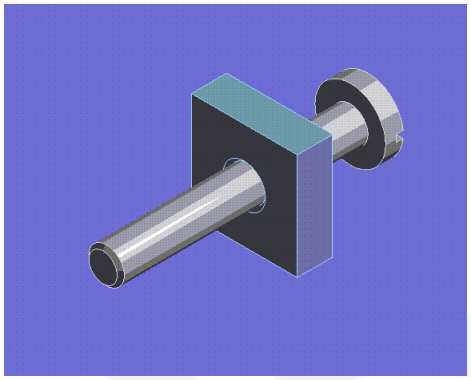
The pitch property in the details view allows you to define the screw pitch (in length unit) per turn. Negative values define a left-handed screw.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Joint"

Definition	
Connection Type	Body-Body
Type	Screw
Suppressed	No
Pitch	10. mm
Reference	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Solid
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Solid
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	All
Stops	

Example:



Constant Velocity Joint (Homokinetic Joint)

This joint type is only supported by the Rigid Dynamics solver.

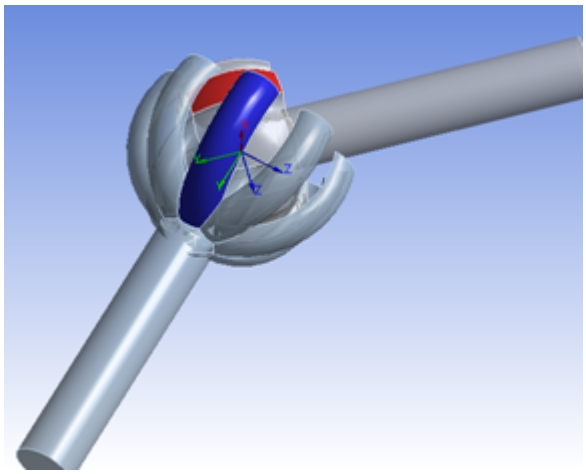
The two shaft orientations are given by the y-axes on the reference and the mobile sides.

Constrained degrees of freedom:

UX, UY, UZ

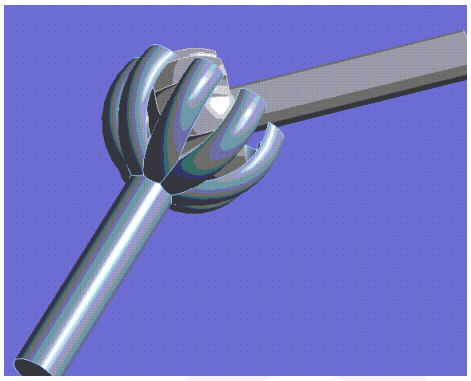
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



$\omega_{Y_{Reference}} = \omega_{Y_{Mobile}}$

Example:



Distance Joint

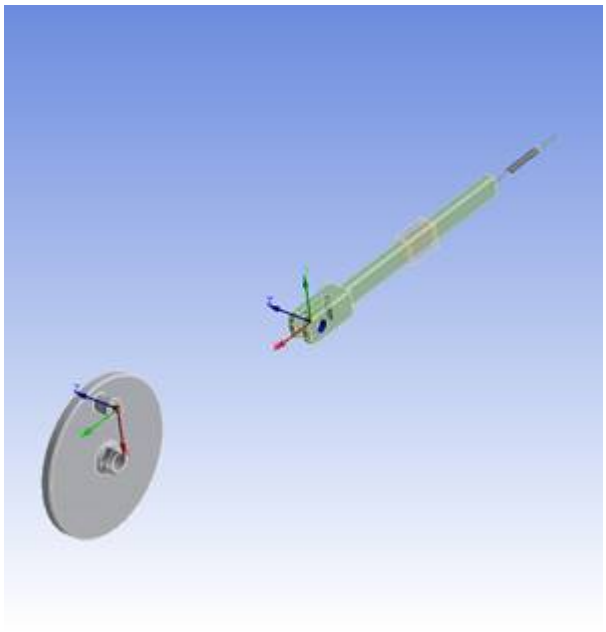
This joint type is only supported by the Rigid Body Dynamics solver.

The length of the joint is specified by the distance between the reference and mobile coordinates systems. Note that as the length must not be equal to zero, and the override option must be used so the reference and mobile are not coincident.

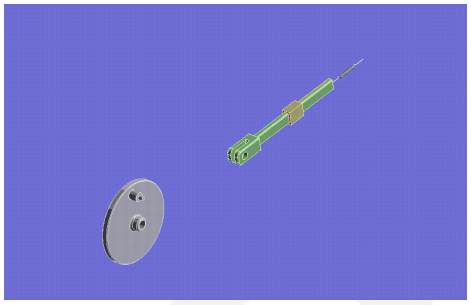
Constrained degrees of freedom:

UX, UY, UZ, ROTX, ROTY, ROTZ are free. Only the distance between reference and mobile coordinates systems is constrained.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Example:



General Joint

Constrained degrees of freedom:

Fix All, Free X, Free Y, Free Z, and Free All.

A general joint has six degrees of freedom, three translations and three rotations, all of which can potentially be characterized by their rotational and translational degrees of freedom as being free or constrained by stiffness. The **Primitive Type** property enables easier selection of fixed or free degrees of freedom. Based on the selected **Primitive Type** option, joint degrees of freedom will be set as following:

Primitive types:

None

All the degrees of freedom are set to fixed by default. You can free the X translation, free the Y translation, free the Z translation and free all rotations. All the translational degrees of freedom can be controlled individually to be fixed or free but there are no individual

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

controls for rotational degrees of freedom. You can either set all rotations fixed, or just one of them (X, Y or Z) free or all free.

Parallel

- Translation X: Free
- Translation Y: Free
- Translation Z: Free
- Rotations: Free Z

In-Plane

- Translation X: Free
- Translation Y: Free
- Translation Z: Fixed
- Rotations: Free All

In-Line

- Translation X: Fixed
- Translation Y: Fixed
- Translation Z: Free
- Rotations: Free All

Orientation

- Translation X: Free
- Translation Y: Free
- Translation Z: Free
- Rotations: Fix All

Also, similar to a bushing, you can enter matrix data for the **Stiffness Coefficients** and **Damping Coefficients** in the **Worksheet**. Coupled terms (off diagonal terms in the matrix) are only allowed when all DOFs are free.

Note:

For a specified joint, if you duplicate it, and/or change the **Type** property to a joint that also supports **Stiffness Coefficients** and **Damping Coefficients** entries in the **Worksheet**, the application automatically retains all worksheet content populated for the DOFs that are set as free for the new joint.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Samcef Solver Comment Object (p. 1295):

GENERAL

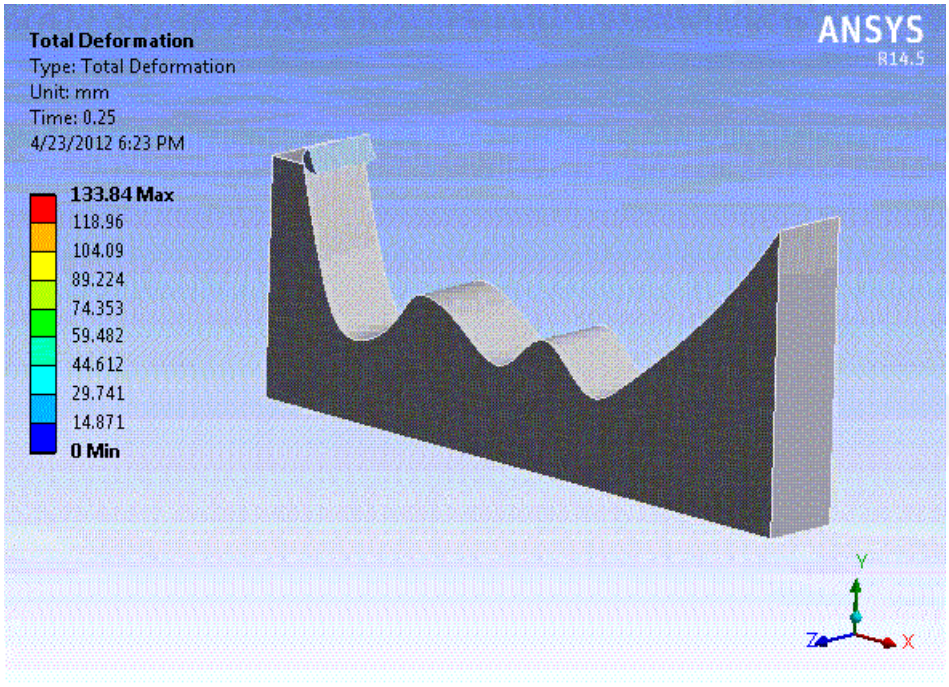
Point on Curve Joint

This joint type is only supported by the Rigid Dynamics solver.

Constrained degrees of freedom:

UY, UZ, ROTX, ROTY, ROTZ if rotations are fixed. UY, UZ if rotations are free.

Example:



Samcef Solver Comment Object (p. 1295):

CURVE or **SLIDER** (depending on the Samcef joint type you wish to use)

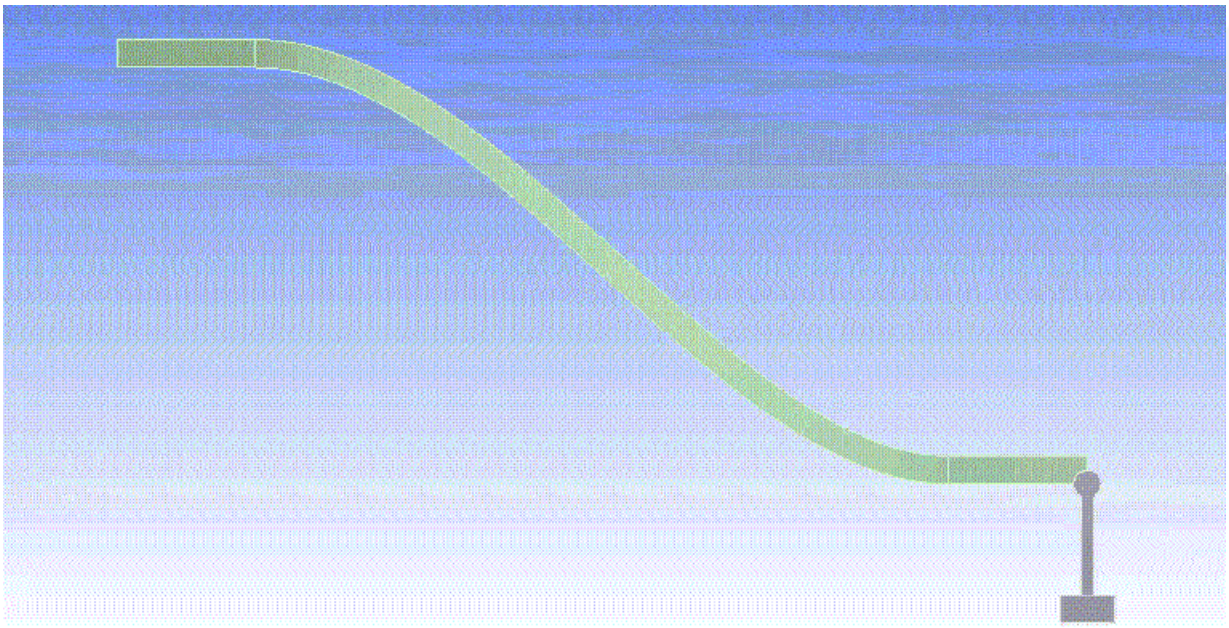
A point on curve joint has one or four degrees of freedom, depending on whether the rotations are fixed or free.

If the rotations are fixed, the point on curve joint has only one degree of freedom, which is the coordinate on the curve. UY and UZ are always equal to zero. ROTX, ROTY, and ROTZ are driven so that the mobile coordinate system of the joint always follows the reference curve.

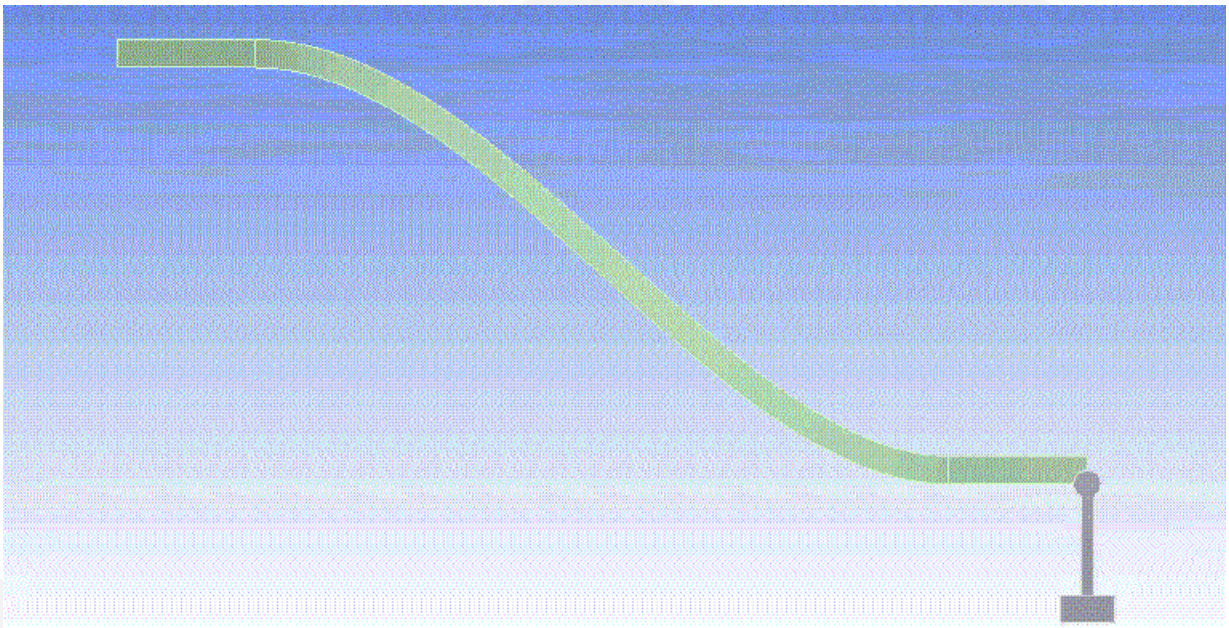
For a point on curve joint, the X axis is always tangent to the reference curve, and the Z axis is always normal to the orientation surface of the joint, pointing outward.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



If the rotations are free, the point on curve joint has four degrees of freedom.



Scoping

You can scope a point on curve joint to a single curve or multiple reference curves. You can have one or more orientation surfaces. The mobile coordinate system has to be scoped to a vertex, and the joint coordinate system has to be positioned and oriented such that:

- The origin is on the curve.
- The X axis is tangent to the curve.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The Z axis is the outer normal to the surface.

Important:

When a model contains a Point On Curve (p. 1312) joint, the Configure and Assemble options (p. 51) are disabled for all the joints. This is also the case for a redundancy analysis that includes a **Point On Curve** joint.

Note:

- The assembly phase may result in minor adjustments to ensure that the mobile coordinate system is properly positioned.
- If multiple curves are used, it is recommended that the curves have a curvature continuity. Otherwise, the fixed rotations options will lead to rotational velocity jumps in the point on curve that may affect other connected joints. In this situation, results may not be fully determined, and they may vary with the model topology.

Imperfect Joint Types

The following joint types are specialized joints dedicated to model clearance in revolute or cylindrical joints (see Radial Gap Stop (p. 1363)). These joint types are only supported by the Rigid Dynamics Solver:

In-Plane Radial Gap

Spherical Gap

Radial Gap

In-Plane Radial Gap

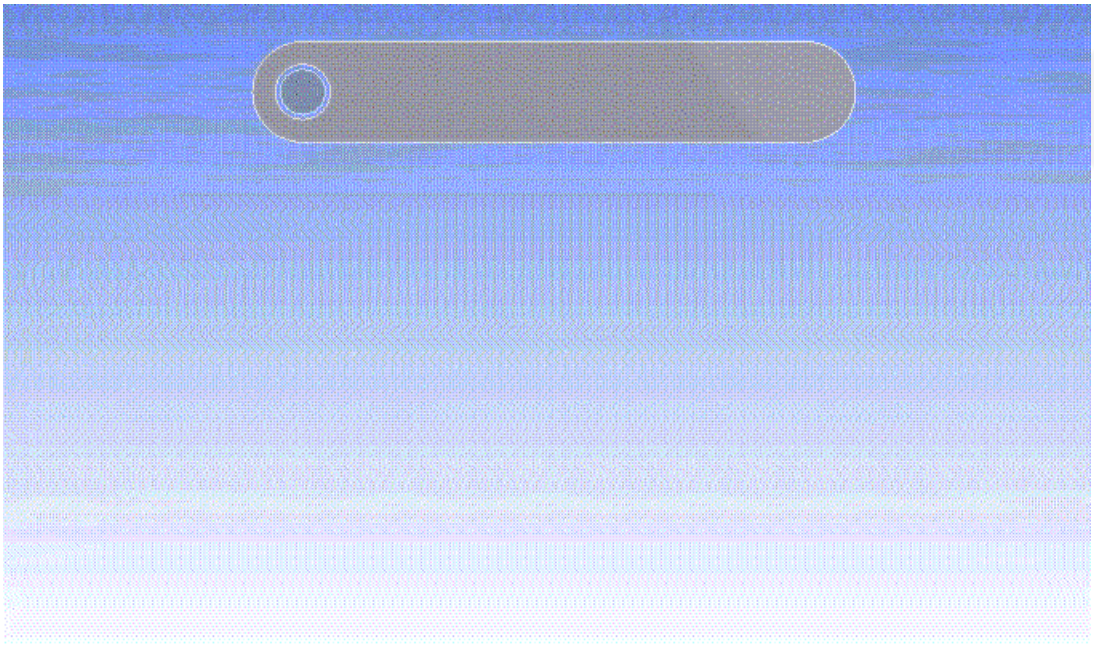
Constrained degrees of freedom:

UZ, ROTX, ROTY (similar to planar joint)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Example:

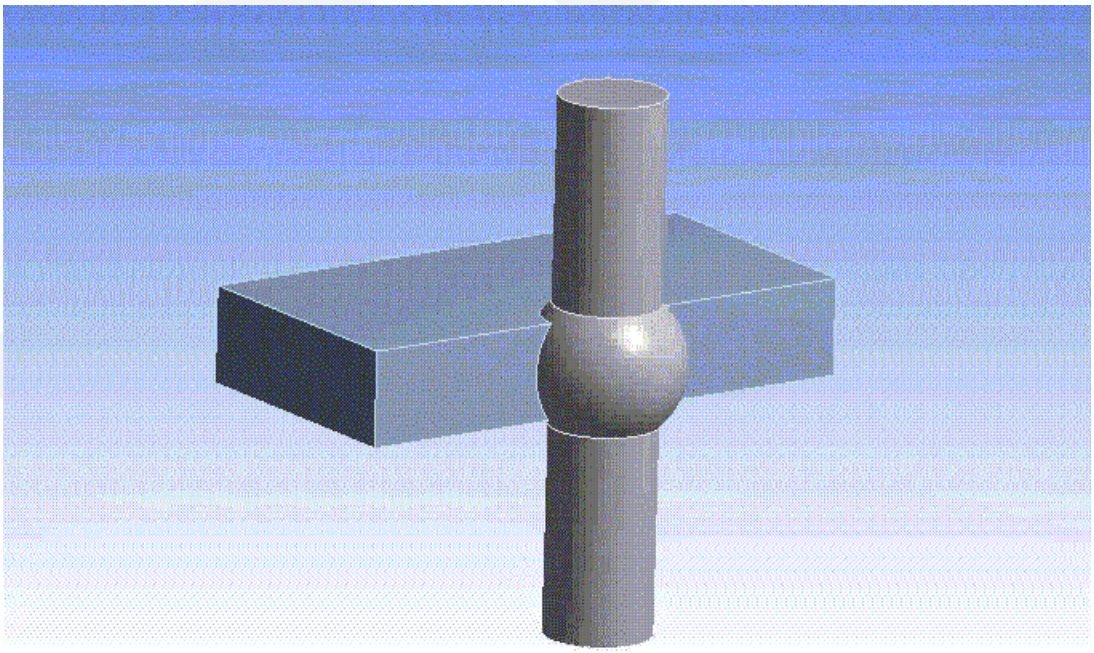


Spherical Gap

Constrained degrees of freedom:

UX, UY, UZ (similar to spherical joint)

Example:



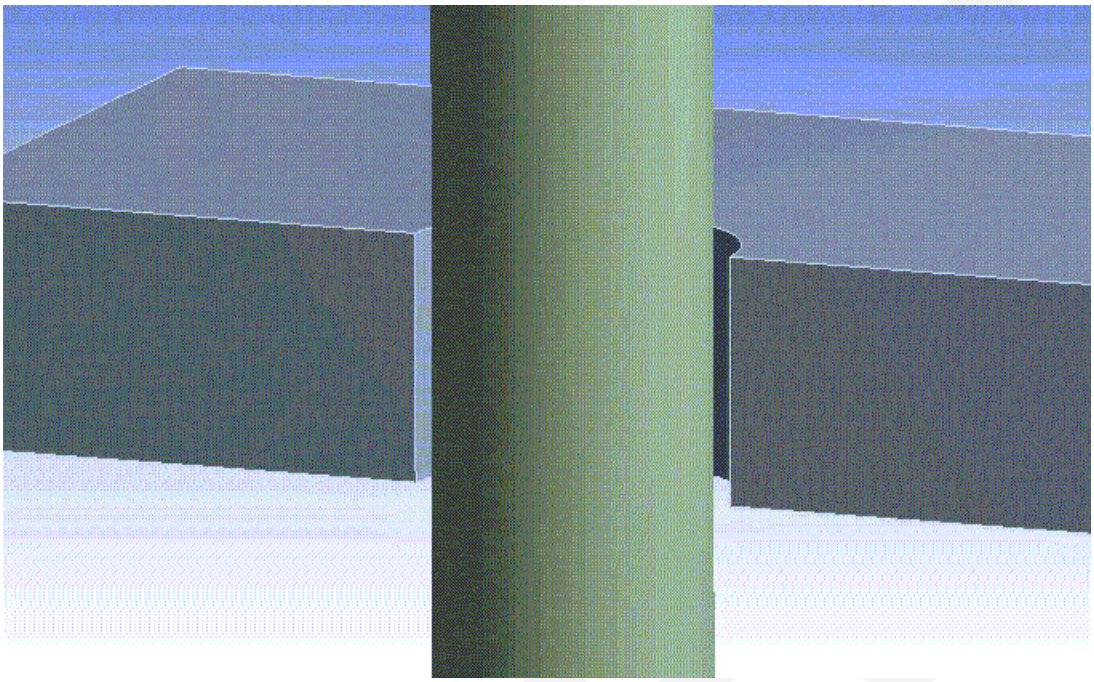
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Radial Gap

Constrained degrees of freedom:

fix or free UZ

Example:



Joint Properties

This section describes the **Details** pane properties associated with a **Joint** object.

Category	Property Name and Description
Definition	**Connection Type** The **Connection Type** property specifies the joint as either a **Body-Body** scoping (multiple faces) or a **Body-Ground** scoping (multiple faces). When defined as **Body-Body**, you need to define **Reference** category and **Mobile** category properties. When you specify the **Connection Type** as body-to-ground, the application assumes that the reference element of the joint is grounded (fixed). **Type** The **Type** property provides a drop-down list of joint type options. See the Specifying Remote Points (p. 1089) section for descriptions of each type. **Note:** The General (p. 1298) joint enables you to specify each degree of freedom as being either **Fixed** or **Free**.

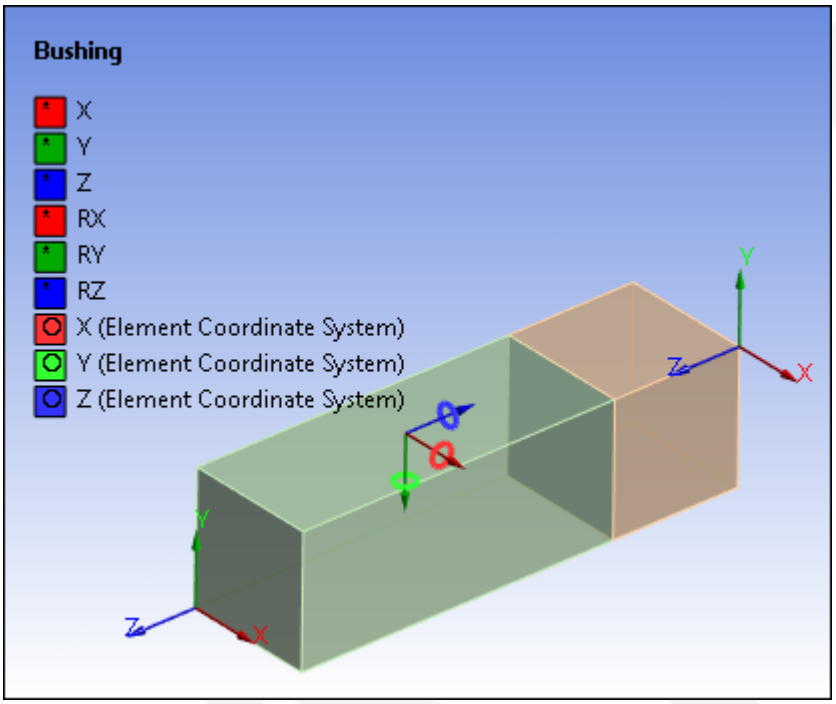
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

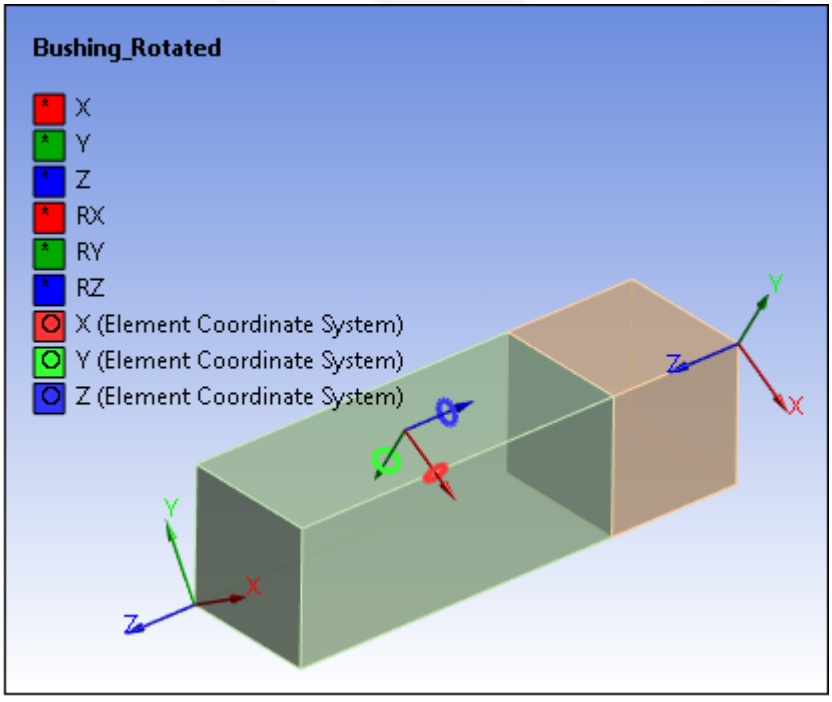
Published: 2023-10-23

Category	Property Name and Description
	Formulation You use this property to specify a desired element type for the Bushing Joint. The options for this property include **MPC** (Multi-Point Constraint) and **Bushing**. These options enable you to specify the underlying element type for the Bushing Joint. The **MPC** option uses the MPC184 element and the **Bushing** option uses the COMBI250 element. **Important:** The **Bushing** option is only supported for the following analysis types: • All Substructure (p. 781) analyses when the **Joint** as included in the **Condensed Part**. • Modal and Harmonic Response. • Static Structural when you active Beta Options. When you specify the **Bushing** option, the **Element Coordinate System** property also displays. As needed, specify a user-defined coordinate system. The default setting for this property is **Global Coordinate System**. **Important:** When you set the **Formulation** property to **Bushing**: • The **Element Coordinate System** is the first coordinate system used to apply rotation. • This is followed by nodal rotations per promoted Remote Points (**Reference** or **Mobile**), such as the **NROTAT** command that rotates nodal coordinate systems into the active system. In addition, rotations from these remote points modify the element matrix (see the **Input Data** topic of the COMBI250 element reference). • This is then followed by the normal processing sequence for rotations. Here is an example of Element Coordinate System annotations ("rings" about each axis), prior to and after rotation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description





Solver Element Type (MAPDL Solver Only)

The **Solver Element Type** property enables you to specify the type of element to use in a fixed joint. Options include:

- **Program Controlled** (default): When selected, an Mechanical APDL joint element is used unless a rigid body exists in the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	• **Joint Element**: this selection uses the MPC184 element. This option enables you to use the Joint probe. In addition, this option may allow you to experience convergence improvements if you are attaching a shell or line body to a node or vertex. • **Contact/Direct**: this selection uses the TARGE170 element for Body-Body joints and a DOF constraint for Body-Ground joints. This option helps to avoid solver pivoting as well as MPC overconstraint. **Torsional Stiffness** The **Torsional Stiffness** property defines the measure of the resistance of a shaft to a twisting or torsional force. You can add torsional stiffness only for cylindrical (p. 1299) and revolute (p. 1299) joints. **Torsional Damping** The **Torsional Damping** property defines the measure of resistance to the angular vibration to a shaft or body along its axis of rotation. You can add torsional damping only for cylindrical (p. 1299) and revolute (p. 1299) joints. **Suppressed** Includes or excludes the joint object in the analysis. **Element APDL Name** This optional property enables you to manually define an APDL parameter (in the input file) and assign its value to the element of the Joint. This facilitates easy programmatic identification of the Joint's element for later use/reference in a Commands (p. 1310) object.
Reference	**Scoping Method** This property enables you to choose to scope using a **Geometry Selection** (default), **Named Selection**, or a user-defined **Remote Point**. **Note:** If you scope a joint to a user-defined **Remote Point**, it is required that the remote point be located at the origin (0.0, 0.0, 0.0) of the **Reference Coordinate System** of the remote point. **Applied By** This property specifies the joint as a **Remote Attachment** (default) or a **Direct Attachment**. The **Remote Attachment** option uses either a user-defined or a system-generated **Remote Point** as a scoping mechanism. **Remote Attachment** is the required **Applied By** property setting if the geometry scoping is to a single face or multiple faces, a single edge or multiple edges, or multiple vertices. The **Direct Attachment** option enables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	you to scope directly to a single vertex (Geometry) or a node (using an individually selected node or a node-based Named Selection) for flexible bodies (only) on your model. **Direct Attachment** is not allowed if scoped to solid bodies, as they do not have rotational degrees of freedom. **Note:** Direct Attachment is not allowed for the Explicit Dynamics solver. **Scope (or Reference Component or Remote Point)** Based on the selected **Scoping Method**, this property displays as either " **Scope** ", " **Reference Component** ", or " **Remote Points** ". When **Geometry Selection** is selected as the Scoping Method, this property displays with the label " **Scope** " and enables you to define the geometry to which the joint is applied. Once a geometry is selected, click in the **Scope** field and then click **Apply**. When **Named Selection** is selected as the **Scoping Method**, this property provides a drop-down list of available user-defined Named Selections. When **Remote Point** is selected as the Scoping Method, this property displays with the label " **Remote Points** ". This property provides a drop-down list of available user-defined Remote Points. This property is not available when the **Applied By** property is specified as **Direct Attachment**. **Body** This read-only property displays the corresponding part/geometry name. **Coordinate System** The scoping of a joint must be accompanied by the definition of a joint coordinate system. This coordinate system defines the location of the joint. It is imperative that the joint coordinate system be fully associative (p. 1192) with the geometry, otherwise, the coordinate system could move in unexpected ways when the **Configure** tool is used to define the initial position of the joint (see the Applying Joints (p. 2303) section). A warning message is issued if you attempt to use the **Configure** tool with a joint whose coordinate system is not fully associative. Under the **Reference** category, the **Coordinate System** property provides a default **Reference Coordinate System**. This coordinate system accompanies a joint when the joint is added to the tree. This applies for joints whose **Connection Type** is either **Body-Ground** or **Body-Body**. When a joint is added, an associated coordinate system is automatically generated at a location based on the selected geometry (face, edge, or vertex). You can modify the Reference Coordinate System's orientation axis by modifying the details of the **Reference Coordinate System** object contained in the joint object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	Additional information about Modifying Joint Coordinate Systems (p. 1326) is also available, including the following topics: • Modify Coordinate System Geometry Scoping (p. 1299) • Change Coordinate System Orientation (p. 1328) Scoping a joint directly to a vertex or a node using the **Direct Attachment** option fixes the coordinate system to that location. Note that the **Reference Coordinate System** property displays automatically and is read-only. **Behavior** For remote attachments, use the **Behavior** property to specify the scoped geometry as either **Rigid**, **Deformable**, or **Beam**. If the **Scope Method** property of the Joint is set to **Remote Point**, the Joint will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties. Refer to the Geometry Behaviors (p. 1095) section for more information. **Formulation** When the **Behavior** property is set to either **Rigid** or **Deformable**, you use this property to specify the contact algorithm used for a particular reference computation. If you set the **Scope Method** property to Remote Point (p. 1090), the Joint uses the **Formulation** defined for the **Remote Point**. Property options include: • **MPC** (default): This option creates multipoint constraint equations internally during the Mechanical APDL solution to tie the bodies together. • **Lagrange Multiplier**: This option enforces zero penetration when the contact is closed, making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. This formulation helps to overcome over-constraint problems better than the MPC formulation. For more information, see the Selecting a Contact Algorithm (KEYOPT(2)) topic in the *Mechanical APDL Contact Technology Guide*. **Relaxation Method** This property is only available when the **Formulation** property is set to **MPC**. You use this property to eliminate overconstraint present in the joint reference condition. Property options include **No** (default) and **Yes**. If you set the **Scope Method** property to **Remote Point**, the joint uses the setting of the **Relaxation Method** property defined for the **Remote Point**. For more information, see the Overconstraint Detection and Elimination section of the *Mechanical APDL Contact Technology Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	Pinball Region Use the **Pinball Region** property to define where the joint attaches to face(s) if the default location is not desirable. By default, the *entire* face is tied to the joint element. This may not be desirable, warranting the input of a **Pinball Region** setting, for the following reasons: • If the scoping is to a topology with a large number of nodes, this can lead to an inefficient solution in terms of memory and speed. • Overlap between the joint scoped faces and other displacement type boundary conditions can lead to over constraint and as a result, solver failures. **Note:** • The **Pinball Region**, **Behavior**, **Formulation**, and **Relaxation Method** settings are applicable to underlying bodies that are flexible. • If a Joint's **Reference** and **Mobile** category are scoped to separate Remote Points (p. 1089), the **Pinball Region**, **Behavior**, **Formulation**, and **Relaxation Method** properties for each category become read-only and are set to the respective remote points. • The **Pinball Region**, **Behavior**, **Formulation** and **Relaxation Method** properties are not applicable when the **Applied By** property is set to **Direct Attachment**.
Mobile	**Scoping Method** This property enables you to choose to scope using a **Geometry Selection** (default), **Named Selection**, or a user-defined **Remote Point**. **Note:** If you scope a joint to a user-defined **Remote Point**, it is required that the remote point be located at the origin (0.0, 0.0, 0.0) of the **Reference Coordinate System** of the remote point. **Applied By** This property specifies the joint as a **Remote Attachment** (default) or a **Direct Attachment**. The **Remote Attachment** option uses either a user-defined or a system-generated **Remote Point** as a scoping mechanism. **Remote Attachment** is the required **Applied By** property setting if the geometry scoping is to a single face or multiple faces, a single edge or multiple edges, or a single vertex or multiple vertices. The **Direct**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	Attachment option allows you to scope directly to a single vertex (Geometry) or a node (using an individually selected node or a node-based Named Selection) to flexible bodies (only) on your model. Direct Attachment is not allowed if scoped to solid bodies, as they do not have rotational degrees of freedom. **Note:** Direct Attachment is not allowed for the Explicit Dynamics solver. **Scope (or Mobile Component or Remote Point)** Based on the selected **Scoping Method**, this property displays as either " **Scope** ", " **Mobile Component** ", or " **Remote Points** ". When **Geometry Selection** is selected as the Scoping Method, this property displays with the label " **Scope** " and enables you to define the geometry to which the joint is applied. Once a geometry is selected, click in the **Scope** field and then click **Apply**. When **Named Selection** is selected as the **Scoping Method**, provides a drop-down list of available user-defined Named Selections. When **Remote Point** is selected as the Scoping Method, this property displays with the label " **Remote Points** ". This property provides a drop-down list of available user-defined Remote Points. This property is not available when the **Applied By** property is specified as **Direct Attachment**. **Body** This property is available under both the **Reference** and **Mobile** categories. This read-only property displays the corresponding part/geometry name. **Coordinate System** The **Mobile** category provides the support for the relative motion between the parts of a joint. A **Mobile Coordinate System** is automatically defined but is only displayed in the tree when the **Initial Position** property is set to **Override**. Scoping a joint directly to a vertex or a node using the **Direct Attachment** option fixes the coordinate system to that location. When scoping directly to a node or vertex using the **Direct Attachment** option, the default setting for the **Initial Position** property is **Override** even though the **Initial Position**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property Name and Description
	property doesn't display in the Details. Rather, the Coordinate System automatically displays and is read-only. **Note:** For the Mechanical APDL solver, Body-Ground joints use the Coordinate System defined on the **Mobile** side if Override is active, when scoped to a Remote Point, or when using the Direct Attachment option. **Initial Position** This property applies to remote attachments only (direct attachments fix the coordinate system). It provides a drop-down list with the options **Unchanged** and **Override**. The **Unchanged** option indicates the use of the same coordinate system for the **Reference** category and the **Mobile** category and the **Override** option causes a **Coordinate System** property to display in the **Mobile** category with the default setting **Mobile Coordinate System**. **Caution:** If you are scoping a joint to a Remote Point (p. 1095), you cannot scope the **Initial Position** setting of a Joint's **Mobile** category as **Unchanged**. This is also true when the **Direct Attachment** option is used because the **Initial Position** property is not available (**Override** is active). **Behavior** For remote attachments, use the **Behavior** property to specify the scoped geometry as either **Rigid**, **Deformable**, or **Beam**. If the **Scope Method** property of the Joint is set to **Remote Point**, the Joint will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties. Refer to the Geometry Behaviors (p. 1095) section for more information. **Formulation** When the **Behavior** property is set to either **Rigid** or **Deformable**, you use this property to specify the contact algorithm used for a particular reference computation. If you set the **Scope Method** property to Remote Point (p. 1090), the Joint uses the **Formulation** defined for the **Remote Point**. Property options include: • **MPC** (default): This option creates multipoint constraint equations internally during the Mechanical APDL solution to tie the bodies together.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	• **Lagrange Multiplier**: This option enforces zero penetration when the contact is closed, making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. This formulation helps to overcome over-constraint problems better than the MPC formulation. For more information, see the Selecting a Contact Algorithm (KEYOPT(2)) topic in the *Mechanical APDL Contact Technology Guide*. **Relaxation Method** This property is only available when the **Formulation** property is set to **MPC**. You use this property to eliminate overconstraint present in the joint mobile condition. Property options include **No** (default) and **Yes**. If you set the **Scope Method** property to **Remote Point**, the joint uses the setting of the **Relaxation Method** property defined for the **Remote Point**. For more information, see the Overconstraint Detection and Elimination section of the *Mechanical APDL Contact Technology Guide*. **Pinball Region** For remote attachments, use the **Pinball Region** property to define where the joint attaches to face(s) if the default location is not desirable. By default, the *entire* face is tied to the joint element. This may not be desirable, warranting the input of a **Pinball Region** setting, for the following reasons: • If the scoping is to a topology with a large number of nodes, this can lead to an inefficient solution in terms of memory and speed. • Overlap between the joint scoped faces and other displacement type boundary conditions can lead to over constraint and as a result, solver failures. **Note:** • The **Pinball Region**, **Behavior**, **Formulation**, and **Relaxation Method** properties are not visible when the **Applied By** method is **Direct Attachment**. • The **Pinball Region** and **Behavior** settings are applicable to underlying bodies that are flexible. • If a Joint's **Reference** and **Mobile** category are scoped to separate Remote Points (p. 1090), the **Behavior** and ***Pinball Region*** properties for each category become read-only and are set to the respective remote points.
Stops	See the Joint Stops and Locks (p. 1360) section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Modifying Joint Coordinate Systems

For either **Reference** or **Mobile** joint coordinate systems, both the location and the orientation of the coordinate system can be changed as shown below.

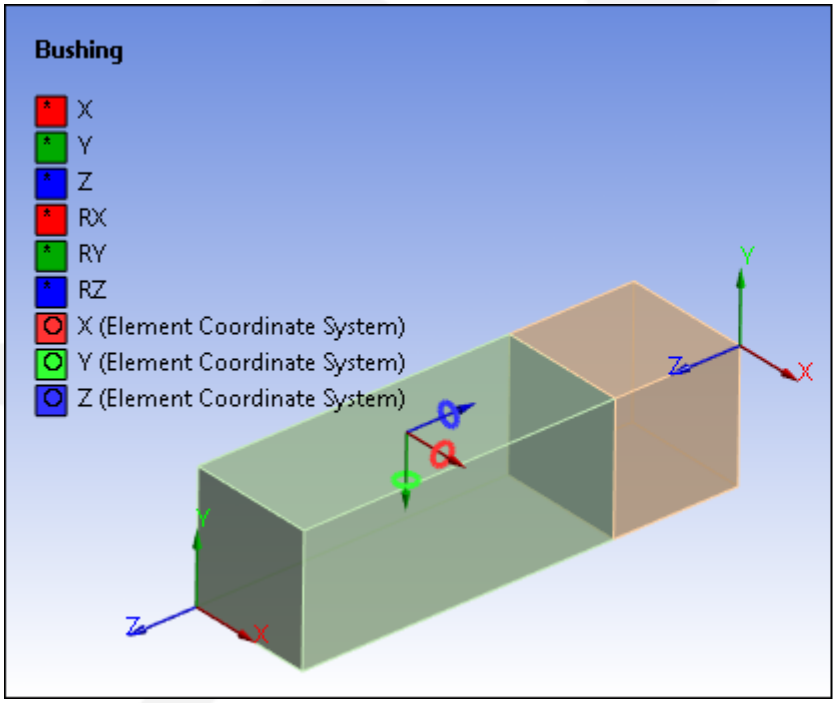
Important:

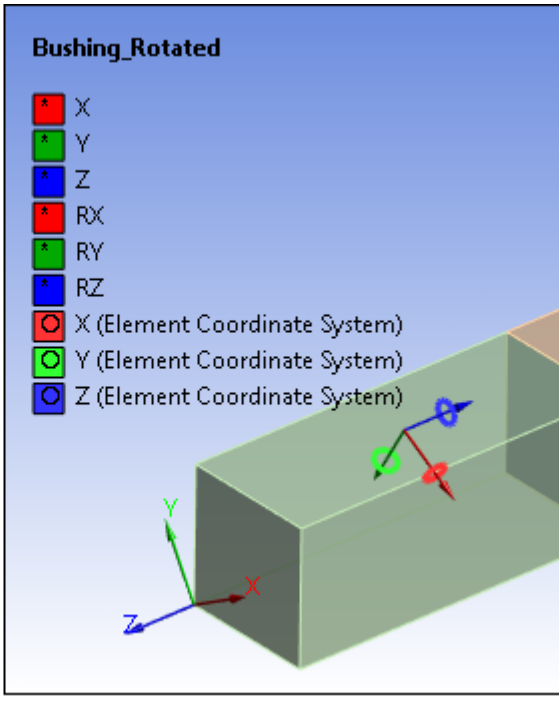
For a **Bushing** joint when you set the **Formulation** property to **Bushing**, the application uses the element COMBI250. As a result:

- The **Element Coordinate System** is the first coordinate system used to apply rotation.
- This is followed by nodal rotations per promoted Remote Points (**Reference** or **Mobile**), such as the **NROTAT** command that rotates nodal coordinate systems into the active system. In addition, rotations from these remote points modify the element matrix (see the Joint Stiffness (p. 1331) section as well as the **COMBI250 Input Data** topic in the COMBI250 section of the *Mechanical APDL Element Reference*).

  In addition, rotations from these remote points modify the element matrix (see the Joint Stiffness section as well as the **Input Data** topic of the COMBI250 element reference).
- This is then followed by the normal processing sequence for rotations.

Here is an example of Element Coordinate System annotations ("rings" about each axis), prior to and after rotation.



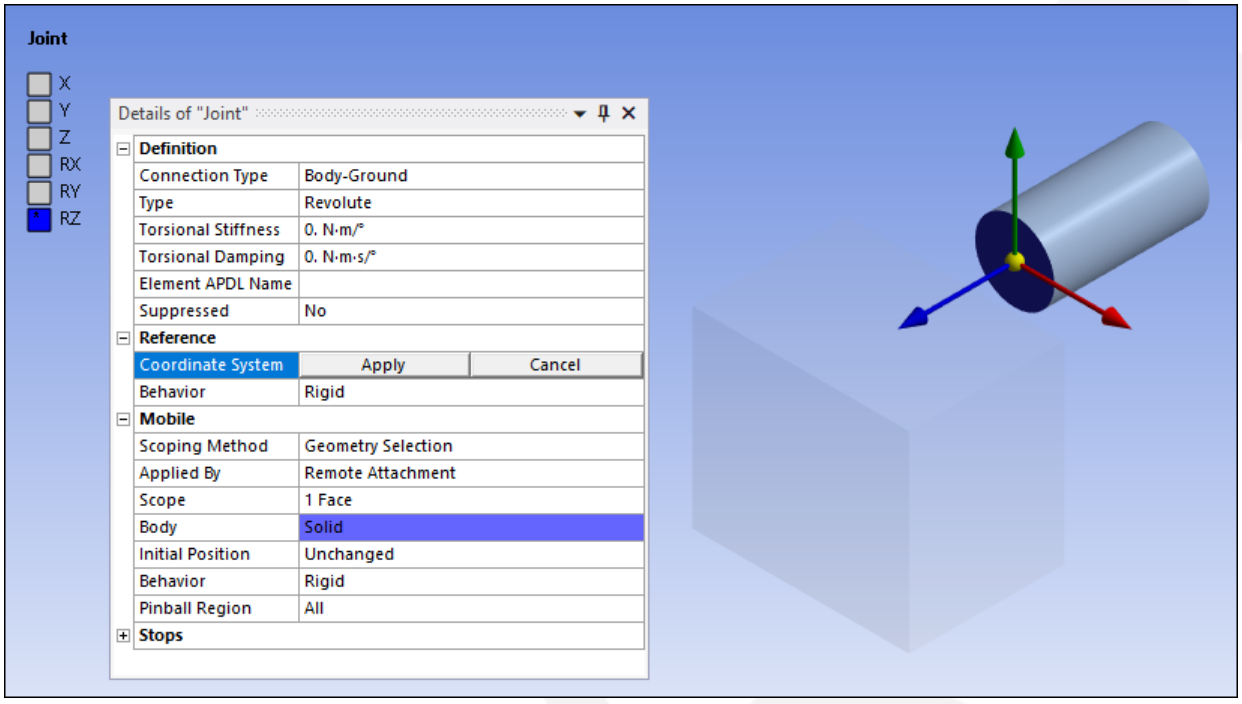


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

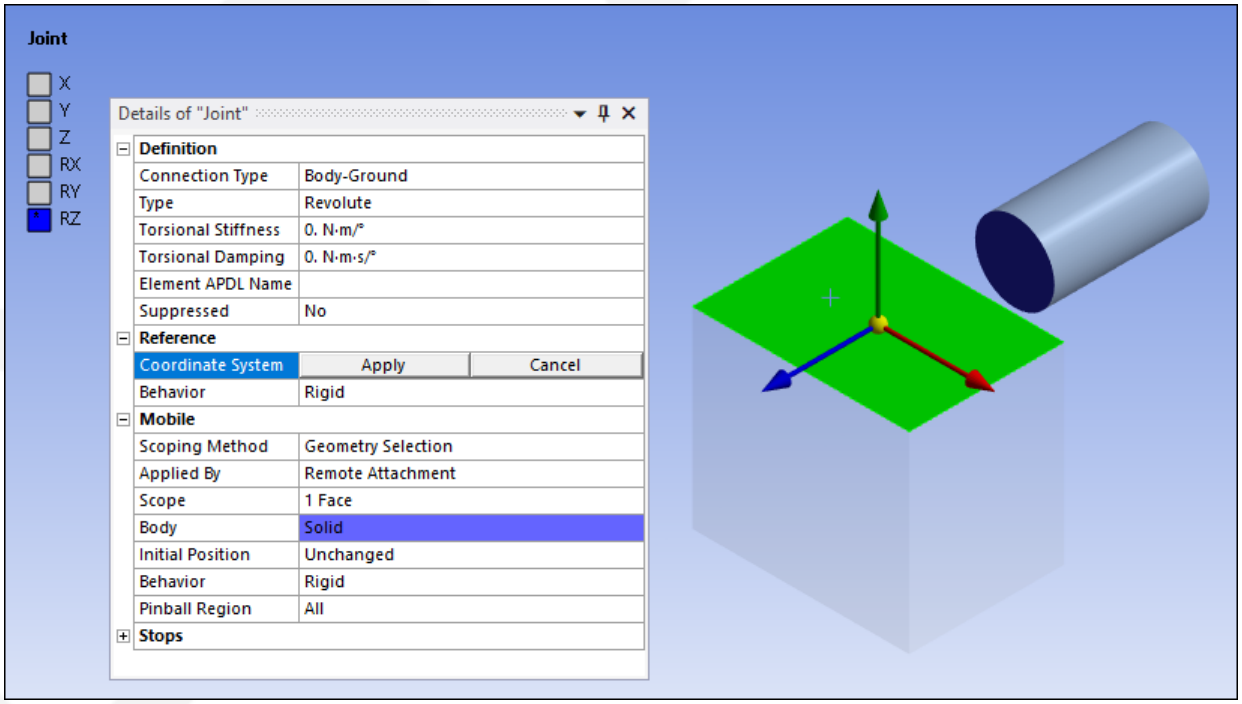
Published: 2023-10-23

To move a joint coordinate system to a particular face:

1. Highlight the **Coordinate System** field in the Details view of the **Joint** object. The origin of the coordinate system will include a yellow sphere indicating that the movement "mode" is active.

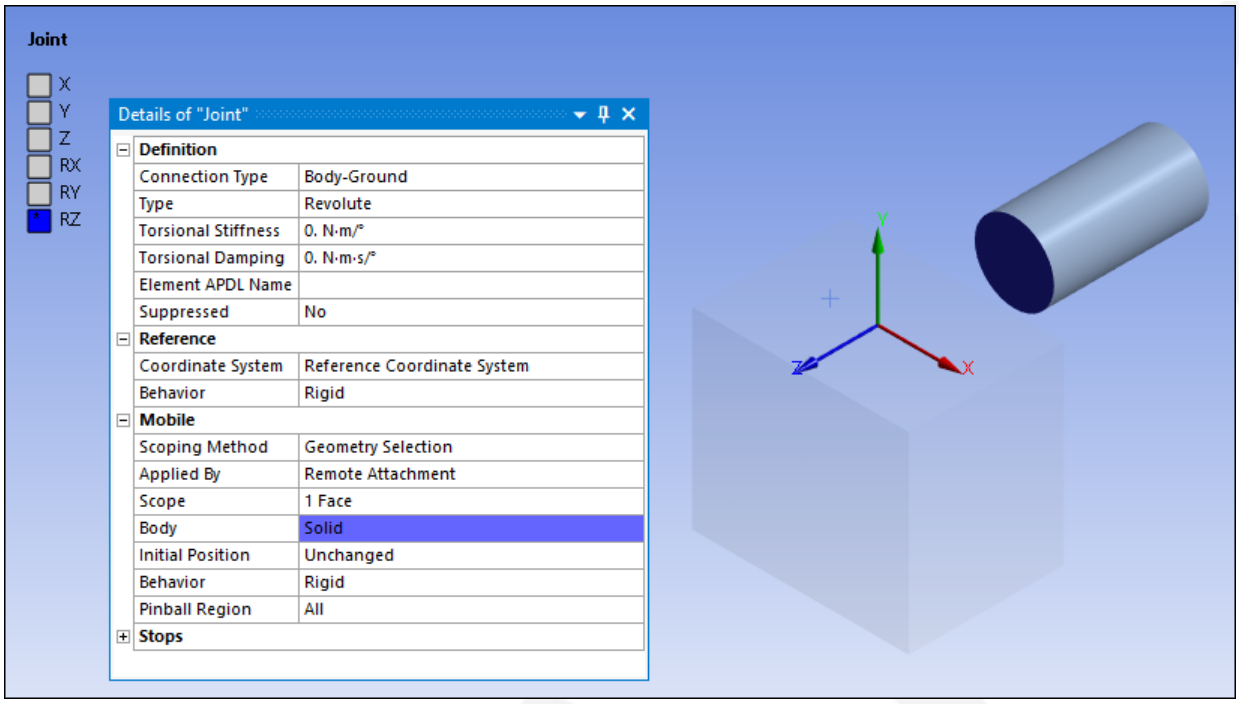


2. Select the face that is to be the destination of the coordinate system. The coordinate system in movement mode relocates to the centroid of the selected face.



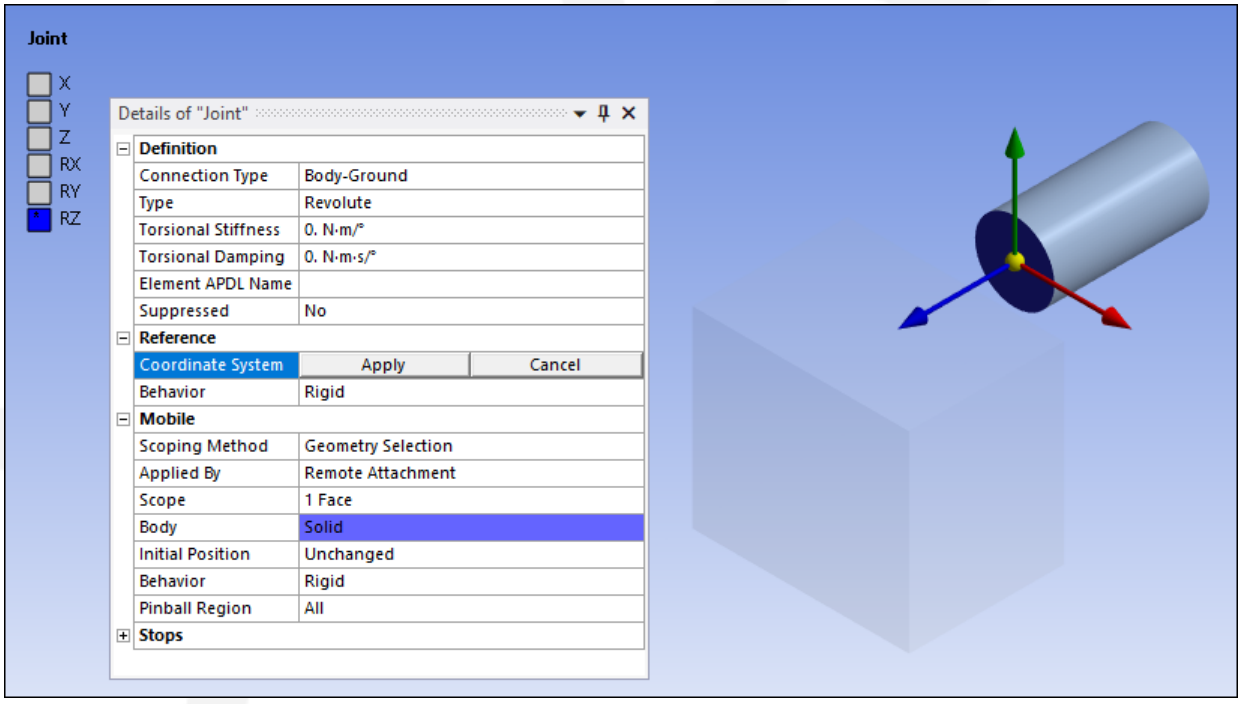
3. Click the **Apply** button. The image of the coordinate system changes from movement mode to a permanent presence at the new location.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



To change the orientation of a joint coordinate system:

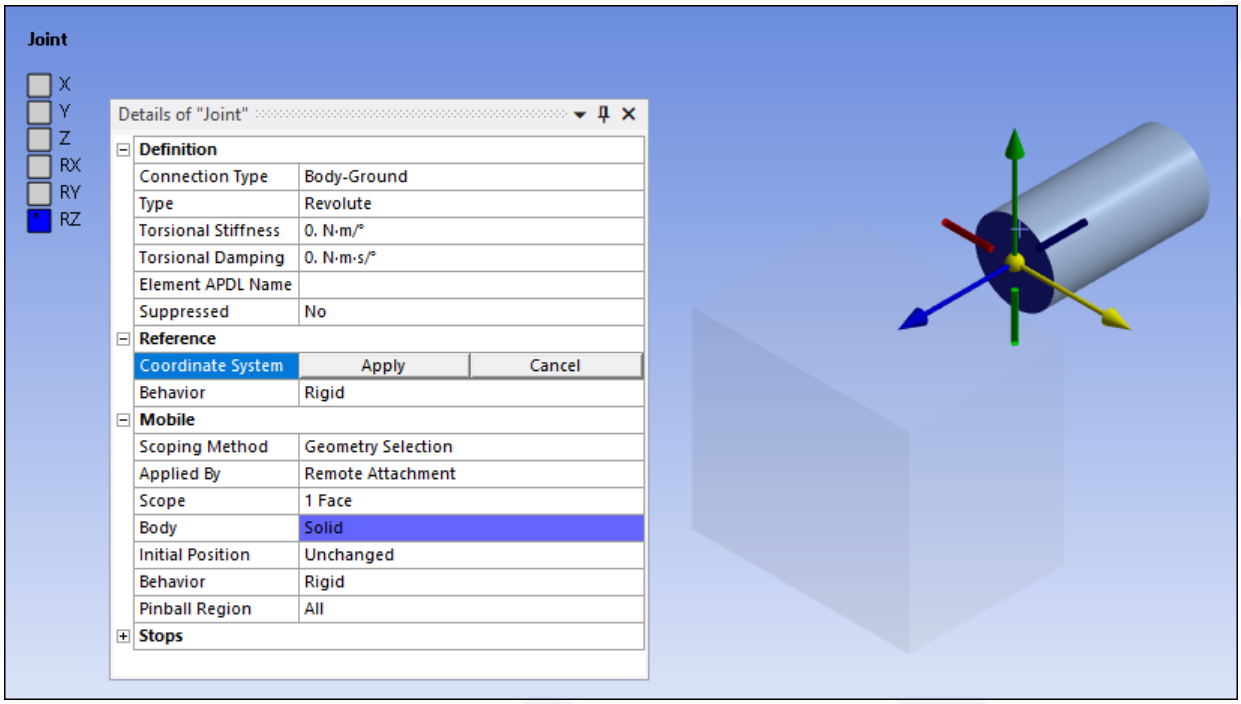
1. Highlight the **Coordinate System** field in the Details view of the **Joint** object. The origin of the coordinate system will include a yellow sphere indicating that the movement "mode" is active.



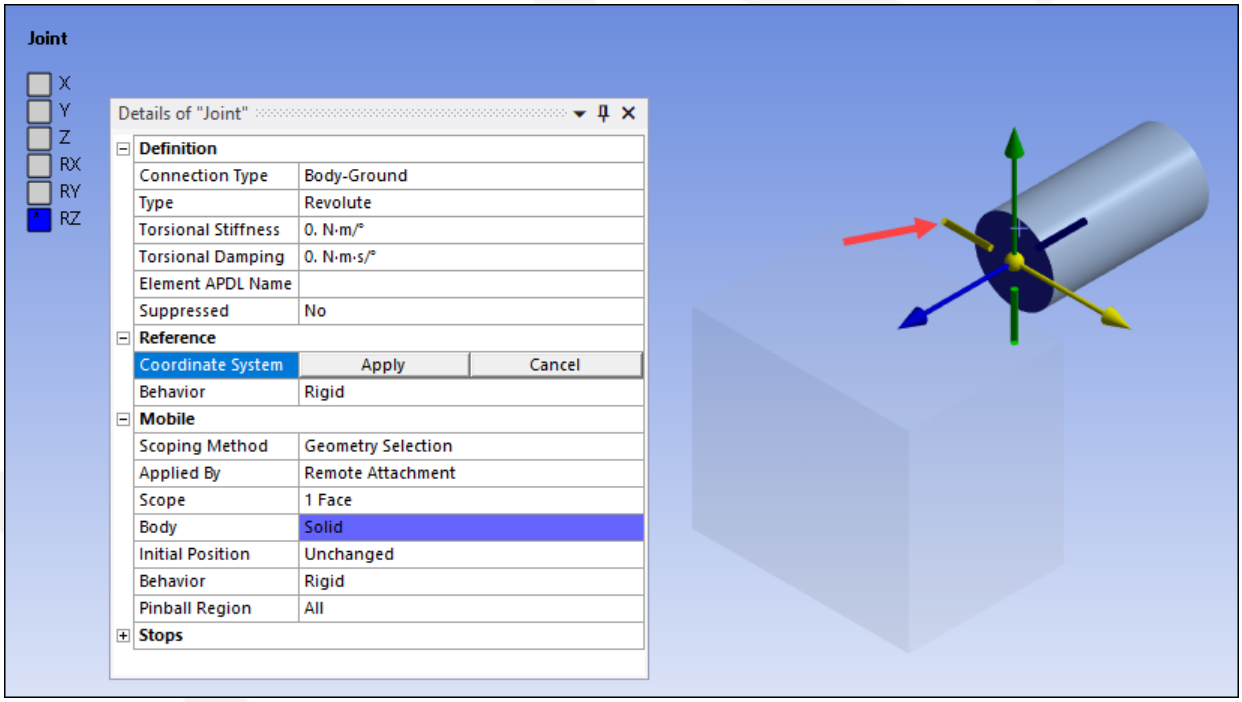
2. Select any of the axis arrows you wish to change. Additional "handles" are displayed for each axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

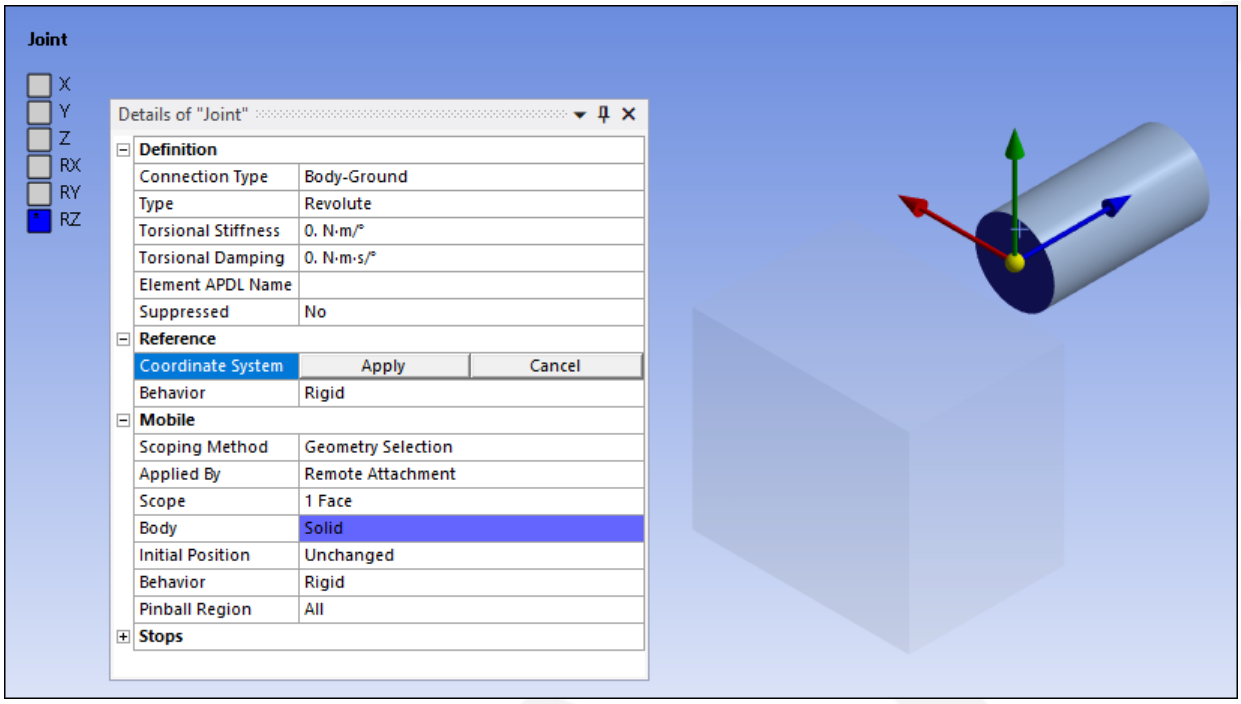


3. Select the handle or axis representing the new direction to which you want to reorient the initially selected axis.

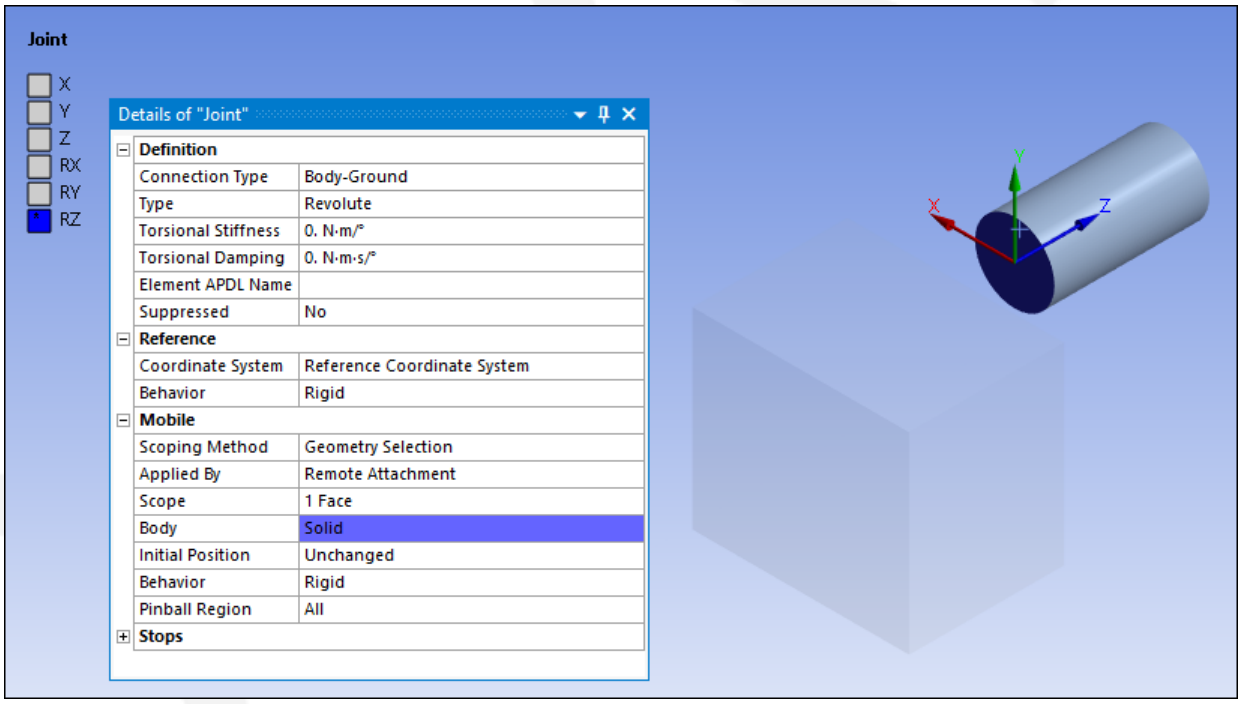


The axis performs a flip transformation (p. 1195).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



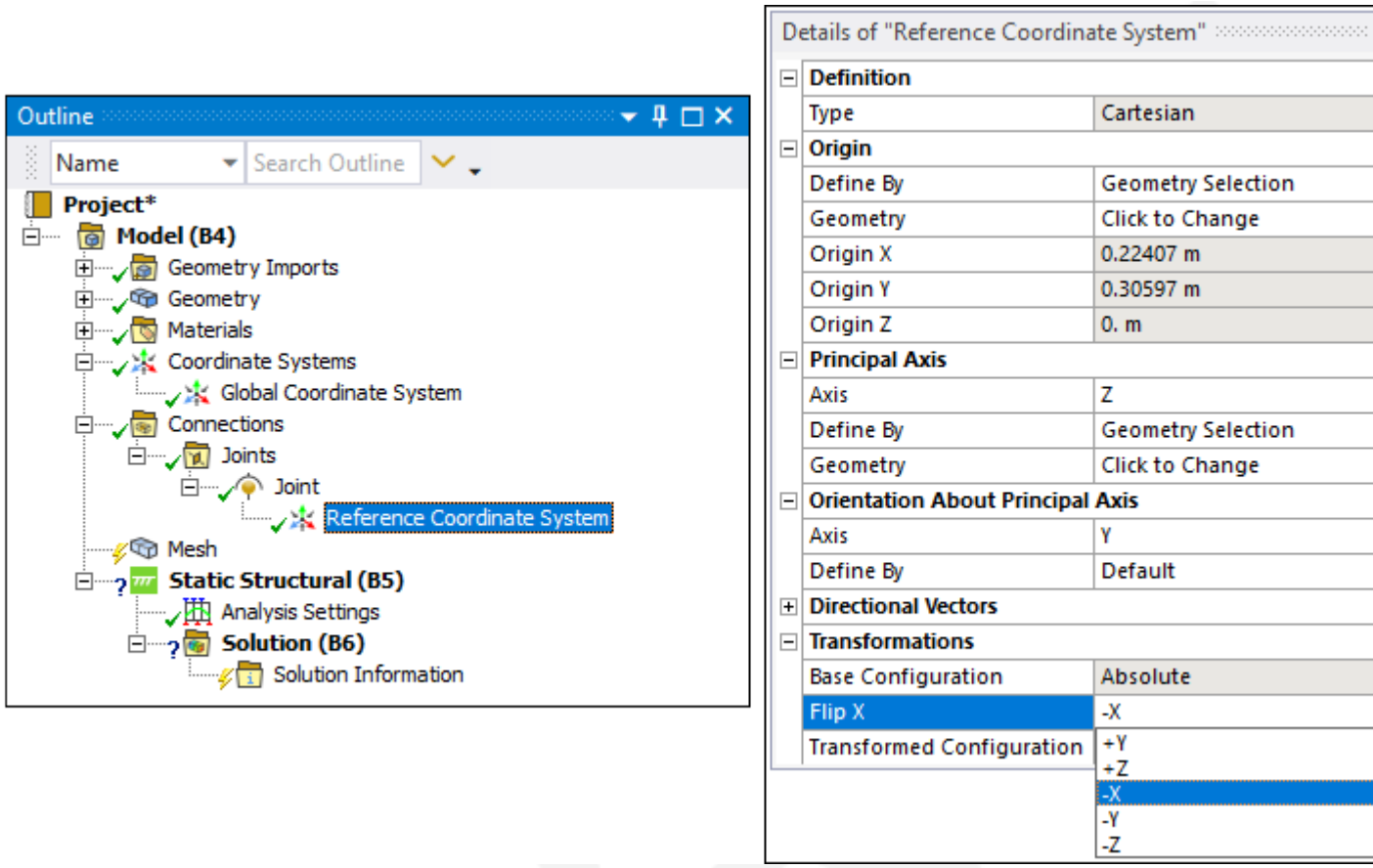
4. Click the **Apply** button. The image of the coordinate system changes from movement mode to a permanent presence at the new orientation.



You can change or delete the status of the flip transformation by highlighting the **Reference Coordinate System** object or a **Mobile Coordinate System** object and making the change or deletion under the **Transformations** category in the Details pane of the child joint coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



When selecting either a **Reference Coordinate System** object or a **Mobile Coordinate System** object, various settings are displayed in the Details view. These are the same settings that apply to all coordinate systems, not just those associated with joints. See the following section on coordinate systems: Initial Creation and Definition (p. 1192) for an explanation of these settings.

Joint Stiffness

For Bushing (p. 1303), General (p. 1310) Joints, and Radial Gaps (RBD only), Mechanical enables you to solve analyses with linear and nonlinear joint stiffness using the features of the **Worksheet**. For these joint types, the **Worksheet** provides the entry options for **Constant** and **Tabular** data.

Linear or nonlinear stiffness and damping behavior is associated with the free or unrestrained components of relative motion of the joint elements. That is, the DOFs are free. For a General Joint, you must specify the DOFs as **Free** in order to make entries in the **Worksheet** matrix.

Element Specification

Generally, Joint Stiffness calculations use the joint element MPC184. See the *Mechanical APDL Element Reference* for additional information as well as the *Mechanical APDL Material Reference* section for MPC184 Joint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When you specify a **Bushing** joint with the **Bushing** formulation, the application uses the bushing element COMBI250. See its *Mechanical APDL Element Reference* section for additional information.

Important:

For the MPC184 and COMBI250 element types, the application may transform the input matrix coefficients based on the sequence of the following coordinate system and node rotation definitions. The application first uses:

1. The **Element Coordinate System** to apply rotation.
2. Then **nodal rotations** per promoted **Remote Points** (**Reference** and **Mobile**). For example, the **NROTAT** command rotates nodal coordinate systems into the active system.

   Note:

   The rotation from these remote points modifies the element matrix.

3. Lastly, the application performs the **normal processing** sequence for rotations.

Linear Joint Stiffness

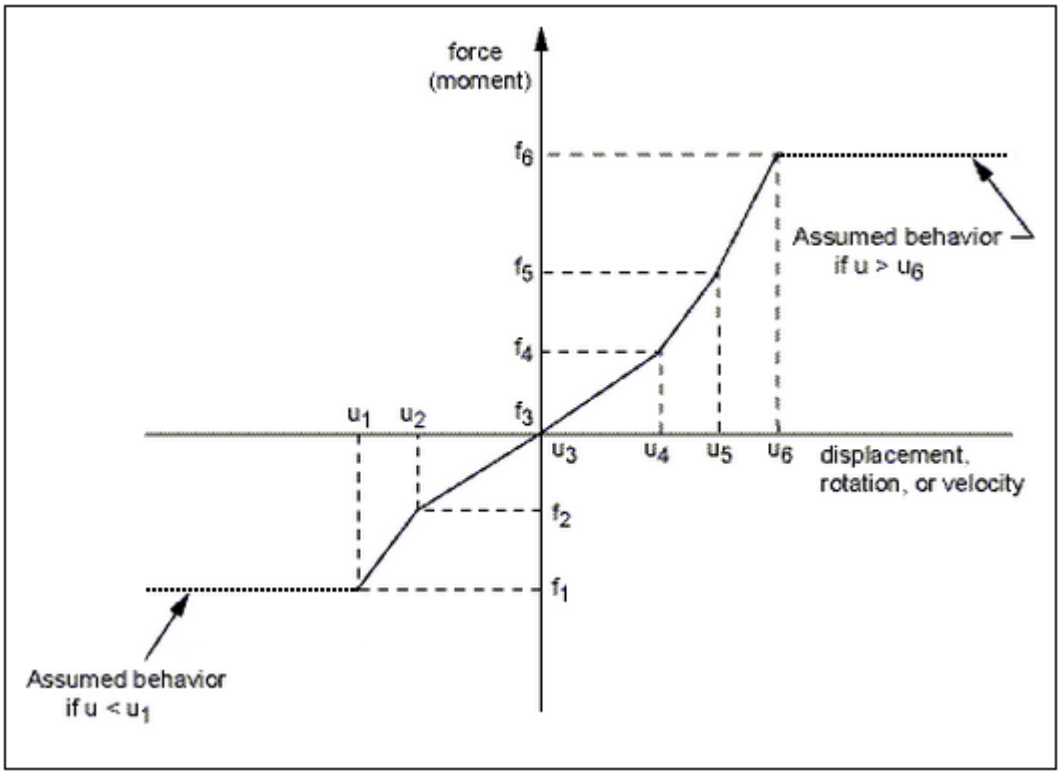
In the case of linear stiffness or linear damping, the values are specified as coefficients of a 6 x 6 elasticity table matrix. Joint Stiffness calculations use the joint element MPC184 and therefore only the appropriate coefficients of the stiffness or damping matrix are used in the joint element calculations.

Nonlinear Joint Stiffness

For nonlinear joint stiffness, relative displacement (rotation) versus force (moment) values are calculated. For nonlinear damping behavior, velocity versus force behavior is specified. You specify nonlinear damping behavior by supplying velocity versus damping force (or moment). The following illustration represents a nonlinear stiffness or damping curve. Note that the Mechanical APDL solver and the Rigid Dynamics Solver assume that there is no added stiffness past the extents.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



See the Material Behavior of Joint Elements topic of the Connecting Multibody Components with Joint Elements section in the *Mechanical APDL Multibody Analysis Guide* for additional details about how this feature related to the Mechanical APDL.

Worksheet

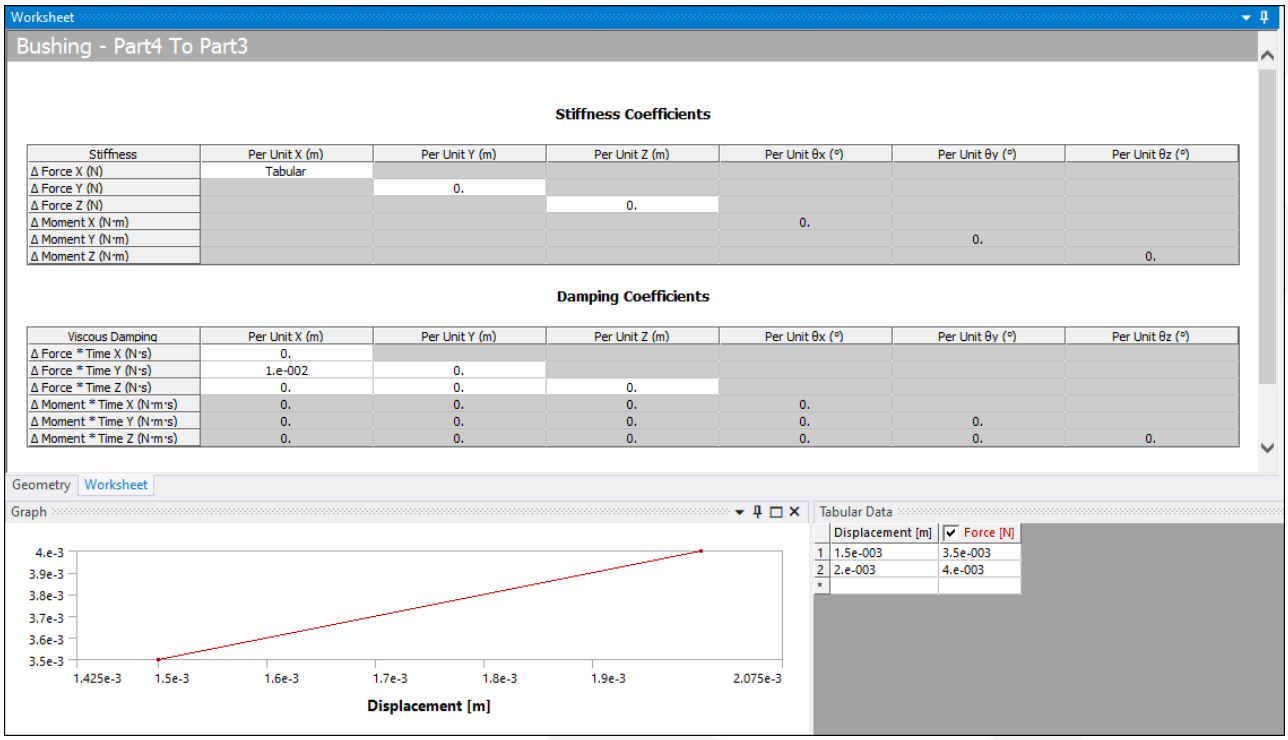
Using the **Worksheet**, you can define **Stiffness Coefficients** in **Constant** or **Tabular** format.

Important:

You cannot edit entries that depend upon an angle unit when the active unit is set to degrees (°). To modify these values, change the active angular unit to Radians.

Nonlinear Joint Stiffness is supported by **Tabular** data entries only and the entries must be made diagonally. In addition, **Damping Coefficients** entries only support constant values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

- The Mechanical APDL solver does not support a mixture of **Constant** and **Tabular** data entries in the **Stiffness Coefficients** matrix. That is, you cannot mix linear and nonlinear stiffness.
- For a linear analysis, such as a Modal analysis, it is recommended that you specify constant values in the **Stiffness Coefficients** matrix. In the event you do specify stiffness values through **Tabular Data** (Displacement vs Force), the application uses the stiffness defined by the slope of the curve at origin (0,0). Also note that if the **Tabular Data**:
  - Starts with the entry 0,0 (Displacement, Force), the application uses the next non-zero entry in the table to compute the stiffness and then uses it throughout the analysis.
  - Starts with a non-zero entry after the origin (0,0), the application uses zero as the stiffness value for the entire analysis.
- The Ansys Rigid Dynamics Solver does support the combination of **Constant** and **Tabular** data entries.
- The **Report Preview** (p. 174) feature does not display table entries from the nonlinear joint stiffness matrix.
- The Ansys Explicit Dynamics solver does not support tabular input (and therefore non-linear stiffness behavior).
- For the Ansys Rigid Dynamics Solver, the use of non-symmetric force/displacement (or torque/rotation) curves is not recommended as it may lead to incorrect results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Joint Friction

Mechanical APDL Solver

Using the Mechanical APDL solver, you can specify friction for a spherical joint.

Rigid Dynamics Solver

For revolute, cylindrical, translational, point on curve, spherical, slot, universal, general, in-plane radial gap, spherical gap, and radial gap joints, Mechanical enables you to introduce frictional behavior in the joint when using the Rigid Dynamics solver.

Friction Model

Joint friction is based on the sliding mode of Coulomb's friction model:

$$F_T = \mu F_N \tag{89}$$

where:

F_T = the resisting tangential force

μ = the friction coefficient

F_N = the normal force

Depending on the joint type, the friction effect leads to a resisting force or torque. Additional geometric information is also required to compute the effect of joint friction in revolute and cylindrical joints. Computation of the normal force depends on the joint type and is described below.

The friction coefficient is set as a constant in the user interface. However, a command exists that allows you to introduce a friction coefficient with an expression (p. 1355).

When the **Joint Friction Type** property is set to **Program Controlled**, **Friction with Sliding/Sticking Transitions**, or **Forced Friction Sliding** the frictional joints are handled as *kinematic* joints, meaning that they are constraint equation based. Two events exist that correspond to the sliding-to-sticking and sticking-to-sliding friction transitions.

When the velocity is lower than a velocity tolerance ($v \le v_{tol}$), the sliding-to-sticking event is active and the following system is solved:

$$\begin{pmatrix} M & B_N^T + \mu B_T^T & B_T^T \\ B_N & 0 & 0 \\ B_T & 0 & 0 \end{pmatrix} \begin{pmatrix} \ddot{q} \\ \lambda_N \\ \lambda_T \end{pmatrix} = - \begin{pmatrix} F \\ \ddot{g}_N \\ \ddot{g}_T \end{pmatrix} \tag{90}$$

where:

M is the inertia matrix and F is the forces vector

g_N and g_T are respectively the normal and tangent constraint equations

B_N and B_T are the Jacobians of the normal and tangent constraints, respectively

During the sliding phase, the traditional sliding Coulomb friction is written as

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\begin{pmatrix} M & B_N^T + \mu B_T^T \\ B_N & 0 \end{pmatrix} \begin{pmatrix} \ddot{q} \\ \lambda_N \end{pmatrix} = - \begin{pmatrix} F \\ \ddot{g}_N \end{pmatrix} \tag{91}$$

When the Forced Frictional Sliding option is not selected, and under the condition $F_T - \mu F_N \leq 0$, with F_N and F_T the normal and tangent forces, the sticking-to-sliding event is active.

When the sliding velocity is low, the friction coefficient is ramped between 0 and its nominal value using the following equation:

$$\mu = \mu_0 \frac{v_t}{v_{tol}} \tag{92}$$

where:

μ = the friction coefficient used in the force or torque calculation

μ_0 = the value of the friction coefficient you have specified in Mechanical

v_t = the tangential velocity

v_{tol} = the velocity tolerance

The velocity tolerance is computed internally, and is based on system velocities and a numerical tolerance. Since the numerical tolerance is problem-dependent, it is possible to change its value using a command (p. 1355).

The following joint friction topics are further discussed in this section:

Joint Friction Definitions

Joint Types

Joint Friction Type

Notes

Joint Friction Definitions

Friction Coefficient

Defines the value of the friction coefficient (unit less, available on the revolute, cylindrical, and translational joints)

Radius

The value of the radius in the specified length unit. Used to compute resisting torque.

Outer Radius

The value of the outer radius in the specified length unit. Used to compute the axial friction torque for a revolute joint.

Effective Radius

The value of the effective radius in the specified length unit. Used to compute the torsional friction torque in a translational joint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

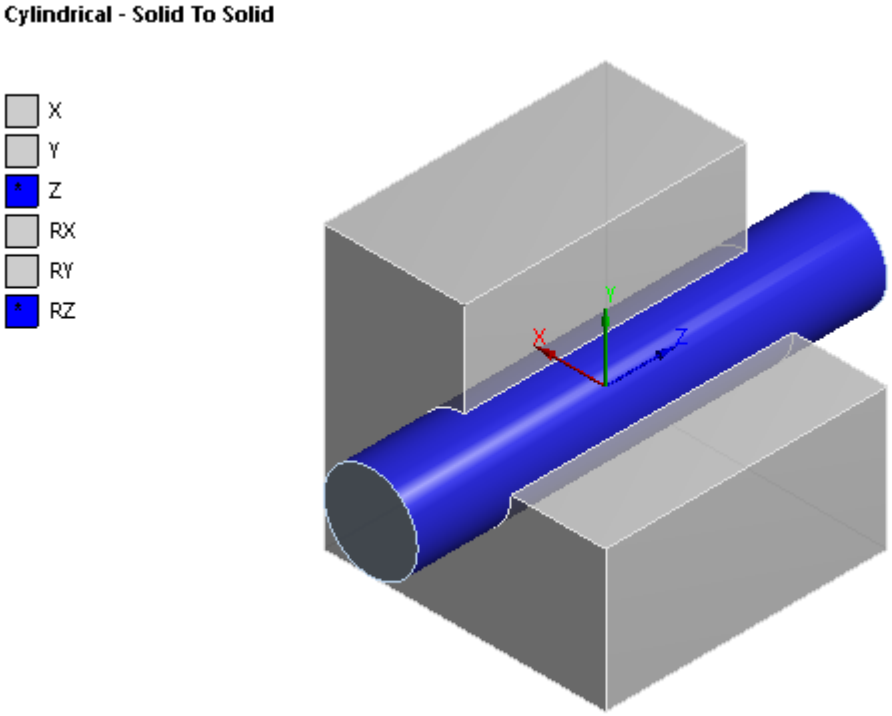
Effective Length

The value of the effective length in the specified length unit. Used to compute the bending force and/or a bending torque.

The pictures below show the definition of geometric properties for joints in typical situations. The effect of normal force is shown by a red arrow.

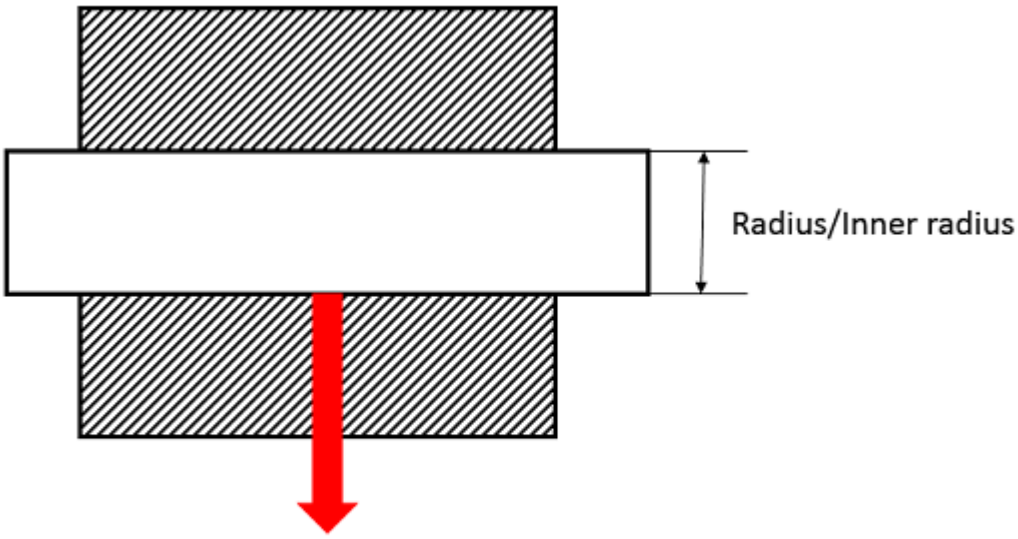
Cylindrical Joint

The typical situation for a cylindrical joint is a pin in a hole. A perfect joint (not gap between the pin and hole) is considered:



Tangential friction is due to radial forces (F_x and F_y) acting between the pin and the hole internal face. The tangential friction force leads to a resisting friction torque along the z-axis of the revolute. The friction torque is proportional to the tangential force via the pin radius.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

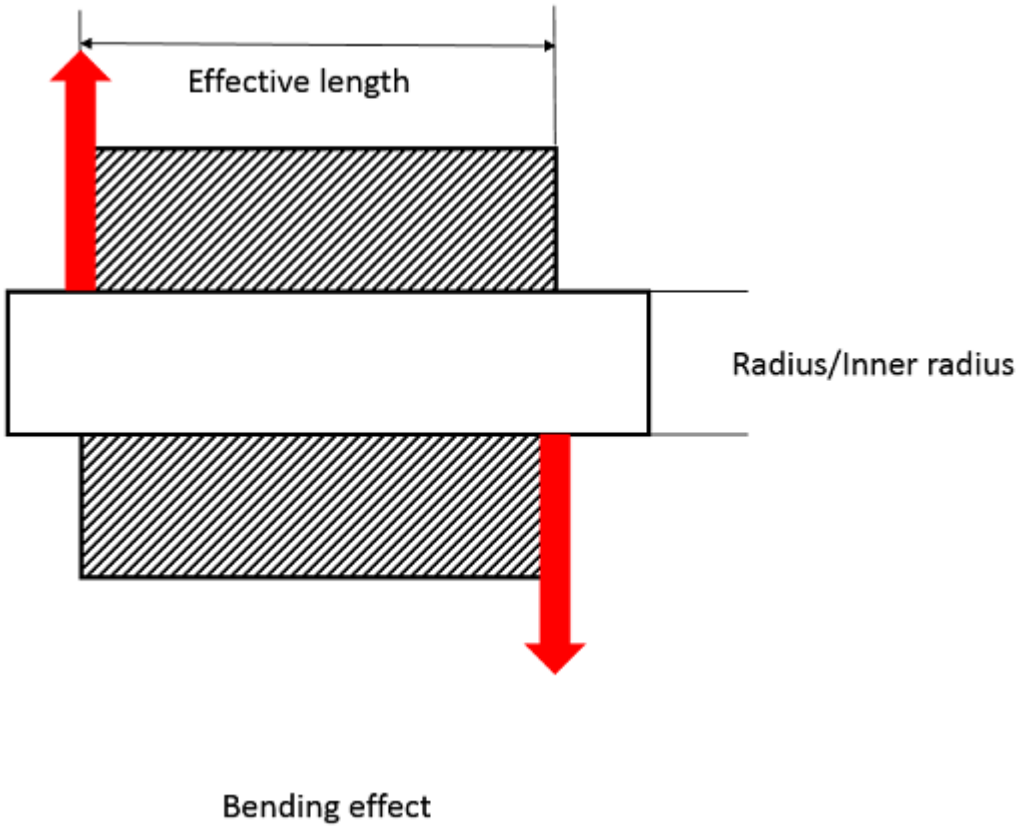


Radial effect

The bending effect (reaction moment M_x and M_y) leads to two opposed forces. The effective length allows the program to compute the normal force from the moments M_x and M_y.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



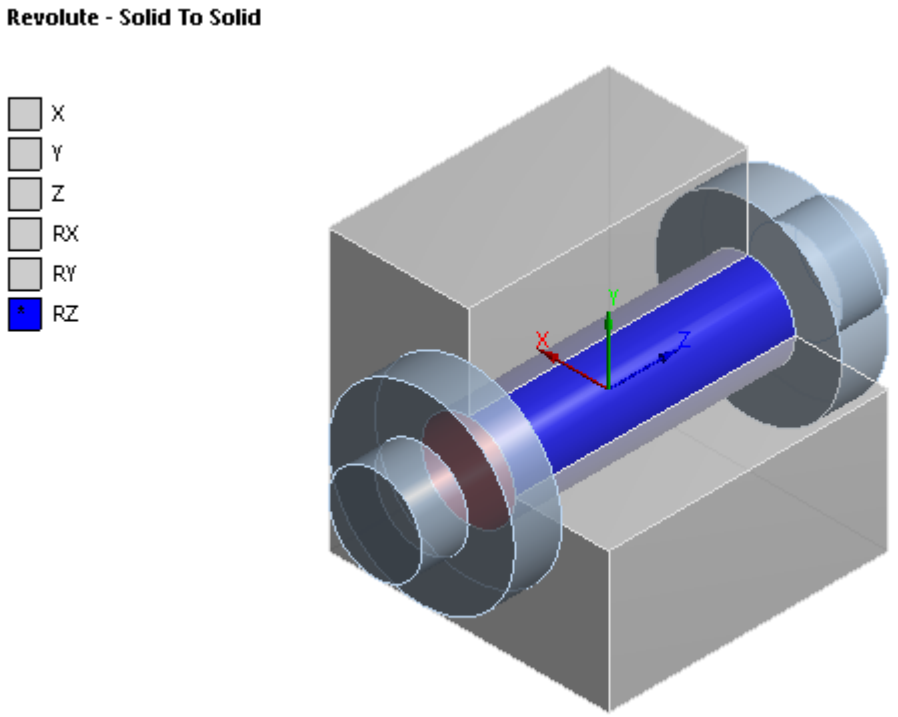
Note:

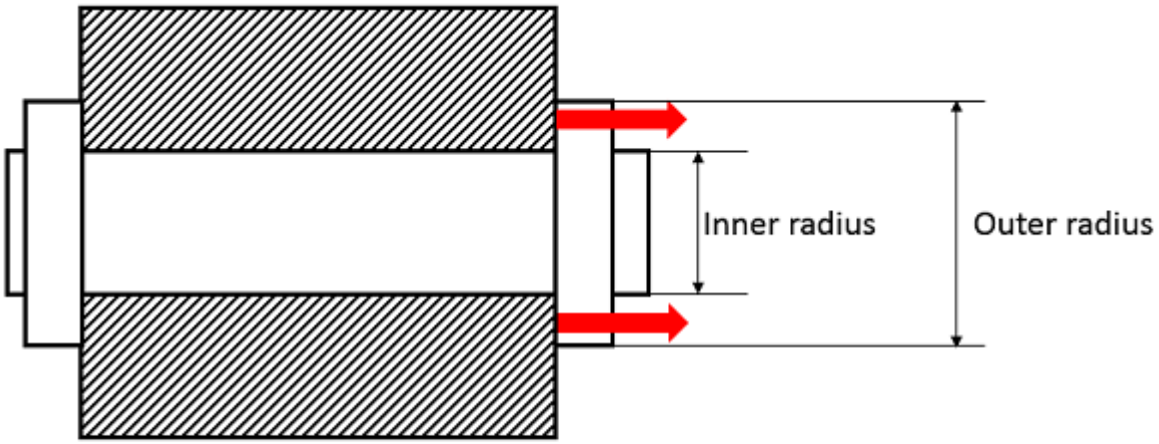
The same definitions also apply to Revolute, Translational, and Point on Curve joints. In the Translational joint, the main axis is x and not z.

Revolute Joint

A revolute joint is also a pin in a hole, but there are two flanges to prevent sliding on the sides. The axial force (along the revolute z-axis) leads to a resisting torque along the axis. An effective radius is used to convert the resisting force to the equivalent torque.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



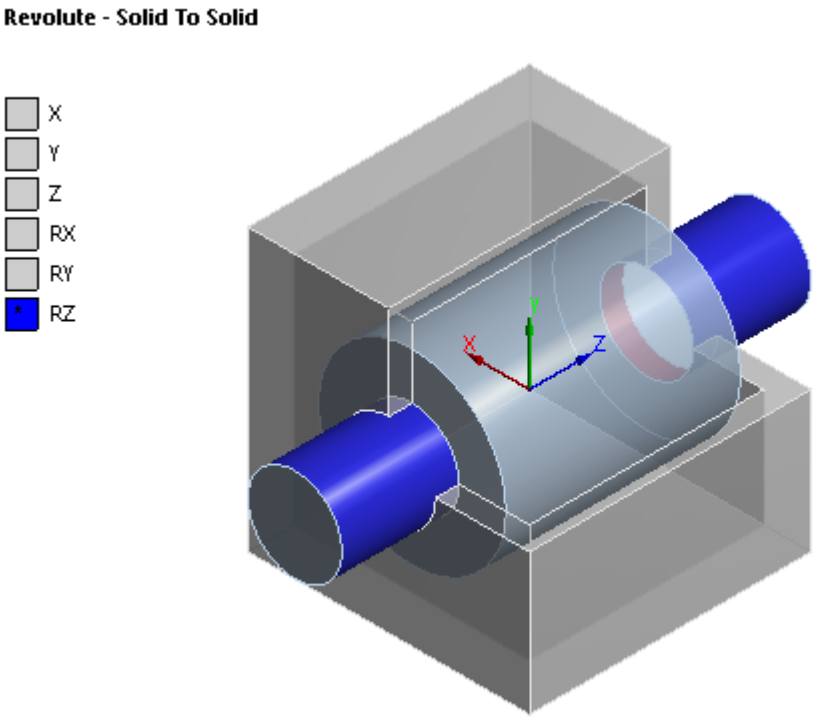


Axial effect

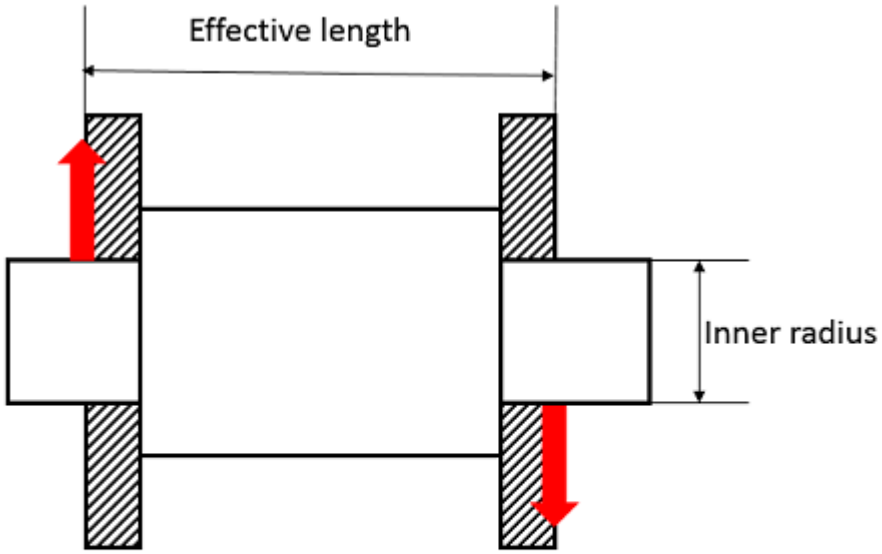
Another way to model a revolute joint is with a single flange on the pin between two side walls:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



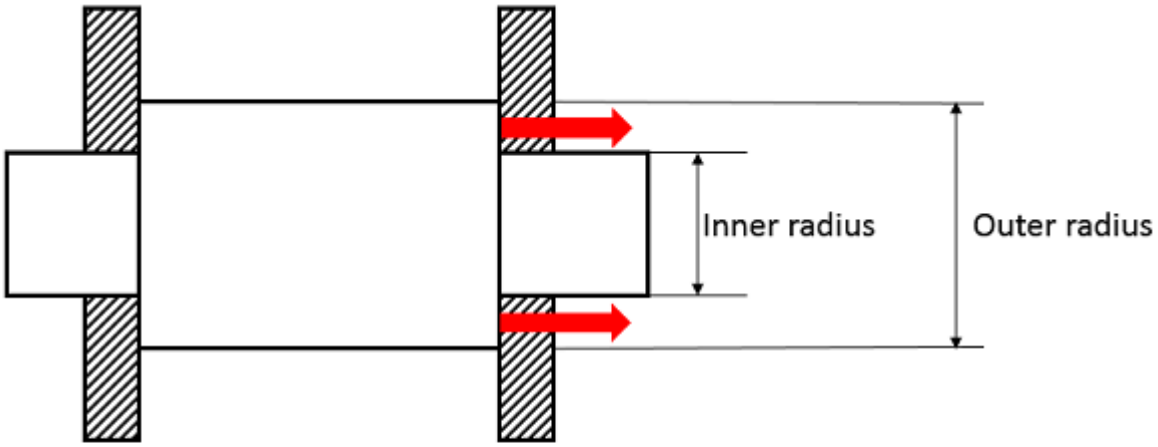
The computation of bending effect is similar to the cylindrical joint except that the effective length is given by the distance between the two walls:



Bending effect

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The axial effect is due to contact between the flange and the wall:



Axial effect

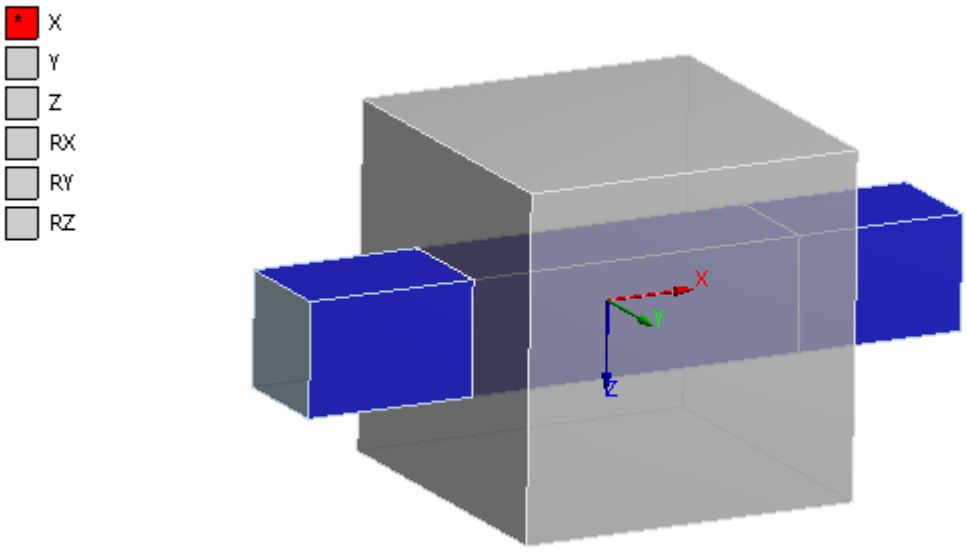
Translational Joint

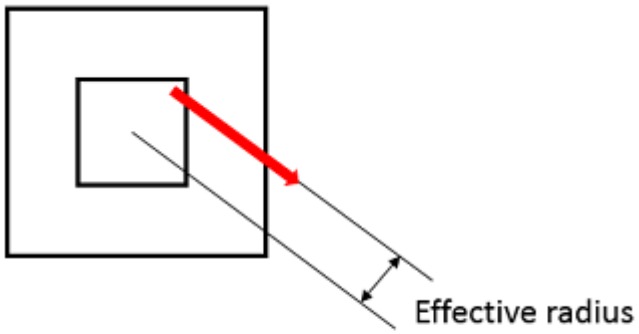
Translational joints typically have a rectangular cross-section. An equivalent effective radius is used to simplify the computation of the torsion effect (moment along translational x-axis).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Translational - Solid To Solid



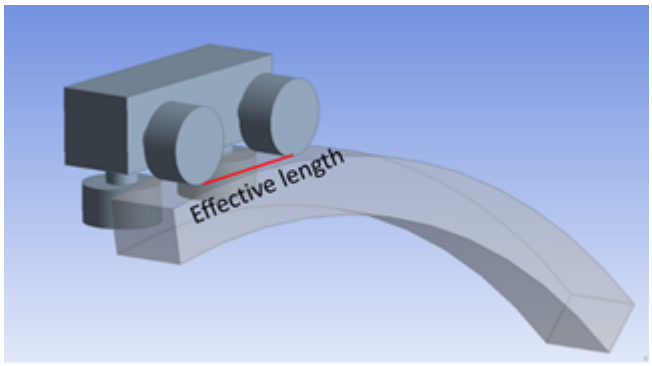


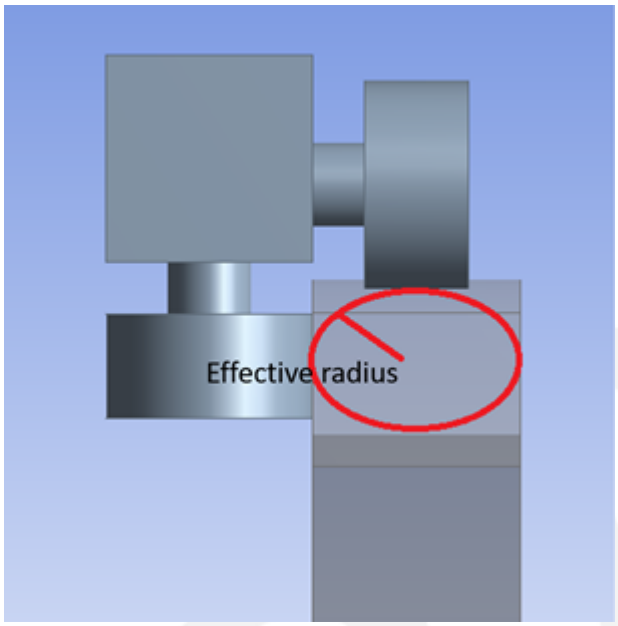
Torsional effect

Point on Curve Joint

Point on Curve joints are similar to translational joints.

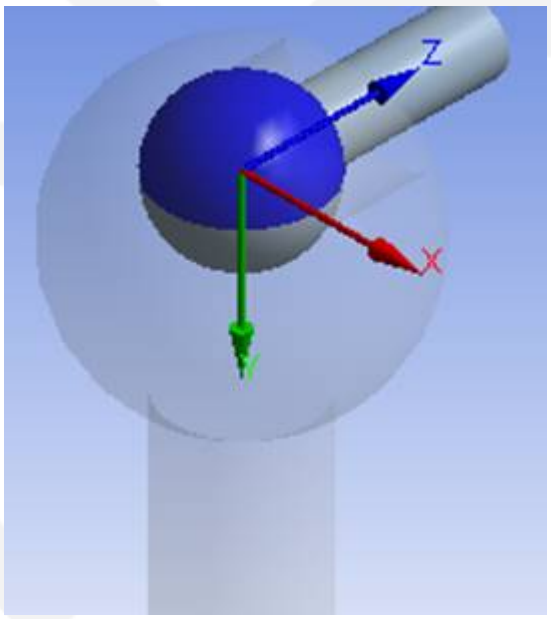
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Spherical Joint

The typical situation for a spherical joint is a ball and socket.

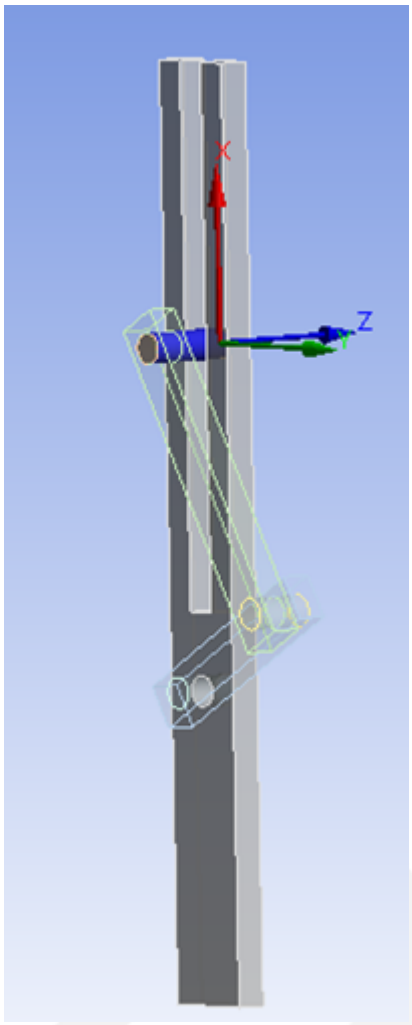


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Slot Joint

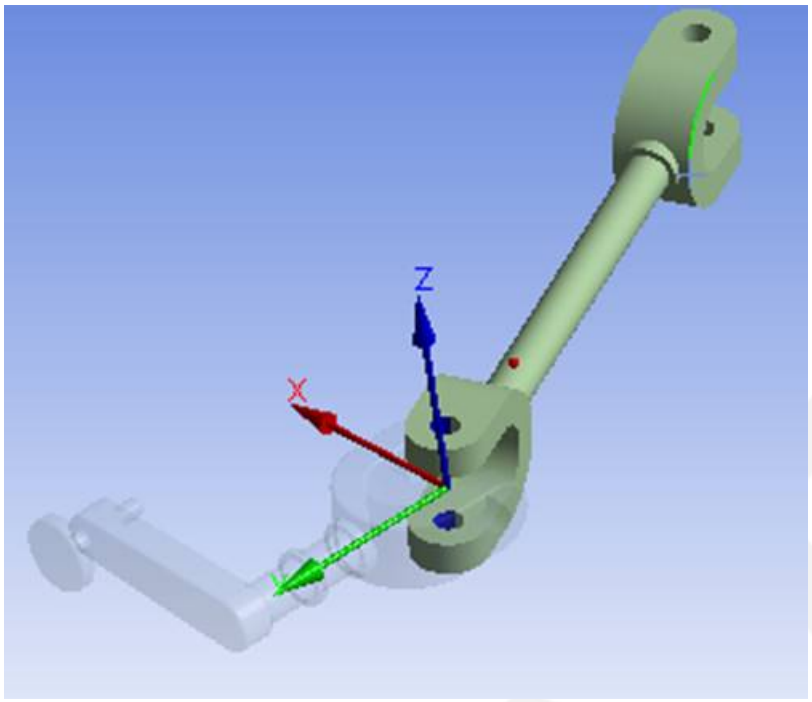
Slot joints are similar to both translational and spherical joints.



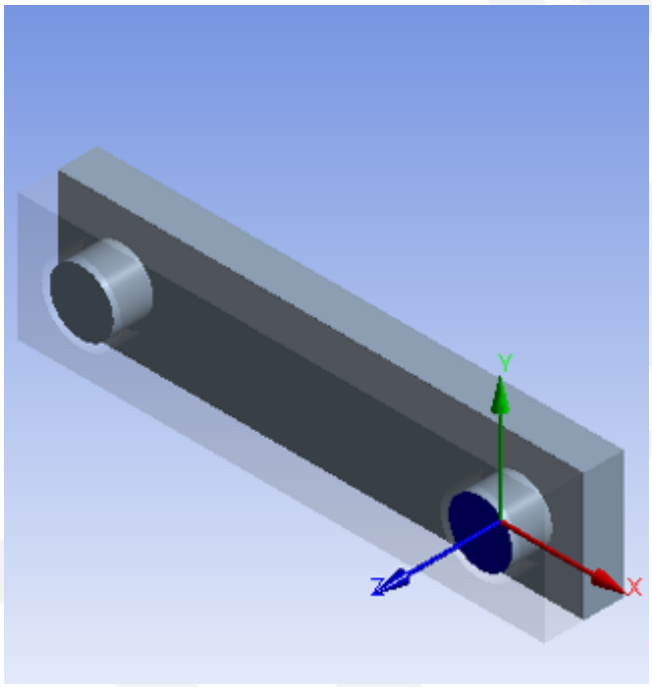
Universal Joint

Universal joints enable two rotations along x and z axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



In-Plane, Spherical, and Radial Gap Joints



General Joint

Friction is allowed in general joints in the following cases:

- The joint has a single translational degree of freedom and/or a single rotation degree of freedom. In this case, the joint is similar to a translational, revolute or cylindrical joint.
- The joint has a single translational degree of freedom and/or its rotations are all free. In this case the joint is similar to a slot or spherical joint.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

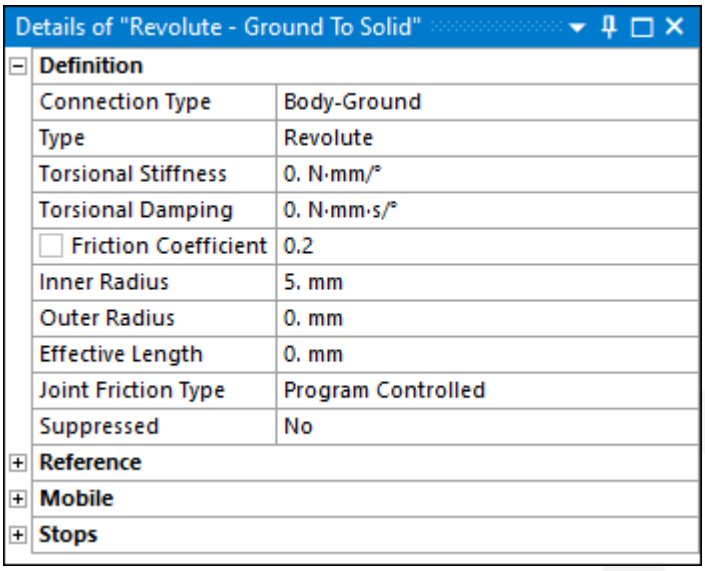
Published: 2023-10-23

For more information, see Material Behavior of Joint Elements.

Joint Types

Revolute Joint

In order to compute the resisting friction torque, the revolute joint is considered as a cylinder pin assembly. The cylinder is characterized by its inner radius, outer radius, and an effective length. These lengths are shown in the following figure:



The joint friction creates a resisting torque along the joint z-axis.

The tangential friction torque along the z-axis is computed as:

$$T_{\text{tangential}}=\mu r F_N \tag{93}$$

Where:

$F_N=\sqrt{F_x^2+F_y^2}$ = the normal force

μ = the friction coefficient

r = the cylinder radius

The effect of the joint axial force (along the joint z-axis) is evaluated as:

$$T_{\text{axial}}=\mu\times\frac{\left(r+r_{\text{outer}}\right)}{2}F_z \tag{94}$$

Where r_{outer} = the outer radius.

The bending moment is computed as:

$$T_{\text{bending}}=\frac{2\mu r\sqrt{M_x^2+M_y^2}}{l_{\text{eff}}} \tag{95}$$

Where l_{eff}= the effective length.

The resultant friction moment is given by:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$T_{\text{fric}} = T_{\text{tangential}} + T_{\text{axial}} + T_{\text{bending}} \quad (96)$$

Note:

The bending torque is computed only if the effective length is greater than 0; the axial torque is computed only if the outer radius is greater than 0.

Translational and Point on Curve Joints

In the case of the translational and point on curve joints, a resisting force is created along the joint x-axis. An effective radius and effective length are needed to compute the torsional friction force and the bending friction force, respectively (see the figure below).

Details of "Translational"	
Definition	
Connection Type	Body-Body
Type	Translational
Friction Coefficient	0.
Radius	0. mm
Effective Length	0. mm
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Mobile	
Stops	

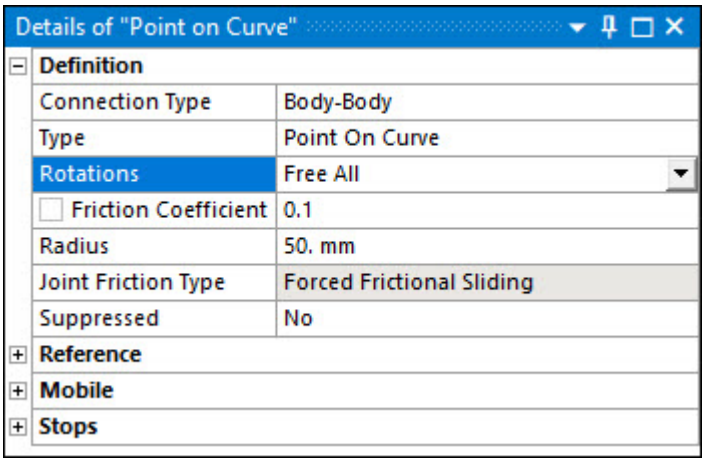
The torsional and bending effects are computed for the point on curve joint if all rotations are fixed.

Details of "Point on Curve"	
Definition	
Connection Type	Body-Body
Type	Point On Curve
Rotations	Fix All
Friction Coefficient	0.1
Radius	50. mm
Effective Length	100. mm
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Mobile	
Stops	

However, when all rotations are free, the friction length is not available and the radius is taken into account for computing a friction moment as described for spherical joints (p. 1350).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The tangential friction force is computed as:

$$F_{\text{fric}}=\mu F_N \tag{97}$$

where:

$F_N=\sqrt{F_y^2+F_z^2}$ = the normal force

μ = the friction coefficient

The bending friction force is given by:

$$F_{\text{bending}}=\frac{2\mu\sqrt{M_y^2+M_z^2}}{l_{\text{eff}}} \tag{98}$$

where l_{eff}= the effective length.

The torsional friction force is evaluated as:

$$F_{\text{torsional}}=\mu\times\frac{M_x}{r_{\text{eff}}} \tag{99}$$

where r_{eff} = the effective radius.

The friction force is computed as:

$$F_{\text{fric}}=F_{\text{tangential}}+F_{\text{bending}}+F_{\text{torsional}} \tag{100}$$

Note:

The bending force is computed only if the effective length is greater than 0; the torsional force is computer only if the effective radius is greater than 0.

Cylindrical Joint

For a cylindrical joint, both a resisting force and a resisting torque are created along the joint z-axis. Similarly to the revolute joint, the joint is considered as a cylinder pin assembly. An effective length is required for computing the bending friction force and torque:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Joint"	
Definition	
Connection Type	Body-Ground
Type	Cylindrical
Torsional Stiffness	0. N·mm/°
Torsional Damping	0. N·mm·s/°
Friction Coefficient	0.1
Radius	40. mm
Effective Length	40. mm
Joint Friction Type	Program Controlled
Suppressed	No
Reference	
Mobile	
Stops	

The friction force and friction torque are computed similarly to the translational and revolute joint, respectively but without the axial and torsional effects:

$$F_{\text{fric}} = F_{\text{tangential}} + F_{\text{bending}} \quad (101)$$

$$T_{\text{fric}} = T_{\text{tangential}} + T_{\text{bending}} \quad (102)$$

Note:

The bending force and torque are computed only if the effective length is greater than 0.

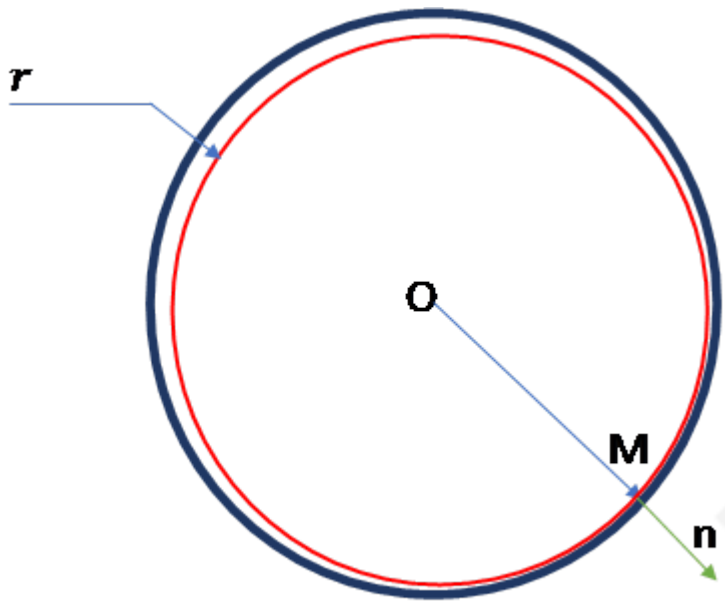
Spherical Joint

Details of "Joint"	
Definition	
Connection Type	Body-Body
Type	Spherical
Friction Coefficient	0.1
Radius	100. mm
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Mobile	

Spherical joints are considered imperfect, so there is a gap between the inner and the outer spheres. If *O* is the center of the joint and the contact occurs at one point *M*, the vector $\overrightarrow{OM}$ is defined as $\overrightarrow{OM} = r \frac{\vec{F}_N}{\|\vec{F}_N\|}$ as shown in the following figure:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Where:

- $\vec{F}_N$ is the constrained force of the joint.
- $\vec{n}$ is a normal vector at the contact point *M*, defined as $\vec{n}= \frac{\vec{F}_N}{\|\vec{F}_N\|}$

A linear velocity is defined as $\vec{v}= \vec{\omega} \times \vec{n}$. A friction force can be calculated as $\vec{F}_T= -\mu \left\|\vec{F}_N\right\| \frac{\vec{v}}{\|\vec{v}\|}$.

Finally, the friction moment is given by $\vec{M}_T=r \vec{F}_T \times \vec{n}$.

Note that similar to revolute joints where the friction force is uniformly distributed around the joint so that the resulting force sums to zero, only the friction torque is considered in the computation. The friction force $\vec{F}_T$ is not considered nor reported.

Only Forced Sliding Friction is allowed for Spherical joints.

Slot Joint

Details of "Joint"

Definition	
Connection Type	Body-Body
Type	Slot
Friction Coefficient	0.5
Radius	100. mm
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Mobile	
Stops	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For slot joints, a resistant friction force is computed along the X axis. The calculation of this force is similar to translational joints but without bending or torsional effects. Furthermore, a friction moment is computed, similarly to spherical joints, which resists the joint rotations.

Universal Joint

Details of "Joint"	
Definition	
Connection Type	Body-Body
Type	Universal
Friction Coefficient	0.4
Radius	5. mm
Joint Friction Type	Forced Frictional Sliding
Suppressed	No
Reference	
Mobile	
Stops	

For universal joints, two resistant moments are computed along X and Z axis, respectively. The calculation of this friction moment is similar to revolute joints but without axial or bending effects.

In-Plane, Spherical, and Radial Gap Joints

For imperfect joints, the **Friction Coefficient** property appears under the **Stops** definition group:

Details of "Joint"	
Definition	
Connection Type	Body-Body
Type	Radial Gap
Translation X	Free
Translation Y	Free
Translation Z	Fixed
Rotations	Free All
Suppressed	No
Reference	
Mobile	
Stops	
Radial Gap	Stop
Inner Diameter	8. mm
Outer Diameter	10. mm
Height	20. mm
Restitution	1.
Friction Coefficient	0.4

The friction in imperfect joints is considered under the assumption of **pure sliding friction**. The friction tensor is evaluated similarly to a *Forced Frictional Sliding contact*.

General Joint

If the joint has a single translational degree of freedom, a friction force is computed along this axis similarly to translational joints.

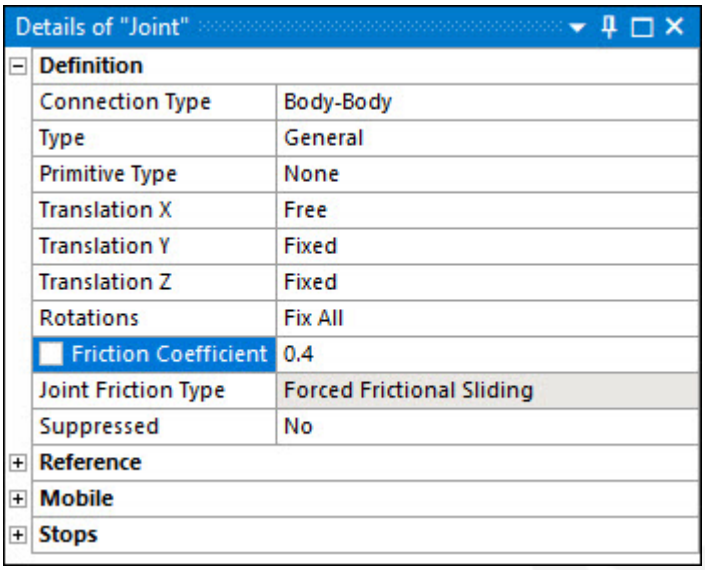
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If the joint has a single rotation degree of freedom, a friction moment is computed similarly to revolute joints.

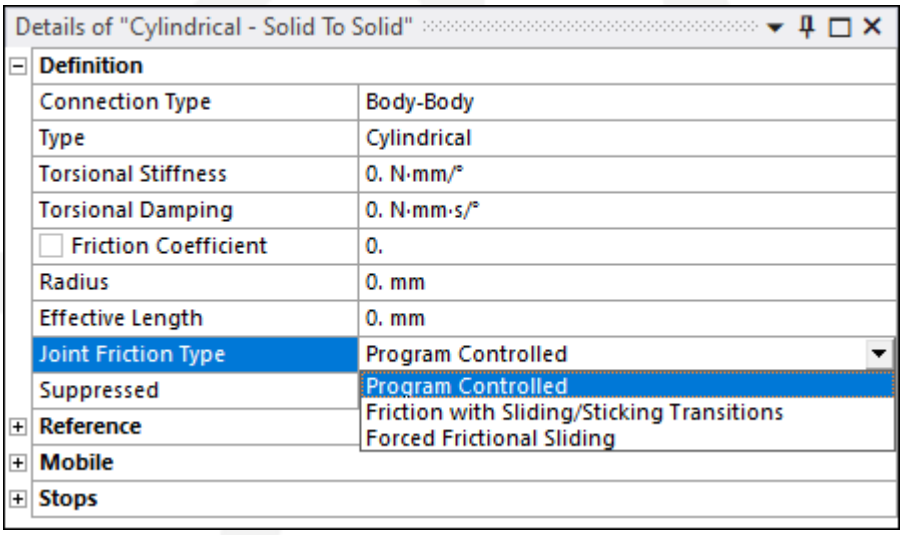
No bending, axial or torsional effects are evaluated for general joints.

When all the rotations are set free, a friction moment is computed similarly to spherical joints.



Joint Friction Type

This option allows you to choose between performing a classic friction analysis with sliding-to-sticking and sticking-to-sliding transitions by setting its value to Friction with Sliding/Sticking Transitions or Program Controlled. You can perform a pure sliding friction analysis by setting its value to Forced Frictional Sliding.

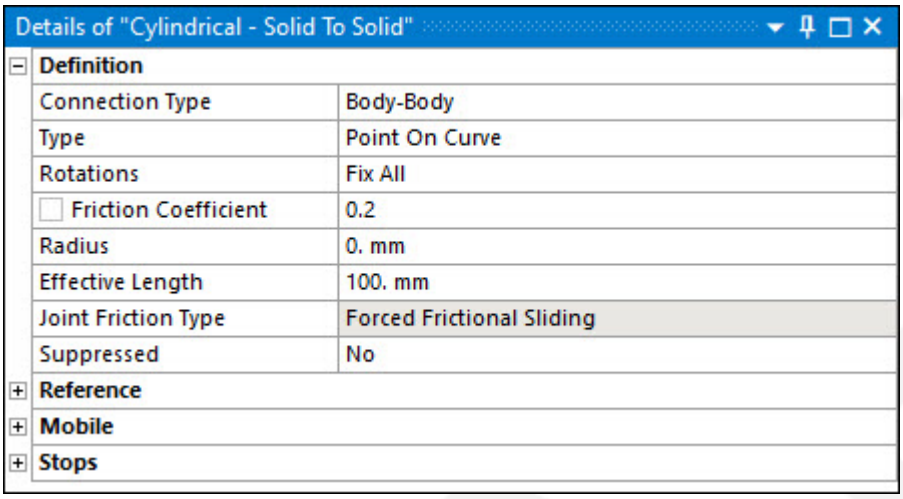


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For revolute, translational and cylindrical joints, its default value is Program Controlled.

Note:

- For point-on-curve, spherical, slot, universal, and general joints, only Forced Frictional Sliding is available.



- The Forced Frictional Sliding behavior enables a smoother simulation, since there is no need to search for transition events. This might save simulation time if advanced friction modelling is not required.

Notes

Computation of Friction Force and Torque

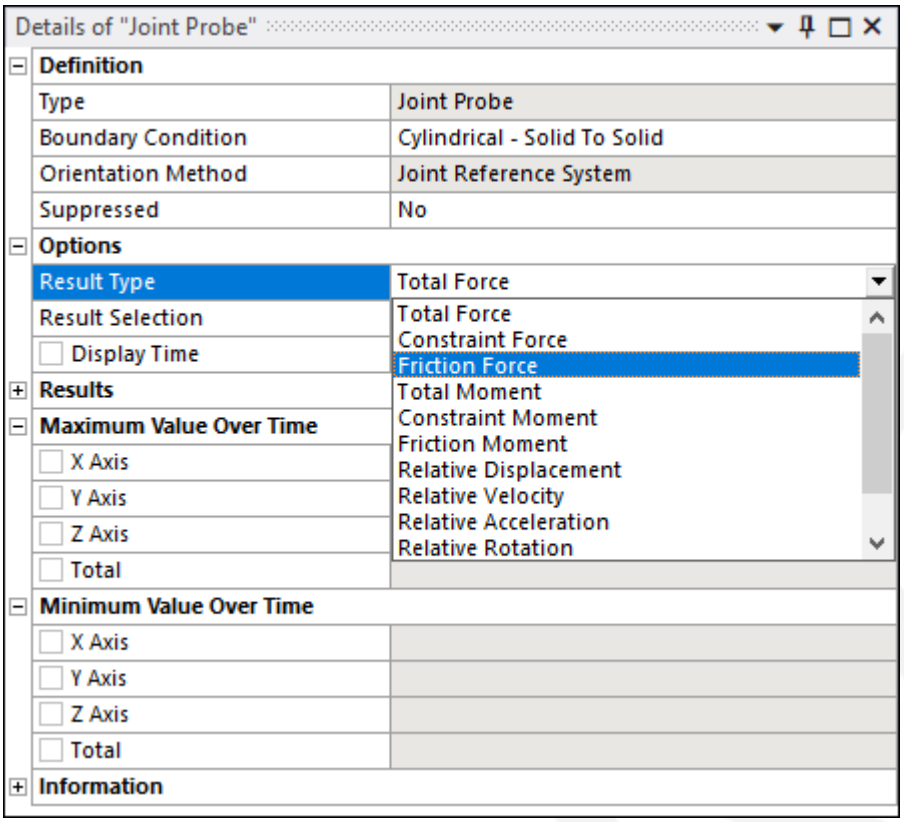
The forces and moments that are used to evaluate the frictional effects are taken at the beginning of the time step. The resisting force and torques have a sign that is opposed to that of the velocity which is also evaluated at the beginning of the time step. In other words, the friction force and torque, during a given time step, are evaluated with the forces and torques of the previous time step and have the opposite sign of the velocity evaluated at the previous time step.

Postprocessing Joint Friction

For translational, revolute, cylindrical, point on curve, spherical, slot, universal and general joints, the Joint Probe (p. 2118) allows you to verify the friction force and the friction moment. Friction force is available for translational, cylindrical, point on curve, slot, general, and all imperfect joints. Friction moment is available for revolute, cylindrical, point on curve with free rotations, spherical, slot, universal, general, and all imperfect joints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

Joint Friction is only supported by the Ansys Rigid Dynamics Solver.

Setting Joint Friction With a Command

The command **`SetFrictionVariable(var)`** replaces the constant value already given to the friction coefficient with the expression given by *`var`*.

For example, to set $\mu=\mu_0+\mu_1\exp\left(-\alpha^*\mathrm{vel}\right)$

Example 6: Setting Variable Joint Friction

```
Joint = CS_Joint.Find(_jid)
Var = CS_Variable()
u0 = 0.1
u1 = 0.2
alpha = 0.5
Var.SetFunc('u0+u1*exp(-alpha*time)',0)
Var.AddInputMeasure(Joint.GetVelocityMeasure())
Joint.SetFrictionVariable(Var)
```

The command has no effect if no value for the friction coefficient has been provided in the UI.

Setting the Friction Tolerance

The command **`SetFrictionTolerance(tol)`** sets the friction tolerance.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example 7: Setting Friction Tolerance

```
Joint = CS_Joint.Find(_jid)
Joint = Joint.SetFrictionTolerance(1e-4)
```

Setting the Static Friction Coefficient with a Command

The static friction coefficient is by default set to the dynamic friction coefficient. The command **SetStaticFrictionCoefficient(muS)** changes its value to muS.

Example 8: Setting the Static Friction Coefficient

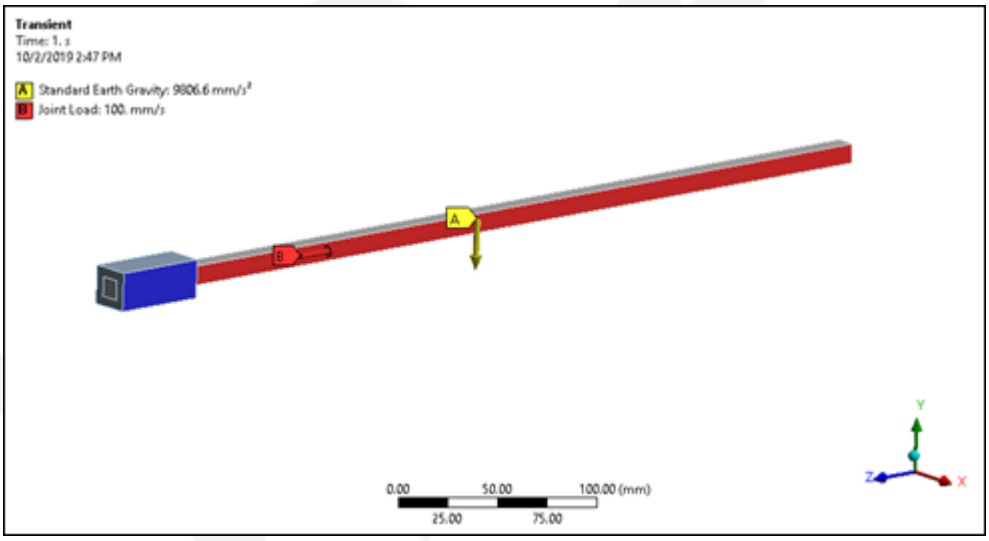
```
Joint = CS_Joint.Find(_jid)
Joint.SetStaticFrictionCoefficient(0.2)
```

Bending Moment for Translational Joints

When using a non-zero length when defining the friction in a joint, the bending effect is proportional to the constraint moment existing in the joint. This moment is always defined as the moment expressed at the reference coordinate system of the joint. For joints that have translational degrees of freedom, such as the translational or cylindrical joint, the reference coordinate system and mobile coordinate system are not always at the same location, and the constraint force $\vec{F}_M$ generates a moment at the reference coordinate system:

$$\vec{M}_R = \vec{M}_M + \overrightarrow{MR} \times \vec{F}_M$$

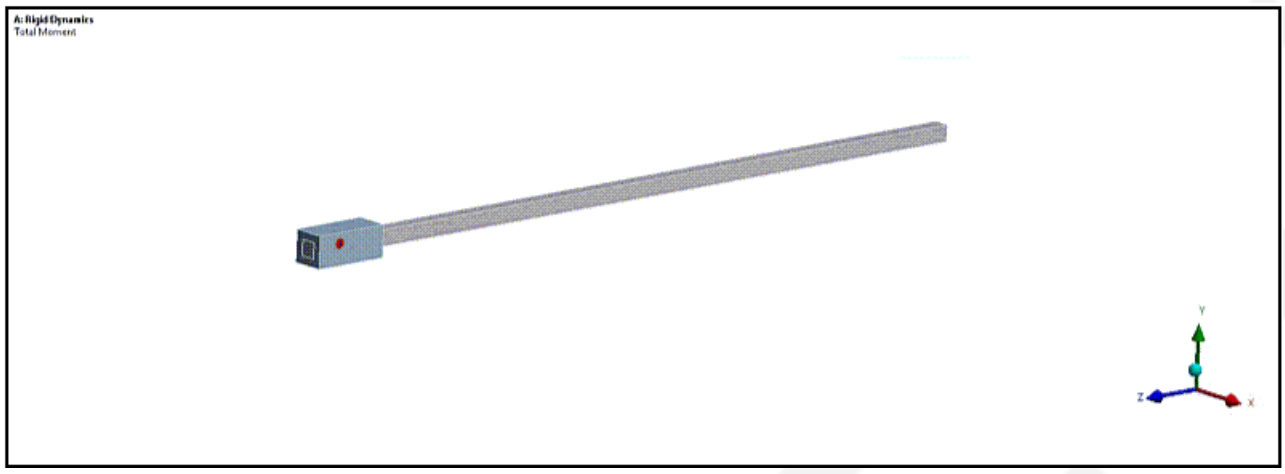
In this example, the blue mass slides on a fixed red bar, under vertical gravity.



As the mass moves to the right the reported moment at the reference coordinate system, which stays at the end of the blue bar, increases linearly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



If a length has been specified in the translational joint, the friction bending forces will increase with the position. In this example, this is not the expected effect.

However, if the blue mass is grounded and the red bar is sliding in it, the moment reported in the reference coordinate system is also increasing and so should the friction bending forces.

You must choose the reference and mobile coordinate systems and effective length properly to represent the physics, as the "joint" abstraction can be sometimes misleading.

Note:

This does not apply to the point on curve joint, as this joint uses an intermediate coordinate system called a *floating reference coordinate system*, which is always coincident with the mobile coordinate system. This floating reference is used to measure the torque, therefore the vector $\overrightarrow{MR}$ is always zero.

Manual Joint Creation

This section examines the steps to manually create joints. Refer to the Automatic Joint Creation (p. 1359) section of the Help for a discussion about how to create joints automatically.

To add a joint manually:

1. The **Joints** object is a child object of the **Connections** object. The **Connections** object is typically generated automatically. As needed, highlight the **Model** object in the tree and choose the **Connections** option from the Model Context Tab (p. 48) once you have imported your model.

2. Highlight the **Connections** object and open either **Body-Ground** menu or the **Body-Body** menu from the Connections Context Tab (p. 50) and then select your desired Joint Type (p. 1298). The new joint object becomes the active object in the tree.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Once inserted and active, there are a number of joint properties that require definition. For a detailed description of each of these properties, refer to the Joint Properties (p. 1316) Help section.

Note:

You can pre-select a vertex or node (**Body-Ground**) or two vertices or nodes (**Body-Body**) and then insert a Joint to automatically create a directly attached joint.

Tip:

The **Body Views** option on the Context tab displays the **Reference** and **Mobile** bodies in separate windows with appropriate transparencies applied. You have full body manipulation capabilities in each of these windows.

---

4. Once you have defined the desired joint properties, you may wish to use the **Configure** tool. The **Configure** tool is activated by selecting the **Configure** option on the Context Tab. This feature positions the Mobile body according to the joint definitions. You can then manipulate the joint interactively (for example, rotate the joint) directly on the model. The notes section (p. 1359) shown below provides additional information about the benefits and use of the Configure feature (as well as the Assemble feature).

   In addition, refer to the Example: Configuring Joints (p. 1380) Help section for an example of the use of the **Configure** tool.

Note:

- The **Configure** option is not supported for Joints scoped as a **Direct Attachment**.
- The **Set** option on the Context tab locks the changed assembly for use in the subsequent analysis.
- The triad position and orientation may not display correctly until you click the **Set** option.
- The **Revert** option on the Context tab restores the assembly to its original configuration from DesignModeler or the CAD system.

---

5. It is suggested that you consider the following:

   - Renaming the joint objects (p. 1365) based on the type of joint and the names of the joined geometry.
   - Display the **Joint DOF Checker** (p. 1368) and modify joint definitions if necessary.
   - Create a redundancy analysis to interactively check the influence of individual joint degrees of freedom on the redundant constraints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Configure and Assemble Tools Notes

The **Configure** and **Assemble** tools are a good way to exercise the model and joints before starting to perform a transient analysis. They are also a way to detect locking configurations.

The **Assemble** tool performs the assembly of the model, finding the closest part configuration that satisfies all the joints.

The **Configure** tool performs the assembly of the model, with a prescribed value of the angle or translational degree of freedom that you are configuring.

For the **Assemble** tool, all the joints degrees of freedom values are considered to be free. For the **Configure** joint, the selected DOF is considered as prescribed.

In both cases, the solver will apply all constraint equations, solve the nonlinear set of equations, and finally verify that all of them are satisfied, including those having been considered as being redundant. The violation of these constraints is compared to the model size. The model size is not the actual size of the part – as the solver does not use the actual geometry, but rather a wireframe representation of the bodies. Each body holds some coordinate systems – center of mass, and joint coordinate systems. For very simple models, where the joints are defined at the center of mass, the size of the parts is zero. The violation of the constraint equations is then compared to very small reference size, and the convergence becomes very difficult to reach, leading the Configure tool or the Assemble tool to fail.

Automatic Joint Creation

This section discusses the automatic joint creation in Mechanical. You can also create joints manually as discussed in Manual Joint Creation (p. 1357) section.

Creating Joints Automatically

You can direct Mechanical to analyze your assembly and automatically create fixed joints (p. 1298) and/or revolute joints (p. 1299).

To create joints automatically:

1. Insert a **Connection Group** object under the **Connections** folder using the Context tab option or using the **Insert** option from the context menu (right mouse click) for this folder.
2. From the Details view of the **Connection Group** object, choose **Joint** from the **Connection Type** drop-down menu.
3. Select some bodies in the model based on the **Scoping Method**. The default is **Geometry Selection** scoped to **All Bodies**.
4. Configure the types of joints (fixed and/or revolute) you want Mechanical to create automatically through the appropriate **Yes** or **No** settings in the Details view. These properties will be applied

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

only to scoped geometries for this connection group. You can set defaults for these settings using the **Options** dialog box (p. 217) under **Connections** (p. 219).

Note:

When both the **Fixed Joints** and **Revolute Joints** properties are set to **Yes**, the revolute joints have priority; the search for revolute joints will be processed first followed by the search for fixed joints.

---

5. Choose **Create Automatic Connections** from the context menu (right mouse click) for the **Connection Group**. Appropriate joint types are created and appear in the tree as objects under the **Joints** folder. Each joint also includes a reference coordinate system that is represented as a child object to the joint object.

Note:

For automatic joint detection, the **Search Across** property only supports options **Parts**, **Assemblies**, and **Anywhere**. The **Assemblies** and **Anywhere** options are only available with model assembly.

---

6. Display the **Joint DOF Checker** (p. 1368) or the redundancy analysis and modify joint definitions if necessary.

Joint Stops and Locks

This section describes the use of joint stops and locks. Topics include:

- Overview (p. 1360)
- Stop and Lock Properties (p. 1361)
- Joint Types Support for Stops and Locks (p. 1362)
- Solver Implications (p. 1363)
- Radial Gap Stop (p. 1363)
- Coefficient of Restitution (p. 1365)

Overview

Stops and Locks are optional constraints that may be applied to restrict the motion of the free relative degree(s) of freedom (DOF) of most types of joints. Any analysis that includes a valid joint type (p. 1362) can involve Stops and/or Locks. For the applicable joint types, you can define a minimum and maximum (min, max) range inside of which the degrees of freedom must remain.

Definition:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

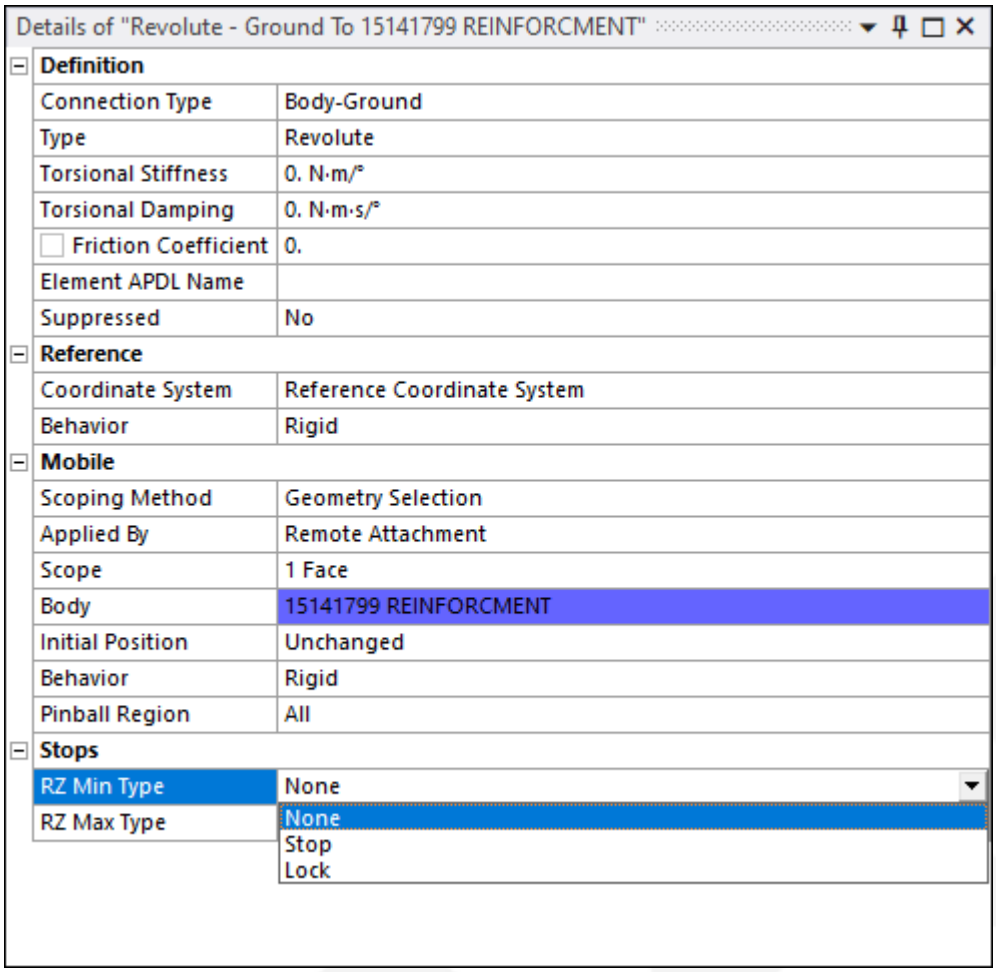
- **Stop**: A Stop is a computationally efficient abstraction of a real contact, which simplifies geometry calculations. For Stops, a shock occurs when a joint reaches the limit of the relative motion.
- **Lock**: A Lock is the same as a Stop except that when the Lock reaches the specified limit for a degree of freedom the Lock becomes fixed in place.

> **Warning:**
>
> *When using a structural system with the MAPDL solver, use Joint Stops sparingly. The solver treats the stop constraint internally as a "must be imposed" or "hard" constraint and no contact logic is used. As a result, during the given iteration of a substep, the stop constraints activate immediately if the application detects a violation of a stop limit. Depending upon the nature of the problem, the stop constraint implementation may cause the solution to trend towards an equilibrated state that may not be readily apparent to you. In addition, do not use stops to simulate zero-displacement boundary conditions. You should also avoid specifying stops on multiple joints. Finally, do not use joint stops as a substitute for contact modeling. Whenever possible, you need to use node-to-node or node-to-surface contact modeling to simulate limit conditions.*

Stop and Lock Properties

For joints with free relative DOFs, the Details pane displays a group of options labeled **Stops**. This grouping displays the applicable free DOFs (UX, UY, UZ, ROTX. etc.) for the joint type from which you specify the constraint as a **Stop** or a **Lock** (as shown below). By default, no Stop or Lock is specified, as indicated by the default option, **None**. You can select any combination of options. For stops and locks, the minimum and maximum values you enter are relative to the joint's coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Joint Types Support for Stops and Locks

Stops and Locks are applied to the following joint types.

Joint Type	Stop/Lock Ansys Rigid Dynamics	Stop/Lock Mechanical APDL	Stop/Lock Ansys Explicit Dynamics
Revolute	Yes	Yes	Yes
Cylindrical	Yes	Yes	Yes
Translational	Yes	Yes	Yes
Slot	Translational	Translational	Translational
Universal	Yes	Yes	No
Spherical	No (Radial Gap is available through Spherical Gap joint)	Yes	No
Planar	Yes	Yes	Yes
General	Translational, Radial Gap	Translational	Translational
Bushing	Translational, Radial Gap	Translational	Not supported
Spherical Gap	Radial Gap stop is always defined	Not supported	Not supported
In-Plane Radial Gap	Radial Gap stop is always defined	Not supported	Not supported

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Joint Type	Stop/Lock Ansys Rigid Dynamics	Stop/Lock Mechanical APDL	Stop/Lock Ansys Explicit Dynamics
Radial Gap	Radial Gap stop is always defined, translation along Z if free	Not supported	Not supported

Note:

- When using the Mechanical APDL solver, Stops and Locks are active only when **Large Deflection** (p. 1435) is set to **On** (under Analysis Settings). This is because Stops and Locks make sense only in the context of finite deformation/rotation. If **Large Deflection** is **Off**, all calculations are carried out in the original configuration and the configuration is never updated, preventing the activation of the Stops and Locks.
- It is important to apply sensible Stop and Lock values to ensure that the initial geometry configuration does not violate the applied stop/lock limits. Also, applying conflicting boundary conditions (for example, applying Acceleration on a joint that has a Stop, or applying Velocity on a joint that has a Stop) on the same DOF leads to non-physical results and therefore is not supported.

Solver Implications

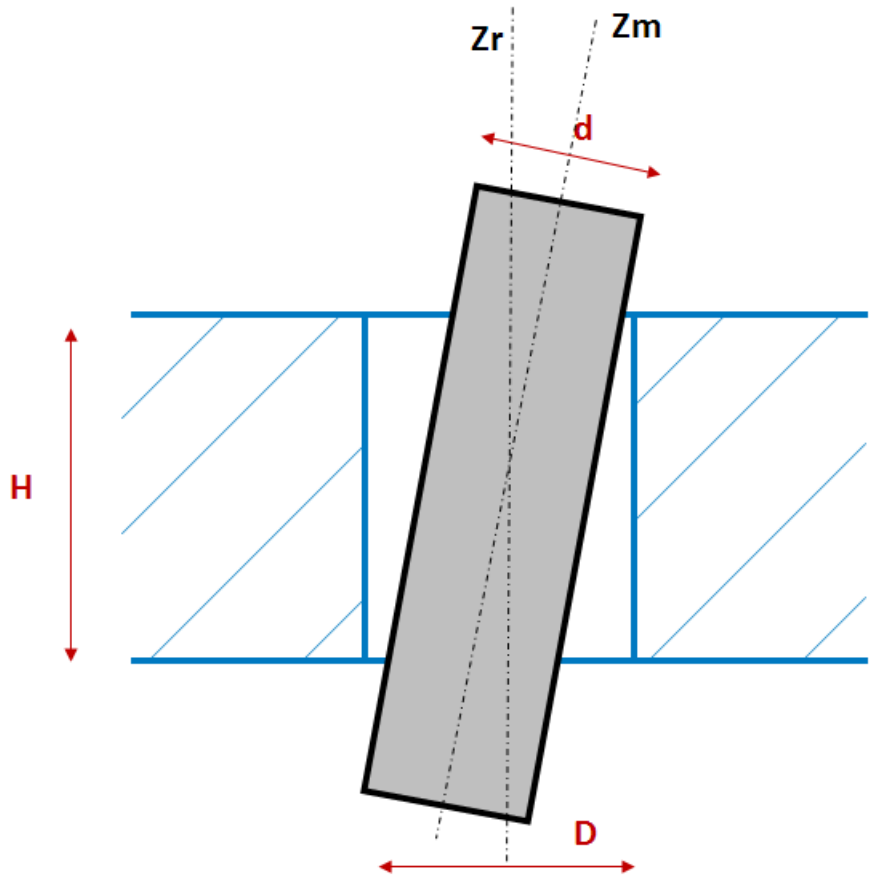
Stops and Locks are available for the Ansys Explicit Dynamics, Ansys Rigid Dynamics, and Mechanical APDL solvers, but are handled differently in certain circumstances by the three independent solvers.

- For the **Ansys Rigid Dynamics** solver the shock is considered as an event with no duration, during which the forces and accelerations are not known or available for postprocessing, but generate a relative velocity "jump".
- For the Mechanical APDL solver the stop and lock constraints are implemented via the Lagrange Multiplier method. The constraint forces due to stop and lock conditions are available when stop is established
- For the Ansys Explicit Dynamics solver the stop/lock event is checked during the timestep. If it is active the timestep is split up such that the free motion is still considered for the first part. The second part of the timestep is computed using either the reversed velocity (while taking into account the restitution factor) or the new fixed DOF. If other stop/locks events are active also, they will be addressed during the next cycle in the solver. Due to the small timesteps in an explicit analysis, this approximation is typically negligible.

Radial Gap Stop

To model clearance in revolute or cylindrical joints, a special type of stop called a radial gap stop can be used. A radial gap stop limits the relative rotation of either the X or Y rotation, limiting the Z axis tilt of the joint's mobile coordinate system with respect to the Z axis of the reference coordinate system. This stop idealizes a revolute joint with a gap between the inner and the outer cylinder that allows the shaft to translate and tilt in the outer cylinder, as shown on the following figure:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Where:

d is the inner diameter.

D is the outer diameter.

H is the height of the joint.

Important Notes:

- The Outer Diameter is considered to be on the reference side of the joint, so you might have to flip reference and mobile on the joint to properly define a radial gap.
- The shaft is considered to be infinitely long.
- If the joint allows relative translations, the center of the shaft will shift with these translations. The radial gap accounts for this center shift.
- The principal axis of the radial gap is Z, meaning that the tilt occurs along the X and Y rotations of the gap.
- Radial gap stops do not support tilt angles greater than 1 rad.
- Radial gap is always included on imperfect joint types (spherical gap, in-plane radial gap, and radial gap)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Coefficient of Restitution

For the **Ansys Rigid Dynamics** and **Ansys Explicit Dynamics** solvers, Stops require you to set a coefficient of restitution value. This value represents the energy lost during the shock and is defined as the ratio between the joint's relative velocity prior to the shock and the velocity following the shock. This value can be between 0 and 1. For a restitution value of zero, a Stop is released when the force in the joint is a traction force, while a Lock does not release. A restitution factor equal to 1 indicates that no energy is lost during the shock, that is, the rebounding velocity equals the impact velocity (a perfectly elastic collision).

The coefficient of restitution is not applicable to the stops on the joints when using the Mechanical APDL solver.

Ease of Use Features

The following ease of use features are available when defining joints:

- Renaming Joint Objects Based on Definition (p. 1365)
- Remote Connection Display (p. 1366)
- Joint Legend (p. 1366)
- Disable/Enable Transparency (p. 1367)
- Hide All Other Bodies (p. 1367)
- Flip Reference/Mobile (p. 1368)
- Joint DOF Checker (p. 1368)
- Redundancy Analysis (p. 1368)
- Model Topology (p. 1369)

Renaming Joint Objects Based on Definition

When joints are created, Mechanical automatically names each of the joint objects with a name that includes the type of joint followed by the names of the joined parts included as child objects under the **Geometry** object folder. For example, if a revolute joint connects a part named **ARM** to a part named **ARM_HOUSING**, then the object name becomes **Revolute - ARM To ARM_HOUSING**.

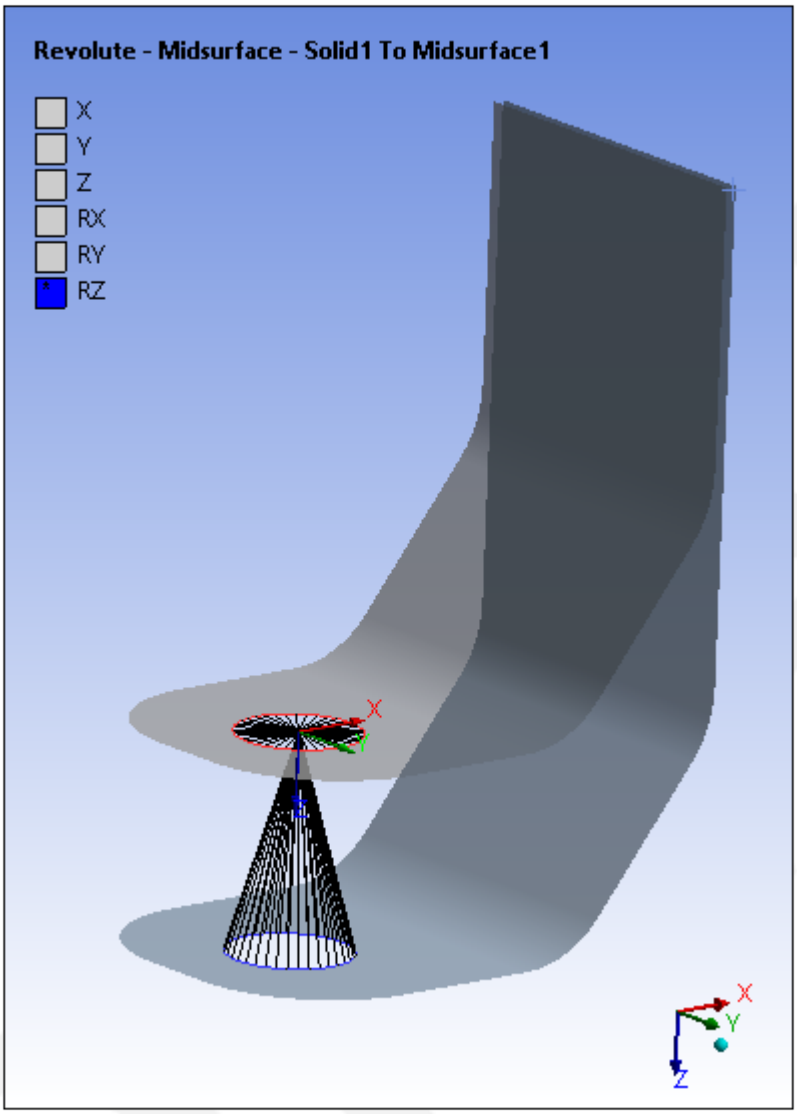
The automatic naming based on the joint type and geometry definition is by default. You can however change the default from the automatic naming to a generic naming of **Joint**, **Joint 2**, **Joint 3**, and so on by opening the **Options** dialog and setting **Auto Rename Connections** to **No** under the **Connections** (p. 219) group.

If you then want to rename any joint object based on the definition, click the right mouse button on the object and choose **Rename Based on Definition** from the context menu. You can rename all joints by clicking the right mouse button on the **Joints** folder then choosing **Rename Based on Definition**. The behavior of this feature is very similar to renaming manually created contact regions. See Renaming Contact Regions Based on Geometry Names (p. 1271) for further details including an animated demonstration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Remote Connection Display

When defined as a **Remote Attachment**, joints are considered to be a remote boundary condition (p. 1735), and can make use of remote points that were either specifically defined or created internally by the application. To do a visual check, you can display the connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab). An example is illustrated here of a spring scoped to two edges.

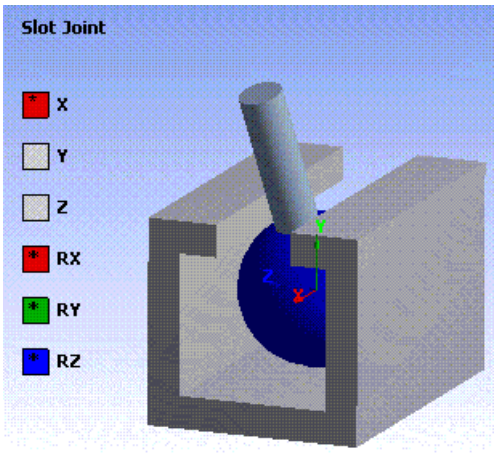


Joint Legend

When you highlight a joint object, the accompanying display in the **Geometry** window includes a legend that depicts the free degrees of freedom characteristic of the type of joint (p. 1298). A color scheme is used to associate the free degrees of freedom with each of the axis of the joint's coordinate system shown in the graphic. An example legend is shown below for a slot joint (p. 1301).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

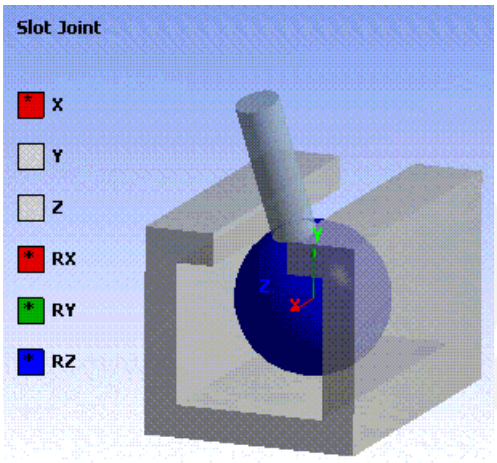
Published: 2023-10-23



You can display or remove the joint legend using **View> Legend** from the main menu.

Disable/Enable Transparency

The **Enable Transparency** feature enables you to graphically highlight a particular joint that is within a group of other joints, by rendering the other joints as transparent. The following example shows the same joint group presented in the Joint Legend (p. 1366) section above but with transparency enabled. Note that the slot joint alone is highlighted.



To enable transparency for a joint object, click the right mouse button on the object and choose **Enable Transparency** from the context menu. Conversely, to disable transparency, click the right mouse button on the object and choose **Disable Transparency** from the context menu. The behavior of this feature is very similar to using transparency for highlighting contact regions. See Controlling Transparency for Contact Regions (p. 1269) for further details including an animated demonstration.

Hide All Other Bodies

You can hide all bodies except those associated with a particular joint.

To use this feature, click the right mouse button on the object and choose **Hide All Other Bodies** from the context menu. Conversely, to show all bodies that may have been hidden, click the right mouse button on the object and choose **Show All Bodies** from the context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Flip Reference/Mobile

For body-to-body joint scoping, you can reverse the scoping between the **Reference** and **Mobile** sides in one action. To use this feature, click the right mouse button on the object and choose **Flip Reference/Mobile** from the context menu. The change is reflected in the Details view of the joint object as well as in the color coding of the scoped entity on the joint graphic. The behavior of this feature is very similar to the **Flip Contact/Target** feature used for contact regions. See Flipping Contact and Target Scope Settings (p. 1272) for further details including an animated demonstration.

Joint DOF Checker

Once joints are created, fully defined, and applied to the model, a **Joint DOF Checker** calculates the total number of free degrees of freedom. The number of free degrees of freedom should be greater than zero in order to produce an expected result. If this number is less than 1, a warning message is displayed stating that the model may possibly be overconstrained, along with a suggestion to check the model closely and remove any redundant joint constraints.

To display the **Joint DOF Checker** information, highlight the **Connections** object and click the **Worksheet** button. The **Joint DOF Checker** information is located just above the **Joint Information** heading in the worksheet.

Redundancy Analysis

This feature enables you to analyze an assembly held together by joints. This analysis will also help you to solve over constrained assemblies. Each body in an assembly has a limited degree of freedom set. The joint constraints must be consistent to the motion of each body, otherwise the assembly can be locked, or the bodies may move in unwanted directions. The redundancy analysis checks the joints you define and indicates the joints that over constrain the assembly. To analyze an assembly for joint redundancies:

1. Right-click the **Connections** object, and then select **Redundancy Analysis** to open a worksheet with a list of joints.
2. Click **Analyze** to perform a redundancy analysis. All the over constrained joints are indicated as redundant.
3. Click the Redundant label, and then select **Fixed** or **Free** to resolve the conflict manually.

   or

   Click Convert Redundancies to Free to remove all over constrained degrees of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Click **Set** to update the joint definitions.

Note:

Click Export to save the worksheet to an Excel/text file.

Important:

When a model contains a **Point On Curve** (p. 1312) joint, the **Configure** and **Assemble** options (p. 51) are disabled for all the joints. This is also the case for a redundancy analysis that includes a **Point On Curve** joint.

Model Topology

The Model Topology worksheet provides a summary of the joint connections between bodies in the model. This feature is a convenient way of verifying and troubleshooting a complex model that has many parts and joints. The Model Topology worksheet displays the connections each body has to other bodies, and the joint through which these bodies are connected. Additional information for the joints is provided, including the joint type and the joint representation for the rigid body solver (i.e. whether the joint is based on degrees of freedom or constraint equations).

To display the model topology, right-click the **Connections** object, and then select **Model Topology**. The **Model Topology** worksheet displays in the **Data View**. The content of the worksheet can be exported as a text file using the **Export** button.

Joints based on degrees of freedom are labeled either Direct or Revert in the Joint Direction column of the Model Topology table. Direct joints have their reference coordinate system on the ground side of the topology tree. Revert joints have their mobile coordinate system on the ground side. This information is useful for all post-processing based on python scripting, where internal data can be retrieved. For reverted joints, some of the joint internal results need to be multiplied by -1.

Refer to the Ansys Rigid Dynamics Theory Manual for more information on model topology and selecting degrees of freedom.

Detecting Overconstrained Conditions

Overconstrained conditions can occur when more constraints than are necessary are applied to a joint's degrees of freedom. These conditions may arise when rigid bodies are joined together using multiple joints. The overconstraints could be due to redundant joints performing the same function, or contradictory motion resulting from improper use of joints connecting different bodies.

- For the Transient Structural analysis type, when a model is overconstrained, nonconvergence of the solution most often occurs, and in some cases, overconstrained models can yield incorrect results.
- For the Rigid Dynamics analysis type, when a model is overconstrained, force calculation cannot be done properly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The following features exist within Mechanical that can assist you in detecting possible overconstrained conditions:

- Use the Joint DOF Checker (p. 1368) for detecting overconstrained conditions before solving (highlight the **Connections** object and select the Worksheet option on the Context tab). In the following example, the original display of the Joint DOF Checker warns that the model may be overconstrained.

  Joint DOF Checker

  6 Bodies x 6 DOF = +36
  4 Fixed constraint x 6 DOF = -24
  2 Revolute joint x 5 DOF = -10
  1 Translational joint x 5 DOF = -5
  Free DOF = -3 DOF

  Warning:
  This model is possibly over constrained. Consider modifying joint definitions.
  Number of overconstraints = 3

  After modifying the joint definitions, display the Joint DOF Checker again, which shows that the overconstrained condition has been resolved.

  Joint DOF Checker

  6 Bodies x 6 DOF = +36
  4 Fixed constraint x 6 DOF = -24
  1 Revolute joint x 5 DOF = -5
  1 Slot joint x 2 DOF = -2
  1 Cylindrical joint x 4 DOF = -4
  Free DOF = +1 DOF
  Number of overconstraints = 0

- After solution, you can highlight the **Solution Information** object, then scroll to the end of its content to view any information that may have been detected on model redundancies that caused overconstrained conditions. An example is presented below.

```
Analysis of model redundancies:
Joint Revolute - Part 4 To Part 3 shows some constraint redundancies
X Translation constraint equation was at time redundant
Y Translation constraint equation was at time redundant
Z Translation constraint equation was at time redundant
X Rotation constraint equation was at time redundant
Y Rotation constraint equation was at time redundant
```

Example: Assembling Joints

This section illustrates the details of assembling geometry using an example of a three-part a pendulum joint model.

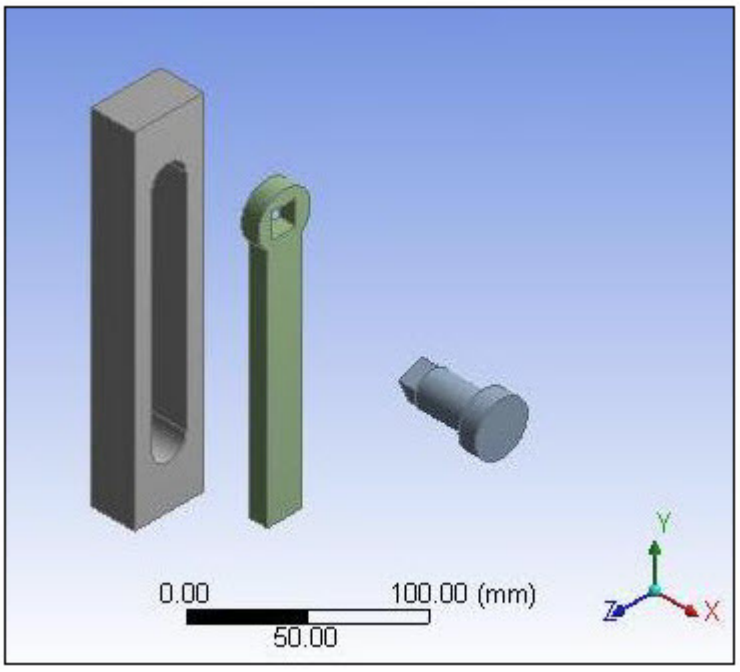
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The Assemble feature enables you to bring in CAD geometry that may initially be in a state of disassembly. After importing the CAD geometry, you can actively assemble the different parts and Set them in the assembled configuration for the start of the analysis.

The geometry shown for the example in Figure 22: Initial Geometry (p. 1371) was imported into a Rigid Dynamics analysis System.

Figure 22: Initial Geometry



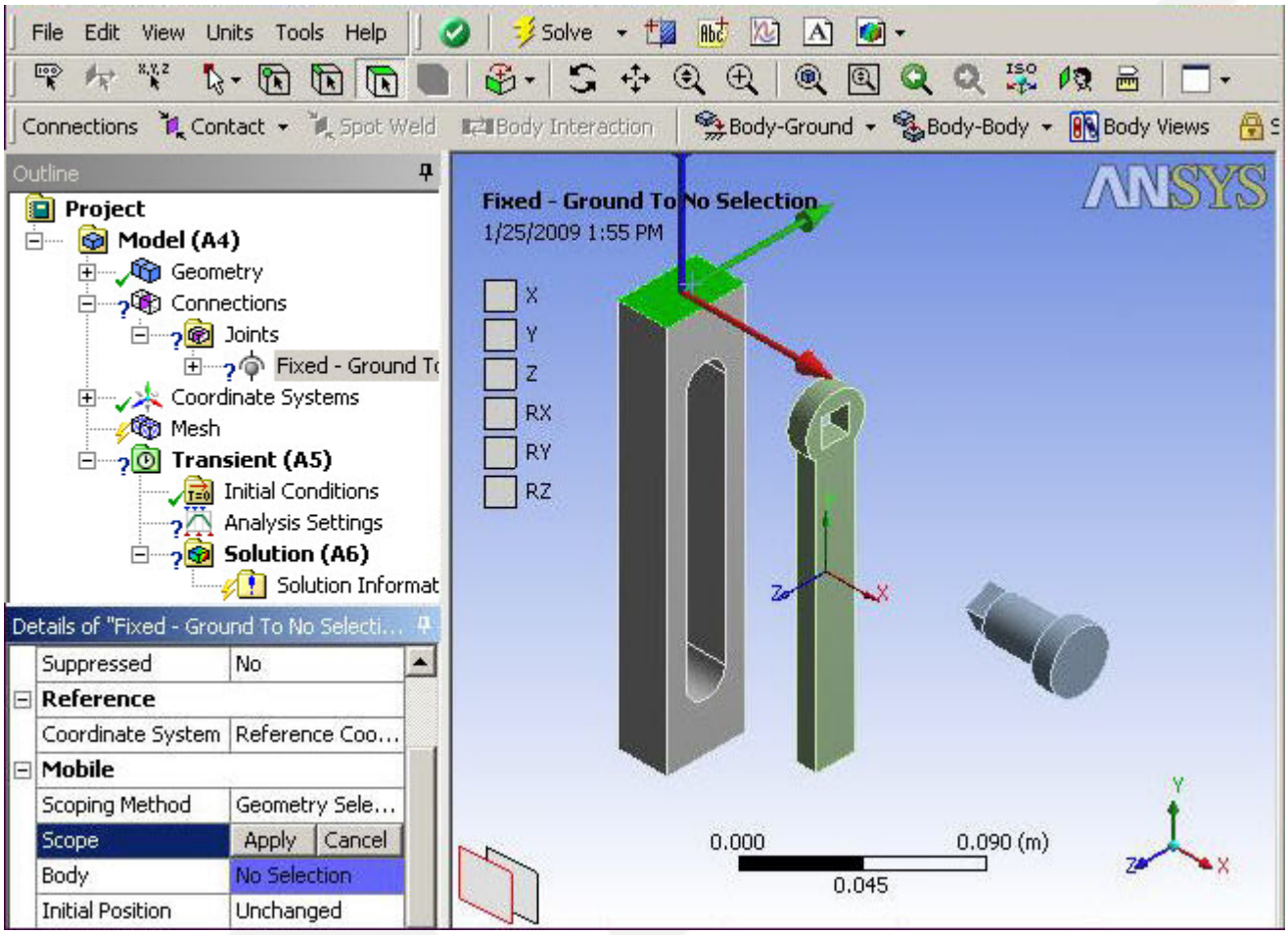
This geometry consists of three bodies. In Figure 22: Initial Geometry (p. 1371) they are (from left to right) the Basis, the Arm, and the PendulumAxis. These three bodies have been imported completely disjointed/separate from each other.

The first step to orient and assemble the bodies is to add a Body-Ground Fixed joint to the body named Basis. To do this:

1. Select the **Connections** object from the Outline.
2. From the **Connections** Context tab, open the **Body-Ground** drop-down menu and select **Fixed**.
3. Select a flat external face on the Basis body as seen in Figure 23: Selecting a Face for a Body-Ground Fixed Connection (p. 1372).
4. In the Details view under Mobile, click in the Scope field and select **Apply**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 23: Selecting a Face for a Body-Ground Fixed Connection

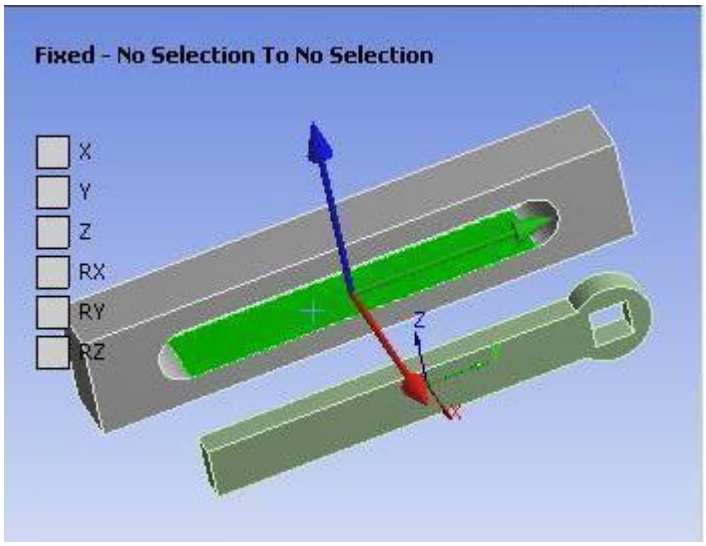


Next, you need to join the PendulumAxis to the Basis. Since they are initially disjoint, you need to set two coordinate systems, one for the Basis and the other for the PendulumAxis. Additionally, to fully define the relative position and orientations of the two bodies, you must define a fixed joint between them. To do this:

1. From the **Connections** Context tab, open the **Body-Body** drop-down menu and select **Fixed**.
2. Highlight the face on the Basis as shown below.

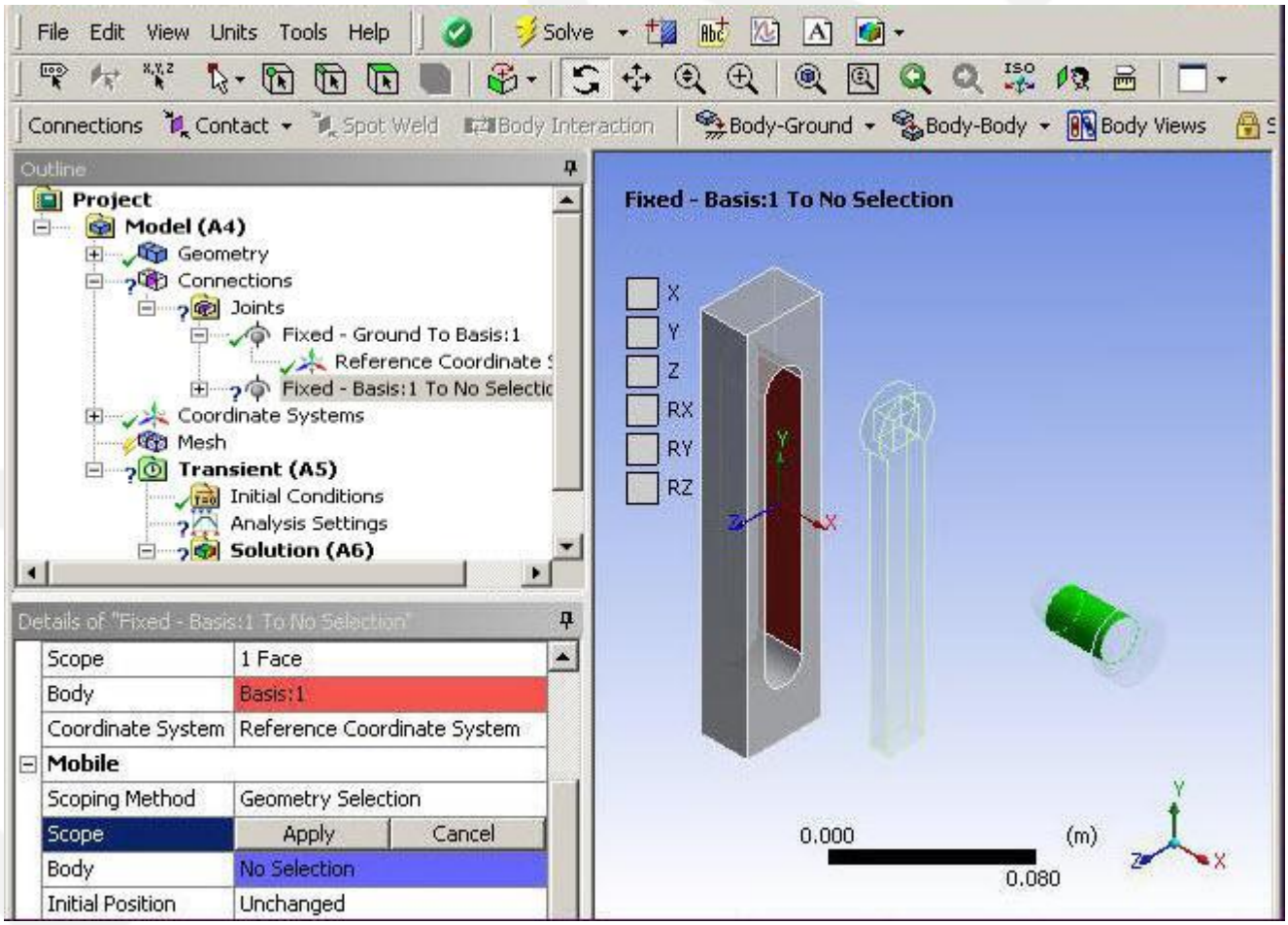
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



3. In the Details view, click the Scope field under Reference and select **Apply**.
4. Select the cylindrical face on the PendulumAxis.
5. In the Details view, select the Scope field under Mobile and select **Apply**.

Figure 24: Creating the Reference Mobile System



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

6. Also, change the Initial Position value under Mobile from **Unchanged** to **Override**.

Now, the joint has two coordinate systems associated with it: A Reference and a Mobile coordinate system.

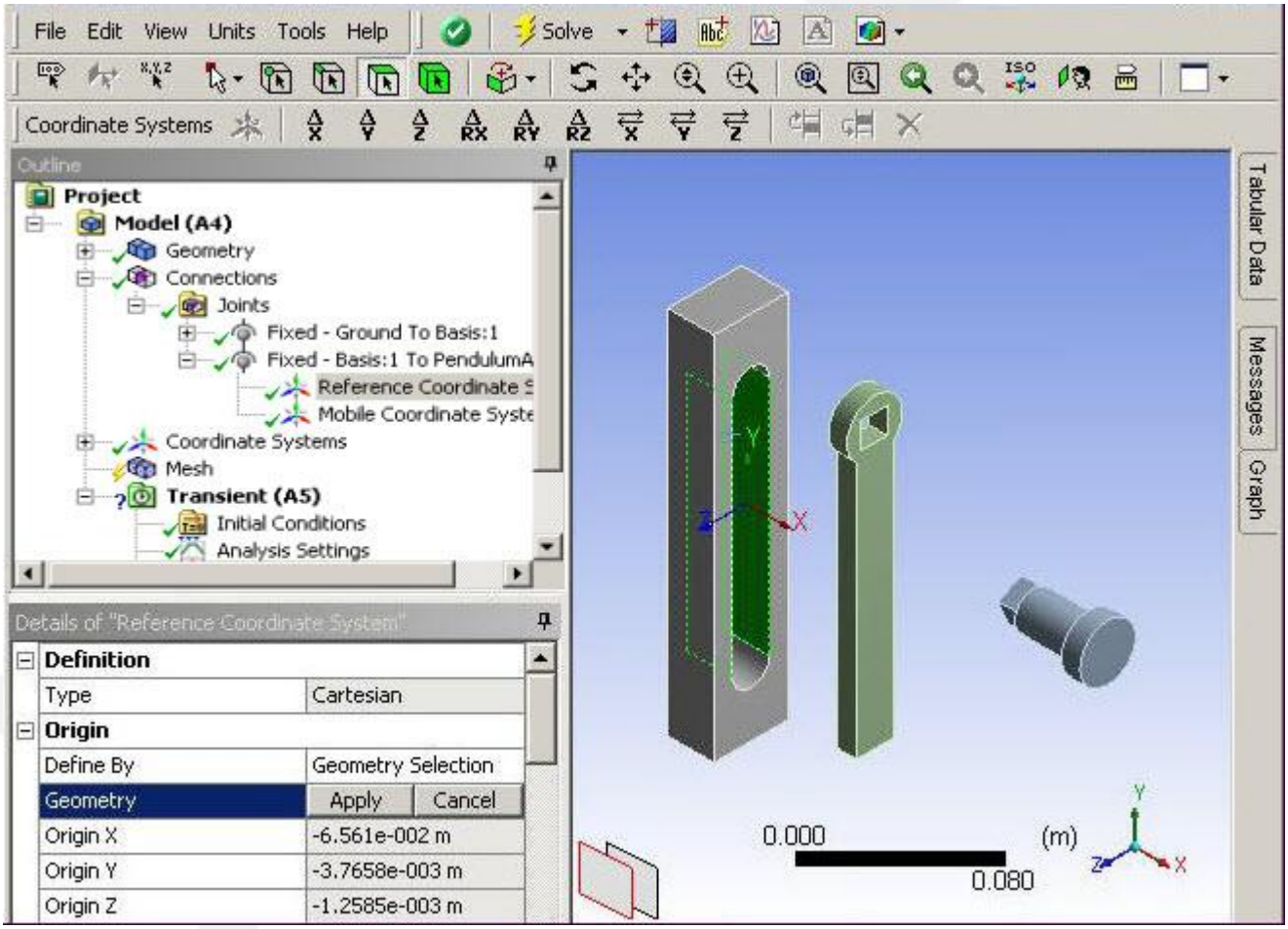
Next, you must associate the Reference and the Mobile Coordinate Systems to the respective bodies with the appropriate orientations. To associate the Reference Coordinate System to the respective bodies:

1. In the Outline, highlight Reference Coordinate System.

2. In the Details view, click the box next to Geometry under Origin.

3. Select the two internal rectangular faces on the Basis as shown in Figure 25: Creating the Reference Coordinate System (p. 1374) and in the Details view, select Apply. This will center The Reference Coordinate System at the center of the hole on the Basis.

Figure 25: Creating the Reference Coordinate System



To associate the Mobile Coordinate System to the respective bodies:

1. Highlight the Mobile Coordinate System (this coordinate system is associated with the Basis).

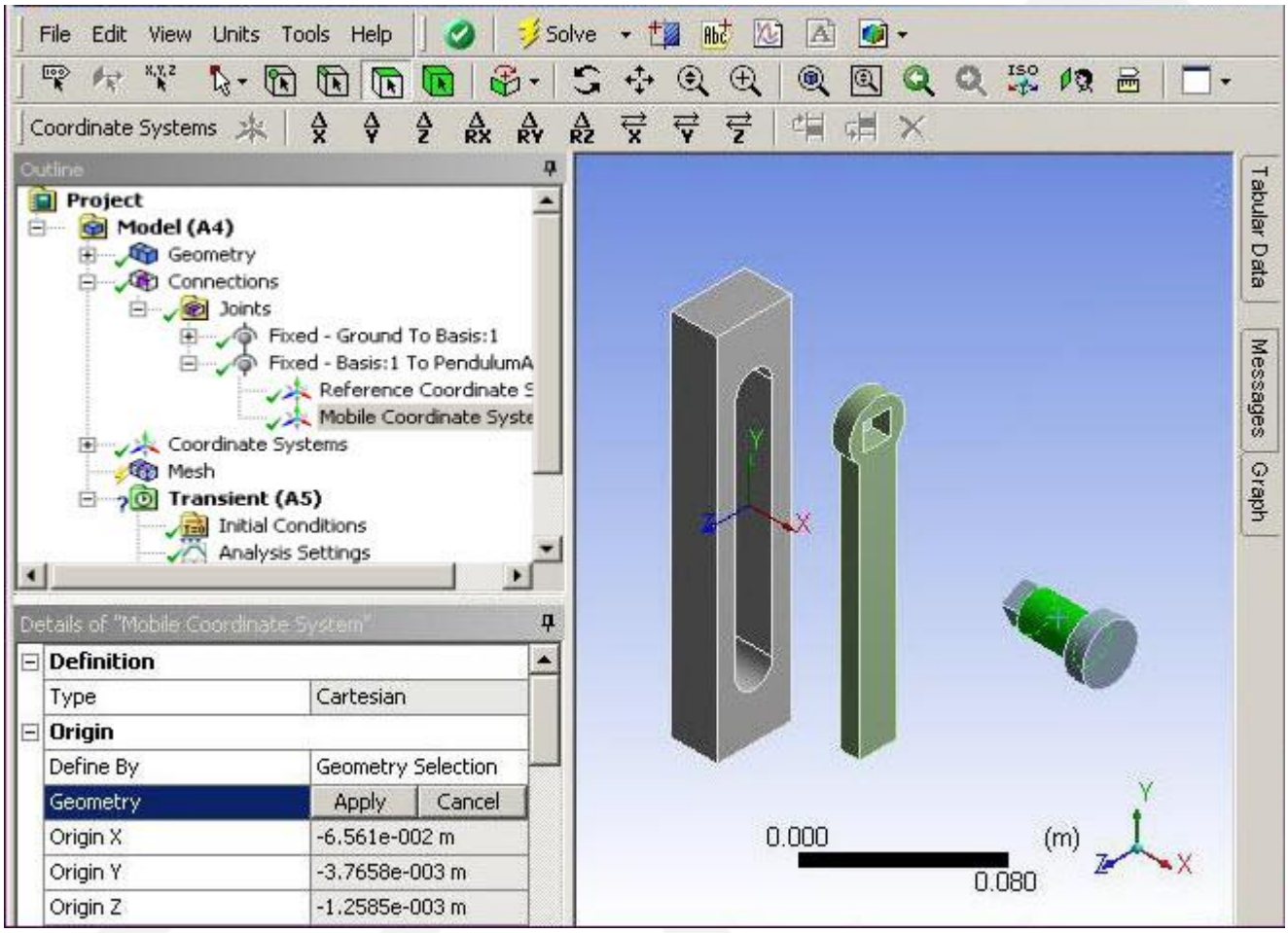
2. In the Details view, click in the Geometry field under Origin.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. Select the cylindrical surface on the PendulumArm.
4. In the Details view, click **Apply**.

Figure 26: Creating the Mobile Coordinate System

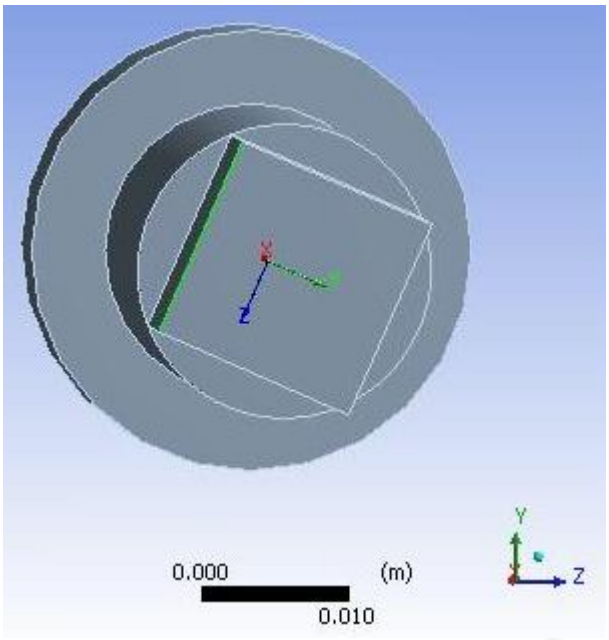


Next, you will need to orient the PendulumAxis coordinate system so that it is oriented correctly in the assembly:

1. In the Mobile Coordinate System associated with the PendulumAxis, click in the box next to Geometry under Principal Axis (set to Z).
2. Select one of the vertical edges on the PendulumAxis such that the Z axis is parallel to it as shown in Figure 27: Orienting the Pendulum Axis (p. 1376). In the Details view, click **Apply**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 27: Orienting the Pendulum Axis



3. With Mobile Coordinate System highlighted in the Outline, select the x-offset button on the Context tab.

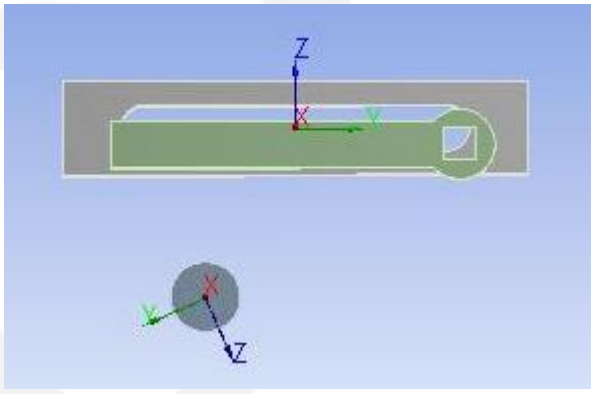
4. In the Details view, enter an Offset X value of 2.5mm to align the faces of the PendulumAxis with the Basis.

Note:

The transformations available allow you to manipulate the coordinate systems by entering offsets or rotations in each of the 3 axis.

The two coordinate systems that were just defined should look similar to the figure below.

Figure 28: Oriented Coordinate Systems

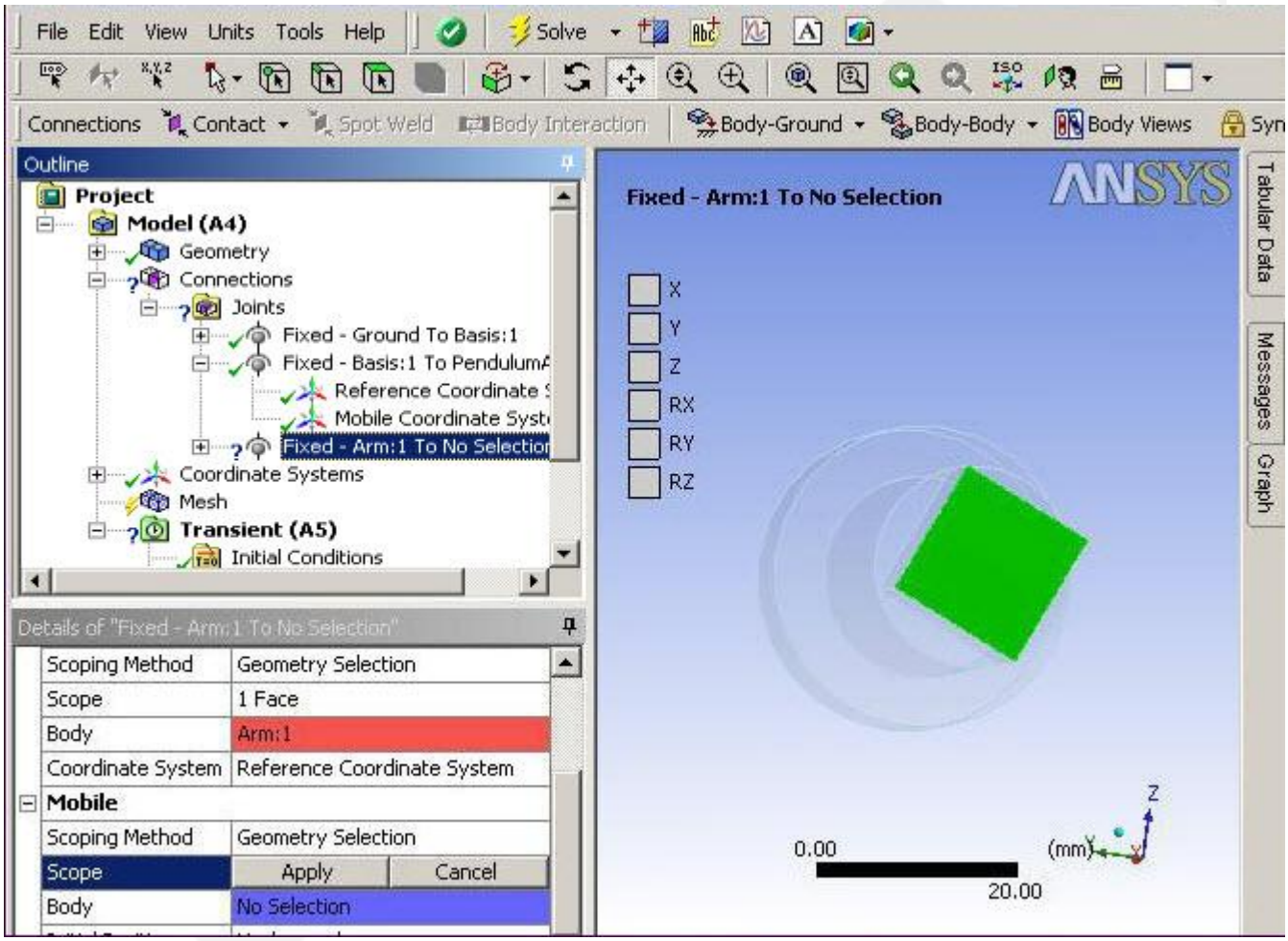


Next, you will need to define the coordinate systems to join the Arm to the PendulumAxis during assembly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the Context tab , select **Body-Body** > **Fixed**.
2. To define the Reference Scope, choose one of the faces of the Arm that will be connected to the PendulumAxis then select **Apply**.
3. Now, configure the Mobile Scope by selecting the flat end face of the PendulumAxis as shown in Figure 29: Scoping the Mobile Coordinate Systems (p. 1377), then select **Apply**.

Figure 29: Scoping the Mobile Coordinate Systems



4. Set the Initial Position under Mobile from **Unchanged** to **Override**.
5. Finally, set the Origin of the Reference Coordinate System to the center of the hole in the Arm using the same procedure described above for the Basis.

Next, you will need to offset the Coordinate System associated with the Arm so that the faces on the Arm are aligned with the end face of the PendulumAxis.

1. With Reference Coordinate System highlighted, select the x-offset button on the Context tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

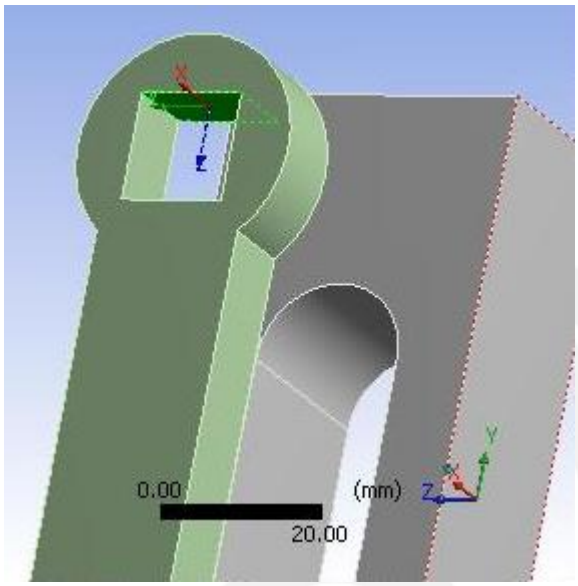
2. Enter an Offset X value of -5mm.

Note:

The transformations available allow you to manipulate the coordinate systems by entering offsets or rotations in each of the 3 axis.

3. Next, highlight the Mobile Coordinate System. This coordinate system is associated with the Arm. Click the box next to Geometry under Origin

4. Select the flat surface on the PendulumArm and click **Apply**.



Now you will need to orient the PendulumAxis so that its faces are aligned with the faces on the Arm during the Assemble process.

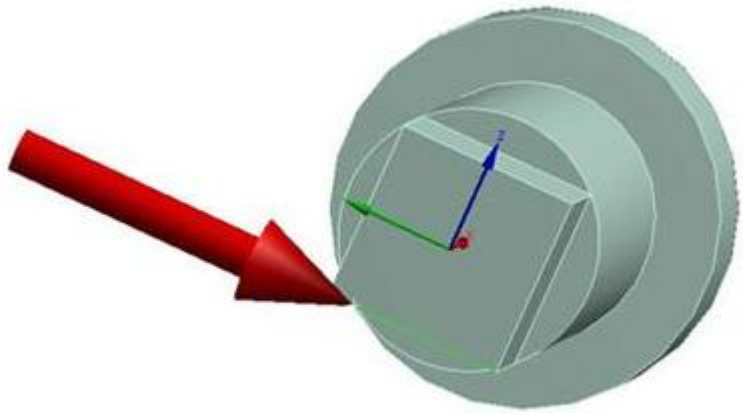
1. Highlight the Mobile Coordinate System that is assigned to the PendulumAxis.

2. From the Details view, click the in the Geometry field under Principal Axis and select an edge of the PendulumAxis as shown in the figure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Figure 30: Choose an Edge to Orient the PendulumAxis Geometry



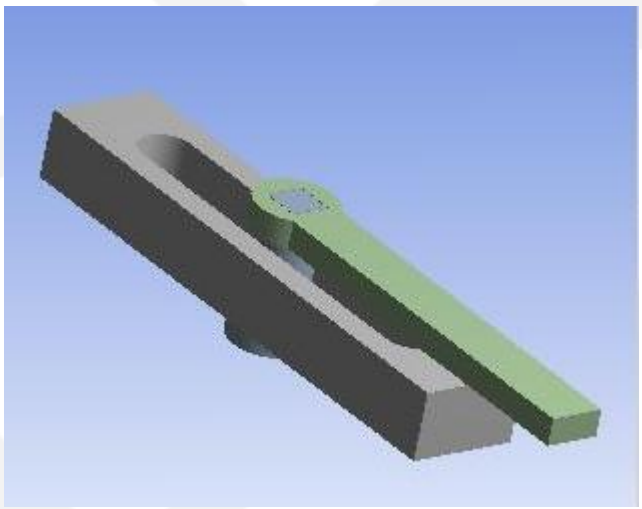
3. Under Principal Axis In the Details view, select Apply in the Geometry field to orient the PendulumAxis to this edge.

Now that the three bodies have been oriented and aligned, they are ready to be assembled.

1. In the Outline, highlight Connections.
2. From the Context tab, select **Assemble**.

The parts should snap together in place and resemble Figure 31: Assembled Geometry (p. 1379). If the geometry you're attempting to assemble has not snapped into place as expected, you should retrace your previous steps to make sure that the coordinate systems are properly oriented. If your assembly has been successfully performed, then click Set on the Context tab to place the assembly in its assembled position to start the analysis.

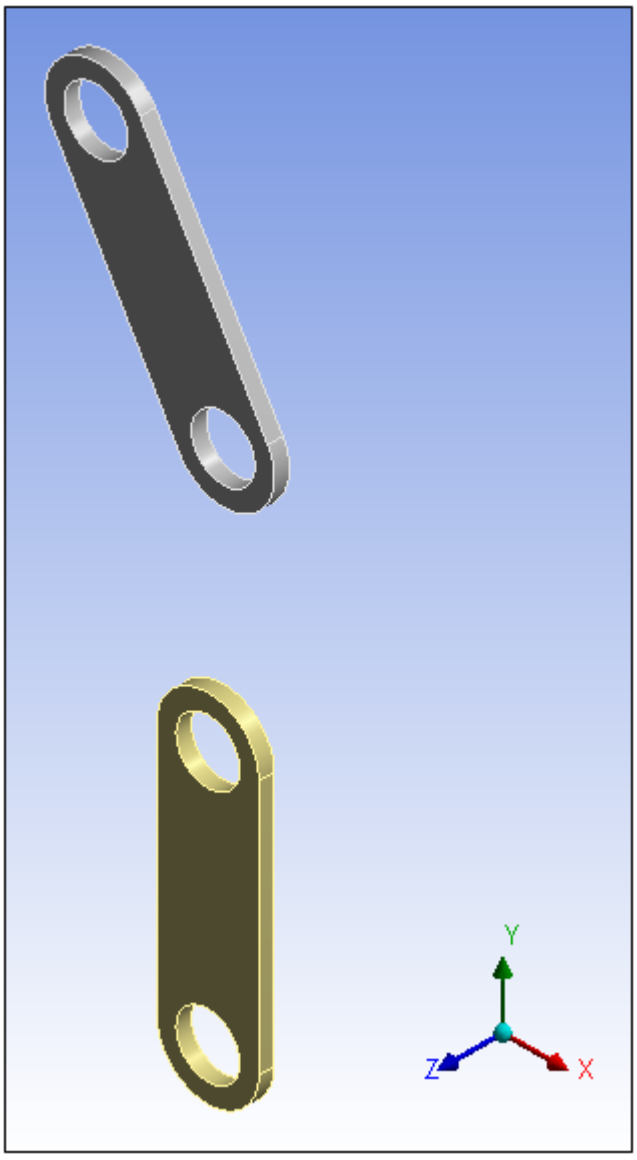
Figure 31: Assembled Geometry



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example: Configuring Joints

This section presents an example of some common joint configuration steps for a model of a pendulum created from two links, as illustrated below:



To achieve the desired result, two revolute (p. 1299) joints were created and configured:

- The first joint is intended to allow rotation of the top link's upper hole referenced to a stationary point (Body-Ground Revolute Joint).
- The second joint is intended to allow rotation of the bottom link's upper hole referenced to the top link's lower hole (Body-Ground Revolute Joint).

Tip:

Make sure that the **Auto Rename Connections** preference (Options > Connections > Default) is set to **Yes**.

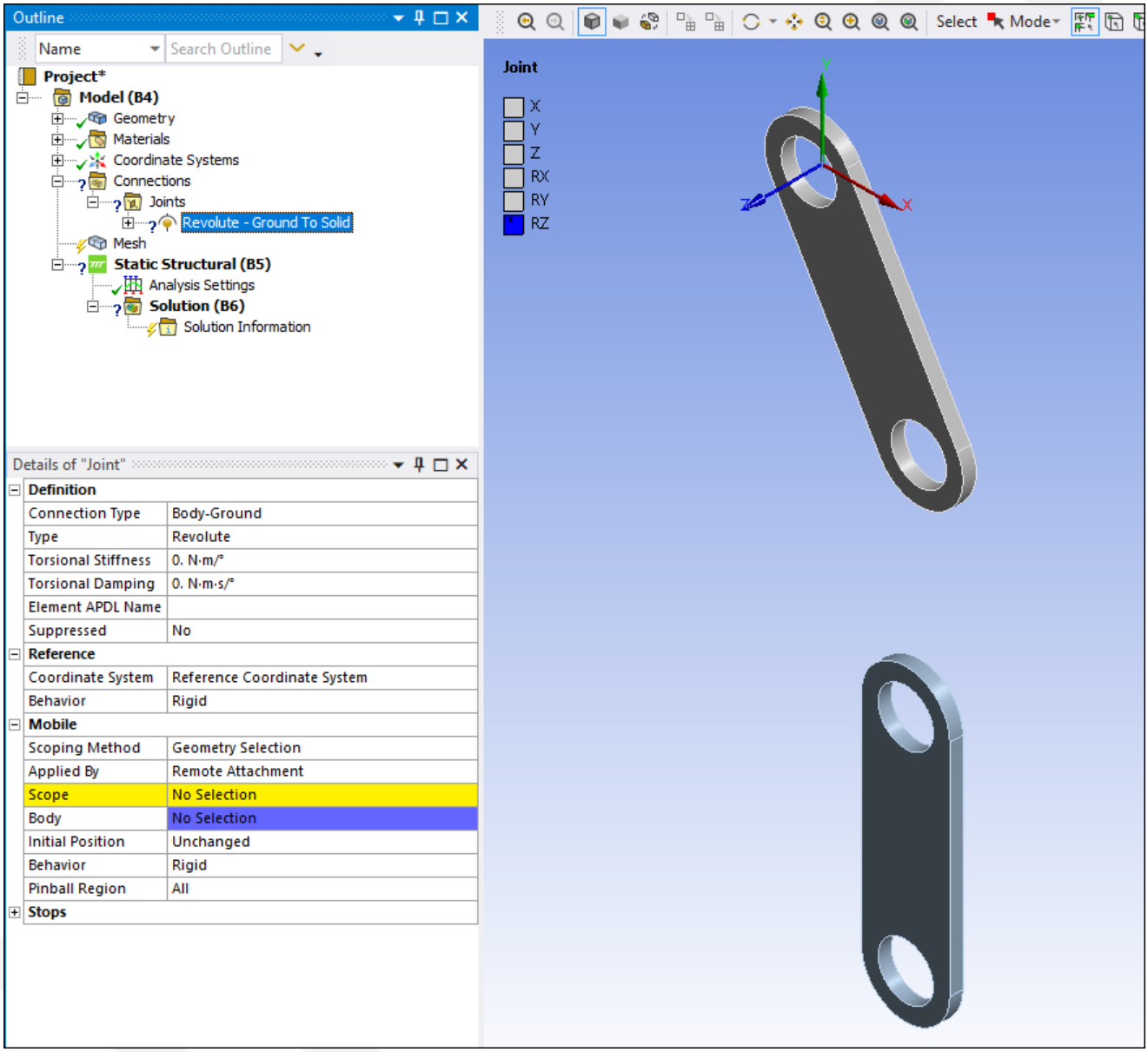
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The following steps illustrate the steps of a common joint configuration:

1. After attaching the model to Mechanical, create the first revolute joint.

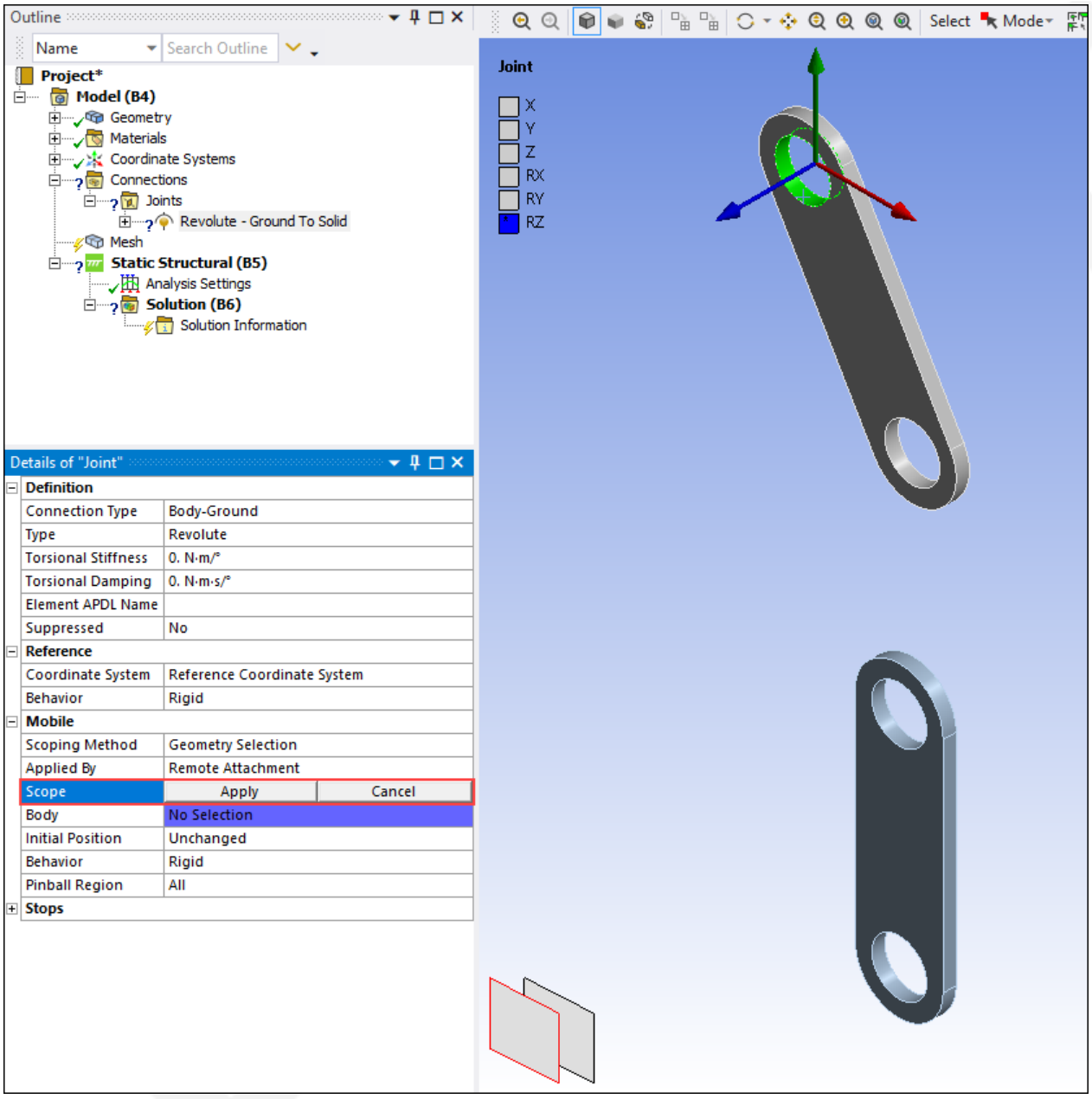
   Select the **Connections** object, open the **Body-Ground** drop-down menu, (Connections (p. 50) Context Tab, and select **Revolute**. The new joint object becomes the active object in the Outline.



2. Scope the **Mobile** side of the first revolute joint to the top link's upper hole.

   Select the inner surface of the upper hole and then under **Mobile** category in the Details view, select the **Scope** field and click the **Apply** button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Create the second revolute joint.

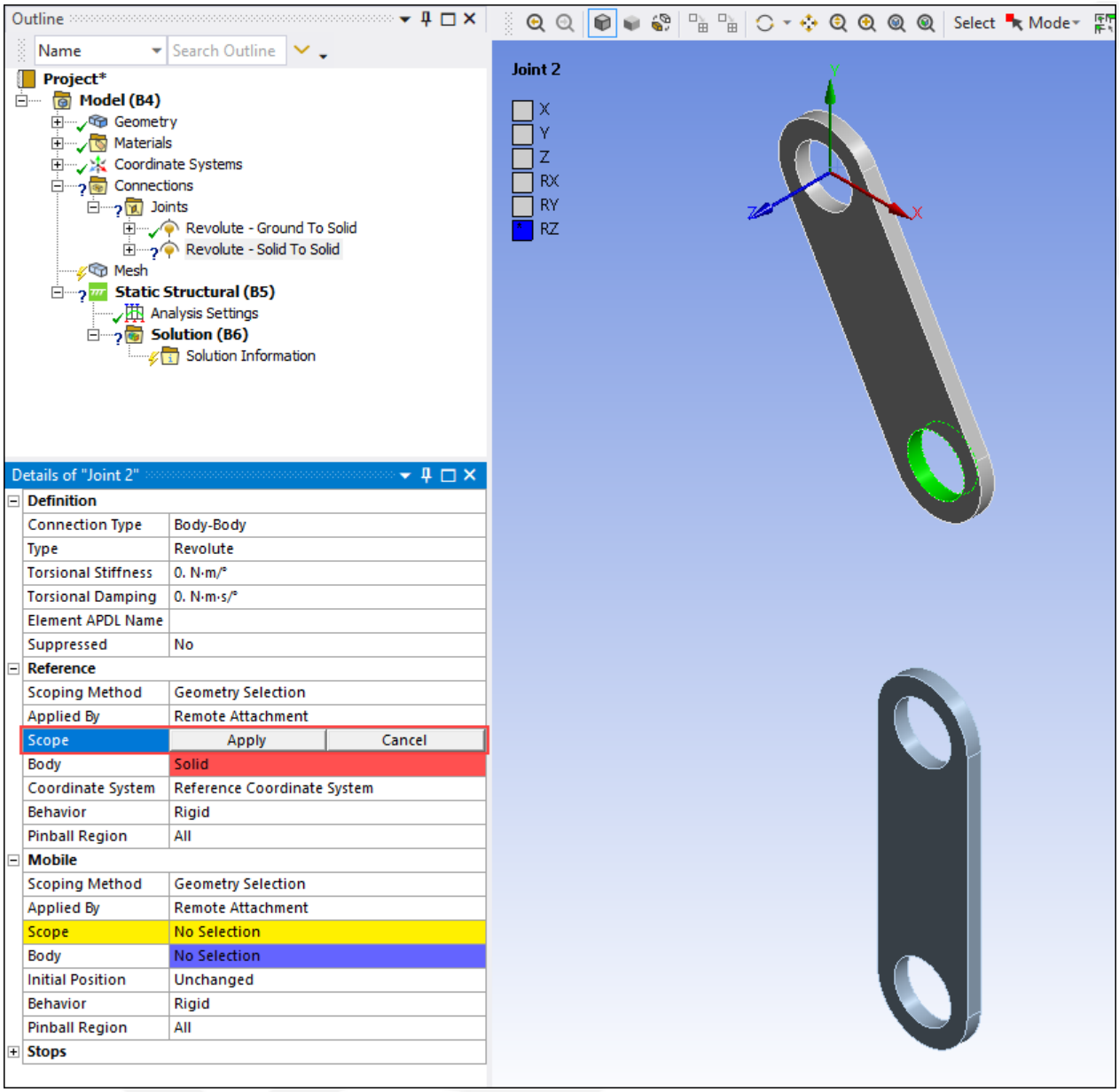
   Open the **Body-Body** drop-down menu from the Connections (p. 50) Context Tab and select **Revolute**. The new joint object becomes the active object in the Outline.

4. Scope the **Reference** side of the second joint to the top link's lower hole.

   Select inner surface of hole and the under **Reference** category in the Details, select the **Scope** field and click the **Apply** button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

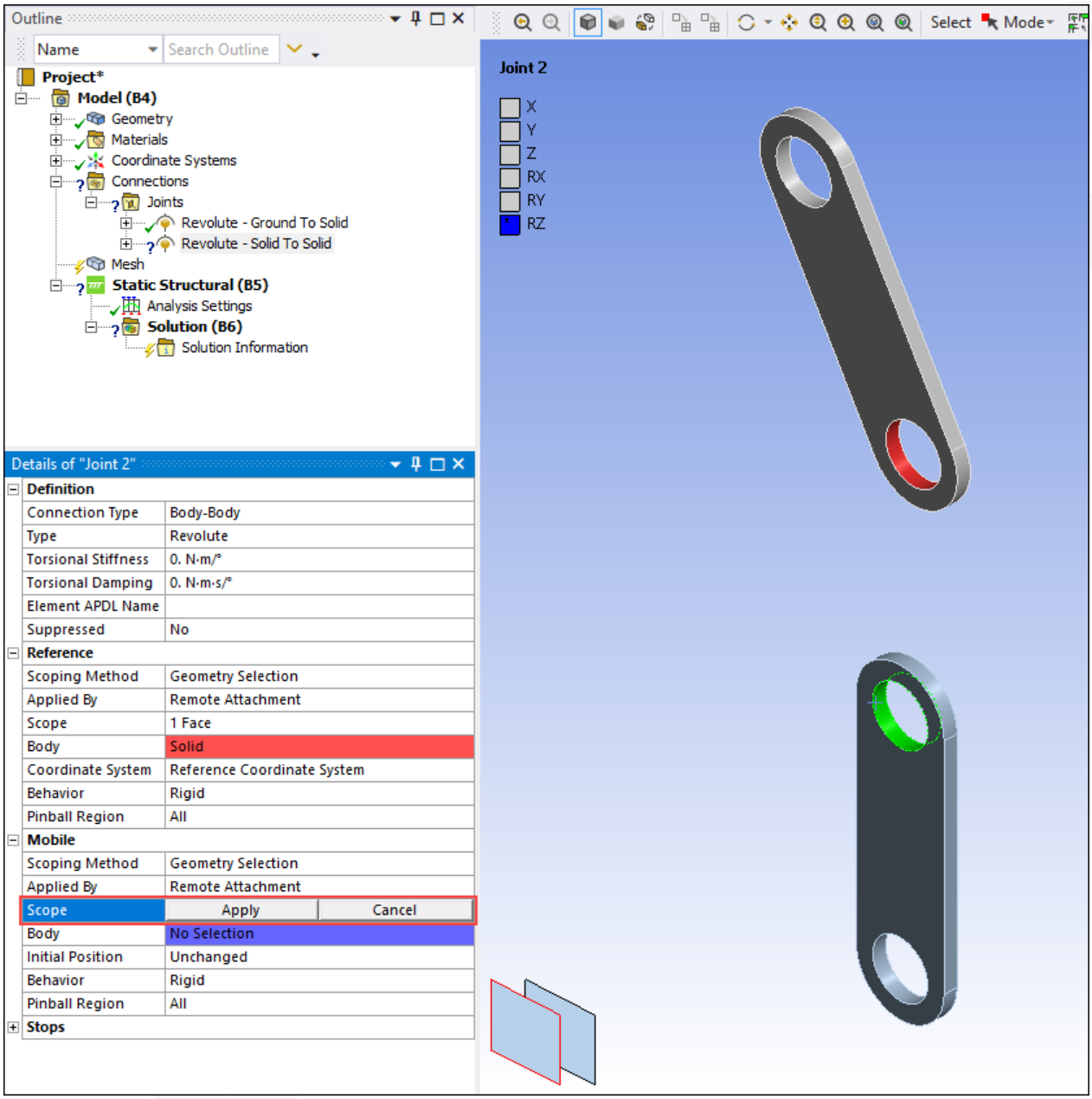
Published: 2023-10-23



5. Scope the mobile side of the second joint to the bottom link's upper hole.

   Select inside surface of hole, then under **Mobile** category in the Details view, select the **Scope** field and click the **Apply** button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

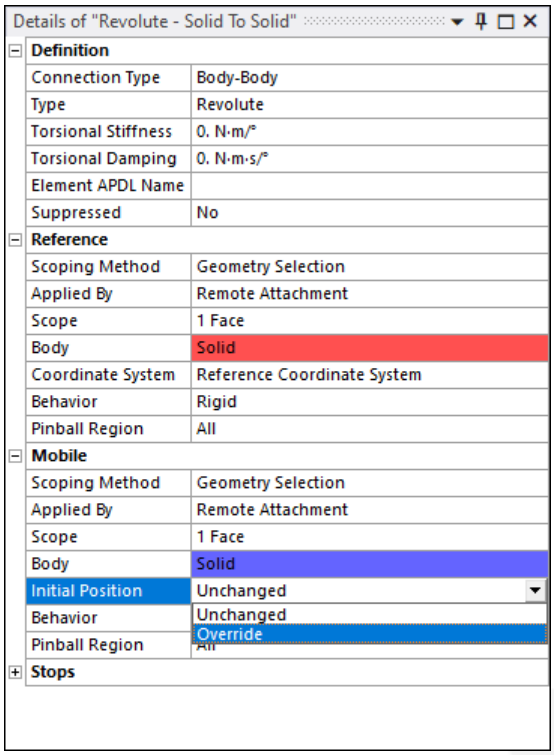


6. The two holes that will form the second joint need to be properly aligned. To align the holes, you need to indicate that the two holes need to match. To achieve this, first create a coordinate system for the mobile side of the second joint, and then align the **Mobile** and **Reference** coordinate systems.

   Highlight the second joint, **Revolute - Solid To Solid**, in the tree and select **Override** from the drop-down menu of the **Initial Position** property. Note that a new **Coordinate System** property displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

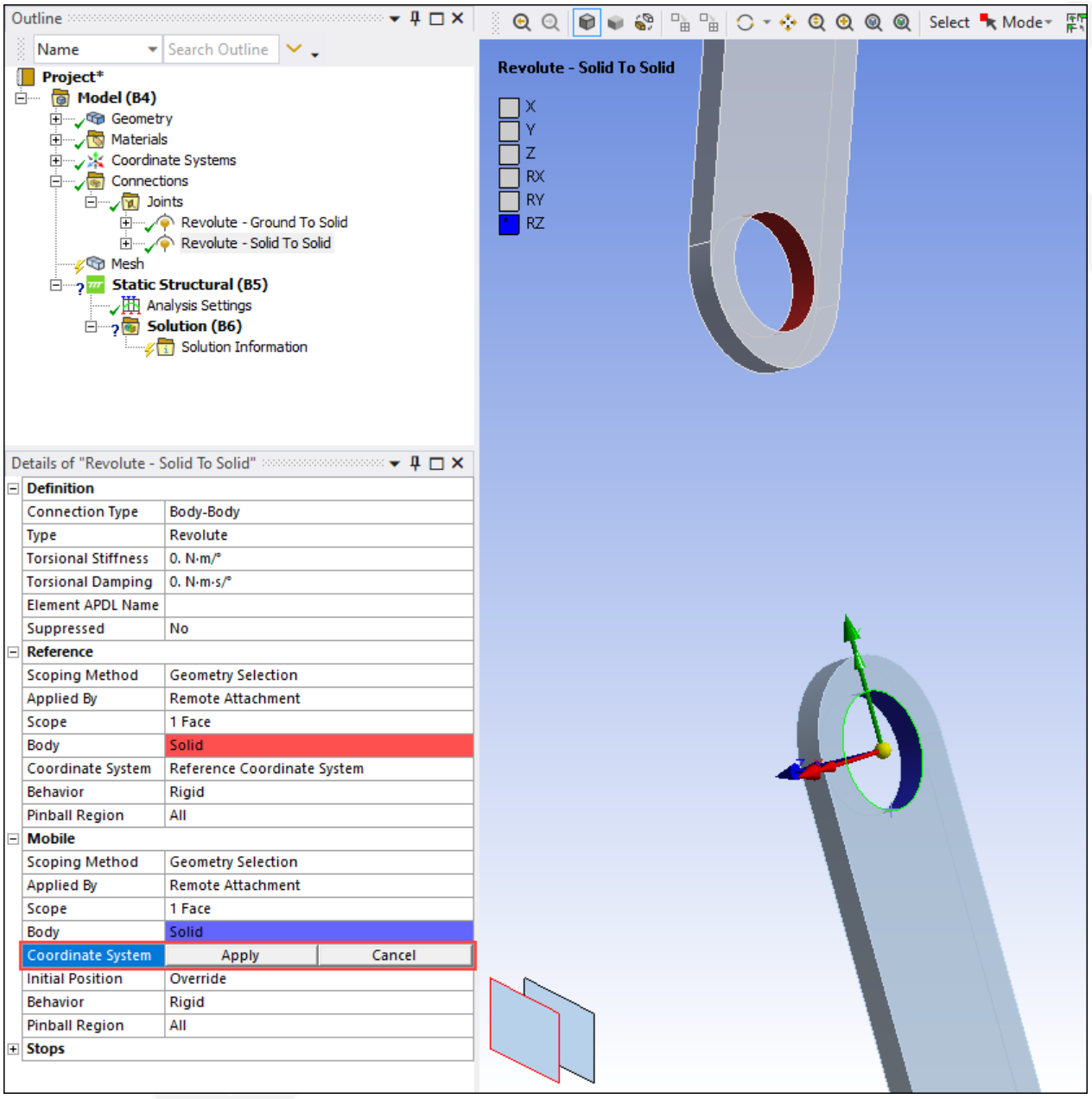


7. Scope the new mobile coordinate system to the back edge of the bottom link's upper hole.

   Select the back edge of the bottom link's upper hole, then under **Mobile** category, select the **Coordinate System** field, and then click the **Apply** button.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

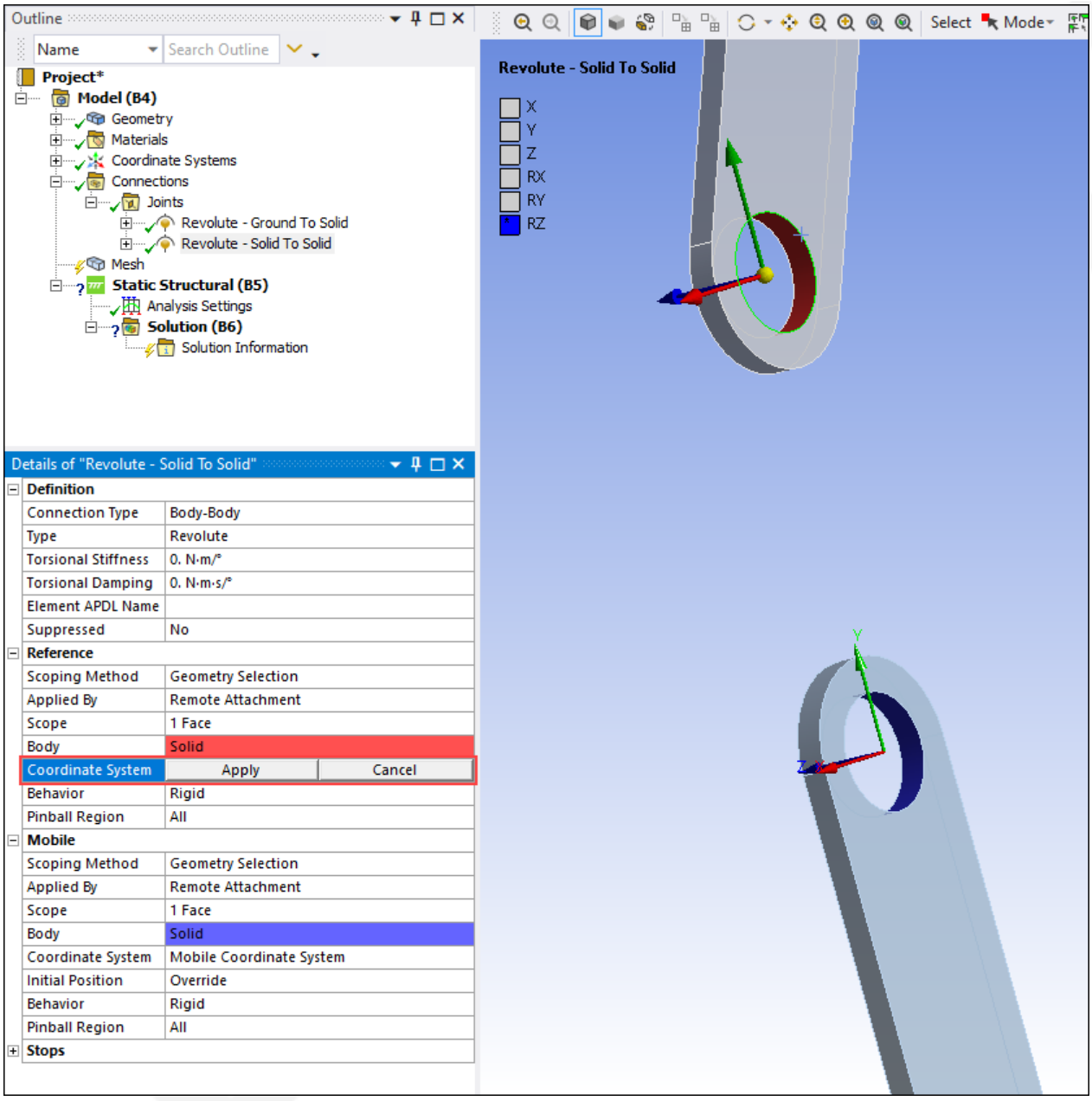


8. Scope the existing Reference Coordinate System to the back edge of the top link's lower hole.

   Select the back edge of the top link's lower hole, and then under **Reference** category, select the **Coordinate System** field and then click the **Apply** button.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

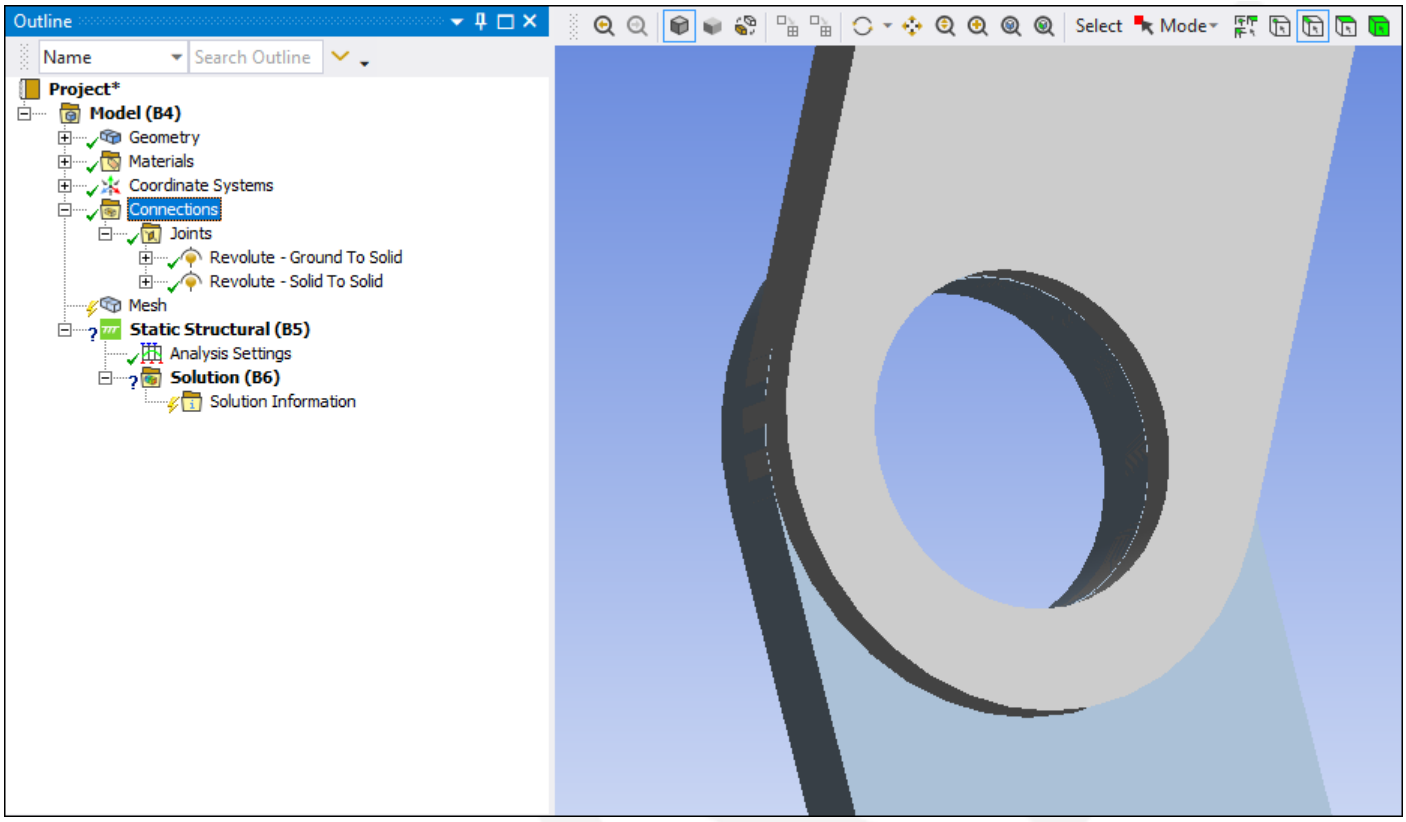
Published: 2023-10-23



The above steps have correctly assigned the coordinate systems so that the holes can be aligned and the revolute joint can operate properly.

To verify, highlight the **Connections** object in the tree and select the **Assemble** option on the Context tab.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

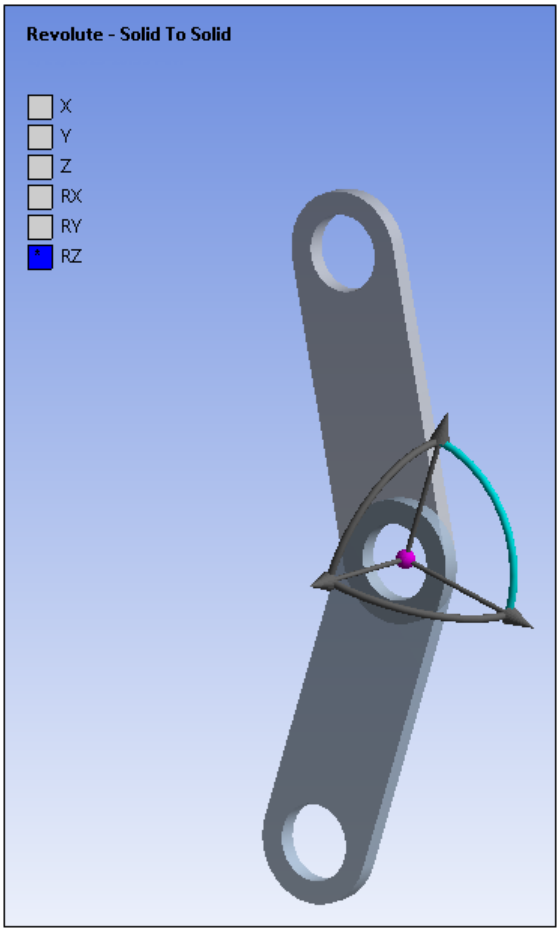


9. Establish the initial position of each joint.

   Highlight the body-to-body joint object in the tree and select the **Configure** option on the Context tab. The joint is graphically displayed according to your configuration. In addition, a triad appears with straight lines representing translational degrees of freedom and curved lines representing rotational degrees of freedom. Among these, any colored lines represent the free degrees of freedom for the joint type. For the joint that is being configured, the translational displacement degrees of freedom always follow the Geometry units rather than the current Mechanical units.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



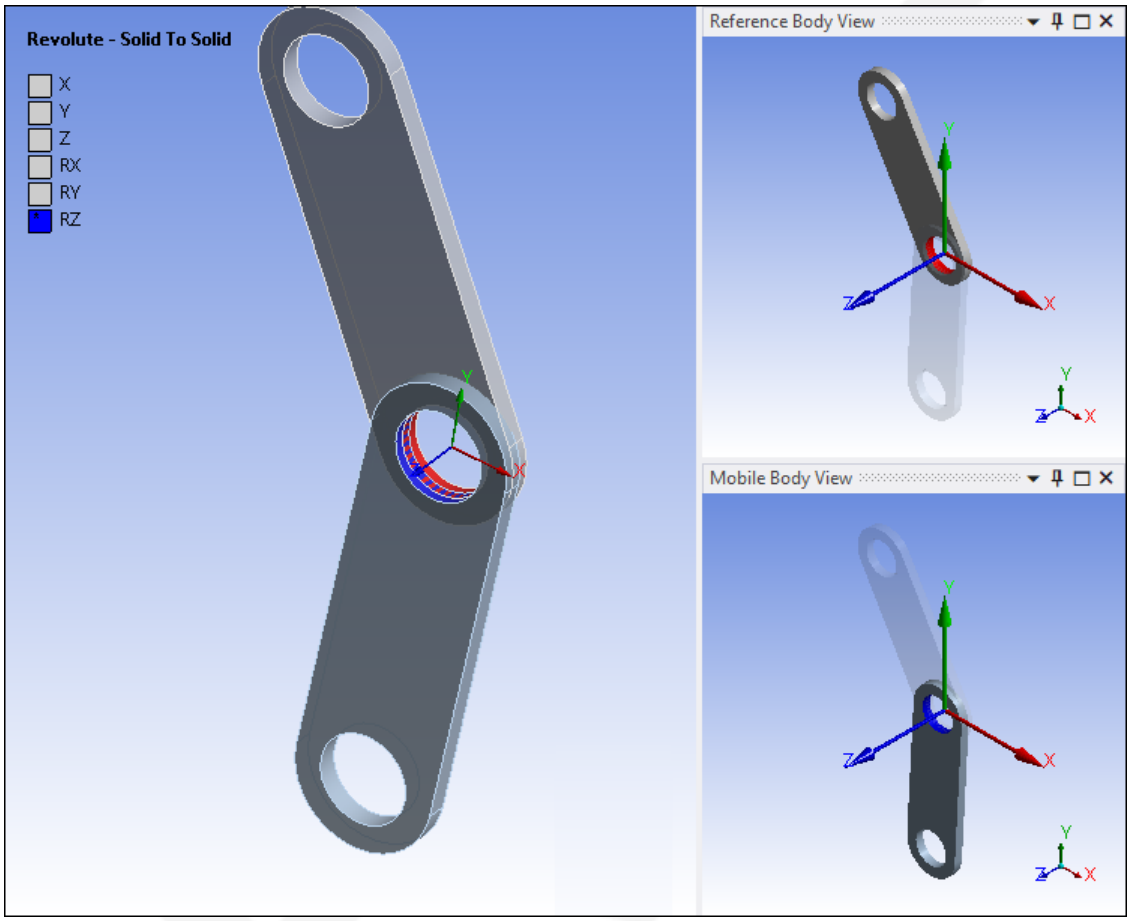
By dragging the mouse cursor on a colored line, the joint will move allowing you to set the initial position of the joint through the free translational or rotational degrees of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For rotations, holding the **[Ctrl]** key while dragging the mouse cursor will advance the rotation in 10 degree increments. You can also type the value of the increment into the Delta field on the Context tab. Selecting the **Configure** option again cancels the joining and positioning of the joint.

10. Create the joints.

    After configuring a joint's initial position, click the **Set** option to create the joint. At this point, you also have the option of returning the configuration to the state it was in before joint creation and upon attaching to Mechanical by selecting the **Revert** option.



Constraints on joints such as stops or locks are fully supported by the Configure tool.

Note:

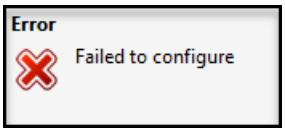
Unexpected results may occur when configuring radial gaps.

Note:

When stops or locks are set on joints and a configure operation leads to the violation of corresponding bounds, an error message, which reads **Fail to Configure** pops up in Mechanical.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



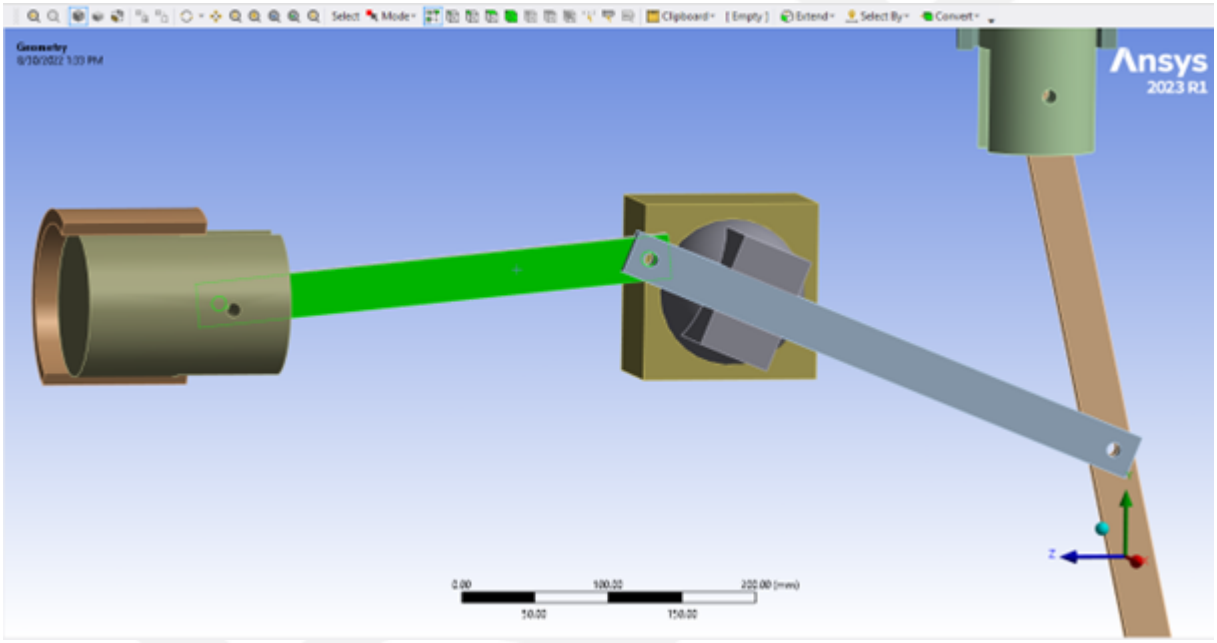
Example: Configuring Bodies

This section describes the steps for configuring a single body within a system of multiple bodies.

In this example, the body to be configured is linked to other bodies in the system by two revolute joints. Hence, applying a transform to this body will result in updating body transforms along the entire chain of bodies.

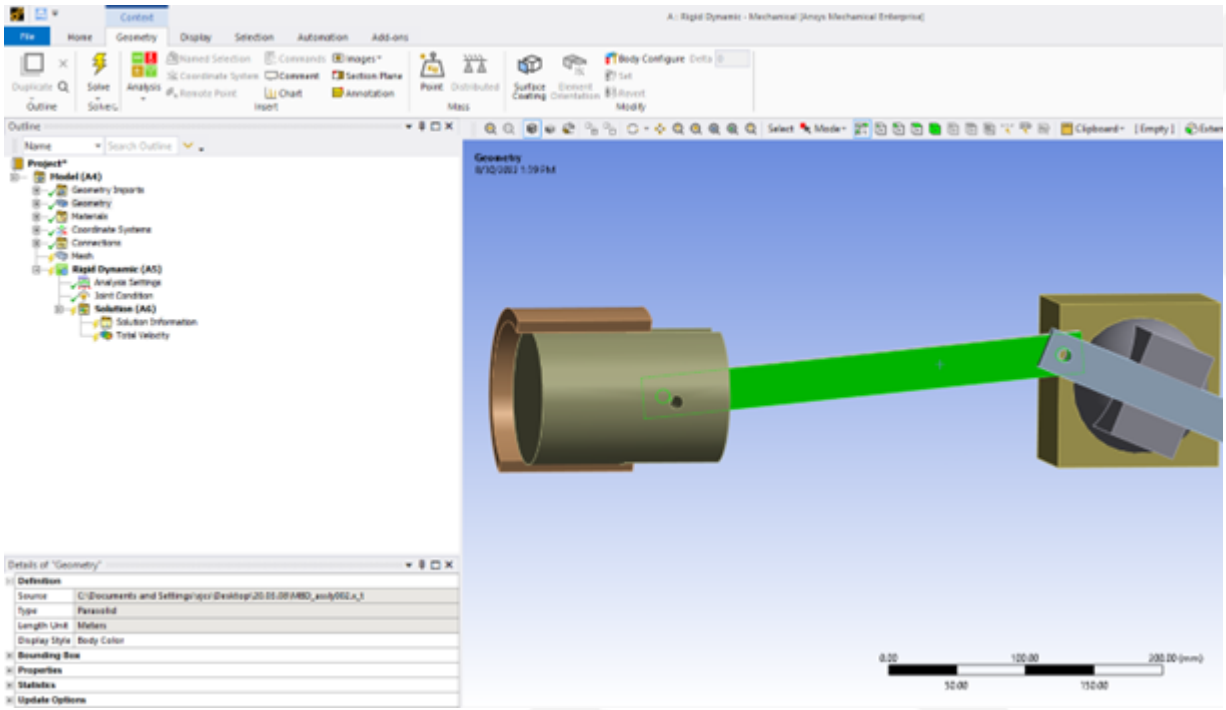
To configure a body, follow these steps:

1. Pick a face, edge, or corner of a body using the **Smart Select** button.



2. In the Mechanical tree, select **Geometry** to make the **Body Configure** button visible.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Click **Body Configure** to make a local coordinate system appear on the selected entity. This coordinate system is acting as a handle on the selected body.

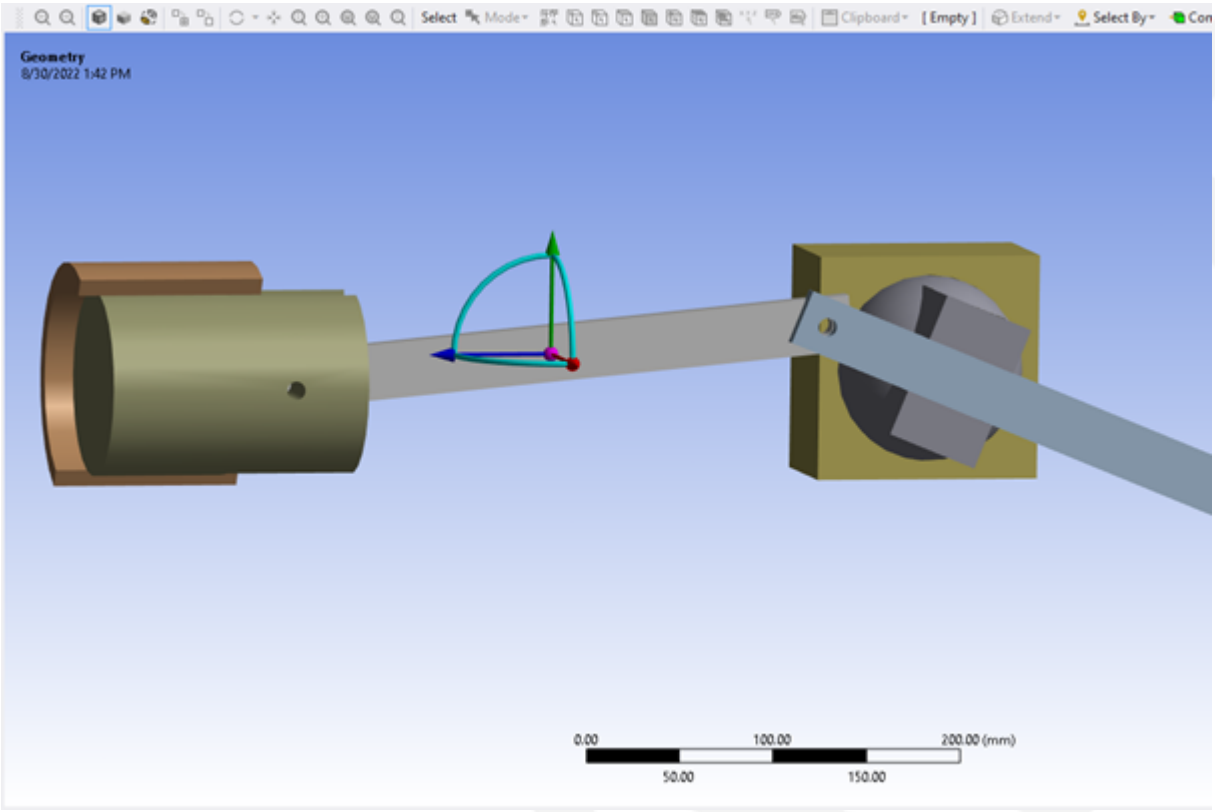
Note:

When selecting a face or edge, the configure coordinate system is oriented with respect to the surface normal. When selecting a body or vertex, the axes of the configure coordinate system are aligned with the global axes, not the inertia axes.

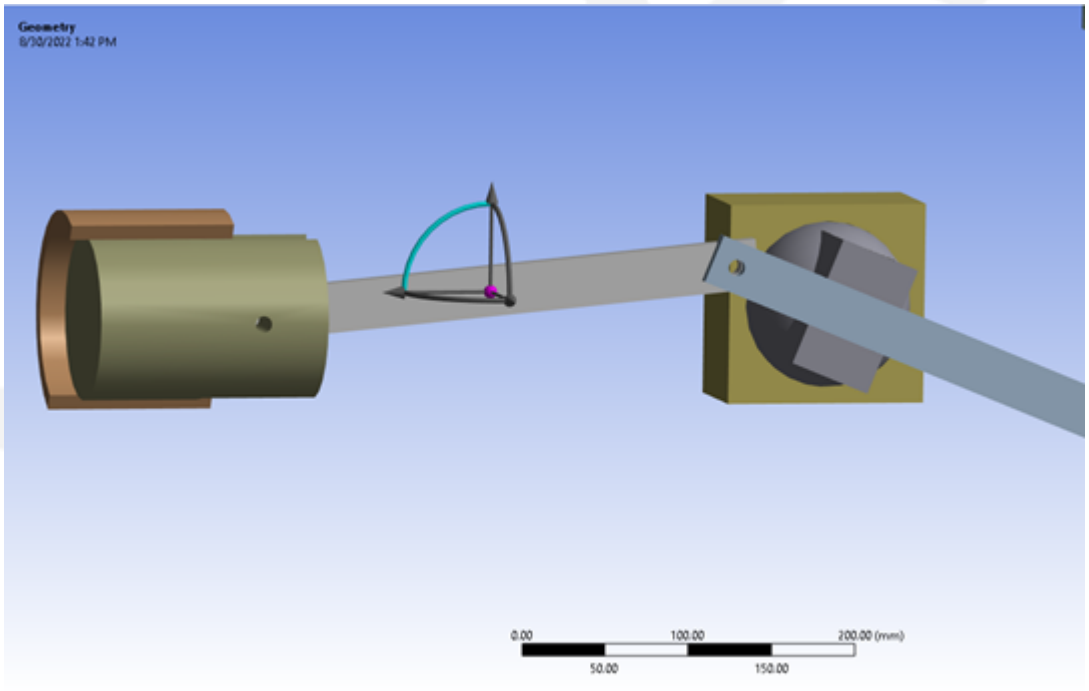
Note:

The origin of the newly created coordinate system is different from the origin of the joint coordinate system, in case one is already attached to the body to configure. The user transform is derived from the origin of the new coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

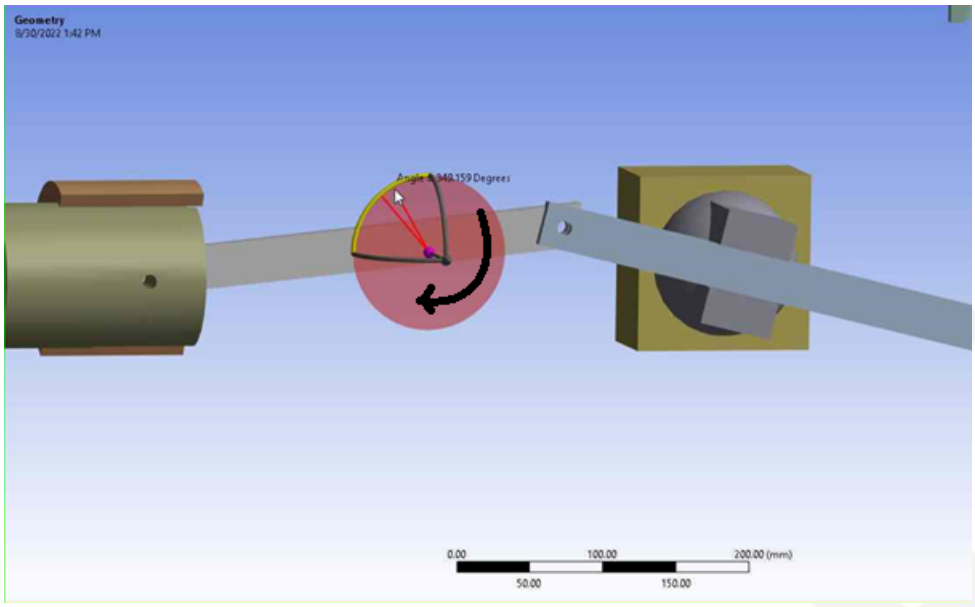


4. From the local coordinate system, pick a degree of freedom in translation or rotation. In the image below, a rotation around the x-axis is selected.



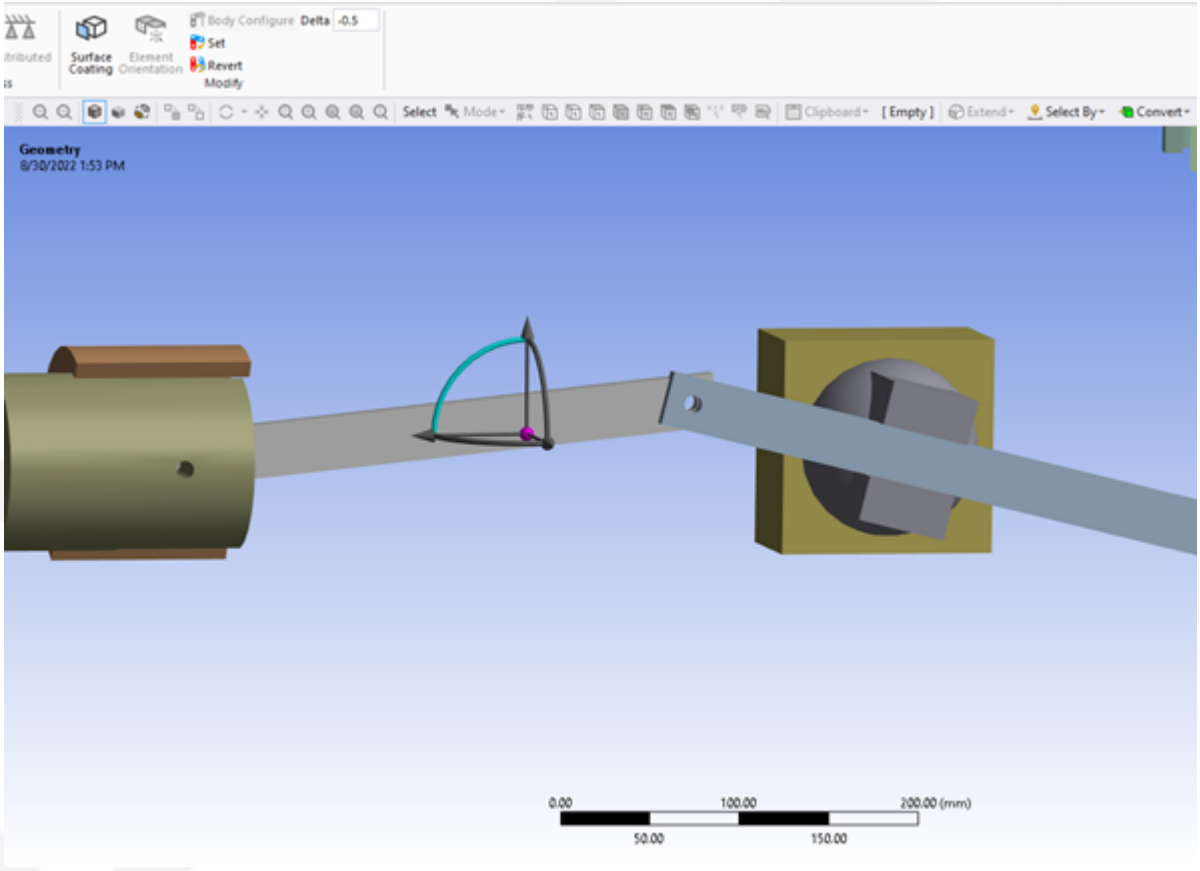
5. Long-press the cursor on the body and drag it along the selected degree of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The selected body must follow the transform you specified and move the entire mechanism accordingly.

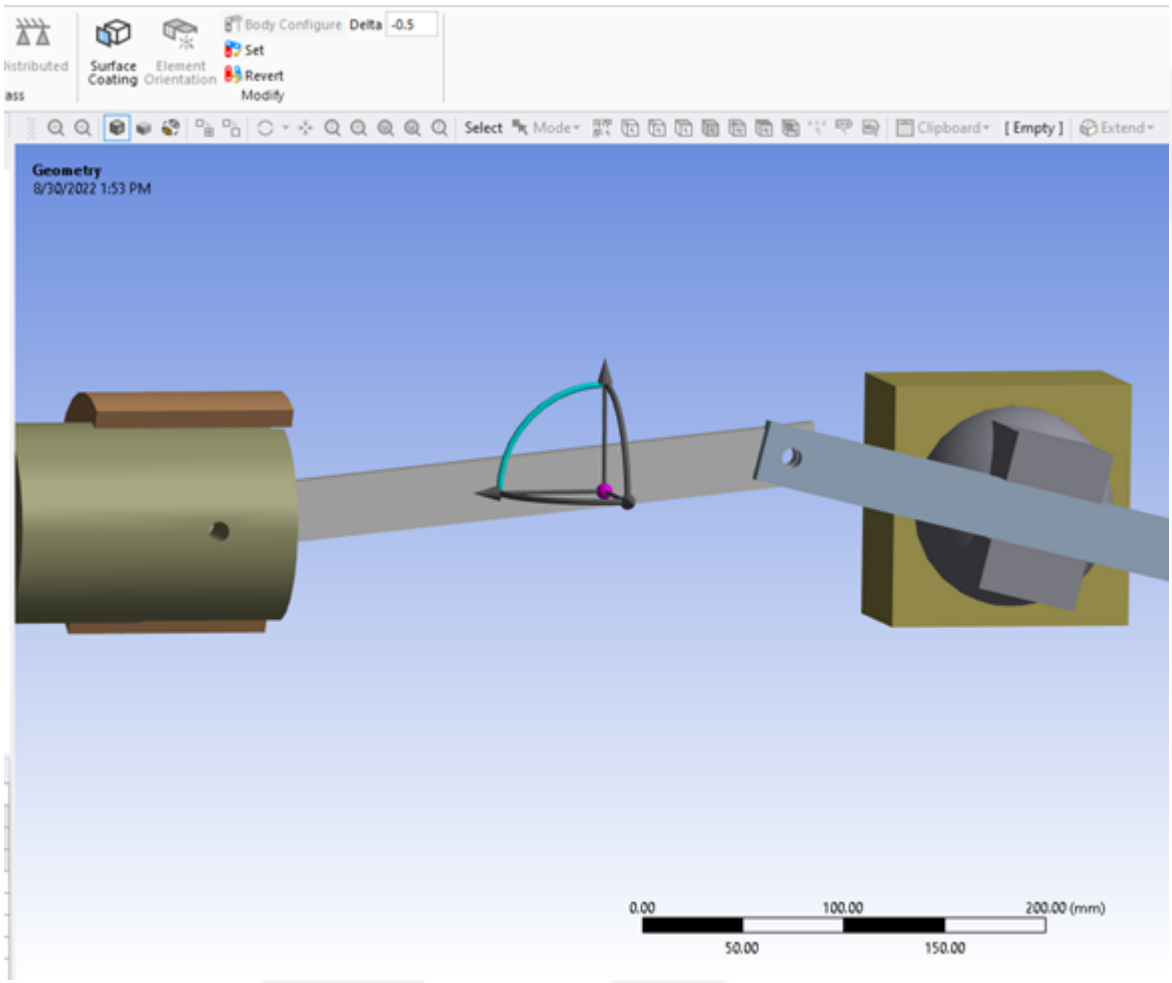
6. For optimal accuracy when applying the transform, enter a value in the **Delta** ribbon. Regarding length, the quantity is expected to be expressed in the unit system currently active in Mechanical. With respect to rotational degrees of freedom, the angle is expected to be expressed in degrees.



7. Once a transform is applied, you can set the model in the resulting configuration or revert changes. Click the **Set** or **Revert** button to perform either action.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Springs

A spring is an elastic element that is used to store mechanical energy and which retains its original shape after a force is removed. Springs are typically defined in a stress free or "unloaded" state. This means that no longitudinal loading conditions exist unless preloading is specified (see below). In Mechanical, the **Configure** (p. 1380) feature is used to modify a Joint. If you configure a joint that has an attached spring, the spring must be redrawn in the Geometry window. In effect, the spring before the Configure action is *replaced* by a new spring in a new unloaded state.

Springs are defined as *longitudinal* and they connect two bodies together or connect a body to ground. Longitudinal springs generate a force that depends on linear displacement. Longitudinal springs can be used as a damping force, which is a function of velocity or angular velocity, respectively. Springs can also be defined directly on a Revolute Joint (p. 1299) or a Cylindrical Joint (p. 1299).

Note:

A spring cannot be applied to a vertex that is scoped to an end release (p. 1409).

The following topics are discussed in this section:

- Applying Springs (p. 1396)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Remote Connection Display (p. 1396)
- Spring Behavior (p. 1397)
- Nonlinear Spring Stiffness (p. 1399)
- Preloading (p. 1400)
- Scoping (p. 1400)
- Spring Length (p. 1400)
- Advanced Features (p. 1401)
- Output (p. 1401)
- Example: Longitudinal Spring with Damping (p. 1402)
- Spring Incompatibility (p. 1404)

Applying Springs

To apply a spring:

1. After importing the model, highlight the **Model** object in the tree and choose the **Connections** option from the Context tab.
2. Highlight the new **Connections** object, open the **Spring** drop-down menu and select either **Body-Ground** or **Body-Body** from the Context tab.

   Note:

   You can pre-select a vertex or node (**Body-Ground**) or two vertices or nodes (**Body-Body**) and then insert a Spring to automatically create a directly attached spring. See the **Scoping** (p. 1400) subsection below.

3. Highlight the new **Spring** object and enter information in the Details view. Note that **Longitudinal Damping** is applicable only to transient analyses.

   Note:

   - The length of the spring connection must be greater than 0.0 with a tolerance of 1e-8 mm.
   - The **Body Views** display is the default display.

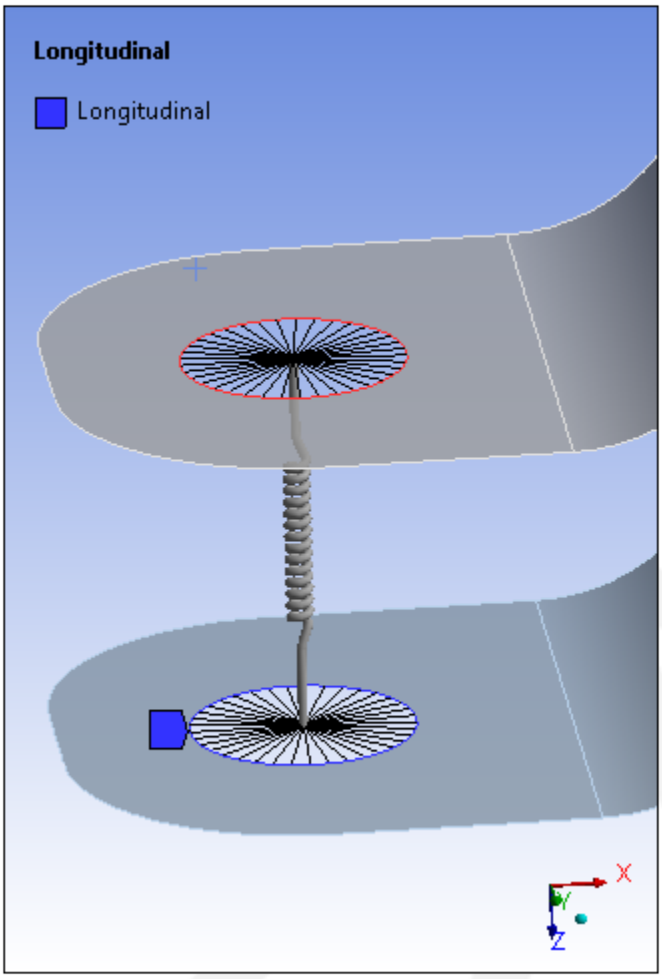
Remote Connection Display

When defined as a **Remote Attachment**, springs are considered to be a remote boundary condition (p. 1735), and can make use of remote points that were either specifically defined or created internally

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

by the application. To do a visual check, you can display the connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab). An example is illustrated here of a spring scoped to two edges.



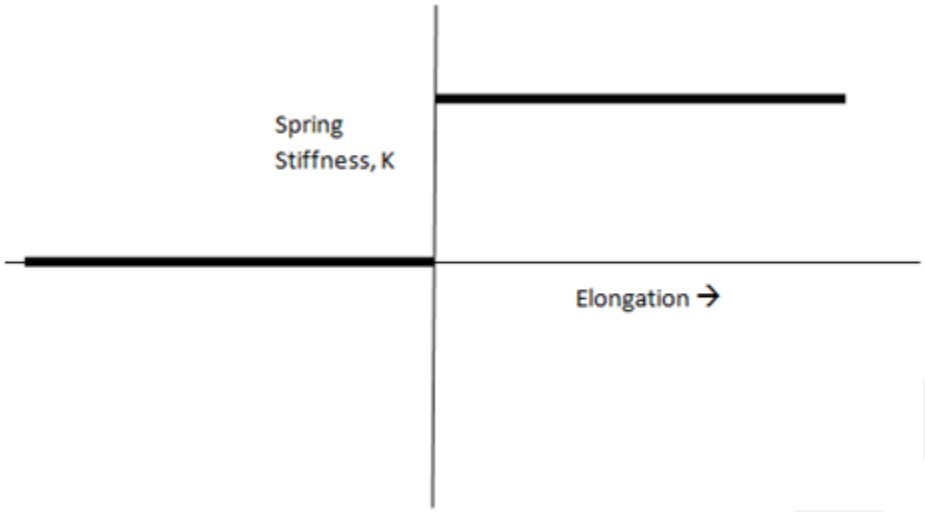
Spring Behavior

The **Spring Behavior** property is modifiable for a Rigid Dynamics (p. 550) and Explicit Dynamics analyses only. For all other analysis types, this field is read-only and displays as **Both**.

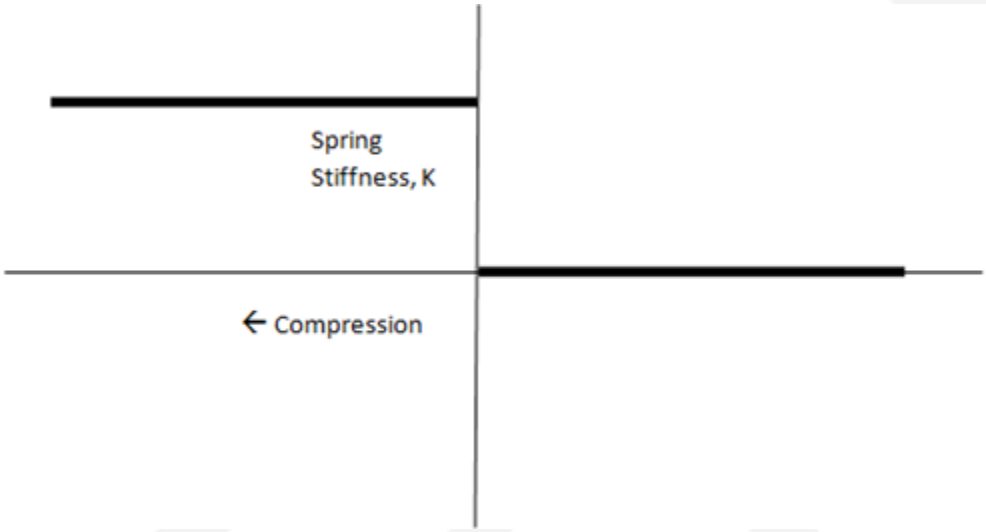
You can define a longitudinal spring to support only tension loads or only compression loads using the **Spring Behavior** property. You can set this property to **Both**, **Compression Only** or **Tension Only**. The tension only spring does not provide any restoring force against compression loads. The compression only spring does not provide resistance against tensile loads. The stiffness of a compression only or tension only spring without any preloads is shown below.

Stiffness Behavior of a Tension Only Spring:

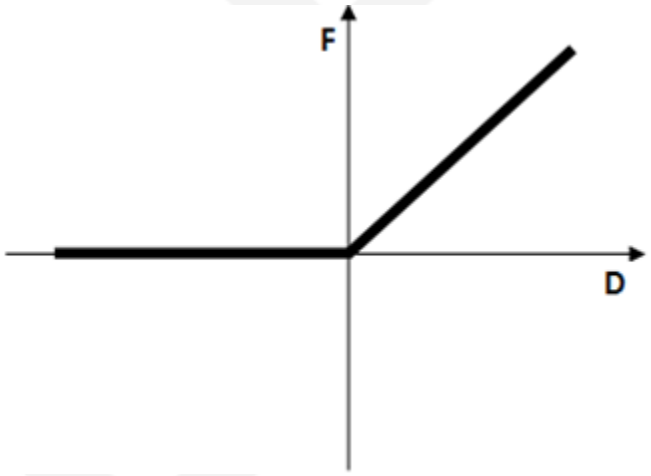
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Stiffness Behavior of a Compression Only Spring:



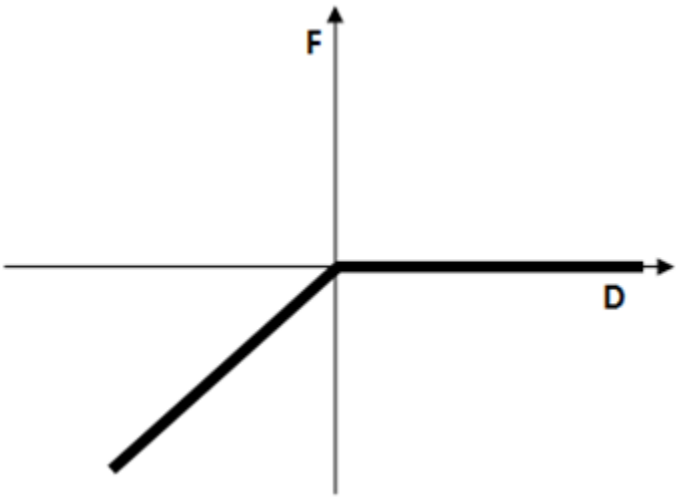
Force Deflection Curve for a Tension Only Spring:



Force Deflection Curve for a Compression Only Spring:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

For the **Compression Only** and **Tension Only** options, the application uses the element LINK180. As a result, the force deflection curve will be nonlinear.

Nonlinear Spring Stiffness

A nonlinear force-deflection curve can be used to simulate multi-rate springs with nonlinear spring stiffness. A linear piecewise curve is used for this purpose.

Note that spring deflection is computed using the distance between the two ends of the spring, minus the initial length. The distance between the two points is never negative, but the deflection can be negative. If you determine that a spring exists with an incorrectly defined nonlinear stiffness, the force-deflection curve may be incorrectly defined as a result of the tabular input for nonlinear stiffness for one or more spring objects. See the details in COMBIN39 element description for more information.

Note:

Support Requirements

- **Tabular Data** requires at least two rows of data.
- The properties **Longitudinal Damping** and **Preload** are not applicable for Springs with nonlinear stiffness.
- For the Mechanical APDL solver, the number of data points used to define the force-deflection curve cannot exceed **20**.

Points to consider for Rigid Dynamics or Explicit Dynamics analyses *only*:

- If a nonlinear stiffness curve is defined with the **Tension Only** option, all points with a negative displacement are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If a nonlinear stiffness curve is defined with the **Compression Only** option, all points with a positive displacement are ignored.

To define a nonlinear force-deflection curve:

1. In the **Spring** object Details view settings, click in the **Longitudinal Stiffness** property.
2. Click the arrow in the Longitudinal Stiffness property then select **Tabular**.
3. Enter displacement and force values in the **Tabular Data** window. A graph showing force vs. displacement is displayed.

Preloading

(Not supported for Explicit Dynamics analyses.)

Mechanical also provides you with the option to preload a spring and create an initial "loaded" state. The **Preload** property in the Details view allows you to define a preload as a length using **Free Length** or to specify a specific **Load**. The actual length is calculated using spring end points from the **Reference** and **Mobile** scoping. For rigid dynamics analyses, the spring will be under tension or compression depending upon whether you specified the free length as smaller or greater than the spring length, respectively. If preload is specified in terms of **Load**, a positive value creates tension and a negative value creates compression. When the spring is linear (defined by a constant stiffness) the Rigid Dynamics solver deduces the spring free length by subtracting the value L=F/K (where F is the preload and K is the stiffness) from the actual spring length. Note that this offset is also applied to the elongation results. When the spring is non-linear (defined by a force/displacement table), this offset is not taken into account.

Spring Length

The read-only property **Spring Length** displays the actual length of the spring which is calculated using the end points from the **Reference** and **Mobile** scoping.

Scoping

You select the **Scope** of springs as body-to-body or body-to-ground using the property of the **Scope** category and you define a spring's end points using the properties of the **Reference** and **Mobile** categories. For body-to-ground property specification, the **Reference** is assumed to be grounded (fixed); scoping is only available on the **Mobile** side. Since this is a unidirectional spring, these two locations determine the spring's line of action and as such the spring's reference and mobile locations cannot be the same as this would result in a spring with zero length.

In addition, the **Reference** and **Mobile** categories provide the scoping property **Applied By**. This property enables you to specify the connection as either a **Direct Attachment** or a **Remote Attachment**. The **Remote Attachment** option (default) uses a Remote Point (p. 1089) as a scoping mechanism. The **Direct Attachment** option enables you to scope directly to a single vertex or a node of the model.

Note:

If specified as a **Remote Attachment**, springs are classified as *remote* boundary conditions. Refer to the Remote Boundary Conditions (p. 1735) section for a listing of all remote boundary conditions and their characteristics.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

You can scope of a spring to a:

- Single face or to multiple faces (applied as a **Remote Attachment** only).
- Single edge or multiple edges (applied as a **Remote Attachment** only).
- Single vertex (can be applied as either a **Remote Attachment** or as a **Direct Attachment**) or multiple vertices (applied as a **Remote Attachment** only).

Note:

A spring cannot be applied to a vertex that is scoped to an end release (p. 1409).

- Single node (applied as a **Direct Attachment** only).

See the Spring Object Reference page of the Help for additional information about the available categories and properties.

Advanced Features

If specified as a **Remote Attachment**, the **Reference** and **Mobile** groups for Springs each include the following advanced properties:

- **Behavior**: This property enables you to specify the scoped geometry as either **Rigid**, **Deformable**, or **Beam**. Refer to the Geometry Behaviors (p. 1095) section for more information.
- **Pinball Region**: The Pinball Region is a radius value (length unit) that defines a region for selecting elements to be used by the solver for the Spring's Reference (Body-Body only) and Mobile scoping.
- **Material**: Select your material from the fly-out menu. Your material must include a constant damping coefficient to account for viscous damping or structural damping of the Spring in the analysis. The default setting is **None**.

Note:

- The **Behavior** setting is applicable to underlying bodies that are flexible.
- If the **Scope Method** property of the Spring is set to **Remote Point**, the Spring will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties.
- When you specify a Material for the spring that includes a constant damping coefficient, based on the analysis type, the application applies damping as structural damping for damped Modal and Full Harmonic Response systems and as viscous damping for MSUP systems.

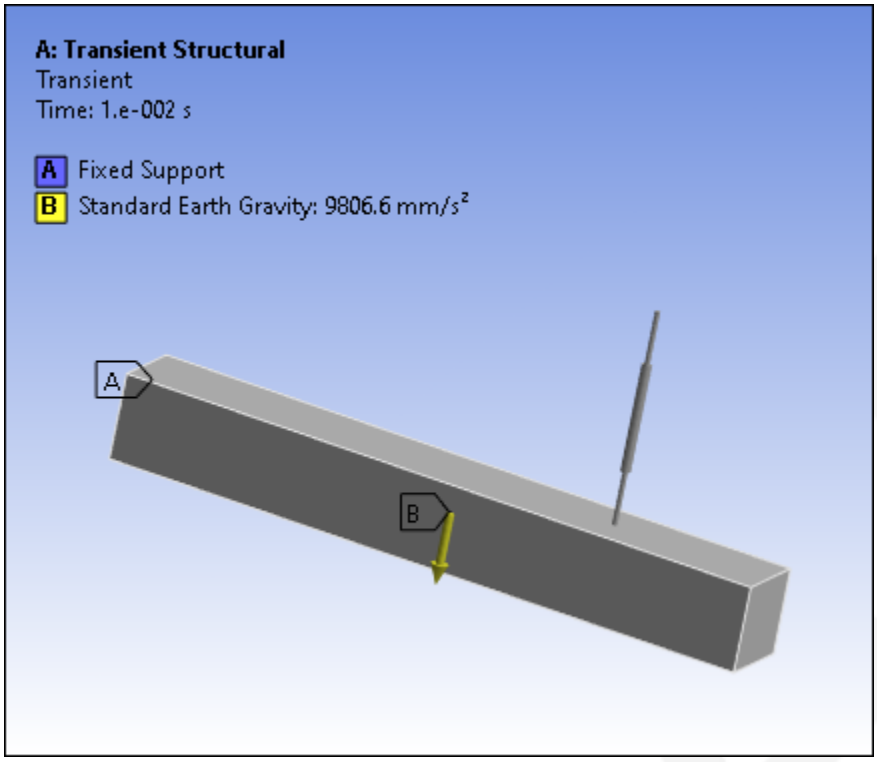
Output

Several outputs are available via a spring probe (p. 2092).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Example: Longitudinal Spring with Damping

This example shows the effect of a longitudinal spring connecting a rectangular bar to ground to represent a damper. A Transient Structural analysis was performed in the environment shown:



The following are the Details pane settings of the **Spring** object:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

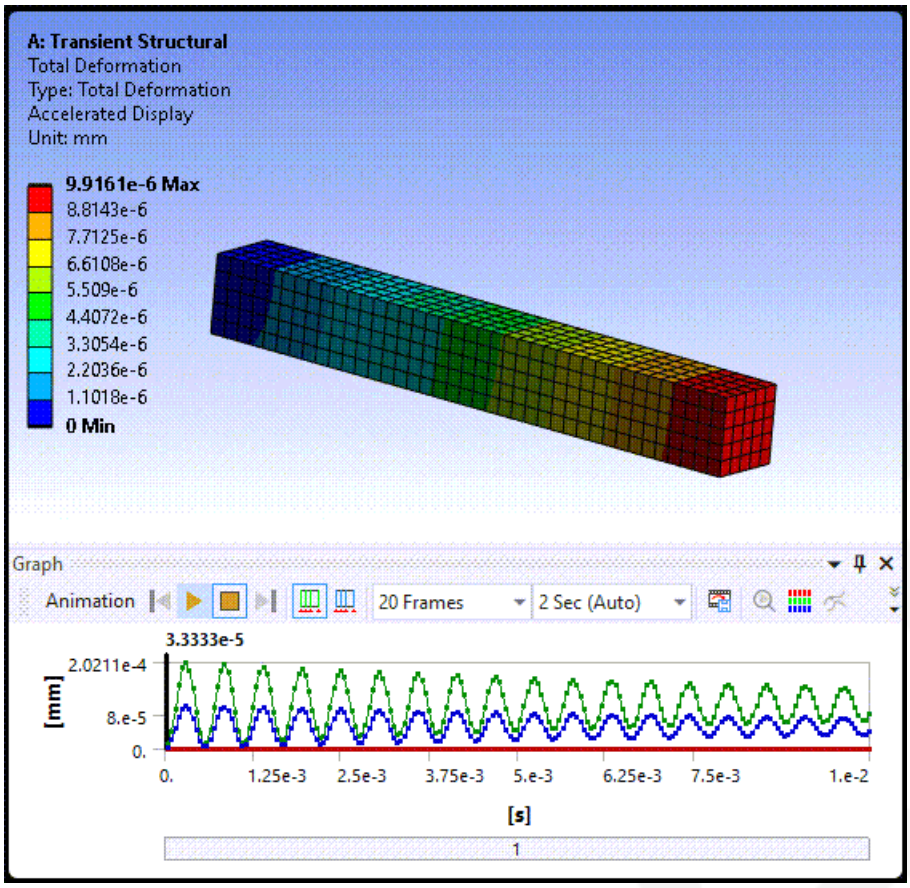
Details of "Longitudinal - Ground To Part 1"

Graphics Properties	
Definition	
Type	Longitudinal
Spring Behavior	Both
Longitudinal Stiffness	1.5 N/mm
Longitudinal Damping	5.e-002 N·s/mm
Preload	None
Suppressed	No
Spring Length	50. mm
Element APDL Name	
Scope	
Scope	Body-Ground
Reference	
Coordinate System	Global Coordinate System
Reference X Coordinate	-70. mm
Reference Y Coordinate	10. mm
Reference Z Coordinate	38.75 mm
Reference Location	Click to Change
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	Part 1
Coordinate System	Global Coordinate System
Mobile X Coordinate	-20. mm
Mobile Y Coordinate	10. mm
Mobile Z Coordinate	38.75 mm
Mobile Location	Click to Change
Behavior	Rigid
Pinball Region	All

Presented below is the **Total Deformation** result:

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Spring Incompatibility

(applicable only to rigid dynamics analyses)

If the preload for a longitudinal spring is a tensile load, then the spring cannot be defined as compression only. Alternatively, if the preload is a compressive load, then the spring cannot be defined as tension only. Should this case occur, the spring will be marked as underdefined and if you attempt to solve such a case, the following error message is displayed: "The preload for a spring is incompatible with its behavior being tension only spring or compression only spring."

Beam Connections

Beam connections use structural beam elements that can carry a bending (flexure) load. The Beam option enables you to specify body-to-body or a body-to-ground connection. This feature supports all structural analyses.

Important:

When you create a beam connection, the application automatically creates internal Remote Points at each end of the beam. The application uses these Remote Points to connect the beam to the scoped body/bodies. If you change the position of a body after creating the beam connections, the positions of the internally created Remote Points do not change.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

They remain in their original position. As a result, the beam's orientation and the new position of the bodies do not align.

Review the Requirements and Default Settings (p. 1407) listed following the Application topic and note the associated Help sections in the References (p. 1407) topic.

Application

To add a **Beam** object:

1. Select the **Connections** folder in the object tree. As needed, add a **Connections** folder by selecting the **Model** object and clicking the **Connections** option on the Model Context Tab (p. 48).
2. On the Connections Context Tab (p. 50), open the **Beam** drop-down menu and select either **Body-Ground** or **Body-Body** to add a circular beam under Connections.
3. In the **Details** View, under **Definition**, click the **Material** fly-out menu, and then select a material for the beam.
4. Enter a beam radius in the **Radius** field.
5. If necessary, modify the **Scope** setting.

   The **Scope** property of the **Scope** category enables you to change the scoping from **Body-Body** to **Body-Ground**. Similar to Springs (p. 1400), this property defines the beam's end points in coordination with the properties of the **Reference** and **Mobile** categories. For body-to-ground property specification, the **Reference** is assumed to be grounded (fixed) and as a result scoping is required on the **Mobile** side only. Because beams define a span, the reference and mobile locations determine a distance and as such the reference and mobile locations cannot be the same.

   In addition, the **Reference** and **Mobile** categories provide the scoping property **Applied By**. This property enables you to specify the connection as either a **Direct Attachment** or a **Remote Attachment**. The **Remote Attachment** option (default) uses a **Remote Point** as a scoping mechanism. The **Direct Attachment** option enables you to scope directly to a single vertex or a node of the model. Direct Attachment is not allowed if scoped to solid bodies, as they do not have rotational degrees of freedom.
6. Under the **Reference** category, for **Body-Body** connections only:

   Specify the **Scoping Method** property as either **Geometry Selection**, **Named Selection**, or **Remote Point**. Based on the selection made in this step, select a:

   - geometry (faces, edges, or vertices) and click **Apply** in the **Scope** property field.

     or...

   - single node (*Direct Attachment Only*) and click **Apply** in the **Scope** property. In order to select an individual node, you need to first generate a mesh on the model, and then select the Node filter on the Graphics Toolbar (p. 97).

     or...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- user-defined node-based named selection (*Direct Attachment Only*) or a user-defined geometry-based named selection (*Remote Attachment Only*) from the drop-down list of the **Named Selection** property.

  or...

- user-defined remote point (*Remote Attachment Only*) from the drop-down list of the **Remote Point** property.

Note:

You can pre-select a vertex or node (**Body-Ground**) or two vertices or nodes (**Body-Body**) and then insert a Beam to automatically create a directly attached beam.

---

7. Specify the following properties as needed. These properties are available under the **Reference** Category (**Body-Body** *or* **Body-Ground** connections) when the **Applied By** property is set to **Remote Attachment**:

   - **Coordinate System**: select a different coordinate system if desired.
   - **Reference X Coordinate**: enter a value as needed.
   - **Reference Y Coordinate**: enter a value as needed.

8. Under **Mobile** Category (**Body-Body** or **Body-Ground** connections):

   a. - geometry (faces, edges, or vertices) and click **Apply** in the **Scope** property field.

      or...

      - single node (*Direct Attachment Only*) and click **Apply** in the **Scope** property. In order to select an individual node, you need to first generate a mesh on the model, and then select the **Node** filter on the Graphics Toolbar (p. 97).

      or...

      - user-defined node-based named selection (*Direct Attachment Only*) or a user-defined geometry-based named selection (*Remote Attachment Only*) from the drop-down list of the **Named Selection** property.

      or...

      - user-defined remote point (*Remote Attachment Only*) from the drop-down list of the **Remote Point** property.

   b. Specify the following properties as needed. These properties are available under the **Mobile** Category (**Body-Body** *or* **Body-Ground** connections) when the **Applied By** property is set to **Remote Attachment**:

      - **Coordinate System**: select a different coordinate system if desired.
      - **Mobile X Coordinate**: enter a location value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Mobile Y Coordinate**: enter a location value.
- **Behavior**: specify this property as either **Rigid**, **Deformable**, or **Beam**. Refer to the Geometry Behaviors and Support Specifications (p. 1095) section for more information.
- **Pinball Radius**: enter a dimension value.

Note:

When defined as a **Remote Attachment**, a beam connection is considered a remote boundary condition (p. 1735), and it can make use of remote points that were either specifically defined or created internally by the application. To do a visual check, you can display the connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Requirements and Default Settings

Note that following when working with beams:

- For **Body-Ground** beam connections, the reference side is fixed. For **Body-Body** beam connections, you must define the reference point for each body.
- The length of the beam connection must be greater than 0.0 with a tolerance of 1e-8 mm.
- The **Body Views** display is the default display.
- Beam connections support structural analyses only. In thermal stress analyses, beam connections are assigned the environment temperature in the structural analysis. You can include a beam in a thermal analysis by creating a line body and as a result providing for temperature transference.

References

- See the Beam Object Reference page of the Help for additional information about the available categories and properties.
- The Beam Probe (p. 2125) results provide you the forces and moments in the beam from your analysis.

Spot Welds

You use the spot weld feature to connect individual solid and surface body parts together to form solid or surface body model assemblies. Structural loads are transferred from one specific body part to another via spot weld connection points. This enables you to simulate of solid and surface body model assemblies.

Weld Group (Beta Feature)

Using a current beta object, Weld Group, you can manage a group of spot welds that connect multiple surface body parts. This object provides properties to either import predefined spot welds from a text file or to manually create individual spot welds.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Spot Weld Details

Spot welds are usually defined in the CAD system and automatically generated upon import, although you can define them manually in Mechanical after the model is imported. Spot welds become hard points in the geometric model. Hard points are vertices in the geometry that are linked together using Beam Elements during the meshing process.

Spot weld objects are located in a **Connection Group** (p. 1212) folder. When selected in the tree, they appear in the **Geometry** window highlighted by a black square around a white dot on the underlying vertices and include an annotation.

If a model contains spot weld features in the CAD system and the **Auto Detect Contact On Attach** is turned on in the Workbench option, **Tools** > **Options**>Mechanical, then **Spot Weld** objects are generated when the model is read into the Mechanical. Spot weld objects will also get generated during geometry refresh if the **Generate Automatic Connection On Refresh** is set to **Yes** in the Details view of the **Connections** folder. This is similar to the way in which Mechanical automatically constructs contacts when reading in assembly models and refreshing the geometry.

You can manually generate spot welds as you would insert any new object into the **Outline**. Either insert a spot weld object from the context menu and then pick two appropriate vertices in the model, or pick two appropriate vertices and then insert the spot weld object.

You can define spot welds for CAD models that do not have a spot weld feature in the CAD system, as long as the model contains vertices at the desired locations. You must define spot welds manually in these cases.

Spot Weld Application

Spot welds do not act to prevent penetration of the connected bodies in areas other than at the spot weld location. Penetration of the joined bodies is possible in areas where spot welds do not exist.

Spot welds transfer structural loads and thermal loads as well as structural effects between solid, surface, and line body parts. Therefore they are appropriate for displacement, stress, elastic strain, thermal, and frequency solutions.

The DesignModeler and SpaceClaim applications can be used to generate spot welds. The only CAD system whose spot welds can be fully realized in Ansys Workbench at this time is NX. The APIs of the remaining CAD systems either do not handle spot welds, or Ansys Workbench does not read spot welds from these other CAD systems.

Note:

Because a **Spot Weld** is a remote boundary condition, it can make use of remote points that were either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

End Releases

For line body models, the End Release feature enables you to free the degrees of freedom (translation and rotation) at a vertex that is shared by two or more edges. You can free the constraint of multiple edges at the vertex's location, however, you must always keep at least one edge from being released.

Review the following:

Application (p. 1409)
Requirements and Limitations (p. 1410)
Examples (p. 1411)

Application

To add an End Release:

1. Add a **Connections** folder if one is not already in the tree, by highlighting the **Model** object and selecting **Connections** from the Model Context Tab (p. 48) or by choosing **Insert >Connections** from the context menu (right-click).
2. Add an **End Release** object by highlighting the **Connections** folder and selecting **End Release** from the Connections Context Tab (p. 50) or by selecting **Insert** > **End Release** from the context (right-click) menu.
3. Specify the **Scoping Method** as **Geometry Selection** (default) or **Named Selection**.
4. Specify the **Vertex Geometry** and the **Edge Geometry**, respectively. These properties support vertex/edge or node/element scoping, either through direct selection or using a properly defined Named Selection.

   Note:

   - Geometric and mesh entities cannot be mixed.
   - The vertex or node must be one of the end points of the selected edges/elements.
   - Element scoping must be done on beam-based line bodies.

5. If you specified more than one edge for the **Edge Geometry** property, use the **Independent Edges** property to specify whether the edges are independent of one another (**Yes**) or fixed together (**No**).
6. Specify the **Coordinate System** as the **Global Coordinate System**, a local user-defined coordinate system, or an **Beam Coordinate System**.

   Note:

   When you select the **Beam Coordinate System** option of the **Coordinate System** property, Mechanical automatically creates an internal coordinate system (not visible in the tree) that orients the **End Release** such that the x-axis is parallel to the edge(s) scoped

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

in the **Edge Geometry** property. This ease-of-use option enables quick definition of the **End Release** when the specified **Edge Geometry** is not aligned with the **Global Coordinate System**.

7. Specify the translational and/or rotational degrees of freedoms in X, Y and Z directions by changing axial properties from **Fixed** to **Free**.
8. Based upon the configuration of your model, specify the connection **Behavior** property as either **Coupled** (default) or **Joint**. This property uses coupling or a general joint, respectively.

Important:

When the application generates the mesh for an analysis that includes a beam End Release, it creates additional nodes at the scoping of the End Release. If you suppress the beam End Release, the mesh still includes these suppressed nodes, and therefore the nodes are not available for post processing. Regenerate the mesh associated with the beam End Release to completely remove the effects of the suppression.

Requirements and Limitations

- The end release feature is only applicable in structural analyses that use the Mechanical APDL solver. If you select a different solver and **End Release** objects are present, the environment folder becomes underdefined.
- An **End Release** object requires that the specified vertex must be contained on a line body and that the vertex is connected to more than one edge.
- A vertex cannot be scoped to more than one **End Release** object.
- Model-to-Model (p. 991) linking does not support upstream **End Release** objects.
- You cannot apply the following boundary conditions to a vertex or an edge that is scoped to an end release. If so, the object becomes underdefined and an error message is generated.
  - **Fixed Support**
  - **Displacement**
  - **Simply Supported**
  - **Fixed Rotation**
  - **Velocity**
- You cannot apply the following remote boundary conditions to a vertex that is scoped to an end release. If so, the object becomes underdefined and an error message is generated.
  - **Remote Displacement**
  - **Remote Force**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Moment**
- **Point Mass**
- **Thermal Point Mass**
- **Spring**
- **Joint**

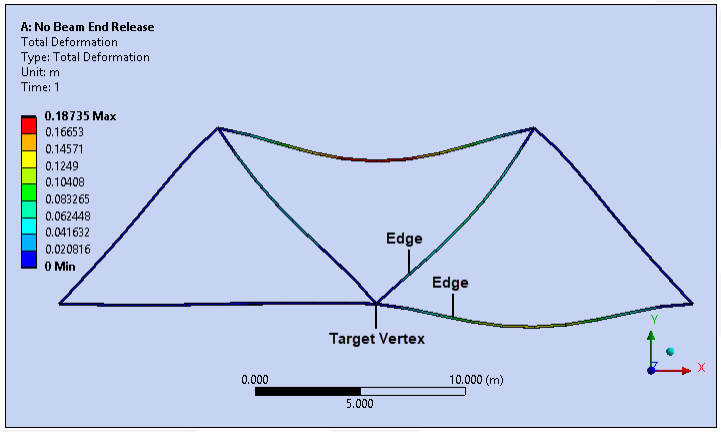
Note:

As stated, an End Release cannot share scoping with the above remote boundary conditions. However, an End Release can be scoped to a geometric entity or element that is also scoped to a remote boundary condition, such as a Joint. For example, you can scope an End Release to the vertex of one end of a line body and a Joint to the vertex of the other end. Given this situation, Ansys recommends that you use remote scoping (**Applied By** property set to **Remote Attachment**) for the shared remote boundary conditions.

Scoping Examples

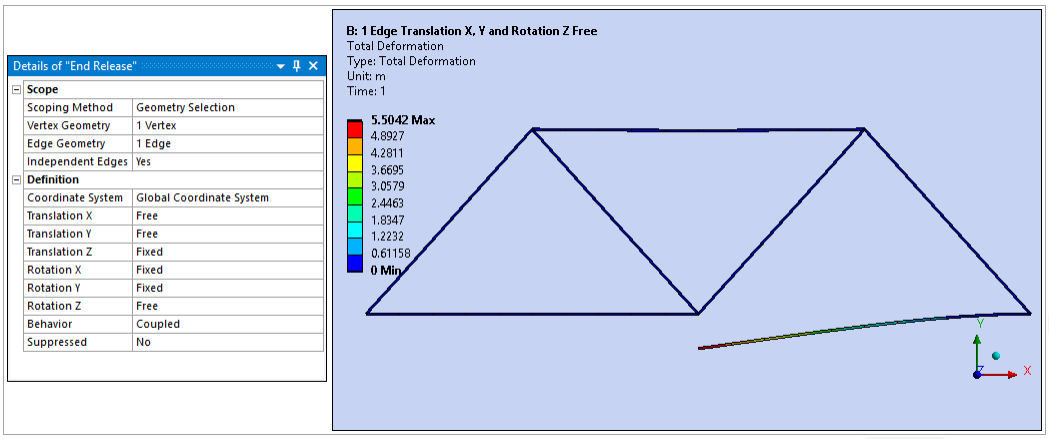
Deformation results for a pin-jointed beam-based truss are shown in the following illustrations. The loading is standard earth gravity. These examples illustrate some basic scoping scenarios.

Here is the Deformation with no **End Release** applied to the truss.

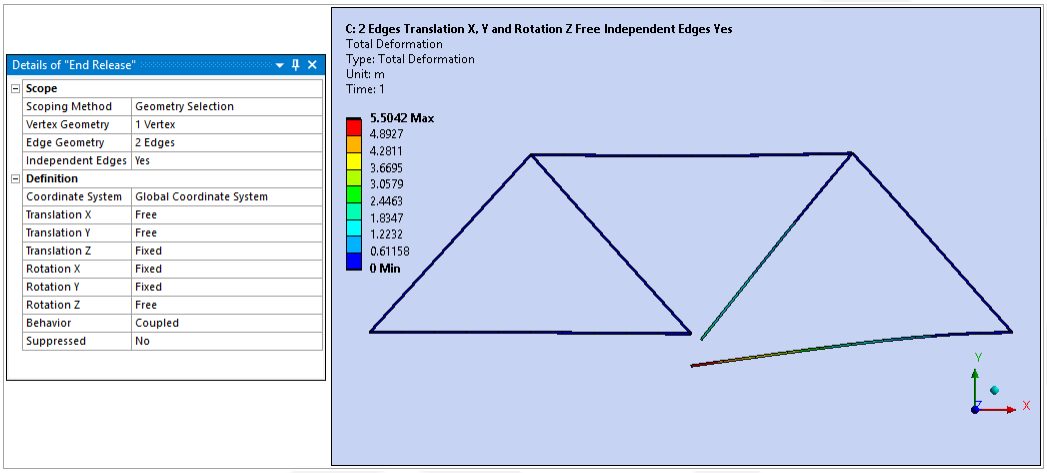


This illustration depicts the **Deformation** of the truss with an **End Release** applied to the vertex and one edge.

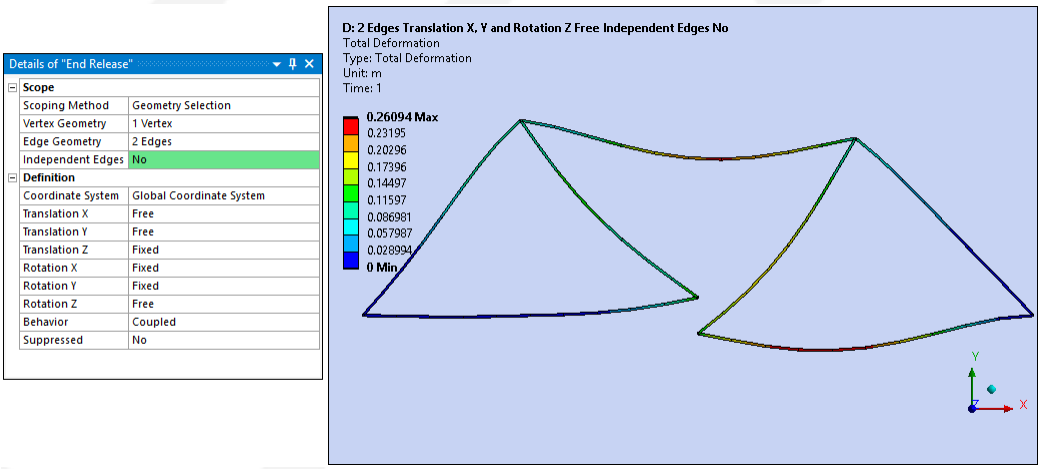
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Here is the **Deformation** with an **End Release** applied to two independent edges.



Here is the **Deformation** with an **End Release** applied to the same two edges except that they are not independent.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Bearings

A bearing is a two-dimensional elastic element used to confine relative motion and rotation of a rotating machinery part. Bearings are a critical support for Rotordynamics analyses and as such, a good bearing design is essential to ensure stability of machinery parts under high speed rotations.

Similar to a spring, a bearing has the structural characteristics of longitudinal stiffness and damping. In addition to these characteristics, bearings are enhanced with coupling stiffness and damping that serve as resistive forces to movement of the machinery part in a rotation plane.

Bearings are supported by all Mechanical analysis types that use the Mechanical APDL solver.

Note:

- The damping characteristics are not applicable to Static Structural, Eigenvalue Buckling, undamped Modal, and Response Spectrum analysis systems.
- While negative stiffness and/or damping characteristics are allowed in all the supported analysis systems, users are cautioned to ensure its proper use, and check the results carefully.
- This boundary condition cannot be applied to a vertex scoped to an End Release (p. 1409).

Scoping Requirements

Bearing scoping is limited to a single face, single edge, single vertex, or an external remote point. Similar to a spring, there is a Mobile side and Reference side for the bearing connection. Based on the Mobile and Reference side selections, a bearing can be made as a bearing connection between **Body to Ground** or **Body to Body**.

Body to Ground Bearing

When specifying the **Connection Type** property as **Body-Ground**, the reference side is assumed to be grounded (or fixed) and the mobile side is set to the scoped entity. Unlike Body-Ground springs, the location of the reference side is set to that of the mobile side because they can be coincident during a linear analysis.

Body to Body Bearing

When specifying the **Connection Type** property as **Body-Body**, both the reference and mobile sides can be set to the scoped geometric entities. Scoped entities of Mobile and Reference should be on two different bodies and the Reference and Mobile locations should be in a selected rotation plane.

For more information about the use of a spring-damper bearing, see COMBI214 - 2D Spring-Damper Bearing in the *Mechanical APDL Theory Reference*.

Apply Bearing

To add a **Bearing**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Add a **Connections** folder if one is not already in the tree, by highlighting the **Model** object and choosing **Connections** from the Model Context Tab (p. 48) or by choosing **Insert**>**Connections** from the context menu (right-click).

2. Add a **Bearing** object by selecting the **Connections** folder and then opening the **Bearing** drop-down menu and then selecting the **Body-Ground** or the **Body-Body** option. You can also right-clicking on the **Connections** folder and selecting **Insert** > **Bearing** from the context menu. This method specifies the **Connection Type** property as **Body-Ground** by default.

3. Under the **Definition** category, specify the **Rotation Plane** property for your model. Selections include:

   - **None** (default)
   - **X-Y Plane**
   - **Y-Z Plane**
   - **X-Z Plane**

4. As required, define the stiffness coefficients (K11, K22, K12, K21) and the damping coefficients (C11, C12, C21, C22). These may be entered as Constant values or using **Tabular Data** entries.

   If you are defining your stiffness and damping coefficients as **Tabular Data**, they are dependent upon **Rotational Velocity** (as provided in the first column of the **Tabular Data** window).

   In addition, when Rotational Velocity-dependent bearings are used for Modal and Full Harmonic Response analyses, the **Coriolis Effect** property (**Analysis Settings**>**Rotordynamics**) must be set to **On, Stationary Reference Frame**.

   The application uses interpolated bearing properties for each **Rotational Velocity** defined in Modal or Full Harmonic Response analyses. When there is no **Rotational Velocity** defined in the system, the first entry of Bearing properties is used.

   See COMBI214 - 2D Spring-Damper Bearing in the *Mechanical APDL Theory Reference* for additional information as well as the example shown below.

5. Specify the following properties under the **Reference** and **Mobile** categories of the Body-Body Bearing and Mobile group of Body-Ground bearing.

   - Specify a **Coordinate System**. This property provides a drop-down list of available coordinate systems. **Global Coordinate System** is the default.
   - Define the **Scoping Method** as **Geometry Selection** (default) or **Named Selection**. The **Scoping Method** may also be specified to a user-defined **Remote Point**, if available.
   - Specify the Connection **Behavior** as either **Rigid** (default), **Deformable**, or **Beam**. If the **Scope Method** property of the Bearing is set to **Remote Point**, the Bearing will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties. The **Behavior** formulation **Coupled** is not supported for Bearings.
   - As needed, specify a **Pinball Region**. Use the **Pinball Region** to define where the bearing attaches to face(s) or edge(s) if the default location is not desirable. By default, the entire face/edge is tied to the bearing element. In the event that this is not desirable, you can

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

choose to enter a **Pinball Region** value. For example, your topology could have a large number of nodes leading to solution processing inefficiencies. Or, if there is overlap between the bearing's scoped faces and another displacement boundary condition, you could experience over-constraint and possible solver failure.

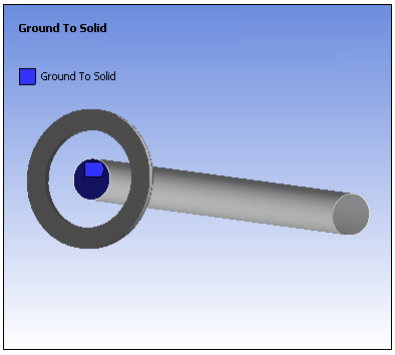
Note:

- The **Pinball Region** and **Behavior** settings are applicable to underlying bodies that are flexible.
- The **Pinball Region** and **Behavior** settings are *not* applicable to a Bearing scoped to the vertex of line body.
- \Because a **Bearing** is a remote boundary condition, it can make use of remote points that were either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

The following examples illustrates Bearings for **Body-Ground** or **Body-Body** with customized Details settings.

Body-Ground

Details of "Ground To Solid"	
Definition	
Connection Type	Body-Ground
Rotation Plane	X-Y Plane
Stiffness K11	100. N/m
Stiffness K22	200. N/m
Stiffness K12	300. N/m
Stiffness K21	400. N/m
Damping C11	10. N-s/m
Damping C22	10. N-s/m
Damping C12	20. N-s/m
Damping C21	30. N-s/m
Suppressed	No
Mobile	
Scoping Method	Geometry Selection
Scope	1 Face
Body	Solid
Coordinate System	Global Coordinate System
Mobile X Coordinate	0.5 m
Mobile Y Coordinate	0.5 m
Mobile Z Coordinate	0. m
Behavior	Rigid
Pinball Region	All

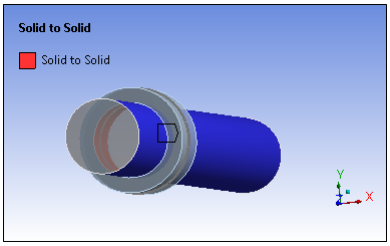


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Body-Body

Details of "Solid to Solid"

Definition	
Connection Type	Body-Body
Rotation Plane	X-Y Plane
Stiffness K11	0.1 N/mm
Stiffness K22	0.2 N/mm
Stiffness K12	0.3 N/mm
Stiffness K21	0.4 N/mm
Damping C11	1.e-002 N-s/mm
Damping C22	2.e-002 N-s/mm
Damping C12	3.e-002 N-s/mm
Damping C21	4.e-002 N-s/mm
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Scope	1 Face
Body	Solid
Coordinate System	Global Coordinate System
Reference X Coordinate	4.3194e-016 mm
Reference Y Coordinate	-4.6703e-015 mm
Reference Z Coordinate	750. mm
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Scope	1 Face
Body	Solid
Coordinate System	Global Coordinate System
Mobile X Coordinate	-1.7278e-016 mm
Mobile Y Coordinate	3.2665e-015 mm
Mobile Z Coordinate	750. mm
Behavior	Rigid
Pinball Region	All



The stiffness characteristics K11, K22, K12, and K21, and damping characteristics C11, C22, C12, and C21 are used to model four spring-damper sets in a plane of a rotating shaft in this example. For more information about the spring-damper orientation, see COMBI214 - 2D Spring-Damper Bearing in the *Mechanical APDL Theory Reference*.

The bearing is created on a face of the shaft that is perpendicular to the Z-axis. As the Z-axis is the rotating axis of the shaft, the X-Y Plane is selected for the Rotation Plane option. While the bearing in this example is defined using Global Coordinate System, it can also be defined with a user-defined local coordinate system. When changing from one coordinate system to another, the Bearing needs the scoping to be updated to desired location for the new coordinate system.

For a bearing to be modeled properly, the location of the reference side and the mobile side must lie in the selected rotation plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Configuring Analysis Settings

The following topics are covered in this section.

Analysis Settings for Most Analysis Types
Steps and Step Controls for Static and Transient Analyses

Analysis Settings for Most Analysis Types

When you define an analysis type (p. 323), an Analysis Settings object is always included in the environment. With this object selected, you can define various solution options in order to customize your analysis, such as enabling large deflection for a stress analysis.

Select from the following **Analysis Settings** object categories for a complete description of their properties and options.

- Step Controls for Static and Transient Analyses
- Step Controls for Harmonic Analysis Types
- Solver Controls
- Restart Analysis
- Restart Controls
- Adaptivity Remeshing Controls
- Creep Controls
- Fracture Controls
- Cyclic Controls
- Radiosity Controls
- Options for Analyses
- Scattering Controls
- Advanced
- Damping Controls
- Nonlinear Controls
- Output Controls
- Analysis Data Management
- Rotordynamics Controls
- Visibility

Note:

Analysis Settings for the following analysis types are described in their respective documents:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- See Analysis Settings for Explicit Dynamics Analyses in the *Explicit Dynamics Analysis Guide*.
- For additive manufacturing:
  - See Establish Thermal Analysis Settings and Establish Structural Analysis Settings in the *LPBF Simulation Guide*.
  - See Establish Thermal Analysis Settings and Establish Structural Analysis Settings in the *DED Simulation Guide*.

Step Controls for Static and Transient Analyses

You use the **Step Controls** category during static and transient dynamic analyses to first specify a multiple step analysis, and then define all associated properties, including those of the Nonlinear Controls (p. 1468) and Output Controls (p. 1473), for each step.

Go to a section topic:

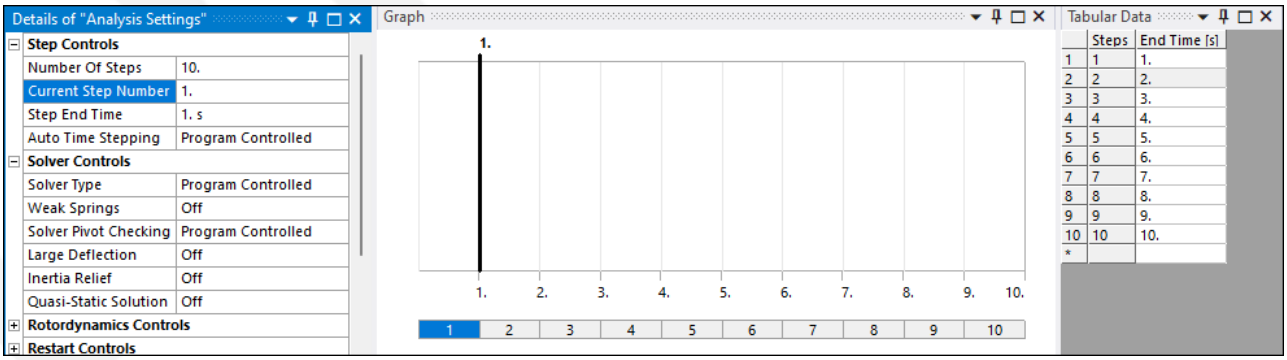
- Specifying Step Controls (p. 1418)
- Specify Settings for Multiple Substeps Simultaneously (p. 1421)
- Display All Analysis Settings in the Worksheet (p. 1423)
- Activating and Deactivating Loads (p. 1423)

Specifying Step Controls

The **Step Controls** category of the **Analysis Settings** object includes the properties described below. The display of certain properties can depend upon the selections that you make, the type of analysis you are performing, as well as the use of a particular feature. Note that most **Step Controls** (as well as **Nonlinear Controls** and **Output Controls**) are "step aware," that is, property settings can be different for each step.

Number of Steps

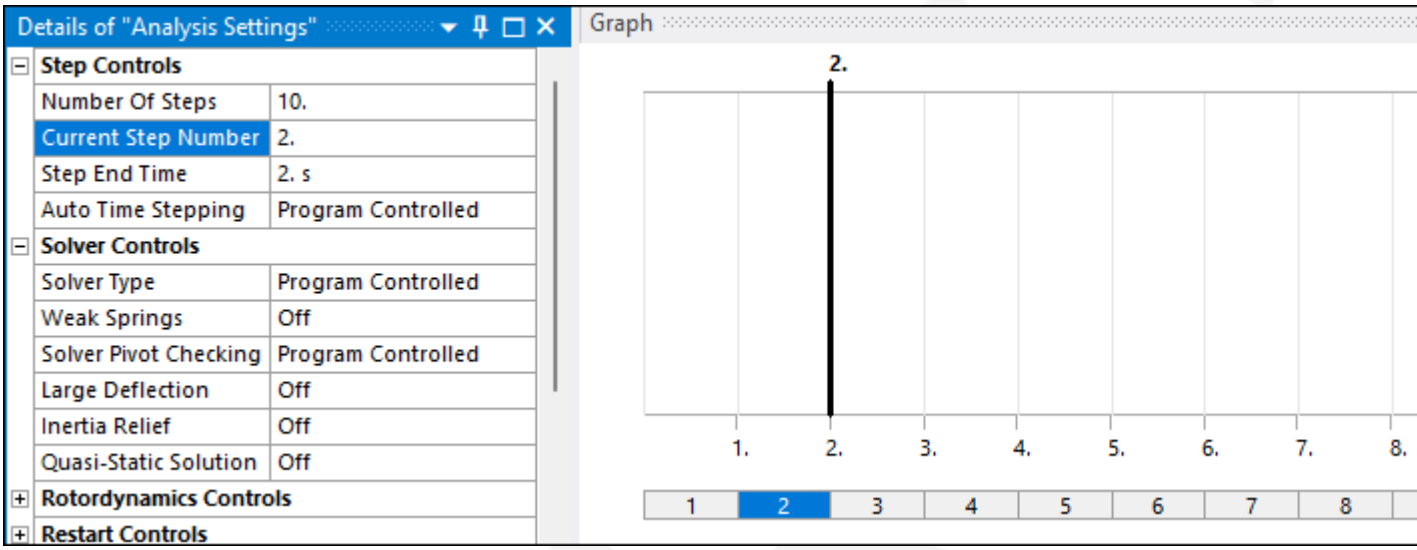
Use this property to define the desired number of steps for your analysis. The **Graph** and **Tabular Data** windows automatically update with the specified number of steps.



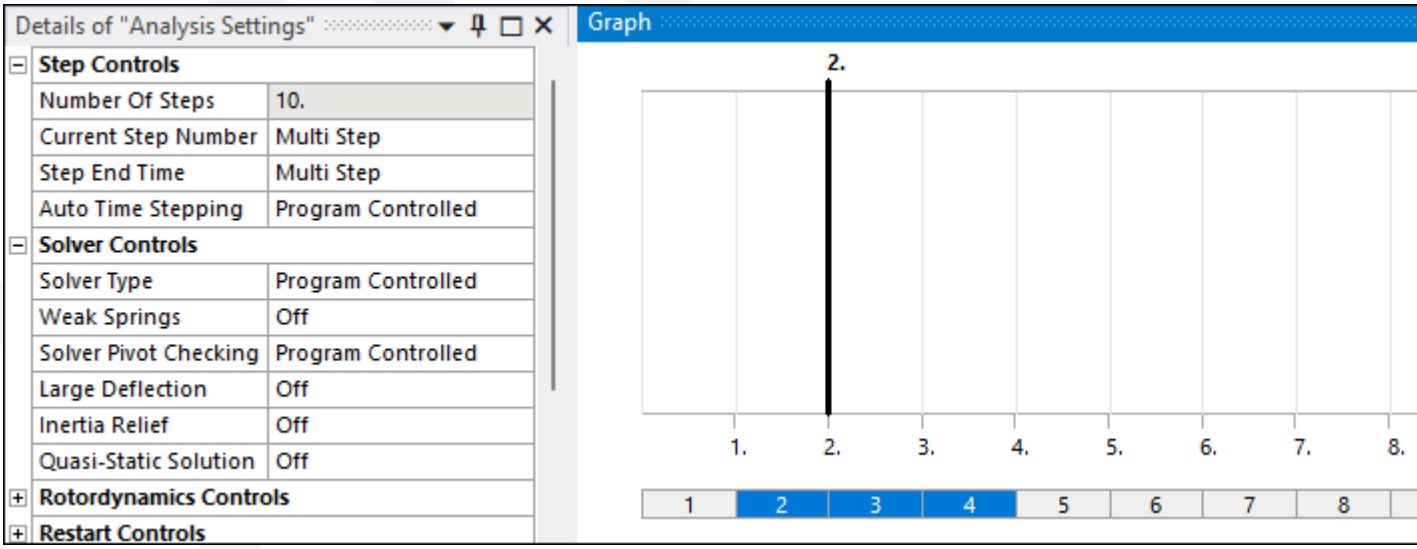
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Current Step Number

This property displays the currently selected step. The default setting is **1**. To select a particular step, you can 1) change the value of this property, 2) select a time value in the **Graph** window (or the step bar displayed below the chart), or 3) select a row in the **Step** column of the **Tabular Data** window. When you change the value of this property, your entry is reflected in the **Graph** and **Tabular Data** windows. An example is illustrated below for a **Current Step Number** value of **2**.



You can select multiple steps in the **Graph** window using the **[Shift]** and **[Ctrl]** keys. When you select multiple steps, the **Current Step Number** property displays the value **Multi Step**. For this type of selection, any changes affect all of the selected steps.



Note:

The selection of multiple steps is not supported for an Explicit Dynamics or a Rigid Body Dynamics analysis. If multiple steps are highlighted in the **Graph** window and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

changes made to the **Analysis Settings**, the changes will affect the last selected step only.

Step End Time

This property displays the end time of the **Current Step Number**. When multiple steps are selected this property displays the value **Multi Step**.

Auto Time Stepping

Automatic time stepping is available for static and transient analyses, and is especially useful for nonlinear solutions. This property is also described in detail in the Automatic Time Stepping (p. 1493) section.

This property provides a drop-menu with the following options:

- **Program Controlled** (default): The Mechanical application automatically switches time stepping on and off as needed. A check is performed on non-convergent patterns. The physics of the simulation is also taken into account. The **Program Controlled** settings are presented in the following table:

Auto Time Stepping Program Controlled Settings			
Analysis Type	**Initial Substeps**	**Minimum Substeps**	**Maximum Substeps**
Linear Static Structural	1	1	1
Nonlinear Static Structural	1	1	10
Linear Steady-State Thermal	1	1	10
Nonlinear Steady-State Thermal	1	1	10
Transient Thermal	100	10	1000

- **On**: You control time stepping by completing the following fields that only appear if you choose this option. No checks are performed on non-convergent patterns and the physics of the simulation is not taken into account.
  - **Initial Substeps**: Specifies the size of the first substep. The default is **1**.
  - **Minimum Substeps**: Specifies the minimum number of substeps to be taken (that is, the maximum time step size). The default is **1**.
  - **Maximum Substeps**: Specifies the maximum number of substeps to be taken (that is, the minimum time step size). The default is **10**.
- **Off**: No time stepping is enabled. You are prompted to enter the **Number Of Substeps**. The default is **1**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Define By

This property displays when you set the **Auto Time Stepping** property to **On** or **Off**. Its drop-down menu contains the options **Time** and **Substep**. The default setting for this property depends upon the analysis type. It enables you to set the limits on load increment in one of two ways. You can specify the **Initial Time Step**, **Minimum Time Step** and **Maximum Time Step** number of substeps for a step or equivalently specify the **Time Substeps**, **Minimum Substeps** and **Maximum Substeps** time step size.

Carry Over Time Step

This property is available when you specify multiple steps. This is useful when you do not want any abrupt changes in the load increments between steps. When this is set to On, the **Initial** time step size of a step will be equal to the last time step size of the previous step.

Time Step

This property displays a time value for the current step.

Time Integration

This property is only available for Coupled Field Transient, Transient Structural, or Transient Thermal analyses. This field indicates whether a step should include transient effects, such as structural inertia or thermal capacitance, or whether it is a static (steady-state) step. This field can be used to set up the Initial Conditions (p. 349) for a transient analysis.

- **On**: Default setting for transient analyses.
- **Off**: This option does not include structural inertia or thermal capacitance in solving this step.

For a **Coupled Field Transient** analysis when the **Time Integration** property is set to **Yes**, the following additional properties display and enable you to specify whether to turn a physics field on or off:

- **Structural Only**: Options include **No** (default) and **Yes**.
- **Thermal Only**: Options include **No** and **Yes** (default) .

Note:

With **Time Integration** set to **Off** during Transient structural analyses, the application does not compute velocity results. Therefore spring damping forces, which are derived from velocity, will equal zero. This is not the case for Rigid Dynamics analyses.

Specify Settings for Multiple Substeps Simultaneously

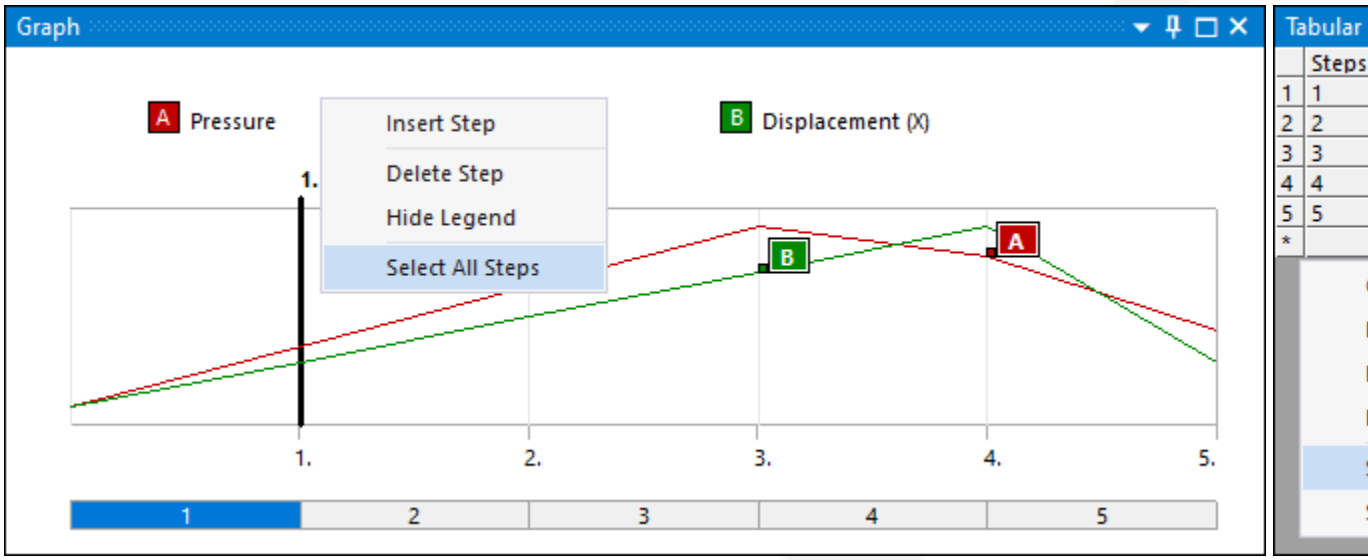
If you want to specify the same analysis setting(s) to several substeps, you can select steps of interest as follows and change the analysis settings properties.

Change All Steps

To specify analysis settings for all the substeps:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. In either the **Graph** or **Tabular Data** window, right-click and select the **Select All Steps** option.



2. In the Details pane, make changes to properties. These changes will automatically apply to all steps.

Change Multiple Steps from Graph Window

From the **Graph** window:

1. Highlight the **Analysis Settings** object.
2. Highlight steps in the **Graph** window using either of the following standard windowing techniques:
   - **Ctrl** key to highlight individual steps.
   - **Shift** key to highlight a continuous group of steps.
3. Specify the analysis settings as needed. These settings will apply to all selected steps.

Change Multiple Steps from Tabular Data Window

From the **Tabular Data** window:

1. Highlight the **Analysis Settings** object.
2. Highlight steps in the **Tabular Data** window using either of the following standard windowing techniques:
   - **Ctrl** key to highlight individual steps.
   - **Shift** key to highlight a continuous group of steps.
3. Click the right mouse button in the window and choose **Select All Highlighted Steps** from the context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Specify the analysis settings as needed. These settings will apply to all selected steps.

Display All Analysis Settings in the Worksheet

The **Worksheet** for the **Analysis Settings** object provides a single display of pertinent settings in the Details for *all* steps.

Worksheet

Analysis Settings

Properties	Step 1	Step 2	Step 3	Step 4	Step 5
Step Controls					
Step End Time	1.	2.	3.	4.	5.
Auto Time Stepping	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Nonlinear Controls					
Force Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Moment Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Displacement Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Rotation Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Energy Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Line Search	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Stabilization	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle
Output Controls					
Stress	Yes	Yes	Yes	Yes	Yes
Back Stress	No	No	No	No	No
Strain	Yes	Yes	Yes	Yes	Yes
Contact Data	Yes	Yes	Yes	Yes	Yes
Nonlinear Data	No	No	No	No	No
Nodal Forces	No	No	No	No	No
Contact Miscellaneous	No	No	No	No	No
General Miscellaneous	No	No	No	No	No
Euler Angles	Yes	Yes	Yes	Yes	Yes
Volume and Energy	Yes	Yes	Yes	Yes	Yes
Store Results At	All Time Points	All Time Points	All Time Points	All Time Points	All Time Points
Advanced					
Inverse Option	No	No	No	No	No
Contact Split (DMP)	Program Controlled	Program Controlled	Program Controlled	Program Controlled	Program Controlle

Geometry | Worksheet

Note:

For Explicit Dynamics, the Worksheet for the **Analysis Settings** object provides a single display of pertinent *step-aware* settings in the Details pane for all steps.

Activating and Deactivating Loads

You can activate (include) or deactivate (delete) a specified load from being used in the analysis within the time span of a step. For most loads, such as pressure or force, deleting the load is the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

same as setting the load value to zero. But for certain loads, such as a specified displacement, this is not the case.

Note:

- Changing the method of how a multiple-step load value is specified (such as Tabular to Constant), the Activation/Deactivation state of all steps resets to the default, Active.
- The activate/deactivate option is only available when the **Independent Variable** property or the **X-Axis** property is set to **Time**.
- The capability to activate and/or deactivate loads is not available for the Samcef solver

To activate or deactivate a load in a stepped analysis:

1. Highlight the load within a step in the **Graph** or a specific step in the **Tabular Data** window.
2. Click the right mouse button and choose **Activate/Deactivate at this step!**.

Note:

For displacements and remote displacements, it is possible to deactivate only one degree of freedom within a step.

Note:

For a **Mode Superposition Transient** analysis, the application does not support deactivation of an **Acceleration** or a **Displacement** defined as a **Base Excitation**.

For Imported loads and Temperature, Thermal Condition, Heat Generation, Voltage, and Current loads, the following rules apply when multiple load objects of the same type exist on common geometry selections:

- A load can assume any one of the following states during each load step:
  - Active: Load is active and data specified during the first step.
  - Reactivated: Load is active and data specified during the current step, but was deactivated during the previous step. A change in step status exists.
  - Deactivated: Load is deactivated at the current step, but was active and data applied during the previous step. A change in step status exists.
  - Unchanged: No change in step status exists.
- During the first step, an active load will overwrite other active loads that exist higher (previously added) in the tree.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

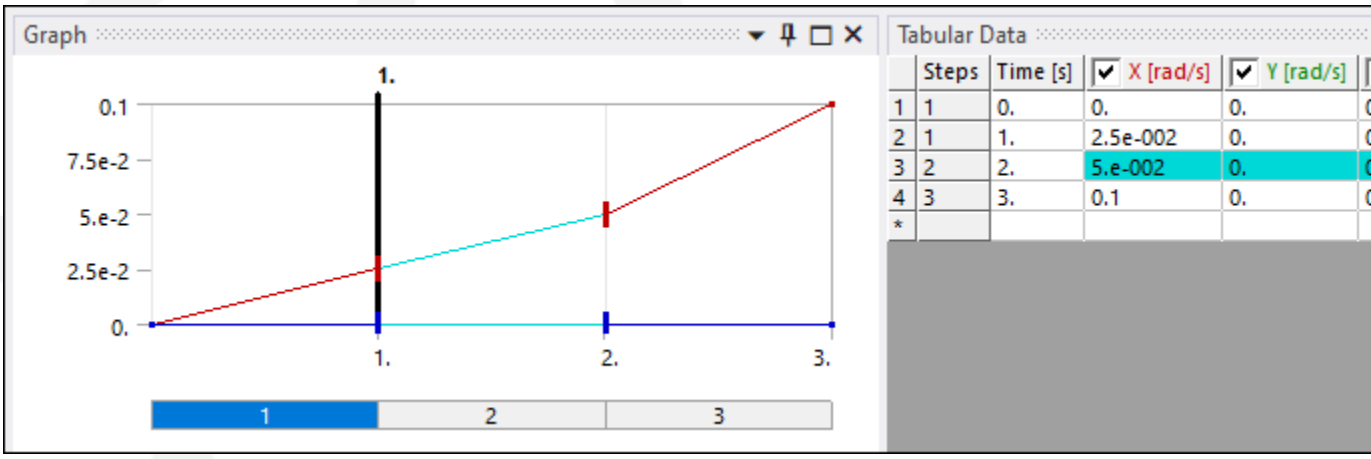
- During any other subsequent step, commands are sent to the solver only if a change in step status exists for a load. Hence, any unchanged loads will get overwritten by other reactivated or deactivated loads irrespective of their location in the tree. A reactivated/deactivated load will overwrite other reactivated and deactivated loads that exist higher (previously added) in the tree.

Note:

Review the following:

- For each load step, if both Imported Loads and user-specified loads are applied on common geometry or mesh selections, the Imported Loads take precedence. See respective Imported Load (p. 1736) for more details.
- Deactivated Imported Loads do not overwrite other reactivated loads or Imported Loads on common geometry or mesh selections even if they exist higher (were previously added) in the Outline.
- For imported loads specified as tables, with the exception of imported displacement and temperature loads, a value of zero is applied in the table where the load is deactivated, and commands are sent to the solver only at the first active load step. Hence these reactivated/deactivated imported loads with tabular loading do not overwrite other unchanged or reactivated/deactivated loads that exist higher (previously added) in the tree.
- For imported loads specified as tables, the data is available outside the range of specified analysis times/frequencies. If the solve time/frequency for a step/sub-step falls outside the specified **Analysis Time/Frequency**, then the load value at the nearest specified analysis time is used.

  The tabular data view provides the equation for the calculation of values through piecewise linear interpolation at steps where data is not specified.



Some scenarios where load deactivation is useful include:

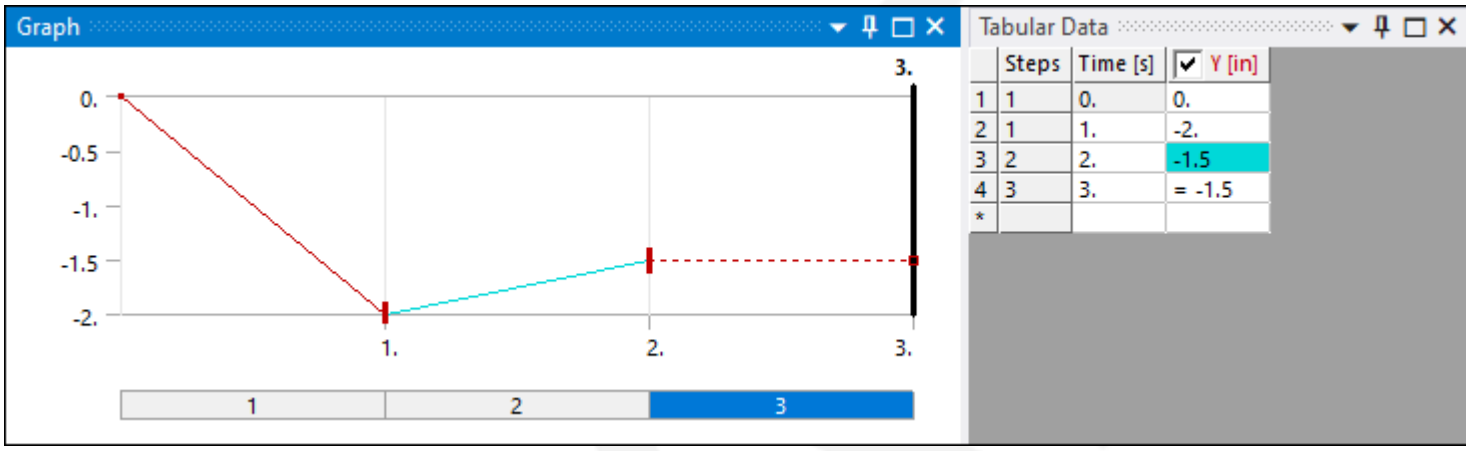
- Springback of a cantilever beam after a plasticity analysis (see example below).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Bolt pretension sequence (p. 1566) (Deactivation is possible by setting **Define By** to **Open** for the load step of interest).
- Specifying different initial velocities for different parts in a transient analysis.

Example: Springback of a cantilever beam after a plasticity analysis

In this case a Y displacement of -2.00 inch is applied in the first Step. In the second step this load is deactivated (deleted). Deactivated portions of a load are shown in cyan in the **Graph** and also have a red stop bars indicating the deactivation. The corresponding cells in the data grid are also shown in cyan.



In this example the second step has a displacement value of -1.5. However since the load is deactivated this will not have any effect until the third step. In the third step a displacement of -1.5 will be step applied from the sprung-back location.

Step Controls for Harmonic Analysis Types

This category enables you to define step controls for Coupled Field Harmonic (p. 380), Harmonic Acoustics, and Harmonic Response (p. 496) analyses that include 1) multiple load steps and 2) loads that vary with load steps or rotational velocity in the form of revolutions per minute (RPMs). You use the properties of this category to define the number of steps and their settings, such as associated RPM values, frequency spacing, minimum frequencies, maximum frequencies, etc.

Important:

During a Mode Superposition Harmonic Response analysis, the deactivation of an Acceleration or Displacement that is defined as **Base Excitation** sends **DVAL** command with the **VALUE** argument set to **0** (zero).

The category properties are described below.

Property	Description
Multiple Steps	This property activates the ability to define load steps. Options include **Yes** and **No** (default). When activated, all associated properties display. When you activate this property,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

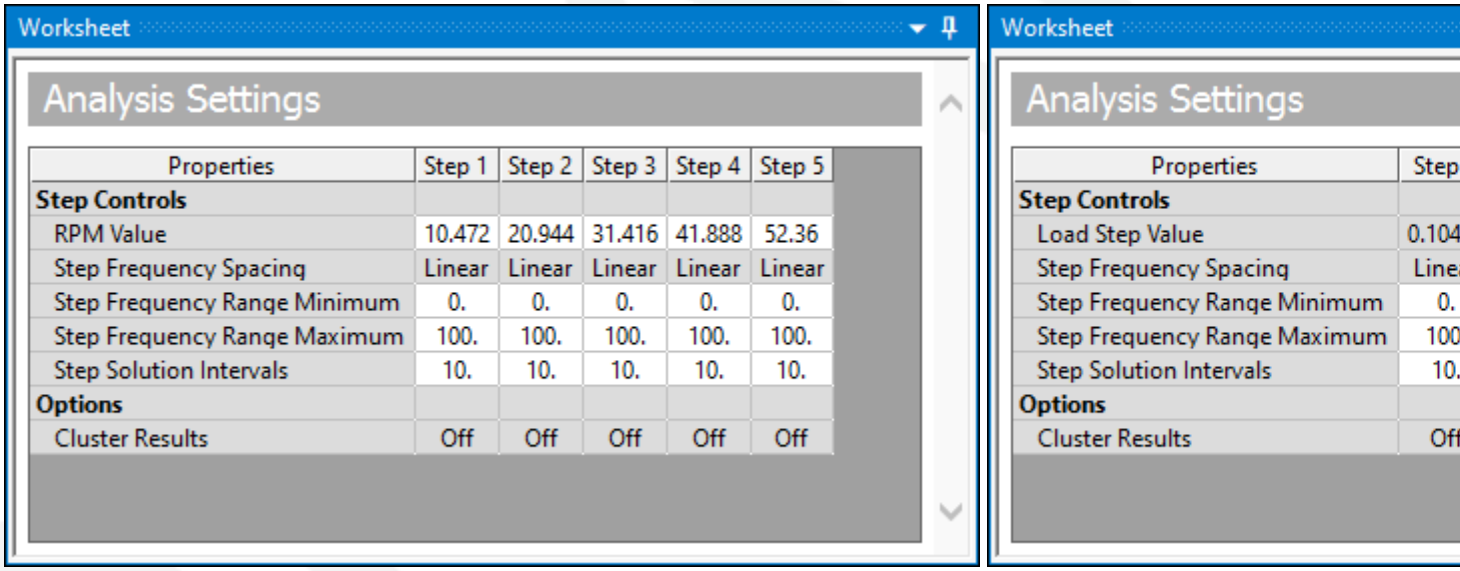
Published: 2023-10-23

Property	Description
	the **Worksheet** window automatically displays. Entries can be made using the **Worksheet**, as described below. **Note:** The application automatically sets this property to **No** and it becomes read-only if you link your **Harmonic Response** analysis to a downstream **Structural Optimization** analysis or if the Future Analysis (p. 1485) property (**Analysis Settings** > **Analysis Data Management**) is set to **Structural Optimization**.
Multiple Step Type	Specify how you want to define step control values, either by **RPM** or **Load Step**.
Number of Steps	You use this property to set the number of load steps for your analysis.
Current Step Number	You use this property to make changes to the load step specified (1, 2, etc.). All subsequent property entries apply only to the load step specified. The default value is 1. Any changes to the properties for the load step are reflected in the corresponding column in the **Worksheet**.
RPM Value	When the **Multiple Step Type** property is set to **RPM**, this property displays. You use this property to specify an RPM value, in radians per second. This value is applied to the value of the **Current Step Number** property.
Step Frequency Spacing	Define how the application performs frequency spacing for the **Current Step** number. Options include **Linear** (default), **Logarithmic**, **1 Octave Band**, **1/2 Octave Band**, **1/3 Octave Band**, **1/6 Octave Band**, **1/12 Octave Band**, **1/24 Octave Band**, and **Order Based**. Using the **Linear** option, harmonic results are reported at uniform frequency intervals for the specified frequency range. The **Logarithmic** option, as well as the **Octave Band** options, defines the frequency spacing by setting an appropriate *LogOpt* key for the **HARFRQ** command. **Important:** The **Order Based** setting: • Is only supported when the **Multiple Step Type** property is set to **RPM**. When selected, all steps are order-based. • Does not support the **Variational Technology** setting for the **Solution Method** property (**Analysis Settings** > Options (p. 1426)).
Step Central Frequency	This option displays when you select an **Octave Band** option for the **Step Frequency Spacing** property, this property also displays. This property requires you to specify the central frequency of octave band.
Orders	This option displays when you set the **Step Frequency Spacing** property to **Order Based**. Enter the orders to evaluate for each RPM step. The property supports individual (including decimal values) and range entries (whole numbers only). Separate individual entries using

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	Description
	a comma (0.5, 10, 15). Separate range values using a hyphen (20-25). You can also combine these entries (0.5, 10, 15, 20-25). The application automatically rounds range values that include a decimal up ($\geq$ 0.5) or down ($\leq$ 0.4).
Step Frequency Range Minimum	You use this property to set the desired minimum frequency for the frequency range for the **Current Step** number. When you select an Octave Band option for the **Step Frequency Spacing** property, this property becomes read-only, and the application calculates its value from the specified **Step Central Frequency**.
Step Frequency Range Maximum	You use this property to set the desired maximum frequency for the frequency range for the **Current Step** number. When you select an Octave Band option for the **Step Frequency Spacing** property, this property becomes read-only, and the application calculates its value from the specified **Step Central Frequency**.
Step Solution Intervals	You use this property to specify the solution intervals for the **Current Step** number. The default value is **10**.

As illustrated below, when you activate Step Controls, the **Worksheet** window displays and enables you to modify or view the step-based settings. When the **Solution Method** property is set to **Mode Superposition**, you can change the setting of the **Cluster Results** under the **Options** category (p. 1447) of the **Analysis Settings**.



Solver Controls

The properties provided by the **Solver Controls** category vary based on the specified Analysis Type. This table denotes which Details view properties are supported for each analysis type. The remainder of the section describes the functions and features of the properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details Properties	Analysis Type																
	Coupled Field Modal (p. 386)	Coupled Field Static (p. 391)	Coupled Field Transient (p. 395)	Static Structural (p. 652)	Static Acoustics	Transient Structural (p. 665)	Rigid Dynamics (p. 550)	Modal (p. 515)	Modal Acoustics	Eigenvalue Buckling (p. 487)	Steady - State Thermal (p. 657)	Transient Thermal (p. 680)	Magnetostatic (p. 546)	Electric (p. 403)	Thermal Electric (p. 661)	Structural Optimization	Substructure Generation (p. 540)
Damped	■							■	■								
Solver Type	■	■	■	■	■	■		■	■	■	■	■	■	■	■	■	■
Mode Reuse								■									
Store Complex Solution								■									
Weak Springs		■	■	■	■	■											
Solver Pivot Checking		■		■	■						■						
Large Deflection		■	■	■	■	■											
Application (App.) Based Settings						■											
Inertia Relief		■		■	■												
Quasi-Static Solution		■		■													
Include Negative Load Multiplier										■							
Time Integration and Constraint Stabilization							■										

Damped

The **Damped** property options include **Yes** and **No** (default). Set the property to **Yes** to enable a damped system where the natural frequencies and mode shapes become complex.

Solver Type

The options of the **Solver Type** property can vary based upon the type of analysis you are performing. In general, the **Solver Type** options include:

- **Program Controlled** (default): this setting enables the application to select the optimal solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Direct**: this setting uses the Sparse solver.
- **Iterative**: this setting uses the PCG or ICCG (for Electric and Electromagnetic analyses) solver.

Note:

- See the **EQSLV** command in the *Mechanical APDL Command Reference* for more information about solver selection.
- See the Linear Equation Solver Memory Requirements section of the *Mechanical APDL Performance Guide* for recommendations about how to manage memory in order to maximize performance.

Based on the analysis type and as described below, additional **Solver Type** property options are available. Got to an analysis type topic:

- Coupled Field Modal Analysis (p. 1430)
- Eigenvalue Buckling Analysis (p. 1431)
- Modal Acoustics Analysis (p. 1431)
- Modal Analysis (p. 1432)
- Structural Optimization (p. 1433)
- Substructure Generation (p. 1433)

Coupled Field Modal Analysis

When the **Damped** property is set to **No**, based on the physics and coupling, the **Solver Type** property options include:

For acoustic coupled analyses:	For structural and electric coupled physics analyses:	For pure Acoustic analyses:
• **Program Controlled** (default) **Unsymmetric**	• **Program Controlled** (default) • **Direct** • **Subspace** • **Unsymmetric**	• **Program Controlled** (default) • **Direct** • **Subspace**

For a Piezoelectric analysis if the **Damped** property is set to **Yes**, the options include:

- **Unsymmetric** (default)
- **Full Damped**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For all other physics, **Full Damped** is the only option.

Note:

For damped systems that have non-viscous damping, Ansys recommends the **Unsymmetric** setting. This setting is more robust and works well for higher frequencies.

Eigenvalue Buckling Analysis

For an Eigenvalue Buckling Analysis (p. 487), the **Solver Type** options include:

- **Program Controlled** (default)
- **Direct**
- **Subspace**

The **Program Controlled** option uses the **Direct** solver. Refer to the **BUCOPT** command for additional information about buckling analysis solver selection.

Modal Acoustics Analysis

If the **Damped** property is set to **Yes**, **Full Damped** is the only supported option for this analysis.

For a Modal Acoustics Analysis when the **Damped** property is set to **No**, **Solver Type** property options include:

- **Program Controlled** (default)
- **Direct**
- **Subspace**
- **Unsymmetric**

Note:

When you have **Fluid Solid Interface** (p. 1648) object defined in your model or the **Element Type** property is set to **Coupled** in the Acoustic FSI Definition of a **Physics Region** object, you must select **Unsymmetric** or **Damped** solver type to proceed with the solution.

Based on your configuration, select your solver type based on the following:

Element Type	Symmetric Formulation Setting	FSI Load	Required Solver Type
Coupled	Off	No	Unsymmetric/Damped
Coupled	On	No	Direct/Subspace/Damped

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Uncoupled	Off	No	Direct/Subspace/Damped
Uncoupled	On	No	Direct/Subspace/Damped
Coupled	Off	Yes	Unsymmetric/Damped
Coupled	On	Yes	Direct/Subspace/Damped
Uncoupled	Off	Yes	Unsymmetric/Damped
Uncoupled	On	Yes	Direct/Subspace/Damped

Modal Analysis

For a Modal Analysis (p. 515) when the **Damped** property is set to **No**, **Solver Type** options include the common settings, **Program Controlled** (default), **Direct**, and **Iterative** as well as the following:

- **Unsymmetric**
- **Supernode**
- **Subspace**

The **Direct**, **Iterative**, **Unsymmetric**, **Supernode**, and **Subspace** types are used to solve a modal system that does not include any damping effects (the **Damped** property is set to **No**). Except for the **Unsymmetric** option, the solver types are intended to solve Eigen solutions with symmetric mass and stiffness. For a large model, the **Iterative** solver is preferred over the **Direct** solver for its efficiency in terms of solution time and memory usage.

During a Modal analysis the:

- **Direct** solver uses the Block Lanczos extraction method that employs an automated shift strategy, combined with a Sturm sequence check, to extract the number of eigenvalues requested. The Sturm sequence check ensures that the requested number of eigen frequencies beyond the user supplied shift frequency (FREQB on the **MODOPT** command) is found without missing any modes. See the Block Lanczos help in the Eigenvalue and Eigenvector Extraction section of the *Mechanical APDL Theory Reference*.
- **Supernode** solver is recommended for extracting a large number of modes. Selecting **Supernode** as the Solver Type automatically sets the **Limit Search to Range** property to **Yes** in the Options (p. 1455) category. This selection also displays the **Range Minimum** and **Range Maximum** properties and requires a **Range Maximum** frequency entry. Alternatively, you may reset the **Limit Search to Range** property to **No** to find all of the possible modes without any restrictions on the frequency range.
- **Subspace** solver (unlike the **Direct** solver) does not perform Sturm sequence check by default (STRMCK is OFF by default in **SUBOPT** command), making it relatively faster than **Direct** solver and also has reasonable accuracy. In addition, the **Subspace** solver supports distributed parallelism allowing you to take advantage of a distributed architecture to perform faster computations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Unsymmetric** solver (for modal systems with unsymmetric mass and/or stiffness) is required for solving the Eigen solutions. See the Help for the **MODOPT** command in the *Mechanical APDL Command Reference* for more information about solver selection.

If the **Damped** property is set to **Yes**, the **Solver Type** property options include:

- **Program Controlled** (default)
- **Full Damped**
- **Reduced Damped**

The **Reduced Damped** solver is preferred over the **Full Damped** solver for its efficiency in terms of solution time. However, the **Reduced Damped** solver is not recommended when high damping effects are present because it can become inaccurate.

Structural Optimization

The Structural Optimization analysis **Solver Type** options include:

- **Program Controlled** (default)
- **Sequential Convex Programming**
- **Optimality Criteria**

Substructure Generation

For the Substructure Generation (p. 540) analysis, options include **Program Controlled** and **Direct**. The **Program Controlled** option defaults to **Direct**, the only supported **Solver Type**.

Mode Reuse (Modal Analyses Only)

The **Mode Reuse** property is only available for Modal analyses when you specify the **Solver Type** as **Reduced Damped**. This property allows the solver to compute complex eigensolutions efficiently during subsequent solve points by reusing the undamped eigensolution that is calculated at the first solve point. The default setting is **Program Controlled**. Set this property to **Yes** to enable it or **No** to disable the property.

Store Complex Solution (Modal Analyses Only)

This property is only available for a Modal Analysis and only when the **Damped** property is set to **Yes** and the **Solver Type** is set to **Reduced Damped**. It allows you to solve and store a damped modal system as an undamped modal system.

Weak Springs

For stress simulations, such as thermal-stress or press fit modeling, the addition of weak springs can facilitate a solution by preventing numerical instability, while not having an effect on real world engineering loads. The following **Weak Springs** settings are available in the Details view:

- **Program Controlled**: Mechanical determines if weak springs will facilitate the solution, then adds a standard weak spring stiffness value accordingly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **On**: Mechanical always adds a weak spring stiffness. Choosing **On** causes a **Spring Stiffness** option to appear that allows you to control the amount of weak spring stiffness. Your choices are to use the standard stiffness mentioned above for the **Program Controlled** setting of **Weak Springs** or to enter a customized value. The following situations may prompt you to choose a customized stiffness value:

  a. The standard weak spring stiffness value may produce springs that are too weak such that the solution does not occur, or that too much rigid body motion occurs.

  b. You may judge that the standard weak spring stiffness value is too high (rare case).

  c. You may want to vary the weak spring stiffness value to determine the impact on the simulation.

  The following **Spring Stiffness** settings are available:

  – **Program Controlled** (default setting): Adds a standard weak spring stiffness (same as the value added for the **Program Controlled** setting of **Weak Springs**).

  – **Factor**: Adds a customized weak spring stiffness whose value equals the **Program Controlled** standard value times the value you enter in the **Spring Stiffness Factor** field (appears only if you choose **Factor**). For example, setting **Spring Stiffness Factor** equal to 20 means that the weak springs will be 20 times stronger than the **Program Controlled** standard value.

  – **Manual**: Adds a customized weak spring stiffness whose value you enter (in units of force/length) in the **Spring Stiffness Value** field (appears only if you choose **Manual**).

- **Off** (default setting): Weak springs are not added. By default, the application does not include weak springs for the solution.

The default setting for this property can be modified using the **Options** dialog box. See the Specifying Options (p. 217) section of the Help under Analysis Settings and Solution (p. 238).

Solver Pivot Checking

In the case of an underconstrained model or contact related issue, an ill conditioned solver matrix will produce an error message in the solver and abort the solution. This property instructs the solver how to handle such an instance. Options include:

- **Program Controlled** (default): Allows the solver to determine the response.
- **Warning**: Instructs the solver to continue upon detection of the condition and attempt the solution.
- **Error**: Instructs the solver to stop upon detection of the condition and issue an error message.
- **Off**: Pivot checking is not performed.

The default setting for this property can be modified using the **Options** dialog box. See the Specifying Options (p. 217) section of the Help under Analysis Settings and Solution (p. 238).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Large Deflection

This property, applicable to Static Structural and Transient Structural analyses, determines whether the solver should take into account large deformation effects such as large deflection, large rotation, and large strain. Set **Large Deflection** to **On** if you expect large deflections (as in the case of a long, slender bar under bending) or large strains (as in a metal-forming problem).

Important:

When using hyperelastic material models, you must set the **Large Deflection** property to **On**.

Application (App.) Based Settings

The **App. Based Settings** property is available for Full Transient Structural analyses only. Based on the **TINTP** command, this property provides the options listed below. The selection of these options depends upon the purpose of your simulation. For a detailed description of each option, see the Transient Dynamic Analysis Settings Based on Application section of the *Mechanical APDL Structural Analysis Guide*.

- **Impact**
- **High Speed Dynamics**
- **Moderate Speed Dynamics** (default)
- **Low Speed Dynamics**
- **Quasi-Static**

  For Quasi Static application-based settings, the Backward Euler algorithm is used. In addition, damping energy and the work done by any external loading condition is taken into account. This can be seen through the transient solution options **TRNOPT** command.

- **User Defined**

When you select the **User Defined** option, the associated property **Amplitude Decay Factor** displays. The default value for this property is 0.1.

Inertia Relief (Linear Static Structural Analyses Only)

Calculates accelerations to counterbalance the applied loads. Displacement constraints on the structure should only be those necessary to prevent rigid-body motions (6 for a 3D structure). The sum of the reaction forces at the constraint points will be zero. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated.

Requirements

The use of the **Inertial Relief** property requires the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- This option applies only to the linear static structural analyses. Nonlinearities, elements that operate in the nodal coordinate system, and axisymmetric or generalized plane strain elements are not allowed.
- Models with both 2D and 3D element types or with symmetry boundary constraints are not recommended. Loads may be input as usual. Displacements and stresses are calculated as usual.
- Symmetry models are not valid for an inertia relief analysis.
- If the **Inertial Relief** property is set to **On**, then any analysis linked to the Static Structural analysis is invalid. This includes a Static Structural analysis linked to the following analysis types:
  - Pre-stressed Modal
  - Eigenvalue Buckling
  - Pre-stressed FULL Harmonic Response
  - Pre-stressed MSUP Harmonic Response
  - Pre-stressed MSUP Transient
  - Pre-stressed Response Spectrum
  - Pre-stressed Random Vibration

Include Negative Load Multiplier (Eigenvalue Buckling Analysis Only)

For an Eigenvalue Buckling Analysis, this property allows you to select the range of eigenvalues (load multipliers) to be extracted. The default option **Program Controlled** and the option **Yes** will extract both the negative and positive eigenvalues (load multipliers). The **No** setting extracts only positive eigenvalues (load multipliers).

Quasi-Static Solution

The **Quasi-Static Solution** property is only available for **Coupled Field Static** and **Static Structural** analyses. Options for this property include **On** and **Off** (default). For analyses that are quasi-static in nature, and that fail to converge, you can set this property to **On** in order to achieve convergence. See the Application (App.) Based Settings (p. 1435) property description for additional information about this option.

Time Integration Type (Rigid Dynamics Solver Only)

This feature is applicable to a **Rigid Dynamics** analysis.

The Details view **Solver Controls** options for the **Time Integration Type** include:

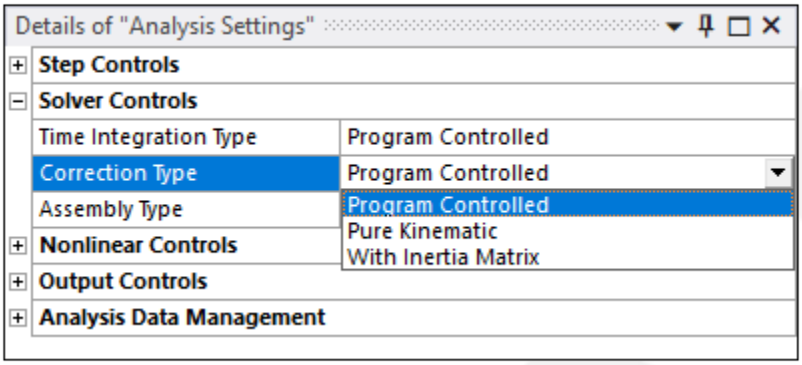
- **Time Integration Type** field. Available time integration algorithms include:
  - **Program Controlled** (default setting): The application selects the most appropriate method based on the current model. If the model contains only rigid bodies, 4th order Runge-Kutta is used. If the model contains flexible bodies (Condensed Parts), the Stabilized Generalized Alpha option is selected automatically.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

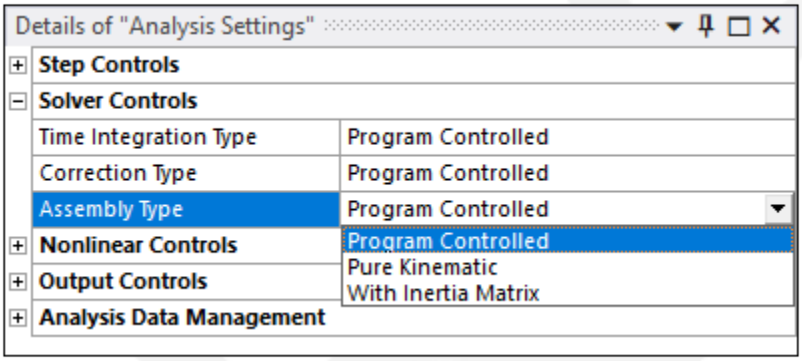
Published: 2023-10-23

  - **Runge-Kutta 4**: Fourth Order Runge-Kutta.
  - **Implicit Generalized Alpha**: Implicit time integration based on the Generalized-α method.
  - **Stabilized Generalized Alpha**: Implicit time integration based on the Stabilized Generalized-α method.
  - **MJ Time Stepping**: Moreau-Jean time-stepping integration.

- **Correction Type**: **Program Controlled** (default), **Pure Kinematic**, or **With Inertia Matrix**.



- **Assembly Type**: **Program Controlled** (default), **Pure Kinematic**, or **With Inertia Matrix**.



Restart Analysis

Note:

- This group is displayed in the Details view only if restart points (p. 1815) are available. Restart points can be generated by adjusting the settings in the **Restart Controls** (p. 1438) group. You will also need to set **Delete Unneeded Files**, under the **Analysis Data Management** (p. 1484) group to **No** so that restart point files are retained after a solve.
- When you use the **Specify** option of the **Load Step** property to specify a load step value during a Static Structural analysis, the **Program Controlled** setting of the **Auto Time Stepping** property (p. 1418) for the specified load step will not set any sub-step or time increment information. In this scenario, the application uses the sub-steps or time increment value specified for the solution not including restart.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The ability to restart a solution is not supported if you switch from a shared memory solution to a Distributed Solution (p. 1804).

The **Restart Analysis** group is available for the following analysis types:

- Static Structural (p. 652)
- Transient Structural (p. 665)
- Coupled Field Analyses (p. 376)

These control whether to use restart points in subsequent solution restarts. If restart points should be used, **Load Step**, **Substep** and **Time** help reveal the point's identity in the calculation sequence.

Note:

When using a modal system database from a version prior to the most current version of Mechanical, it is possible to encounter incompatibility of the `file.esav`, created by a linked static structural system. This incompatibility can cause the modal system's solution to fail. In the event you experience this issue, use the **Clear Generated Data** feature and resolve the static structural system.

The **Restart Analysis** category properties include:

- **Restart Type**: By default, Mechanical tracks the state of restart points and selects the most appropriate point when set to **Program Controlled**. You may choose different restart points by setting this to **Manual**, however. To disable solution restarts altogether, set it to **Off**.
- **Current Restart Point**: This option lets you choose which restart point to use. This option is displayed only if the **Restart Type** property is set to **Manual**.
- **Load Step**: Displays the **Load Step** of the restart point to use. If no restart points are available (or all are invalid for a **Restart Type** of **Program Controlled**) the display is **Initial**.
- **Substep**: Displays the **Substep** of the restart point to use. If no restart points are available (or all are invalid for a **Restart Type** of **Program Controlled**) the display is **Initial**.
- **Time**: Displays the time of the restart point to use.

Restart Controls

The properties of this category control the creation of **Restart Points** and enable you to manage the files that the solver writes for each restart point. See the Solution Restarts (p. 1815) section for more information about the restart capability and how it relates to Restart Points.

The **Restart Controls** category properties include:

- **Generate Restart Points**: Enables the creation of restart points.

  Program Controlled: Instructs the program to select restart point generation settings for you. This setting is equivalent to setting the **Load Step** and the **Substep** properties below to **Last**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Manual: Enables access to the detailed settings for restart point generation.

Off: Restricts any new restart points from being created.

- **Load Step**: This property enables you to specify the load step used to create restart points. Options include **Last**, **All** (default), and **Specify**[1].

  The **All** option obtains restart points for all load steps. The **Last** option to obtains a restart point in the last load step only. And the **Specify** option enables you to enter the desired **Load Step** value in the **Load Step Number** property in order to generate restart points for the specific load step.

  Note:

  Do **not** restart an analysis from the end of a load step and then select the **Specify** option to save the restart points for the following load step. For example, do not restart an analysis from the end of a load step 2 and set the **Specify** option to **Load Step 3**.

- **Substep**: Specifies how often the restart points are created within a load step. Options include:

  Last (default): Write the file for the last substep of the load step only.

  All: Write the files for all substeps of the load step.

  Specified Recurrence Rate: Enter a number N, in the **Value** field, to generate restart points for a specified number of substeps per load step.

  Equally Spaced Points: Enter a number N, in the **Value** field, to generate restart points at equally spaced time intervals within a load step.

- **Maximum Points to Save Per Step**: Specifies the maximum number of files to save for the load step. Choose one of the following options:

  An entry of **0** specifies the option **All**. The maximum number of files for one run is 999. If this number is reached before the analysis is complete, the analysis continues but the application no longer writes any files, except if you are using the Nonlinear Adaptive Region (p. 1707) condition. When this condition is specified, the application begins to overwrite the files, beginning with the oldest restart file, when this maximum is met.

  Enter a positive number to specify the maximum number of files to keep for each load step. When the maximum number has been written for each load step, the first file of that load step will be overwritten for subsequent substeps.

  Note:

  If you want to interrupt the solution in a linear transient analysis, by default, the interrupt will be at load step boundaries only (as opposed to nonlinear analyses where interrupts occur at substeps). However, if you want to interrupt a solution to a linear transient analysis on a substep basis, set the following: **Generate Restart Controls** = **Manual**,

[1] Not supported for Coupled Field Static analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Load Step = **All**, **Substep** = **All**, and **Maximum Points to Save Per Step** = 1. These settings allow you to accomplish the interrupt on a substep basis without filling up your disk with restart files.

- **Retain Files After Full Solve**: When restart points are requested, the necessary restart files are always retained for an incomplete solve due to a convergence failure or user request. However, when the solve completes successfully, you have the option to request to either keep the restart points by setting this field to **Yes**, or to delete them by setting this field to **No**. You can control this setting here in the Details view of the **Analysis Settings** object, or under **Tools> Options** in the **Analysis Settings and Solution** (p. 238) preferences list. The setting in the Details view overrides the preference setting.

  Note:

  Retain Files After Full Solve has interactions with other controls. Under the Analysis Data Management (p. 1484) category, setting the **Future Analysis** property to **Prestressed** forces the restart files to be retained. Similarly, setting the **Delete Unneeded Files** property to **No** implies that restart files are to be retained.

- **Combine Restart Files**: This property enables you to select whether the multi-frame restart files will be automatically combined after the solution. Setting this property to Yes, prior to initiating a solve, enables you to restart your analysis with a different core count of processors. Options include:

  Program Controlled (default): this option does not send any commands to the solver. It uses the Mechanical APDL solver default setting (No).

  Yes: this option issues the command **DMPOPTION, RNNN, YES.** This is the required setting if you wish to combine result files in a downstream system.

  No: this option issues the command **DMPOPTION, RNNN, NO**.

  Note:

  The **Combine Restart Files** property requires significant processing time if there is a large number of restart files to combine.

Adaptivity Remeshing Controls

The **Adaptivity Remeshing Controls** category is included in the Analysis Settings (p. 1417) for Static Structural (p. 652) analyses and is only available when you have inserted **and** specified one of the following objects:

- Nonlinear Adaptive Region (p. 1707)
- Geometry based Adaptivity (p. 1707)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

All category properties are described below. The availability of properties depends upon how you specify the condition.

Important:

All properties, except for the **Refinement Algorithm** and **Minimum Element Size** properties, are step dependent and can therefore be modified on a per step basis using the Step Controls (p. 1418) property **Current Step Number**.

Refinement Algorithm

This property is visible when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to **Energy** or **Box**. The options for this property include **General Remeshing** (default) and **Mesh Splitting**. When you specify the **Mesh Splitting** option, the application hides all of the other **Adaptivity Remeshing Controls** properties, unless there is another **Nonlinear Adaptive Region** object with **Criterion** property set to **Mesh**.

Note:

The **General Remeshing** option is not supported for high order 2D elements and the **Mesh Splitting** option is not supported for high order tetrahedral elements.

Remeshing Gradient

This property is visible when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to **Mesh**, or when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to either **Energy** or **Box** and the **Refinement Algorithm** property to **General Remeshing**. This property controls the new mesh sizing gradient during remesh. The options for this property include:

- **No Gradient**: This option does not maintain the mesh-size gradient but generates a uniform mesh.
- **Average Gradient**: This option generates a new mesh and retains the same local size as the source mesh.
- **Perfect Shape Gradient** (default for 2D analysis): This option generates a new mesh while retaining the same local size as the source mesh. It also provides additional compensation for mesh sizing in the event the size of existing elements changes during the solution process as a result of large deformations. The option strives to refine your mesh at the location of distorted elements. Distorted elements generally originate from deformation during the solution process.
- **Practical Shape Gradient** (default for 3D analysis): This option is only available for 3D analysis. This option is similar to the **Perfect Shape Gradient** option, but it assumes that perfect mesh quality is not required. Minor distortions are not refined.

See the Local Sizing and Gradient Control section of the *Mechanical APDL Advanced Analysis Guide* for additional information. In particular, how these options correspond to the Value (**VAL**) entries.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Minimum Element Size

For the **Geometry Based Adaptivity** condition, you use this property to further filter elements based on size, in addition to the elements already included in the remesh at the particular substep. The default value for this property is the same as the value of the Defeature Size property of the Mesh object, whether user-defined or application generated.

Review the **NLMESH** section of the *Mechanical APDL Command Reference* for more information.

Boundary Angle

This property is visible when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to **Mesh**, or when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to either **Energy** or **Box** and the **Refinement Algorithm** property to **General Remeshing**. It defines the boundary angle threshold in degrees. You use this option to retain the source mesh geometry features. It is valid for both 2D and 3D analyses. The default value is **15**° for 3D analyses and **10**° for 2D analyses.

In a 3D analysis, this value is the dihedral angle (the angle between the normal vectors from two neighboring surface facets). In a 2D analysis, this value is the 2D patch boundary edge normal angle. If the edge angle or dihedral angle is larger than the specified threshold, the node shared by 2D edges or edges shared by 3D facets are retained during remeshing.

See the Boundary-Angle and Edge-Angle Control section of the *Mechanical APDL Advanced Analysis Guide* for additional information.

Edge Splitting Angle

For *3D analyses only*, this property is visible when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to **Mesh**, or when you set the **Criterion** property of the **Nonlinear Adaptive Region** object to either **Energy** or **Box** and the **Refinement Algorithm** property to **General Remeshing**. It defines the "edge angle threshold" to split patch segment. The edge angle is defined as angle between adjacency surface segment edges sharing a node. If the edge angle is larger than specified edge angle threshold, the segment will be split and the node will be automatically treated as hard node to be retained. The default edge angle threshold is **10.0**°. The larger edge angle threshold tends to improve result mesh quality, but may loss feature nodes. The entry range is **0**° to **80**°.

See the Boundary-Angle and Edge-Angle Control section of the *Mechanical APDL Advanced Analysis Guide* for additional information.

Number of Sculpted Layers

This read-only property, in combination with its sub-properties, defines the number of sculpting layers, beginning with the detected seed elements. It is available and valid for both 2D and 3D analyses. This property is specified, based on the **Criterion** property setting of the Nonlinear Adaptive Region (p. 1707) object and the value of one, or both, of the following properties depending upon whether you have more than one **Nonlinear Adaptive Region** object specified:

- **Quality Improvement (NSL)**: Controls remeshing to remove distortion. Visible when the **Criterion** property is set to **Mesh**. The default value for this property is **2** for 3D analyses and **10** for 2D analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Refinement (NSL)**: Controls element refinement. Visible when the **Criterion** property is set to **Energy** or **Box**. The default value for this property is **2** for 3D analyses and **1** for 2D analyses.

This option helps to detect the remeshing regions from whole model. Generally, a larger value leads to larger remeshing regions and tends to unite multiple isolated regions. A larger value also tends to create a better remesh quality (and increases mapping and solution overhead accordingly). Value of **0** is not recommended because the remeshed regions would contain only detected seed elements, resulting in many small cavities within the remeshed regions (especially if the specified skewness threshold [**NLADAPTIVE**] is relatively large).

See the Sculpting Layers Control section of the *Mechanical APDL Advanced Analysis Guide* for additional information.

Global Size Ratio

This read-only property, in combination with its sub-properties, defines the global sizing ratio for remeshing. This property is specified, based on the **Criterion** property setting of the Nonlinear Adaptive Region (p. 1707) object and the value of one, or both, of the following properties depending upon whether you have more than one **Nonlinear Adaptive Region** object specified:

- **Quality Improvement (GSR)**: Controls remeshing to remove distortion. Visible when the **Criterion** property is set to **Mesh**. The default value is **1.0**. The default value results in the new mesh having a size similar to that of the original mesh.

- **Refinement (GSR)**: Controls element refinement. Visible when the **Criterion** property is set to **Energy** or **Box**. The default value is **0.75**.

These entries enable the target mesh to have the same average as the source mesh. The target mesh will get finer or become more coarse if the value is less than **1.0** or larger than **1.0** respectively. Generally, the desired setting greater than or equal to **0.7**.

See the Global Sizing Control section of the *Mechanical APDL Advanced Analysis Guide* for additional information.

Remeshing Tolerance

This read-only property, in combination with its sub-properties, defines the tolerance for accepting new mesh. This property is specified, based on the **Criterion** property setting of the Nonlinear Adaptive Region (p. 1707) object and the value of one, or both, of the following properties depending upon whether you have more than one **Nonlinear Adaptive Region** object specified:

- **Quality Improvement (RT)**: Controls remeshing to remove distortion. Visible when the **Criterion** property is set to **Mesh**. The default value is **0.05**.

- **Refinement (RT)**: Controls element refinement. Visible when the **Criterion** property is set to **Energy** or **Box**. The default value is **0.5**.

Note:

Review the **NLMESH** section of the *Mechanical APDL Command Reference* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Aggressive Remeshing

This property helps to create a mesh with improved shape metrics. Note that it may change certain global remeshing control parameters and as a result may increase remeshing time. Options for this property include **On** and **Off** (default). This property is visible when the:

- **Criterion** property is set to **Mesh**.

  Or...

- **Criterion** property is set to either **Energy** or **Box** *and* the **Refinement Algorithm** property is set to **General Remeshing** (default).

Generate Restart Points

This property enables you to create restart points. The options include **Program Controlled** (default) and **Manual**. The **Program Controlled** options instructs the application to select restart points for you. when you select the **Manual** setting, the **Load Step** and **Substep** properties display. The **Load Step** property includes the options **Last** (default) and **All**. **Last**. The **Substep** property is read-only and the default setting is also **Last**. This property employs the **RESCONTROL** command.

Retain Files After Full Solve

When you request restart points, the application retains all necessary restart files for an incomplete solve due to a convergence failure or user request. For a successful solution, this property choose whether to keep or delete restart files. Options for this property include **Yes** (default) and **No**.

Mechanical APDL References

Adaptivity Controls are applied using the **NLMESH** command. Control options and commands are listed below.

Remesh Control	MAPDL Command
Refinement Algorithm	**NLMESH**,REFA
Remeshing Gradient	**NLMESH**,GRAD
Minimum Element Size	**NLMESH**,ELSZ,VAL1
Boundary Angle	**NLMESH**,BDRA
Edge Splitting Angle	**NLMESH**,AEDG
Number of Sculpted Layers	**NLMESH**,NLAY
Global Size Ratio	**NLMESH**,SRAT
Remeshing Tolerance	**NLMESH**,QTOL
Aggressive Remeshing	**NLMESH**,AGGR

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Creep Controls

Creep is a rate-dependent material nonlinearity in which the material continues to deform under a constant load. You can perform an implicit creep analysis for a static or transient structural analysis. **Creep Controls** are available in the Details view of the analysis settings for these two environments only after you have selected a creep material for at least one prototype in the analysis.

The **Creep Controls** group is available for the following analysis types:

- Static Structural (p. 652)
- Transient Structural (p. 665)

Creep controls are step-aware, meaning that you are allowed to set different creep controls for different load steps in a multi-step analysis. If there were multiple load steps in the analysis before you chose the creep material, then choosing the creep material will set the **Creep Controls** properties to their default value.

The **Creep Controls** group includes the following properties:

- **Creep Effects**: The default value is **Off** for the first load step and **On** for all the subsequent load steps. You may change it according to your analysis.
- **Creep Limit Ratio** (available only if **Creep Effects** is set to **On**): This property issues the Mechanical APDL CUTCONTROL command with your input value of creep limit ratio. (Refer to the **CUTCONTROL** command description for details). The default value of **Creep Limit Ratio** is 1. You are allowed to pick any non-negative value.

Fracture Controls

Fracture controls make sure that the effect of cracks is included in the solution. The **Fracture Controls** category becomes visible under **Analysis Settings** when a **Fracture** object is inserted via the **Model** object. It is only visible for Static Structural and Transient Structural analyses.

The primary property, **Fracture**, is set to **On** by default to compute fracture parameters. By default, the parameters SIFS, J-Integral, and VCCT are computed. Also when set to **On**, the following additional properties display:

- **SIFS**: The default setting is **Yes**. Set this control to **No** to exclude the SIFS parameter results computation.
- **J-Integral**: The default setting is **Yes**. Set this control to **No** to exclude the J-Integral parameter results computation.
- **Material Force**: The default setting is **No**. Set this control to **Yes** to compute the Material Force parameter results.
- **T-Stress**: The default setting is **No**. Set this control to **Yes** to compute the T-Stress parameter results.
- **C*-Integral**: this property is visible only when a creep material is used and the **Creep Effects** property of the Creep Controls (p. 1445) is set to **On**. The default setting is **Yes**. Set this control to **No** to exclude the computation of the C*-Integral parameter results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Cyclic Controls

The Harmonic Index Range setting within the **Cyclic Controls** category is only used in a Modal (p. 515) analysis that involves cyclic symmetry (p. 1147) to specify the solution ranges for the harmonic index. The setting appears if you have defined a Cyclic Region for this analysis.

- The **Program Controlled** option solves all applicable harmonic indices.
- The **Manual** option exposes additional fields that allow you to specify a range of harmonic indices for solution from the **Minimum** value to the **Maximum** value in steps of the **Interval** value.

Note:

Static Structural (p. 652) cyclic symmetry solutions always use all harmonic indices required for the applied loads.

Radiosity Controls

The **Radiosity Controls** group is available for Coupled Field, Steady-State Thermal, Transient Thermal, and Thermal Electric analyses.

The following settings within the **Radiosity Controls** category are used in conjunction with the Radiation (p. 1609) boundary condition when defining surface-to-surface radiation for thermal related analyses that use the Mechanical APDL solver. These settings are based on the **RADOPT** command in Mechanical APDL.

- **Radiosity Solver**
- **Flux Convergence**
- **Maximum Iteration**
- **Solver Tolerance** (dependent upon the unit of measure)
- **Over Relaxation**

For 3D analyses, selections for the **Radiosity Solver** property includes the **Program Controlled** (that defaults to the **Iterative Jacobi** solver setting), the **Direct** solver, **Iterative Jacobi** solver, and the **Iterative Gauss-Seidel** solver. For 2D analyses, selections are the same, however the **Program Controlled** option defaults to the **Iterative Gauss-Seidel** solver.

View Factors for 3D Geometry

For 3D geometry, the **Hemicube Resolution** setting is also available based on the **HEMIOPT** command in Mechanical APDL. See the View Factor Calculation (3D): Hemicube Method section in the *Mechanical APDL Theory Reference* for further information.

View Factors for 2-D Geometry

For 2–D geometry, the following settings are available and are based on the **V2DOPT** command in Mechanical APDL:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **View Factor Method**
- **Number of Zones**
- **Axisymmetric Divisions**

See the following sections of the Mechanical APDL help for further information on these settings:

- Using the Radiosity Solver Method in the *Thermal Analysis Guide*.
- *Mechanical APDL Theory Reference* sections:
  - Non-Hidden Method
  - Hidden Method
  - View Factors of Axisymmetric Bodies

Options for Analyses

The **Options** category is included in the Details of the **Analysis Settings** object for the following analysis types. Select a link to go to a subsection topic:

- Coupled Field Harmonic, Harmonic Acoustics, and Harmonic Response Options (p. 1447)
- Eigenvalue Buckling Options (p. 1455)
- Modal Acoustics and Modal Options (p. 1455)
- Random Vibration Options (p. 1457)
- Response Spectrum Options (p. 1458)
- Substructure Generation Options (p. 1458)
- Transient Structural (Linked with Modal) Options (p. 1459)

Coupled Field Harmonic, Harmonic Acoustics, and Harmonic Response Options

The **Options** category is available for Coupled Field Harmonic (p. 380), Harmonic Acoustics, and Harmonic Response (p. 496) analyses. However, not all of the properties described below may be applicable for each analysis type.

Go to a property description:

- Frequency Spacing (p. 1448)
- Central Frequency (p. 1448)
- Range Minimum and Range Maximum (Frequency Sweep Range) (p. 1448)
- Solution Intervals (p. 1448)
- User Defined Frequencies (p. 1448)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Solution Method (p. 1449)
- Spin Softening (p. 1454)

Frequency Spacing

This property defines how frequency spacing is performed. Options include: **Linear** (default), **Logarithmic**, **Octave Band**, **1/2 Octave Band**, **1/3 Octave Band**, **1/6 Octave Band**, **1/12 Octave Band**, and **1/24 Octave Band**.

When you select the **Linear** option, harmonic results are reported at uniform frequency intervals for the specified frequency range.

The **Logarithmic** option as well as the **Octave Band** options define the Logarithm Frequency Spacing in a Harmonic Analysis by setting an appropriate *LogOpt* key for the **HARFRQ** command.

Central Frequency

This property displays when you select an **Octave Band** option for the **Frequency Spacing** property. This property requires you to specify the central frequency of octave band.

Range Minimum and Range Maximum (Frequency Sweep Range)

You define a frequency sweep range by defining the **Range Minimum** and **Range Maximum** values when the **Frequency Spacing** property is set to **Linear** or **Logarithmic**. When you set the **Frequency Spacing** property to an *n*th-Octave Band, you must also define a **Central Frequency** value. Based on the **Central Frequency** input, the **Range Minimum** and **Range Maximum** values are calculated and displayed as read-only. This information is useful when requesting a result at a particular frequency.

Solution Intervals

This property defines the number of the solution points between the frequency sweep range. You can request any number of harmonic solutions to be calculated. The solutions are evenly spaced within the specified frequency range, as long as clustering is not active.

For example, if you specify 10 (default) solutions in the range 30 to 40 Hz, the program calculates the response at 31, 32, 33, ..., 39, and 40 Hz. No response is calculated at the lower end of the frequency range.

This property is replaced by the **Cluster Number** property when you are using the **Mode-Superposition** Solution Method and the **Cluster Results** property is set to **Yes**.

User Defined Frequencies

This property enables you to add additional frequency steps to your analysis. These frequency steps are in addition to the steps defined for the **Frequency Spacing** property. Options include: **Off** (default) and **On**. When you specify **Off**, no additional frequency steps are included in the solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Specifying **On** requires you to make entries in the **Tabular Data** window in the **User Defined Frequency Steps** column.

Important:

If your analysis has multiple load steps, this property must be defined for each step.

The application executes the **HARFRQ** command using the *FREQARR* input (one dimensional array) to send data to the solver.

Note:

The **User Defined Frequencies** capability is not supported for the following cases:

- Cyclic Symmetry for a Harmonic Response analysis.
- When the **Solution Method** property is set to **Variational Technology**.

Solution Method

The options of the **Solution Method** property, listed below, vary based on the harmonic-based analysis type. The **Mode Superposition** option is only available for **Harmonic Response** analyses.

- **Program Controlled**: The application selects the best solution method based on the model. Internally, the application chooses either the **Full** or **Variational Technology** solution method.
- **Mode Superposition** (default): See below (p. 1450).
- **Full** (Direct Integration): The **Full** method uses the full system matrices for the solution calculations. It is more thorough but also requires greater processing time and capability.
- **Variational Technology**: Instead of using the full matrices to compute the results, the application computes the solution at the middle of the requested frequency range and then interpolates the system matrices and loading on the entire frequency range to approximate the results across the range.

Note:

This setting is not supported when the **Step Frequency Spacing** property is set to **Order Based**. See the Step Controls for Harmonic Analysis Types (p. 1426) section for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For additional information, see Harmonic Response Analysis Variational Technology Method, and **Variational Technology**, as well as the **HROPT** command in the *Command Reference*.

Note:

The application automatically sets this property to **Mode Superposition** and becomes read-only if you link your **Harmonic Response** analysis to a downstream **Structural Optimization** analysis or if the Future Analysis (p. 1485) property (**Analysis Settings** > **Analysis Data Management**) is set to **Structural Optimization**.

Mode Superposition Method

Mode Superposition is the default method for **Harmonic Response** analyses and generally provides results faster than the **Full** method. Using the **Mode Superposition** method, a modal analysis is first performed to compute the natural frequencies and mode shapes. Then the mode superposition solution is carried out where these mode shapes are combined.

When you specify the **Mode Superposition** option, additional properties become available, as listed below. However, the availability of these properties can differ based on whether the analysis is prestressed as well as based on the settings of other properties. The properties displayed when **Mode Superposition** is selected include:

- Include Residual Vector (p. 1450)
- Cluster Results and Cluster Number (p. 1450)
- Modal Frequency Range (p. 1452)
- On Demand Expansion Option (p. 1452)
- Store Results At All Frequencies (p. 1453)
- Mode Selection Method (p. 1454)

Include Residual Vector

This property is available for a Harmonic Response Analysis Using Linked Modal Analysis System (p. 509) and a stand-alone **MSUP** Harmonic Response Analysis (p. 496) when the **Solution Method** property is set to **Mode Superposition** (default). When set to **On** the application executes the **RESVEC** command and calculates or includes residual vectors. The default setting is **Off**. See the **RESVEC** command documentation in the *Mechanical APDL Command Reference* for additional information.

Cluster Results and Cluster Number

Cluster Results: This property enables the solver to automatically cluster solution points near the structure's natural frequencies ensuring capture of behavior near the peak responses. This results in a smoother, more accurate, response curve.

Cluster Number: This property specifies the number of solutions on each side of a natural frequency. The default is value is **4** (to calculate four solutions). The range of available values is **2** to **20**. The following settings are required to display and define this property:

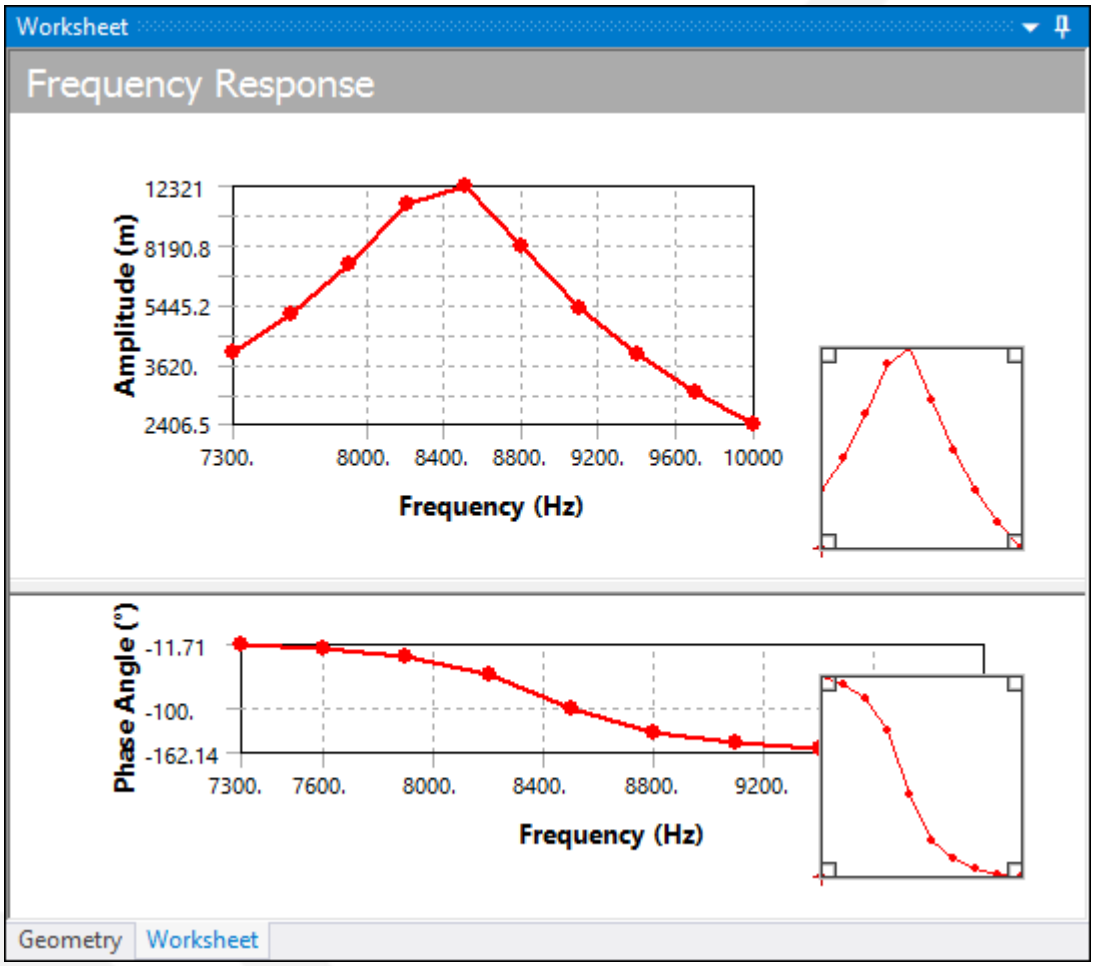
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Solution Method** = **Mode Superposition**
- **Cluster Results** = **Yes**

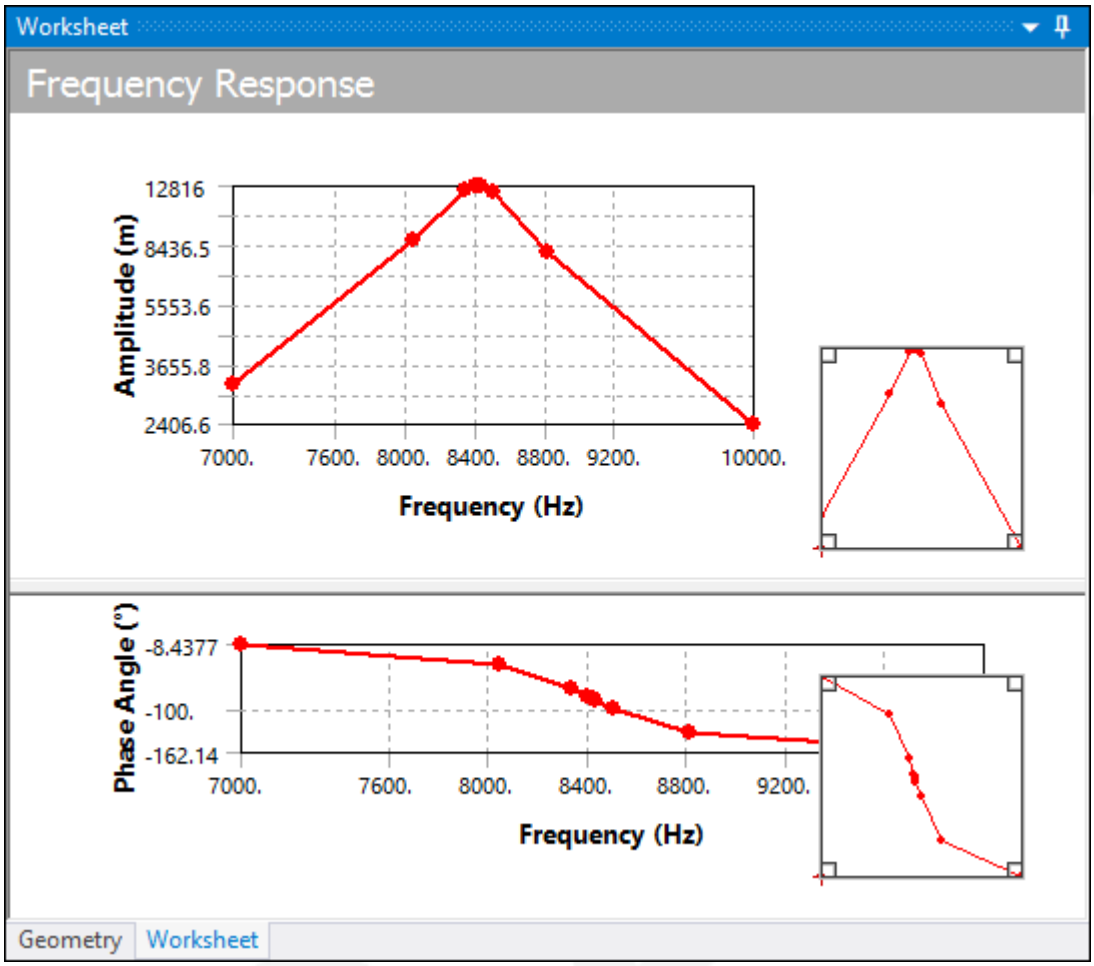
The following example illustrates a comparison of Solution Intervals versus clustering based on the different Solution Methods selections. You may also want to review the Automatic Frequency Spacing in a Harmonic Analysis section for a Harmonic Analysis in the *Mechanical APDL Theory Reference*.

Solution Intervals = 10: Here 10 solutions are evenly spaced within the frequency range. Note how the peak can be missed altogether.



Cluster = 7: Here 7 solutions are performed automatically on either side of each natural frequency capturing the behavior near the peaks.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Modal Frequency Range

Specifies the range of frequencies over which mode shapes will be computed in the modal analysis:

- **Program Controlled**: The modal sweep range is automatically set to 200% of the upper harmonic limit and 50% of the lower harmonic limit. This setting is adequate for most simulations.
- **Manual**: Allows you to manually set the modal sweep range. Choosing **Manual** displays the **Modal Range Minimum** and **Modal Range Maximum** fields where you can specify these values.

On Demand Expansion Option

Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**. The application sets this property as **No** when you request output controls or request fewer frequencies than the number of modes.

When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. That is, the application evaluates the results using the Modal solution data and calculates any other results "on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

demand." This improves solution performance and reduces file size. In addition, the application automatically removes mode shape data from the result file as determined by the **On Demand Mode Shape** preference (see note below).

Furthermore, the application supports the following result types only when this option is active (contour, frequ

- Total and Directional Deformation
- Total and Directional Acceleration
- Total and Directional Velocity
- Maximum, Middle and Minimum Principal Elastic Strain
- Maximum Shear Elastic Strain
- Elastic Strain Intensity
- Normal Elastic Strain/Shear Elastic Strain
- Equivalent Stress
- Maximum, Middle and
- Maximum Shear Stress
- Stress Intensity
- Normal Stress/Shear S
- Force Reaction/Momer

Important:

- **Force Reaction** (p. 2107) and **Moment Reaction** (p. 2107) probes are only supported when scoped to a **Boundary Condition**.
- Only the **Frequency Response** chart options **Deformation**, **Velocity**, and **Acceleration** support Remote Point scoping.
- The **Moment Reaction** probe is not supported on beam or shell bodies or when scoped to Remote Displacements.
- You can change the default setting (**Program Controlled**) of this property using the **On Demand Expansion Option** preference contained in the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.
- Also note that mode shape data is automatically removed from result files. You can change the setting of this process using the **On Demand Mode Shapes** preference, also contained under the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.

Store Results At All Frequencies

Upon solution, harmonic environments store data specified in the Output Controls (p. 1473) for all intervals in the frequency range. Consequently, seeking additional results at new frequencies will no longer force a solved harmonic environment to be resolved. This choice will lead to a better compromise between storage space (results are now stored in binary form in the RST file) and speed (by reducing the need to resort to the solver to supply new results).

If storage is an issue, set the **Store Results At All Frequencies** to **No**. The application retains minimal data with this setting, providing only the harmonic results requested at the time of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

solution. As a result, the **Output Controls** do not control the availability of the results. This option is especially useful with frequency clustering.

Note:

With this option set to **No**, the addition of new frequency or phase responses to a solved environment requires a new solution. Adding a new contour result of any type (stress or strain) or a new probe result of any type (reaction force, reaction moment, or bearing) for the first time on a solved environment requires you to solve, but adding additional contour results or probe results of the same type does not share this requirement; data from the closest available frequency is displayed (the reported frequency is noted on each result). Note that the values of frequency, type of contour results (displacement, stress, or strain) and type of probe results (reaction force, reaction moment, or bearing) at the moment of the solution determine the contents of the result file and the subsequent availability of data. Planning these choices can significantly reduce the need to re-solve an analysis.

Mode Selection Method

This property displays when you are performing a Harmonic Response Analysis Using Linked Modal Analysis System (p. 509). This property enables you to select the modes for the mode expansion. Property options include:

- **None** (default): This option expands all the extracted modes from the modal analysis. All the modes participate in the mode-superposition harmonic response analysis.
- **Modal Effective Mass**: When you select this option, the **Significance Threshold** property also displays. For the **Significance Threshold** property value, the application selects the modes that have a ratio for the Modal Effective Mass to the Total Mass that is greater than this value, in all directions, for expansion. That is, only the selected modes participate in the mode-superposition harmonic response analysis. The application default is **`0.001`**. This feature improves the performance of postprocessing the modal results and reduces the file size.

For more information, see the Mode Selection Based on Modal Effective Mass section of the Mechanical APDL Structural Analysis Guide.

Spin Softening

This property enables you to specify whether to include or exclude Spin Softening effect in the linear perturbation analyses. It is only available for a pre-stressed Harmonic Response analysis when the **Solution Method** is set to **Program Controlled**, **Full**, or **Variational Technology**. The options include **Program Controlled** (default), **Yes**, and **No**. The **Program Controlled** setting selects either to include or exclude Spin Softening in the solution based on whether the rotating reference frame attached to the body is fixed (Yes) or in motion (No). The **Yes** option is invalid if the **Coriolis Effect** property (Rotordynamics Controls (p. 1490)) is set to **On, Stationary Reference Frame**.

For more information, refer to the **PERTURB** command from *MAPDL Command Reference Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Eigenvalue Buckling Analysis Options

For Eigenvalue Buckling (p. 487) analyses, the **Options** category includes the following properties:

- **Max Modes to Find**: You need to specify the number of buckling load factors and corresponding buckling mode shapes of interest. Typically only the first (lowest) buckling load factor is of interest.
- **Keep Pre-Stress Load-Pattern**: This property is available when the upstream structural analysis is nonlinear. This property enables you to keep the loads defined in Static Structural analysis and use them in Eigenvalue Buckling analysis to generate perturbation loads. The default setting is **Yes**. Setting the property to **No** requires you to define a new loading pattern for the Eigenvalue Buckling analysis.

  Note that at least one load is required in the buckling analysis when the prestressed analysis is nonlinear.

Modal Acoustics and Modal Analysis Options

For Modal Acoustics and Modal (p. 515) analyses, the **Options** group includes the following properties:

Max Modes to Find

Specifies the number of natural frequencies to solve for in a modal analysis. The default value is **6**. Set the property to zero (**0**) to specify a value of **All** (find all frequencies). In order to specify this value, you must set the **Limit Search to Range** property to **Yes** and then set the **Solver Type** explicitly to either **Direct**, **Unsymmetric**, or **Subspace** or when the **Program Controlled** option selects one of these solvers.

Limit Search Range

Allows you to specify a frequency range within which to find the natural frequencies. The default is set to **No**. If you set this to **Yes**, you can enter a minimum and maximum frequency value. If you specify a range the solver will strive to extract as many frequencies as possible within the specified range subject to a maximum specified in the **Max Modes to Find** field.

On Demand Expansion Option

Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**.

When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. That is, the application evaluates the results using the Modal solution data and calculates any other results "on demand." This improves solution performance and reduces file size.

Important:

Once the analysis is solved, if you make changes and those changes do not clear the current solution, the application does not reevaluate the setting of the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

property for the **Program Controlled** option (**Yes** or **No**). The previous setting is used.

If the application detects any of the following conditions, it automatically sets the **On Demand Expansion** property to **No**:

- The system is a standalone modal analysis (no downstream systems).
- You specify any Output Controls (p. 1473).
- A downstream system does not support an **On Demand Expansion Option** property setting of **Yes**.
- A downstream system has the On Demand Expansion Option set to either **No** or **Program Controlled** (**No**).
- Damping is present in the analysis.
- Symmetry present in the analysis.

When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application automatically changes the settings of the following properties to improve solution performance and reduces file size.

- The **Store Modal Results** property is set to **No** to reduce mode file size
- Solution files are not recombined when the **Distributed** solution option is active.
- **Expansion Settings** are suppressed because on demand expansion is done "on the fly" to plot displacement results on Condensed Parts.

Note:

- The application automatically removes mode shapes from the result files. You can change the setting of this process using the **On Demand Mode Shapes** preference, also contained under the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.
- You can change the default setting (**Program Controlled**) of this property using the **On Demand Expansion Option** preference contained in the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.

In addition, when the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the property automatically modifies the following properties in any linked downstream Harmonic Response or Transient structural analysis to improve its performance.

- **On Demand Expansion** is set to **Yes**.
- **Expansion Results From** is set to **Harmonic/Transient**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Expansion Settings** are suppressed.

Also note that if you solve the analysis when the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, and you change the setting of the property to **No** manually or as a result of the application setting the **On Demand Expansion** property to **No** because you change a property or a required condition, the application automatically clears the solution data.

Furthermore, there are specific result types (contour, frequency response, and probes) supported when this op

- Total and Directional Deformation
- Maximum, Middle and Minimum Principal Elastic Strain
- Maximum Shear Elastic Strain
- Elastic Strain Intensity
- Normal Elastic Strain/Shear Elastic Strain
- Equivalent Stress
- Maximum, Middle and
- Maximum Shear Stress
- Stress Intensity
- Normal Stress/Shear S

Important:

- Response Spectrum, Random Vibration, and Structural Optimization analyses cannot be linked to an upstream Modal analysis that has the **On Demand Expansion** property set to **Yes**, either manually or from the **Program Controlled** setting.
- Stress and strain cannot be evaluated on Condensed Parts.
- Spring annotations do not update when viewing results.
- You can change the default setting of this property under the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) group in the **Options** dialog.

Spin Softening

Enables you to specify whether to include or exclude Spin Softening effect in the linear perturbation analyses. This field is available when performing a Pre-Stressed Modal analysis. The options include **Program Controlled** (default), **Yes**, and **No**. The **Program Controlled** setting selects either to include or exclude Spin Softening in the solution based on whether the rotating reference frame attached to the body is fixed (Yes) or in motion (No). The **Yes** option is invalid if the **Coriolis Effect** property (Rotordynamics Controls (p. 1490)) is set to **On, Stationary Reference Frame**.

For more information, refer to the **PERTURB** command from *MAPDL Command Reference Guide*.

Random Vibration Options

For Random Vibration (p. 524) analyses, the **Options** category includes the following properties:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Number of Modes to Use

Specifies the number of modes to use from the modal analysis. A conservative rule of thumb is to include modes that cover 1.5 times the maximum frequency in the PSD excitation table.

Exclude Insignificant Modes

When set to **Yes**, this property enables you to not include modes for the mode combination as determined by the threshold value you set by the **Mode Significance Level** property (that automatically displays when **Yes** is selected). The default value of **0** means all modes are selected (same as setting **Exclude Insignificant Modes** to **No**) while a value of **1** means that no modes are selected. The higher you set the threshold, the fewer the modes selected for mode combination.

Response Spectrum Options

For Response Spectrum (p. 533) analyses, the **Options** category includes the following properties:

Number of Modes to Use

Specify the number of modes to use from the modal analysis. It is suggested to have modes that span 1.5 times the maximum frequency defined in input excitation spectrum.

Spectrum Type

Specify either **Single Point** or **Multiple Points**. If two or more input excitation spectrums are defined on the same fixed degree of freedoms, use **Single Point**, otherwise use **Multiple Points**.

Modes Combination Type

Specify a method to be used for response spectrum calculation. Choices are **SRSS**, **CQC**, and **ROSE**. In general, the **SRSS** method is more conservative than the other methods.

The SRSS method assumes that all maximum modal values are uncorrelated. For a complex structural component in three dimensions, it is not uncommon to have modes that are coupled. In this case, the assumption overestimates the responses overall. On the other hand, the CQC and the ROSE methods accommodate the deficiency of the SRSS by providing a means of evaluating modal correlation for the response spectrum analysis. Mathematically, the approach is built upon random vibration theory assuming a finite duration of white noise excitation. The ability to account for the modes coupling makes the response estimate from the CQC and ROSE methods more realistic and closer to the exact time history solution.

Substructure Generation Options

For Substructure Generation (p. 540) analyses, the **Options** category includes the following properties:

Max Modes to Find

Specify the desired number of natural frequencies the solver will use during a substructure generation analysis. The default value is **6**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Limit Search to Range

If desired, specify a frequency range within which to find the natural frequencies. The default is set to **No**. If you set this to **Yes**, the **Range Minimum** and **Range Maximum** properties display. Use these properties to specify the desired frequency range. Note that specifying a range instructs the solver to extract as many frequencies as possible within the specified range. The frequency modes extracted cannot exceed the values specified by the **Max Modes to Find** property.

Number of Load Vectors

Limit the number of load vectors generated during the substructure generation process to improve efficiency and performance. The default setting is **Program Controlled**. This value can be overridden with an integer, it must not be less than the highest load vector number for the loads defined in the analysis..

For more information about this setting, see the **/CONFIG** (**/CONFIG,numsublv**) section of the Mechanical APDL Commands Reference.

Transient Structural (Linked to Modal) Options

For a Transient Structural analysis using a linked Modal (p. 674) system, the **Options** category includes the following properties:

Include Residual Vector

This property is available for a Transient Structural Analysis Using Linked Modal Analysis System (p. 674). When set to **On** the application executes the **RESVEC** command and calculates or includes residual vectors. The default setting is **Off**. See the **RESVEC** command documentation in the *Mechanical APDL Command Reference* for additional information.

On Demand Expansion

This property is available when you are performing a Transient Structural Analysis Using Linked Modal Analysis System (p. 674). Options for this property include **Program Controlled** (default), **Yes**, and **No**. When set to **Program Controlled**, an additional read-only property, **On Demand Expansion**, displays. This property only displays for the **Program Controlled** option and the application determines the value for the property, either **Yes** or **No**. The application sets this property as **No** when you request output controls or request fewer frequencies than the number of modes.

When the **On Demand Expansion Option** property is set to **Yes**, either manually or from the **Program Controlled** setting, the application creates the result file optimally. That is, the application evaluates the results using the Modal solution data and calculates any other results "on demand." This improves solution performance and reduces file size. In addition, the application automatically removes mode shape data from the result file as determined by the **On Demand Mode Shape** preference (see note below).

Furthermore, there are specific result types supported when this option is active:

- Total and Directional Deformation
- Total and Directional Acceleration
- Equivalent Stress
- Maximum, Middle and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Total and Directional Velocity
- Maximum, Middle and Minimum Principal Elastic Strain
- Maximum Shear Elastic Strain
- Elastic Strain Intensity
- Normal Elastic Strain/Shear Elastic Strain
- Maximum Shear Stress
- Stress Intensity
- Normal Stress/Shear S
- Force Reaction/Momer

Important:

- **Force Reaction** (p. 2107) and **Moment Reaction** (p. 2107) probes are only supported when scoped to a **Boundary Condition**.
- The **Moment Reaction** probe is not supported on beam or shell bodies or when scoped to Remote Displacements.
- Loading conditions applied to geometric entities that are scoped to a Reaction probe are not included in the probe's calculation.
- You can change the default setting (**Program Controlled**) of this property using the **On Demand Expansion Option** preference contained in the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.
- Also note that mode shape data is automatically removed from result files. You can change the setting of this process using the **On Demand Mode Shapes** preference, also contained under the **Options (Modal, Harmonic and Transient Mode Superposition)** category of the Analysis Settings and Solution (p. 238) category of the Options (p. 217) dialog.

Mode Selection Method

This property displays when you are performing a Transient Structural Analysis Using Linked Modal Analysis System (p. 674). This property enables you to select the modes for the mode expansion. Property options include:

- **None** (default): This option expands all the extracted modes from the modal analysis. All the modes participate in the mode-superposition transient dynamic analysis.
- **Modal Effective Mass**: When you select this option, the **Significance Threshold** property also displays. For the **Significance Threshold** property value, the application selects the modes that have a ratio for the Modal Effective Mass to the Total Mass that is greater than this value, in all directions, for expansion. That is, only the selected modes participate in the mode-superposition transient dynamic analysis. The application default is **`0.001`**. This feature improves the performance of postprocessing the modal results and reduces the file size.

For more information, see the Mode Selection Based on Modal Effective Mass section of the Mechanical APDL Structural Analysis Guide.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Scattering Controls

The **Scattering Controls** category includes the **Scattered Field Formulation** property. The options for this property include:

- **Program Controlled** (default)
- **Off**: Selecting this option turns scattering controls off.
- **On**: Selecting this option turns scattering controls on and also displays the **Scattering Output Type** property.

  The **Scattering Output Type** property is used to specify the output type for an acoustic scattering analysis. The options for this property include **Total** and **Scattered**. Select the Total option when you wish to output the total pressure field and the **Scattered** option when you want to output the scattered pressure field.

  If you specify an **Incident Wave Location** excitation and set the property to **Outside the Model**, Mechanical automatically sets the **Scattering Output Type** property to **Total**.

  For more information, refer to the **ASOL** and **ASCRES** commands in the *Mechanical APDL Command Reference*.

Advanced

The **Advanced** category includes the following properties.

Far-field Radiation Surface

For a Harmonic Acoustics analysis only, the **Advanced** category includes the property **Far-field Radiation Surface**. Far-field result calculations are based on the Equivalent Source (Maxwell) Surfaces. Therefore, this field controls far-field result definitions and results. The options include:

- **Program Controlled** (default): If your analysis does not include a user-defined Equivalent Source Surface boundary condition object, this setting identifies the Equivalent Source (Maxwell) Surfaces automatically created by the application using the environment option **Create Automatic** > **Far-field Radiation Surface**. In this case, the application applies the surface flag MXWF on them. If the analysis does include a user-defined **Far-field Radiation Surface** object, this settings defined by that object are used.
- **Manual**: This option requires the definition of at least one user-defined **Far-field Radiation Surface** object.
- **No**: This setting invalidates all Equivalent Source (Maxwell) Surface objects and Far-field result objects.

Inverse Option

For Static Structural analyses only, the **Advanced** category includes the property **Inverse Option**. The options for this property include **No** (default) and **Yes**. Selecting the **Yes** option instructs the application to perform an inverse solution beginning at first step. In addition, the **End Step** property displays when you select the **Yes** option. This property enables you to specify at which step the inverse

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

solve routine should stop. Entries for this property (the maximum value) are based on the value specified in the **Number of Steps** property. The default value for this property is **1**.

Contact Split (DMP)

When the **Contact Split (DMP)** property is enabled, the contact conditions for the model are split during the distributed solution to generate several smaller contacts. These contacts are then processed across the available cores. Once the solution is complete, the split pairs are merged. This entire process is automatic. Property options include:

- **Off**: No contact splitting.
- **Program Controlled** (default): The application selects whether to perform contact splitting in order to improve performance.

  Important:

  When using this setting for standalone Static Structural (not linked to any downstream system, such as Modal, etc.) and Transient Structural analyses, the application automatically performs contact pair splitting during the solution.

- **On**: The application splits the contact regions during the solution. Selecting this option displays the additional property **Max Number of Splits**. This entry specifies the maximum number of splits that can occur for each contact region during the distributed solution. A minimum value of 2 is acceptable.

  Note:

  - This feature requires the **Distributed** (p. 1792) option to be enabled.
  - When contact splitting is enabled, different contacts in the model can result in different levels of splitting. The level of splitting is determined by the application during run time.
  - The use of this property may trim some of the contact elements during the solution. This reduces the number of contact elements compared to the original untrimmed contact condition. As a result, contact result items such as Contact Force Reactions and Contact Trackers may provide different results when you turn the **Contact Split (DMP)** property **On** compared to a solution with the property turned **Off**.
  - You can change the default setting of this property using the **Contact Split (DMP)** preference contained in the **Advanced** category of the Analysis Settings and Solution (p. 238) group of the **Options** dialog.

Damping Controls

The properties of the **Damping Controls** category vary based on the type of analysis being performed. Using these properties, you can define the following types of system damping:

- Damping Ratio (**DMPRAT**)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Constant Global Structural Damping Coefficient (**DMPSTR**)
- Global Alpha and Beta Damping (Rayleigh Damping) (**ALPHAD**, **BETAD**)

See the Damping section of the *Mechanical APDLStructural Analysis Guide* for more information.

Supported Analysis Types

The **Damping** category is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Modal (p. 386)
- Coupled Field Transient (p. 395)
- Harmonic Response (p. 496)
- Modal (p. 515)
- Random Vibration (p. 524)/Response Spectrum (p. 533)
- Transient Structural (p. 665)
- Transient Structural Analysis Using Linked Modal Analysis System (p. 674)
- Rigid Body Dynamics (p. 550)

Property Descriptions

Mechanical supports the following system-level damping properties.

Eqv. Damping Ratio From Modal

This **Damping Controls** property is available for a Harmonic Response analysis when the **Solution Method** property is set to **Mode Superposition** (MSUP) and for a Transient Structural analysis linked to a Modal analysis. For these analyses, if the upstream Modal analysis Solver Type is undamped and you define the **Damping Ratio** in the Material Dependent Damping property grouping of Engineering Data, then this property can control the material-based **Damping Ratio** effect in your MSUP Harmonic and MSUP Transient solutions for all options of **Expand Results From** (p. 1477) property. The options for this property include:

- **Yes**: The application includes the material-based **Damping Ratio** (**MP,DMPR**) effect in your MSUP Harmonic or MSUP Transient solution.

Important:

For Reaction Force (p. 2107) calculations, results differ when considering material-based Damping Ratio (**MP,DMPR**) when you have the **Expand From** property set to either **Harmonic Solution** or **Transient Solution**, compared to when you have it set to **Modal Solution**. This difference is a result of the way the element damping nodal loads are calculated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifically, when you set the **Expand From** property to **Harmonic Solution** or **Transient Solution**, the solver considers the **Material Damping Coefficient** as a **Constant Structural Damping Coefficient**. If you refer to *MAPDL Theory Reference*, the element damping matrix of Equation 14-101 is calculated on Equation (14-39) for the **Harmonic Solution** setting and Equation 14-37 for **Transient Solution** setting.

When you set the **Expand From** property to the **Modal Solution** setting, the element damping matrix is obtained by combining the element damping nodal loads written to the mode file with coefficients obtained from Equation (14-42). Those results should be considered as the expected ones when any material-based **Damping Ratio** is defined.

- **No**: The application **does not** include the material-based **Damping Ratio** (**MP,DMPR**) effect in your MSUP Harmonic or MSUP Transient solution.

Important:

The application automatically sets this property to **No** and it becomes read-only if you link a **Harmonic Response** analysis to a downstream **Structural Optimization** analysis or if the Future Analysis (p. 1485) property (**Analysis Settings** > **Analysis Data Management**) is set to **Structural Optimization**.

Note:

- If you define the **Damping Ratio** in the Material Dependent Damping property grouping, the application automatically sets this property to **Yes**.
- When you have a Mode Superposition Harmonic Response analysis that is linked to an upstream Modal analysis and you set the **Damped** property in the Modal analysis to **Yes**, the application automatically hides the **Eqv. Damping Ratio From Modal** property in the harmonic system.

Constant Damping

This property is available for Random Vibration analyses. The default setting is **Program Controlled**. You may also set the property to **Manual**.

Damping Define By

For a Harmonic Response analysis when the **Solution Method** property is set to **Mode Superposition**, this property enables you to specify damping using a **Damping Ratio** (default) or a **Constant**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Structural Damping Coefficient. Based on your selection, an associated property of the same name displays. See the descriptions below.

Important:

The application automatically makes this property read-only if you link a **Harmonic Response** analysis to a downstream **Structural Optimization** analysis or if the Future Analysis (p. 1485) property (**Analysis Settings** > **Analysis Data Management**) is set to **Structural Optimization**.

Damping Ratio

This property specifies the amount of damping in the structure as a percentage of critical damping using the **DMPRAT** command. Note the following conditions:

- If you set this property in conjunction with the **Stiffness Coefficient** and **Mass Coefficient**, the effects are cumulative.
- For a Random Vibration analysis, this property defaults to 0.01 (1%). Set the **Constant Damping** property to **Manual** to specify the value.

Note:

The Engineering Data workspace also includes a Damping Ratio property that you can specify for a material. Mechanical supports material-based damping in addition to damping specified in the application. See the Material Dependent Damping Definition (p. 329) topic in the *Define Engineering Data* section for a listing of the analysis types, and their requirements, that support material-based damping.

Constant Structural Damping Coefficient

This property specifies the amount of constant structural damping data using the **DMPSTR** command. The property is available for:

- Mode Superposition-based Harmonic Response analyses.
- Modal analysis when the:
  - **Damped** property is set to **Yes** and the **Solver Type** is set to either **Program Controlled** or **Full Damped**.
  - **Damped** property is set to **Yes**, the **Solver Type** is set to **Reduced Damped**, and the **Store Complex Solution** property is set to **Yes**.
  - **Damped** property is set to **No** and the **Solver Type** is set to **Unsymmetric**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– **Damped** property is set to **No** and the **Solver Type** is set to **Program Controlled** where the application specified the **Unsymmetric** setting.

Note:

If you parameterize the **Constant Structural Damping Coefficient** property using the **Program Controlled** option, based on the Solver Type selection, damping is added or removed during the design point solution.

If you set this in conjunction with the **Stiffness Coefficient** and **Mass Coefficient**, the effects are cumulative.

Note:

The Engineering Data workspace also includes a Constant Structural Damping Coefficient property that you can specify for a material. Mechanical supports material-based damping in addition to damping specified in the application. See the Material Dependent Damping Definition (p. 329) topic in the *Define Engineering Data* section for a listing of the analysis types, and their requirements, that support material-based damping.

Stiffness Coefficient Defined By

Define the Stiffness Coefficient by entering a value, **Direct**, or by entering a Frequency and a Damping Ratio, **Damping vs. Frequency**.

Stiffness Coefficient (Beta Damping, β)

A coefficient value that is used to define a Beta damping by multiplying it with stiffness. You can enter the value directly or the value can be computed from a damping ratio at a specified frequency. You define a Stiffness Coefficient in the Details view of the **Analysis Settings** object.

Beta Damping can also be specified in Engineering Data. Refer to the **BETAD** command section in the *Mechanical APDL Command Reference* for more information about the Beta Damping Factor.

Note:

The unit for the **Stiffness Coefficient** (Beta Damping) property is seconds. However, the interface in Mechanical as well as Engineering Data displays the value **without** a unit.

Frequency: This property is visible when **Stiffness Coefficient Defined By** is set to **Damping vs. Frequency**. Enter a desired value.

Damping Ratio: This property is visible when **Stiffness Coefficient Defined By** is set to **Damping vs. Frequency**. Enter a desired value. The value of β is not generally known directly, but is calculated from the modal damping ratio, ξi. ξi is the ratio of actual damping to critical damping for a particular mode of vibration, i. If ωi is the natural circular frequency, then the beta damping is related to the damping ratio as $\beta = 2\ \xi i/\omega i$. Only one value of β can be input in a step, so choose the most dominant frequency active in that step to calculate β.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mass Coefficient (Alpha Damping Factor, α)

A coefficient that is used to define an Alpha damping by multiplying it with mass. Beta and Alpha damping factors are collectively called Rayleigh damping.

The Alpha Damping can also be specified in Engineering Data. Refer to the **ALPHAD** command in the *Mechanical APDL Command Reference* for more information about the Alpha Damping Factor.

Note:

The unit for the **Mass Coefficient** (Alpha Damping Factor) property is 1/seconds. However, the interface in Mechanical as well as Engineering Data displays the value **without** a unit.

Numerical Damping

This option is available for a Transient Structural analysis using a linked Modal analysis system. Numerical Damping is also referred to as amplitude decay factor (γ). This property controls the numerical noise produced by the higher frequencies of a structure. Usually the contributions of these high frequency modes are not accurate and some numerical damping is preferable. Options for this property include **Program Controlled** (default) and **Manual**.

The property is accompanied by the **Numerical Damping Value** property. The default value for this property is 0.005 and is read-only when the **Numerical Damping Value** is set to **Program Controlled** setting and can be modified when you use the **Manual** option.

Note:

For Full Transient Structural analysis, you can specify Numerical Damping manually using the **User Defined** option of the **App. Based Settings** (p. 1435) property. The **User Defined** option provides the associated property **Amplitude Decay Factor**. You use this property to specify a Numerical Damping Value.

Material Damping

There are two types of material-based damping, Material Dependent Damping and Constant Damping Coefficient. Material Dependent Damping consists of beta damping and alpha damping. These are defined as material properties in Engineering Data.

Element Damping

Spring damping and Bearing damping are defined in the Details view of the Spring (p. 1395) object and Bearing (p. 1413) object.

Rigid Dynamics Analysis Damping

Numerical Damping Control: (Only available with Implicit Generalized-α time integration.) This option allows you to control the noise produced by high frequencies. When the numerical damping control is enabled, you can directly input the value of the `\rho_inf` coefficient (refer to Implicit Generalized-α Method (p. 639)). The value must be between 0 and 1. The default value is 0.99, meaning no numerical damping. A smaller value reduces the noise produced by high frequencies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You can specify more than one form of damping in a model. In addition to structural damping and material damping, the model can have damping from spring and bearing connection, namely Element Damping (see above). The application formulates the damping matrix as the sum of all the specified forms of damping.

You can specify a **Material** for the spring that includes a constant damping coefficient. Based on the analysis type, the application applies damping as structural damping for damped Modal and Full Harmonic Response systems and as viscous damping for MSUP systems.

Note:

Restrictions of applying damping in each analysis type can be found in Damping section of the Mechanical APDL Structural Analysis Guide.

Nonlinear Controls

This section describes the properties provided by **Nonlinear Controls** category. The properties of this category vary based on analysis type. The subsections listed here describe the **Nonlinear Controls** properties for each supported analysis type.

- Nonlinear Controls for Steady-State, Static, and Transient Analyses (p. 1468)
- Nonlinear Controls for Transient Thermal Analyses (p. 1472)
- Nonlinear Controls for Rigid Dynamics Analyses (p. 1472)

Nonlinear Controls for Steady-State, Static, and Transient Analyses

This section examines the **Nonlinear Controls** as they apply to the following analysis types:

- Coupled Field Static and Transient (p. 376)
- Electric (p. 403)/Magnetostatic (p. 546)/Thermal-Electric (p. 661)
- Steady-State Thermal (p. 657)
- Static Structural (p. 652)/Transient Structural (p. 665)

Go to a property description:

- Newton-Raphson Option (p. 1469)
- Convergence Criterion (p. 1469)
- Line Search (p. 1470)
- Stabilization (p. 1470)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Newton-Raphson Option

For *nonlinear* Static Structural and Full Transient Structural analysis types, the **Newton-Raphson Option** property is available. This property allows you to specify how often the stiffness matrix is updated during the solution process. **Newton-Raphson Option** property options include:

- **Program Controlled** (default setting)
- **Full**
- **Modified**
- **Unsymmetric**

The **Program Controlled** option allows the program to select the **Newton-Raphson Option** setting based on the nonlinearities present in your model. For more information about the additional options, see the Newton-Raphson Option section in the *Mechanical APDL Structural Analysis Guide*. If you experience convergence difficulties, switching to an **Unsymmetric** solver may aid in Convergence.

Convergence Criterion

When solving nonlinear steady-state, static, or transient analyses, an iterative procedure (equilibrium iterations) is carried out at each substep. Successful solution is indicated when the out-of-balance loads are less than the specified convergence criteria. Criteria appropriate for the analysis type and physics are displayed in this grouping. Convergence controls are "step aware". This means that the setting can be different for each step.

The following convergence criteria properties are available:

- Coupled Field Static and Coupled Field Transient:

  If Structural physics is enabled: **Force Convergence**, **Moment Convergence**, **Displacement Convergence**, and **Rotation Convergence**.

  If Thermal physics is enabled: **Heat Convergence** and **Temperature Convergence**.

  If Electric physics (Charge) is enabled: **Voltage Convergence** and **Charge Convergence**.

  If Electric physics (Conduction) is enabled **Voltage Convergence** and **Current Convergence**.

- Electric analysis: **Voltage Convergence** and **Current Convergence**.
- Magnetostatic analysis: **CSG Convergence** and **AMP Convergence**.
- Static Structural analysis and Transient Structural analysis: **Force Convergence**, **Moment Convergence**, **Displacement Convergence**, and **Rotation Convergence**.
- Steady-State Thermal analysis: **Heat Convergence** and **Temperature Convergence**.
- Thermal-Electric analysis: **Heat Convergence**, **Temperature Convergence**, **Voltage Convergence**, and **Current Convergence**.

The following convergence controls are available for each of these properties:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Program Controlled** (default setting): The application sets the convergence criteria. See the **STABILIZE** command reference for application defaults settings for Unstable structures as well as Semi-Implicit Solution Controls.
- **On**: You specify that a convergence criterion is activated. Once activated, additional properties become available and include:
  - **Value**: This is the reference value that the solver uses to establish convergence. The recommended and program controlled setting, **Calculated by solver**, automatically calculates the value based on external forces, including reactions, or you can input a constant value.

    When **Temperature Convergence** is set to **On**, the **Value** field provides a drop-down menu with the options **Calculated by solver** or **User Input**. Selecting **User Input** displays an **Input Value** field you use to enter a value.

    When *any other* convergence property is set to **On**, selecting the **Calculated by solver** field enables you to manually enter a value.
  - **Tolerance** times **Value** determines the convergence criterion
  - **Minimum Reference**: This is useful for analyses where the external forces tend to zero. This can happen, for example, with free thermal expansion where rigid body motion is prevented. In these cases the larger of **Value** or **Minimum Reference** will be used as the reference value.

Note:

If you do not want any convergence options to be turned on, then you may try setting the solution controls to off, using a Commands (ADPL) object.

Line Search

Line search can be useful for enhancing convergence, but it can be expensive (especially with plasticity). You might consider setting **Line Search** on in the following cases:

- When your structure is force-loaded (as opposed to displacement-controlled).
- If you are analyzing a "flimsy" structure which exhibits increasing stiffness (such as a fishing pole).
- If you notice (from the program output messages) oscillatory convergence patterns.

Note:

The Line Search control is "step aware" and can be different for each step.

Stabilization

Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

There are three Keys for controlling nonlinear stabilization:

- **Program Controlled** (default): The application does not issue any request to the solver to activate stabilization.
- **Off**: Deactivate stabilization.
- **Constant**: Activate stabilization. The energy dissipation ratio or damping factor remains constant during the load step.
- **Reduce**: Activate stabilization. The energy dissipation ratio or damping factor is reduced linearly to zero at the end of the load step from the specified or calculated value.

There are two options for the **Method** property for stabilization control:

- **Energy**: Use the energy dissipation ratio as the control (default setting).
- **Damping**: Use the damping factor as the control.

When **Energy** is specified, an **Energy Dissipation Ratio** must be entered. The energy dissipation ratio is the ratio of work done by stabilization forces to element potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

When **Damping** is specified, a **Damping Factor** value needs to be entered. The damping factor is the value that the Mechanical APDL solver uses to calculate stabilization forces for all subsequent substeps. This value is greater than 0.

Note:

The **Damping Factor** value is dependent on the active unit system and may influence the results if unit systems are changed. You may wish to use an initial trial value from a previous run for this entry (such as a run with the Energy Dissipation Ratio as input). See the Controlling the Stabilization Force section of the *Mechanical APDL Structural Analysis Guide* for additional information.

There are three options for Activation For First Substep control:

- **No**: Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached (default setting).
- **On Nonconvergence**: Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached. Use this option for the first load step only.
- **Yes**: Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step Key = **Constant**.

For **Stabilization Force Limit**, a number between 0 and 1 should be specified. The default value is 0.2. To omit a stabilization force check, set this value to 0.

Refer to Unstable Structures in the *Structural Analysis Guide* for assistance with using the stabilization options listed above.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Nonlinear Controls for Transient Thermal Analyses

Nonlinear Formulation

The **Nonlinear Formulation** category controls how nonlinearities are to be handled for the solution. The following options are available:

- **Program Controlled** (default): Mechanical automatically specifies either the **Full** or **Quasi** setting as described below. The **Quasi** setting is based on a default **Reformulation Tolerance** of 5%. The **Quasi** option is used by default except when a **Radiation** load is present or if enthalpy is defined as a material property, then the **Full** option is automatically specified.
- **Full**: Manually sets formulation for a full Newton-Raphson solution.
- **Quasi**: Manually sets formulation based on a tolerance you enter in the **Reformulation Tolerance** field that appears if **Quasi** is chosen.

Note:

The **Quasi** method is an approximation to the **Full** method. It is faster but not as accurate for strong nonlinearities. The **Quasi** method sets up a single fast material table, with equal temperature points between a maximum and a minimum temperature for evaluation of temperature-dependent material properties. Using this option, the solution uses 64 points between the maximum and the minimum temperature of the fast material table. If your material properties contain highly non-linear behavior and 64 points are not sufficient, using **Full** method is recommended.

Nonlinear Controls for Rigid Dynamics Analyses

Energy Accuracy Tolerance

This is the main driver to the automatic time stepping. The automatic time stepping algorithm measures the portion of potential and kinetic energy that is contained in the highest order terms of the time integration scheme, and computes the ratio of the energy to the energy variations over the previous time steps. Comparing the ratio to the **Energy Accuracy Tolerance**, Workbench will decide to increase or decrease the time step. See the Preparing a Rigid Dynamics Analysis (p. 551) section for more information.

When implicit time integration is chosen and automatic time stepping is enabled, the energy accuracy tolerance can be turned off. The time step is then determined by the number of iterations required by Newton-Raphson convergence.

Force Residual Relative Tolerance

Controls the threshold for force residual convergence in Newton-Raphson convergence. The default value is 1.e-7. A smaller value will lead to a smaller residual, but it will require more iterations. The convergence of force residual can be monitored in **Solution Information** using Force Convergence.

This option is only available with the Generalized-α, Stabilized Generalized-α, and MJ Time Stepping time integration methods.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Constraint Equation Residual Relative Tolerance

Controls the threshold for convergence of constraint equation violations in Newton-Raphson convergence. The default value is 1.e-8. The convergence of this criterion can be checked in **Solution Information** using Displacement Convergence.

This option is only available with Generalized-α, Stabilized Generalized-α, and MJ Time Stepping time integration.

Output Controls

The controls of the **Output Controls** category vary based on the type of analysis being performed.

Output Controls give you the ability to specify which type of quantities are written to the result file for use during post-processing. These properties enable you to control the size of the results file which can be beneficial when performing a large analysis.

The following **Output Controls** properties are available in the Details view to be activated (**Yes**) or not (**No**) and included or not included in the results file. Note that these controls *are not* step-aware, meaning that the settings are constant across multiple steps.

- **Stress**. Writes element nodal stresses to the results file. The default value is **Yes**. Available for Coupled Field analyses, Static Structural, Transient Structural, Modal, and Eigenvalue Buckling analysis types.

  Note:

  There are two options to obtain *surface* stress results. You can use the Surface Coating (p. 903) feature to accurately evaluate surface stresses. Or you can use a Command (APDL) (p. 2303) object. To properly define this object, see the *Mechanical APDL Element Reference* for key options to activate surface solution results on eligible elements as well as the Solution Output topic for the steps to obtain the surface solution as tabular data.

- **Back Stress**: Writes element back stresses to the result file. Options include **Yes** and **No** (default). Available for Coupled Field analyses, Static Structural, Transient Structural, Modal, Harmonic Response, and Eigenvalue Buckling analyses.
- **Strain**. Writes element elastic strains to the results file. The default value is **Yes**. Available for Coupled Field analyses, Static Structural, Transient Structural, Modal, and Eigenvalue Buckling analysis types.
- **Contact Data**: Writes element contact data to the result file. Options include **Yes** (default) and **No**. Available for an Additive Manufacturing Process Simulation, Coupled Field analyses, Static Structural, Transient Structural, Modal, Harmonic Response, Eigenvalue Buckling, Steady-State thermal, Transient Thermal, Electric, and Thermal-Electric analyses.

  Note:

  For an Additive Manufacturing Process Simulation, the default setting is **No** in order to reduce the size of the result file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Nonlinear Data**: Writes element nonlinear data to the result file. Options include **Yes** and **No** (default). Available for Coupled Field Static, Coupled Field Transient, Static Structural, and Transient Structural analyses. See the User Defined Results for Nonlinear Analyses (p. 2210) topic in the *User Defined Results for the Mechanical APDL Solver* section for the specific Items supported by the command. These items are accessed through the Solution Quantities and Result Summary (p. 1961) page of the Worksheet following the solution process.

- **Nodal Forces**. Writes elemental nodal forces to the results file. Options include:

  – **No**: No nodal forces are written to the results file. This is the default setting except for a Modal Analysis in which the **Future Analysis** property, under **Analysis Data Management** category, is set to **MSUP Analyses**. In that case, the default setting is **Constrained Nodes**.

  – **Yes**: This option writes nodal forces for all nodes. It is available for Static Structural, Transient Structural, Harmonic Response, Modal, Steady-State Thermal, and Transient Thermal analysis types. This Output Control must be set to **Yes** if you want to use the Mechanical APDL Command **NFORCE**, **FSUM** in Mechanical (via command snippets) because those MAPDL commands will access nodal force records in the result file as well as to obtain Reactions (p. 2107) on the underlying source or target element. For thermal analyses, nodal forces represent heat reactions.

    If **Future Analysis** property, under **Analysis Data Management** category, is set to **Structural Optimization**, **PreStressed & Structural Optimization**, or **MSUP & TopoOptimization**, then the **Nodal Forces** property is automatically set to **Yes** and becomes read-only.

  – **Constrained Nodes**. This option writes nodal forces for constrained nodes only. It is available for a Modal Analysis as well as Mode-Superposition (MSUP) Harmonic Response and Transient analyses that are linked to a Modal Analysis with the **Expand Results From** option set to the **Modal Solution**. This option directs Mechanical to use only the constrained nodes when calculating reaction forces and moments. The advantage is a reduced results file size.

- **Euler Angles**. Requests Euler Angle result values be written to the result file. Available for all analysis types except Response Spectrum, Random Vibration, and Structural Optimization analyses. The default is **Yes**.

Note:

– For a Modal, Eigenvalue Buckling, and an Additive Manufacturing Process Simulation, the default setting is **No** in order to reduce the size of the result file.

– The application may skip the evaluation of Euler Angles if the element coordinate system is aligned with Global Coordinate System.

Important:

Some result types, such as stress and/or strain results, may require Euler angle result data to properly display a result in the desired coordinate system. When the **Euler Angle** property is set to **No**, the result file may still contain the Euler angle results if any of the elemental results are set to **Yes**. However, if the **Euler Angle** property is set to **No** and all the elemental result outputs are also set to **No**, the application produces an error message and no result contours display for the elements in need of Euler angles.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Volume and Energy**. Writes summed volume and energy values to the result file: data that is essential for results such as Volume Probes, Energy Probes, Structural Error, etc. Available for all analysis types except Response Spectrum, Random Vibration, and Structural Optimization analyses. The default is **Yes**.

  Note:

  For a Modal, Eigenvalue Buckling, and an Additive Manufacturing Process Simulation, the default setting is **No** in order to reduce the size of the result file.

- **Calculate Reactions**. Turn **On** for **Nodal Forces** on constraints. Available for Coupled Field Harmonic, Coupled Field Modal, Modal, and Transient (when linked to a upstream Modal system) analysis types.
- **Calculate Velocity and Acceleration**. Writes Velocity and Acceleration results to the result file. The default value is **Yes**. Available for Mode-Superposition (MSUP) Transient Structural analyses only.

  Note:

  During a MSUP Transient analysis, the **Force Reaction** probe contains only the static contribution if you set the **Calculate Velocity and Acceleration** property to **No**. Review the **TRNOPT** command in the *Mechanical APDL Command Reference* for more information.

- **Calculate Thermal Flux**. Available for an Additive Manufacturing Process Simulation, Coupled Field Static, Coupled Field Transient, Steady-State Thermal, and Transient Thermal analysis types.

  Note:

  For an Additive Manufacturing Process Simulation, the default setting is **No** in order to reduce the size of the result file.

- **Heat Generation Rate**: Writes element heat generation rate to the result file. Options include **Yes** (default) and **No**. Available for Coupled Field analyses, Electric, and Thermal-Electric analyses.
- **Keep Modal Results**. Available for Random Vibration analyses *only*. The default value is **No**. This setting removes modal results from the result file in an effort to reduce file size. Setting this property to **Yes** allows you to perform post-processing on results of the Random Vibration solution (for example, Response PSD) via command snippets.
- **Calculate Velocity**. Writes Velocity to the results file. Available for Response Spectrum and Random Vibration analysis types. The default value is **No** for both analysis types.
- **Calculate Acceleration**. Writes Acceleration to the results file. Available for Response Spectrum and Random Vibration analysis types. The default value is **No** for both analysis types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Contact Miscellaneous**. Turn **On** if Contact Based Force Reactions (p. 2117) are desired. The default value is **No**. Available for Coupled Field Static, Coupled Field Transient, Static, and Transient Structural analysis types. Not Available when linked to a Modal analysis.
- **Element Current Density**: Writes element current densities to the result file. Options include **Yes** and **No** (default). Available for Coupled Field Static, Coupled Field Transient, Electric, Thermal-Electric, and Magnetostatic analyses.
- **Electromagnetic Nodal Forces**: Writes element electromagnetic nodal forces to the result file. Options include **Yes** and **No** (default). Available for Magnetostatic analyses.
- **Participation Factor** (**Modal** Analysis Only). This property displays when the **Solver Type** (in **Solver Controls** (p. 1428) category) property is set to **Iterative**. The options for this property include:
  - **Program Controlled** (default): When selected, the application automatically provides the **Participation Factor Summary** based on the following criteria.

    If the **Future Analysis** property of the **Analysis Data Management** category is set to **MSUP Analyses**, the application creates the file, `file.full`, and the **Participation Factor Summary** option becomes available in the drop-down list for the **Solution Output** property (under the **Solution Information** object (p. 1833)).

    If the **Future Analysis** property is set to **None**, the file, `file.full`, is not created and the **Participation Factor Summary** is not available.
  - **Yes**: The **Participation Factor Summary** option available in the **Solution Output** property of the **Solution Information** object.
  - **No**: The **Participation Factor Summary** option is not available in the **Solution Output** property of the **Solution Information** object. This option is not allowed if the **Future Analysis** property of the **Analysis Data Management** category is set to **MSUP Analyses**.
- **General Miscellaneous**. Used to access element miscellaneous records via SMISC/NMISC expressions (p. 2205) for user defined results. Options include **Yes**/**No**. The default value is **No**.

Note:

To ensure that Membrane and Bending Stress results are **not** under-defined, set this option to **Yes**.

Acoustics Analyses Only

Setting this property to **Yes** during an Acoustics analysis causes the application to also display the **Value** sub-property. The **Value** property enables you to select the bodies of your model for which the application sends element-based miscellaneous solution data to the output file. Options include:

- **Program Controlled**: The action of this property depends upon the acoustics analysis type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Harmonic Acoustics

This option issues miscellaneous data for all acoustic bodies if your analysis includes a Far-Field Radiation Surface (that you manually defined or that the application automatically generated), a Free Surface boundary condition, a Diffuse Sound Field excitation, or an acoustic Port.

Modal Acoustics

This option issues miscellaneous data for all acoustic bodies if your analysis includes a Free Surface boundary condition.

- **All Bodies**: This option generates miscellaneous data for all bodies.
- **Acoustic Bodies**: This option generates miscellaneous data for all acoustic bodies.
- **Structural Bodies**: This option generates miscellaneous data for all structural bodies.

Note:

Setting the **General Miscellaneous** property to **No** invalidates all Free Surface boundary conditions as well as all Far-field results.

- **Store Modal Results** (Modal analyses only). This field is only displayed when one of the element-based **Output Controls** is set to **Yes**, such as **Stress**, **Strain**, etc. This implies that element-based results must be expanded and saved to `file.mode`, in addition to displacement results (mode shapes). Depending on the downstream linked analysis, you may want to save these modal stress and/or modal strain results, which are linearly superimposed to get the stress and/or strain results of the downstream linked analysis. This reduces computation time significantly in the downstream linked analysis because no modal stress and/or modal strain results are expanded again. The following options are available:
  - **Program Controlled** (default setting): Let the program choose whether or not the modal results are saved for possible downstream analysis.
  - **No**: Element-based results are not saved to `file.mode` for later use in the downstream analyses. This option is recommended for a linked Harmonic Response analysis due to load generation, which requires that stresses and/or strains are expanded again as a result of the addition of elemental loads in the linked Harmonic Response analysis.
  - **For Future Analysis**: Element-based results are saved to `file.mode` for later use in the downstream linked analyses. This option is recommended for linked Random Vibration and Response Spectrum analyses. For these analysis types, the application does not need to solve elemental results, therefore, this setting improves performance and efficiency.
- **Expand Results From**
  - **MSUP Harmonic Analyses (Linked and Standalone)**. For this analysis type, the **Expand Results From** property displays only when one or more of the following results are set to **Yes**:

Stress	Strain	Calculate React

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Euler Angles **Volume and Energy** **Contact Data**

Activating one or more of these properties implies that one or more of the above results are to be expanded and saved to `file.mode` after the load generation. Depending on the number of modes and number of frequency steps, you may want to save these modal stresses and/or strains after the load generation, which can be linearly superimposed to obtain harmonic stresses and/or strains at each frequency step. The following options are available for this property:

→ **Program Controlled** (default setting): Let the program choose whether or not the stress, strain, and reaction results are expanded and saved for possible downstream analysis. When the **Program Controlled** option is chosen, the read-only Details view property **Expansion** is displayed. This indicates whether the stress, strain and reaction results are expanded from the modal solution or harmonic solution. However, if the **Cluster Results** property is set to **On** and you specify a high number of Modes, the solution is expanded from Harmonic Response analysis. If a lower number of Modes is specified, the solution is expanded from the Modal analysis.

→ **Harmonic Solution**: Stress, strain, and reaction results are not expanded nor saved to `file.mode` after the load generation in the MSUP Harmonic system (linked and standalone). This option is recommended when the number of frequency steps is far less than the number of modes. In this option, the stress, strain, and/or reaction results are expanded from harmonic displacement at each frequency step. In this case, stress, strain, and/or reaction expansion is performed as many times as the number of frequency steps.

Note:

For a MSUP Harmonic Response analysis linked to an upstream Modal Analysis that (1), includes a defined **Constant Damping Coefficient** in the Engineering Data Workspace and (2), the **Eqv. Damping Ratio From Modal** property set to **Yes** (**Analysis Settings** > **Damping Controls** (p. 1462)), you **cannot** expand results from the Harmonic solution unless the elemental results are expanded during the upstream modal solution. You must set the **Stress**, **Strain**, and/or **Calculate Reactions** properties to **Yes** in upstream Modal analysis to expand the elemental results. See the **MXPAND** section of the *Mechanical APDL Command Reference* for more information.

→ **Modal Solution**: Stress, strain, and reaction results are expanded and saved to `file.mode` after the load generation in the MSUP Harmonic system (linked and standalone). This option is recommended when the number of frequency steps is far more than the number of modes. In this option, the stress, strain, and/or reaction results are calculated by linearly combining the modal stresses, modal strains, and/or modal reactions expanded after the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

load generation. In this case, stress, strain, and/or reaction expansion are performed as many times as the number of modes.

Important:

For MSUP Harmonic Response analysis, if you 1) scope a **Force Reaction** probe (p. 2107) to a Remote Point (p. 1089) or a Remote Boundary Condition (p. 1735) and 2) set the **Expand Results From** property of the Harmonic Response analysis to either:

→ **Modal Solution**

Or...

→ **Program Controlled** with the **Expansion** property set to **Modal Solution**. Then the **Force Reaction** probe includes the static contribution only. For more information, see the Reaction Forces topic in the Mechanical APDL Theory Reference.

Refer to Recommended Settings for Modal and Linked Analysis Systems (p. 1482) for further details.

– **Linked Transient Analyses**. For this analysis type, the **Expand Results From** property displays only when one or more of the following properties are set to **Yes**:

Stress	**Strain**	**Calculate Reactions**
Euler Angles	**Volume and Energy**	**Contact Data**

Activating one or more of these properties implies that one or more of the above results are to be expanded and saved to `file.mode` after the load generation. Depending on the number of modes and total number of sub steps/ time steps, you may want to save these modal stresses and/or strains after the load generation, which can be linearly superimposed to obtain transient stresses and/or strains at each time step. The following options are available for this property:

→ **Program Controlled** (default setting): Let the program choose whether or not the stress and strain results are expanded and saved for possible downstream analysis. When the **Program Controlled** option is chosen, the read-only Details view property **Expansion** is displayed. This indicates whether the stress and strain results are expanded from modal solution or transient solution.

→ **Transient Solution**: Stress and strain results are not expanded nor saved to `file.mode` after the load generation in the linked transient analysis system. This option is recommended when the number of time steps accumulated over all the load steps is far less than the number of modes. In this option, the stress and/or strain results are expanded from transient displacement

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

at each time step. In this case, stress and/or strain expansion is performed as many times as the number of time steps.

Note:

For a Transient Structural analysis linked to an upstream Modal Analysis that (1), includes a defined **Constant Damping Coefficient** in the Engineering Data Workspace and (2), the **Eqv. Damping Ratio From Modal** property set to **Yes** (**Analysis Settings** > **Damping Controls** (p. 1462)), you cannot expand from the Transient solution unless the elemental results are expanded during the upstream modal solution. You must set the **Stress**, **Strain**, and/or **Calculate Reactions** properties to **Yes** in upstream Modal analysis to expand the elemental results.

→ **Modal Solution**: Stress and strain results are expanded and saved to `file.mode` after the load generation in the linked transient system. This option is recommended when the number of time steps accumulated over all the load steps is far more than the number of modes. In this option, the stress and/or strain results are calculated by linearly combining the modal stresses and/or modal strains expanded after the load generation. In this case, stress and/or strain expansion are performed as many times as the number of modes.

Important:

For MSUP Transient analysis, if you 1) scope a **Force Reaction** probe (p. 2107) to a Remote Point (p. 1089) or a Remote Boundary Condition (p. 1735) and 2) set the **Expand Results From** property of the Harmonic Response analysis to either:

→ **Modal Solution**

Or...

→ **Program Controlled** with the **Expansion** property set to **Modal Solution**.
Then the **Force Reaction** probe includes the static contribution only. For more information, see the Reaction Forces topic in the Mechanical APDL Theory Reference.

Refer to Recommended Settings for Modal and Linked Analysis Systems (p. 1482) for further details.

- If you are using the Samcef solver interface for your analysis, the **SAI Command** setting can be used to control the SAI codes written in the solver input file. When this setting is set to **Program Controlled**, the SAI codes are taken from the configuration file, stored in *ANSYS_INSTALL_DIR*`\v241\AISOL\WBAddins\SamcefAddin\SamcefArchiveSettings.xml`. If this option is set to **Manual**, the text field **SAI Command List** option is shown, and the SAI ARCH codes written to the input file are taken from this field instead of the configuration file. For more information about the configuration file, see The Samcef Result Storage Configuration File.

- If you are using the ABAQUS solver interface, there are several options to control the output of Elements and Nodes from that solver:

  – The **Nodal/Elemental/Contact/Radiation Outputs** field controls result codes sent to the solver. When set to **Program Controlled**, the codes are provided from the configuration file *ANSYS_IN-*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

STALL_DIR\v241\AISOL\WBAddins\AbaqusAddin\AbaqusArchiveSettings.xml. When set to **Manual**, the text field **Outputs List** appears and allows you manually set the result codes. When set to **All**, the solver stores all results. Note that only the fields valid for the type of analysis you are performing are shown. For more information about the configuration file, see The ABAQUS Result Storage Configuration File.

– The **Output Storage/Output Storage Value** fields define the type of result storage.

– For a modal analysis, the **Mode Selection** field allows you to select all modes for output, or define a subset of modes manually.

Note:

- It is recommended that you **not** change **Output Controls** settings during a Solution Restart (p. 1815). Modifying **Output Controls** settings change the availability of the respective result type in the results file. Consequently, result calculations cannot be guaranteed for the entire solution. In addition, Result file values may not correspond to GUI settings in this scenario. Settings turned off during a restart generate results equal to zero and may affect post processing of results and are therefore unreliable.
- Modification of **Stress**, **Strain**, **Nodal Force**, **Contact Miscellaneous**, and **General Miscellaneous** properties will not invalidate the solution. If you want these Output Controls settings modifications to be incorporated to your solution, clean the solution first.

Multiple Step Properties

In addition, the following settings are step-aware and allow you to define *when* data is calculated and written to the result file for Static Structural, Transient Structural, Rigid Dynamics, Steady-State Thermal, Transient Thermal, and Structural Optimization analyses:

- **Store Results At**. Based on the analysis type, specify this time to be **All Time Points** or **All Iterations** (default setting), **Last Time Point** or **Last Iteration**, **Equally Spaced Points** or **Specified Recurrence Rate**. For Additive Manufacturing simulations, options include **All Layers**, **Last Heating and Cooling Steps**, and **Every N Layers**.
- **Value**. Displayed only if **Store Results At** is set to **Equally Spaced Points** or **Specified Recurrence Rate**.
- **Result File Compression**: This property enables you to generate a compressed result file. Options include **Program Controlled** (default), **Sparse**, **Off**. The **Program Controlled** and **Sparse** settings instruct the application to compress the file.
- **Export Layer End Temperature**. No (default) or Yes. This setting controls whether the temperature of a layer just before a new layer is applied is written out to an AMResults.txt file. Available for Additive Manufacturing thermal analyses only.
- **Export Recoater Interference**. No (default) or Yes. This setting controls whether the z-deformation of a layer just before a new layer is applied is written out to an AMResults.txt file. Available for Additive Manufacturing structural analyses only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Recommended Settings for Modal and Linked Analysis Systems

The following table provides a summary of recommended settings for **Store Modal Results** and **Expand Results From** based on the analysis type.

Analysis Type	Recommended Store Modal Results Settings	Recommended Expand Results From Settings
Modal with no downstream analysis	**No** Stress and strain results not needed to be saved to `file.mode` because there is no downstream analysis.	Not available.
MSUP Harmonic Response analyses (Linked and Standalone)	**No**[a] Stress and strain results from modal analysis are overwritten by stresses and strains which are expanded again in the linked Harmonic Response analysis due to any loads added in the downstream analysis. **Note:** This setting is available for a linked MSUP Harmonic Response analysis only.	**Harmonic Solution** Use when number of frequency steps are far less than the number of modes. This option is not available when the Modal has a nonlinear Pre-Stress environment. **Modal Solution** Use when number of frequency steps are far more than the number of modes. This is the only option available when the Modal analysis has a nonlinear Pre-Stress environment.
Modal linked to downstream Random Vibration analysis	**For Future Analysis** Stress and strain results from modal analysis are expanded and used in the linked random vibration analysis. No stress or strain expansion is needed again because there is no load.	Not available.
Modal linked to downstream Response Spectrum analysis	**No** Stress and strain results are always combined in response spectrum analysis using `file.rst` and `file.mcom`. **Note:** To evaluate summation of element nodal forces	Not available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Type	Recommended Store Modal Results Settings	Recommended Expand Results From Settings
	using **FSUM** in Command Snippet, it is required to save element nodal forces in modal to `file.mode`.	
Modal linked to downstream Structural Optimization analysis	**For Future Analysis** The **Nodal Forces** and the **General Miscellaneous** properties are set to **Yes**.	Program Controlled.
Static Structural linked to downstream Structural Optimization analysis	**For Future Analysis** The **Nodal Forces** property is set to **Yes**.	Program Controlled.
Modal linked to downstream Transient analysis	**No**[a] Stress and strain results from modal analysis are overwritten by stresses and strains which are expanded again in the linked transient analysis due to any loads added in the downstream analysis.	**Transient Solution** Use when number of time steps accumulated over all the load steps is far less than the number of modes. This option is not available when the Modal Analysis has a nonlinear Pre-Stress environment.
		Modal Solution Use when number of time steps accumulated over all the load steps is far more than the number of modes. This is the *only* option available when the Modal Analysis has a nonlinear Pre-Stress environment.

[a] Review the Requirements and Limitations for Modal and Linked MSUP Analysis Systems shown below.

Requirements and Limitations for Modal and Linked MSUP Analysis Systems

If you have an MSUP analysis, either Harmonic Response or Transient, linked to an upstream Modal system, the Modal analysis does not respect the **Expand Results From** property setting when:

- Only Direct FE loads, Remote Force, or Moment are scoped to global remote points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The analysis includes a **Commands (APDL)** object.
- No **Damping Ratio** is specified.
- If the material damping is specified using the **Eqv. Damping Ratio From Modal** property (set to **Yes**) and at least one of the elemental-based **Output Controls** (Stress, Strain, etc.) is also set to **Yes** in the linked Modal analysis.

Instead, it will be driven by the **Store Modal Results** property (of the Modal system):

- Setting this property to **No**, the **Expand Results From** property of the MSUP Harmonic Response or Transient analysis is ignored, even if set to **Modal Solution**. The results would be as if the **Harmonic Solution** or **Transient Solution** option had been set. Given this internal specification, you do not receive the results you expected.
- Setting the property to **For Future Analysis**, the **Expand Results From** property of the MSUP Harmonic Response or Transient analysis is ignored. The results would be as if the **Modal Solution** option had been set. In this case, be sure that you request the same results from the Output Controls settings for both the Modal and downstream MSUP analyses. Given this internal specification, you do not receive the results you expected.

The scenarios are summarized in the following table.

Store Modal Results Property Setting	Expand Result From Property Setting	Actual Expanded Results
No	Expand from Modal	Expanded from Harmonic/Transient
No	Expand from Harmonic/Transient	Expanded from Harmonic/Transient
For Future Analysis	Expand from Modal	Expanded from Modal
For Future Analysis	Expand from Harmonic/Transient	Expanded from Modal

Limitations When Using the Mechanical APDL Solver

The Mechanical application cannot post process split result files produced by the Mechanical APDL solver. Try either of the following workarounds should this be an issue:

- Use **Output Controls** to limit the result file size. Also, the size can more fully be controlled (if needed) by inserting a **Commands** object (p. 2303) for the **OUTRES** command.
- Increase the threshold for the files to be split by inserting a **Commands** object (p. 2303) for the **/CONFIG**,FSPLIT command.

Analysis Data Management

The properties of the **Analysis Data Management** category vary based on the type of analysis being performed. Supported analysis types include:

- Coupled Field Analyses (p. 376)
- Static Structural (p. 652)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Eigenvalue Buckling (p. 487)
- Electric (p. 403)
- Harmonic Response (p. 496)
- Magnetostatic (p. 546)
- Modal (p. 515)
- Random Vibration (p. 524)/Response Spectrum (p. 533)
- Rigid Dynamics (p. 550)
- Steady - State Thermal (p. 657)
- Substructure Generation (p. 540)
- Thermal Electric (p. 661)
- Structural Optimization
- Transient Structural (p. 665)
- Transient Thermal (p. 680)

The **Analysis Data Management** category includes the following properties and options.

Property	Options/Descriptions
Solver Files Directory	This property Indicates the location of the solution files for this analysis. The directory location is automatically determined by the program as detailed in Managing Files (p. 1859). The solution file folder can be displayed using the context **Open Solver Files Directory** menu option. **Open Solver Files Directory Feature** • This right-click context menu option is available when you have an analysis Environment or a Solution object selected. • Once executed, this option opens the operating system's (Windows Only) file manager and displays the directory that contains the solution files for your analysis. • The directory path is shown in the Details View. If a solution is in progress, the directory is shown in the **Solver Files Directory** field. When a solution is in progress, the directory displays in the **Scratch Solver Files Directory** (p. 1487). For a remote solve, it will open the scratch directory until the results download is complete.
Future Analysis	This property defines whether to use the results of the current analysis as loading or as an initial condition in a subsequent downstream analysis. Supported analysis types and the supported downstream systems are described below. **Coupled Field Static** Options include **None** (default) and **Prestressed Analysis**. If a Coupled Field Static analysis is used to provide Pre-Stress effects, this property automatically defaults to the **Prestressed Analysis** setting. It can provide Pre-Stress effects for the following analysis types: • Pre-Stressed (Full) Coupled Field Harmonic

Property	Options/Descriptions
	• Pre-Stressed Coupled Field Modal **Harmonic Response** Options include **None** (default) and **Structural Optimization**. If you select **Structural Optimization**, the application automatically sets other **Analysis Settings** properties to specific read-only settings. Such as the **Multiple Steps** property, the **Solution Method** property (**Mode Superposition**), **Output Controls** properties, and **On Demand Expansion** property. If you link a **Harmonic Response** analysis to a downstream **Structural Optimization** analysis, this property automatically defaults to the **Structural Optimization** setting, becomes read-only, and the same automatic actions as stated above apply. Unlinking these analyses reverts all automatic read-only settings. **Modal** Options include **None** (default), **MSUP Analyses**, **Structural Optimization**, or **MSUP & Structural Optimization**. When linked to a supported analysis type (**Harmonic Response** or **Random Vibration (PSD)** or **Response Spectrum**), this property automatically defaults to the **MSUP Analyses** setting. **Note:** A **Modal** analysis is a prerequisite for the following analysis types: • **Random Vibration (PSD)** • **Response Spectrum** If a Modal analysis is linked with a **Structural Optimization** analysis, this property automatically defaults to the **Structural Optimization** setting. If a **Modal** analysis is linked with a **Harmonic Response**, or a **Random Vibration (PSD)**, or a **Response Spectrum** analysis and a **Structural Optimization** analysis, this property automatically defaults to the **MSUP & Structural Optimization** setting. **Static Structural** Options include **None** (default), **Prestressed Analysis**, **Structural Optimization**, and **PreStressed & Structural Optimization**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	Options/Descriptions
	If a Static Structural analysis is used to provide Pre-Stress effects, this property automatically defaults to the **Prestressed Analysis** setting. It can provide Pre-Stress effects for the following analysis types: • Pre-Stressed (Full) **Harmonic Response** • Pre-Stressed **Modal** • Pre-Stressed **Eigen Value Buckling** **Note:** A Static Structural analysis is a prerequisite for Eigenvalue Buckling analysis (p. 487). If a **Static Structural** analysis is linked with a **Structural Optimization** analysis, this property automatically defaults to the **Structural Optimization** setting. If a **Static Structural** analysis is linked with a **Modal** analysis or a **Harmonic Response** analysis and a **Structural Optimization** analysis, this property automatically defaults to the **PreStressed & Structural Optimization** setting. **Substructure Generation** Options include **None** (default) and **Ansys Rigid Dynamics (Beta)** (visible when you activate Beta Options). When you select the **Ansys Rigid Dynamics (Beta)** option, the application automatically includes the additional files necessary to perform a downstream Rigid Dynamics (p. 550) analysis in the exported file.
Scratch Solver Files Directory	This is a read-only indication of the directory where a solve "in progress" occurs. All files generated after the solution is done (including but not limited to result files) are then moved to the **Solver Files Directory**. The files generated during solves on **My Computer** or files requested from RSM for postprocessing during a solve remain in the scratch directory. For example, an early result file could be brought to the scratch folder from a remote machine through RSM during postprocessing while solving. With the RSM method, the solve may even be computed in this folder (for example, using the **My Computer, Background** Solve Process Settings). The scratch directory is only set for the duration of the solve (with either **My Computer** or **My Computer, Background**). After the solve is complete, this directory is set to blank. As desired, you can specify a unique disk location for this directory using the **Scratch Solver Files Directory** option in the **Analysis Settings and Solution** (p. 238) category of the Options (p. 217) preference settings. Specifying a different disk location for the scratch files enables you take advantage of a faster disk drive.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Options/Descriptions
	The use of the **Scratch Solver Files Directory** prevents the **Solver Files Directory** from ever getting an early result file.
Save MAPDL db	Options include **No** (default) and **Yes**. Some **Future Analysis** settings will require the database file to be written. In these cases this field will be set to **Yes** automatically.
Contact Summary	This property enables you to control where contact pair data is written during the solution process, either to the solver output file or to a contact output file. This ability enables you to limit the contact data written to the solver output file. Property options include **Program Controlled** (default), **Solver Output**, and **CNM File**. If your model includes more than 100 contact pairs, the **Program Controlled** option automatically writes contact pair data to a contact output text file named `file.cnm` (refer to the **CNTR,OUT,YES** command), otherwise the data is written to the solver output file. The **Solver Output** option writes the data as normal to the solver output file. The **CNM File** option writes contact pair data to `file.cnm` regardless of the number of contact pairs in your model.
Delete Un-needed Files	Options include **Yes** (default) and **No**. If you prefer to save all the solution files for some other use you may do so by setting this field to **No**. **Important:** When you are 1) using the Mechanical APDL solver, 2) have the **Distributed** (p. 1791) solve option selected (default), and 3) this property is set to **Yes** (required), the application automatically executes the Mechanical APDL command **/FCLEAN**. This command deletes all unnecessary distributed files from the scratch directory. In addition to this property needing to be set to **Yes**, the **Future Analysis** property (described above) must be set to **None** for the command to execute properly. These requirements apply to each analysis system included in your project. The action of deleting unnecessary distributed files is especially useful for solutions being performed on a remote machine because it eliminates the need to download potentially large files that serve no specific purpose for the postprocessing of your analysis. **Note:** For Static Structural and Transient Structural analyses, setting this property to **Yes** automatically sets the Restart Controls property **Retain Files After Full Solve** property to **Yes** (the default is **No**) and the application does not execute the **/FCLEAN** command. However, changing the **Delete Unneeded Files** property from **Yes** to **No** DOES NOT automatically reset the **Retain Files After Full Solve** property (to **No**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	Options/Descriptions
	If you are using a Samcef or ABAQUS solver interface for your analysis, the **Solver Files Name** setting controls the name of the files generated in the analysis directory. By default, this setting is the name of the solver being used ("samcef" or "abaqus").
Nonlinear Solutions	Read only indication of **Yes** or **No** depending on presence of nonlinearities in the analysis.
Solver Units	You can select one of two options for this property: • **Active System**: This options instructs the solver to use the currently active unit system (determined via the **Units** option in the **Tools** (p. 45) group of the **Home** tab.) for the very next solve. • **Manual**: This option enables you to select the unit system for the solver to use by allowing them access to the second field, "Solver Unit System".
Solver Units System	• If **Active System** is chosen for the **Solver Units** field, then this field is read-only and displays the active system. • If **Manual** is chosen for the **Solver Units** field, this field is a selectable drop-down menu. • If a Magnetostatic analysis is being performed, the field is read only because the only system available to solve the analysis is the mks system. • If a Electric (p. 403), Coupled Field Static (p. 391) (with Electric physics), Coupled Field Harmonic (p. 380), Coupled Field Modal (p. 386), or Thermal-Electric (p. 661), analysis is being performed, only mks and µmks systems can be selected because they are the only systems currently allowed for these analyses.
Max Num of Intermediate Files	This property is for Structural Optimization analyses only. It specifies the number of files you wish to retain. A value of **1** indicates that the generated file is overwritten each iteration. The default value for the property is set to the text string "All Iterations" that equals a setting of zero (**0**). This setting saves the intermediate topology files for all iterations solved. **Note:** The animation of the **Topology Density** results and **Topology Elemental Density** results is based on the number of intermediate results computed and saved during solution. The computation of results is driven by **Store Results At** property of the **Output Controls** (p. 1473) and the intermediate results saved is driven by this property. You can change the default setting for this property using the **Max Num Of Intermediate Files** property under the **Analysis Data Management (Structural Optimization)** category of the **Options** (p. 217) > **Analysis Settings and Solution** (p. 238) preference.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Rotordynamics Controls

The controls of the **Rotordynamics Controls** group vary based on the type of analysis being performed. Supported analysis types include:

- Coupled Field Modal (p. 386)[2]/Prestressed Coupled Field Modal Analysis (p. 388)[2]
- Prestressed Coupled Field Harmonic Analysis (p. 382)[2]
- Coupled Field Static (p. 391)[2]
- Harmonic Response (p. 496)
- Modal (p. 515) (including a prestressed system)
- Static Structural (p. 652)

Coupled Field Static Analysis[2]

The **Coriolis Effect** property is the only **Rotordynamics Controls** property for a **Coupled Field Static** analysis. Options include **Off** (default) and **On, Rotating Reference Frame**.

Coupled Field Modal[2]**, Prestressed Coupled Field Modal, and Modal Analyses**[2]

The following properties control the items that apply to a rotating structure:

- **Coriolis Effect**: Options for this property include **Off** (default), **On, Stationary Reference Frame** for a Modal analysis, and **On, Rotating Reference Frame** for a Coupled Field Modal analysis (including prestressed). Both of the "On" settings are valid *only* if the Damped (p. 1428) property of the **Solver Control** category is set to **Yes**.
- **Campbell Diagram**: Set to **On** if Campbell diagram is to be plotted. The default is **Off**. **On** is a valid choice only if **Coriolis Effect** is turned **On**.
- **Number of Points**: This property is only displayed when **Campbell Diagram** is set to **On**. This property's values indicates the number of solve points for the **Campbell Diagram**. The default value is **2**. A minimum of two (2) solve points is necessary.

  In addition, this value also determines the number of solution **Points** (data rows) displayed in **Tabular Data** for the associated **Rotational Velocity**.

Harmonic Response Analysis

The **Coriolis Effect** property is the only **Rotordynamics Controls** property for a Harmonic Response analysis. The default setting is **Off**. The **On, Stationary Reference Frame** setting is valid only when the **Solution Method** is set to **Full** (the default setting is **Mode Superposition**). When **Coriolis Effect** is set to **On, Stationary Reference Frame**, you can use the Rotating Force (p. 1655) boundary condition in your analysis.

[2] When you specify structural and electric (charge) physics.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Prestressed Coupled Field Harmonic Analysis[2]

The **Coriolis Effect** property is the only **Rotordynamics Controls** property for a prestressed **Coupled Field Harmonic** analysis. Options include **Off** (default) and **On, Rotating Reference Frame**.

Static Structural Analysis

The **Coriolis Effect** property is the only **Rotordynamics Controls** property for a Static Structural analysis. The default setting is **Off**.

Note:

For the above analyses, **except** the coupled field types, when the **Coriolis Effect** property is active, the application applies Coriolis effects in a stationary reference frame.

Visibility

Allows you to selectively display loads in the **Graph** window by choosing **Display** or **Omit** for each available load type. A load must first be applied before the **Visibility** group becomes available/shown under Analysis Settings.

The **Visibility** group is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Steady - State Thermal (p. 657)
- Transient Thermal (p. 680)

Steps and Step Controls for Static and Transient Analyses

The following topics are covered in this section:

Role of Time in Tracking
Steps, Substeps, and Equilibrium Iterations
Automatic Time Stepping
Guidelines for Integration Step Size

Role of Time in Tracking

Time is used as a tracking parameter in *all* static and transient analyses, whether or not the analysis is truly time-dependent. The advantage of this is that you can use one consistent "counter" or "tracker" in all cases, eliminating the need for analysis-dependent terminology. Moreover, time always increases monotonically, and most things in nature happen over a period of time, however brief the period may be.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Obviously, in a transient analysis time represents actual, chronological time in seconds, minutes, or hours. In a static analysis, however, time simply becomes a counter that identifies steps and substeps. By default, the program automatically assigns time = 1.0 at the end of step 1, time = 2.0 at the end of step 2, and so on. Any substeps within a step will be assigned the appropriate, linearly interpolated time value. By assigning your own time values in such analyses, you can establish your own tracking parameter. For example, if a load of 100 units is to be applied incrementally over one step, you can specify time at the end of that step to be 100, so that the load and time values are synchronous.

Steps, Substeps, and Equilibrium Iterations

What is a step?

A step corresponds to a set of loads for which you want to obtain a solution and review results. In this way every static or transient dynamic analysis has at least one step. However there are several scenarios where you may want to consider using multiple steps within a single analysis, that is, multiple solutions and result sets within a single analysis.

A static or transient analysis starts at time = 0 and proceeds until a step end time that you specify. This time span can be further subdivided into multiple steps where each step spans a different time range.

As mentioned in the Role of Time in Tracking (p. 1491) section, each step spans a 'time' even in a static analysis.

When do you need Steps?

Steps are required if you want to change the analysis settings for a specific time period. For example in an impact analysis you may want to manually change the allowable minimum and maximum time step sizes during impact. In this case you can introduce a step that spans a time period shortly before and shortly after impact and change the analysis settings for that step.

Steps are also useful generally to delineate different portions of an analysis. For example, in a linear static structural analysis you can apply a wind load in the first step, a gravity load in the second step, both loads and a different support condition in the third step, and so on. As another example, a transient analysis of an engine might include load conditions corresponding to gravity, idle speed, maximum power, back to idle speed. The analysis may require repetition of these conditions over various time spans. It is convenient to track these conditions as separate steps within the time history.

In addition steps are also required for deleting loads or adding new loads such as specified displacements or to set up a pretension bolt load sequence. Steps are also useful in setting up initial conditions for a transient analysis.

How do you define steps?

See the procedure, "Specifying Analysis Settings for Multiple Steps" located in the Establish Analysis Settings (p. 349) section.

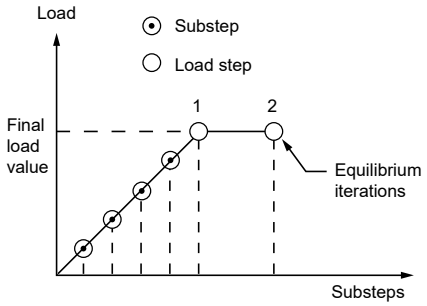
What are substeps and equilibrium iterations?

Solving an analysis with nonlinearities requires convergence of an iterative solution procedure. Convergence of this solution procedure requires the load to be applied gradually with solutions carried

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

out at intermediate load values. These intermediate solution points within a step are referred to as *substeps*. Essentially a substep is an increment of load within a step at which a solution is carried out. The iterations carried out at each substep to arrive at a converged solution are referred to as *equilibrium iterations*.



Automatic Time Stepping

Auto time stepping, also known as time step optimization, aims to reduce the solution time especially for nonlinear and/or transient dynamic problems by adjusting the amount of load increment. If nonlinearities are present, automatic time stepping gives the added advantage of incrementing the loads appropriately and retreating to the previous converged solution (bisection) if convergence is not obtained. The amount of load increment is based on several criteria including the response frequency of the structure and the degree of nonlinearities in the analysis.

The load increment within a step is controlled by the auto time stepping procedure within limits set by you. You have the option to specify the maximum, minimum and initial load increments. The solution will start with the "initial" increment but then the automatic procedure can vary further increments within the range prescribed by the minimum and maximum values.

You can specify these limits on load increment by specifying the initial, minimum, and maximum number of substeps that are allowed. Alternatively, since a step always has a time span (start time and end time), you can also equivalently specify the initial, minimum and maximum time step sizes.

Although it seems like a good idea to activate automatic time stepping for all analyses, there are some cases where it may not be beneficial (and may even be harmful):

- Problems that have only localized dynamic behavior (for example, turbine blade and hub assemblies), where the low-frequency energy content of part of the system may dominate the high-frequency areas.
- Problems that are constantly excited (for example, seismic loading), where the time step tends to change continually as different frequencies are excited.
- Kinematics (rigid-body motion) problems, where the rigid-body contribution to the response frequency term may dominate.

Guidelines for Integration Step Size

The accuracy of the transient dynamic solution depends on the integration time step: the smaller the time step, the higher the accuracy. A time step that is too large introduces an error that affects the response of the higher modes (and hence the overall response). On the other hand too small a time step size wastes computer resources.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

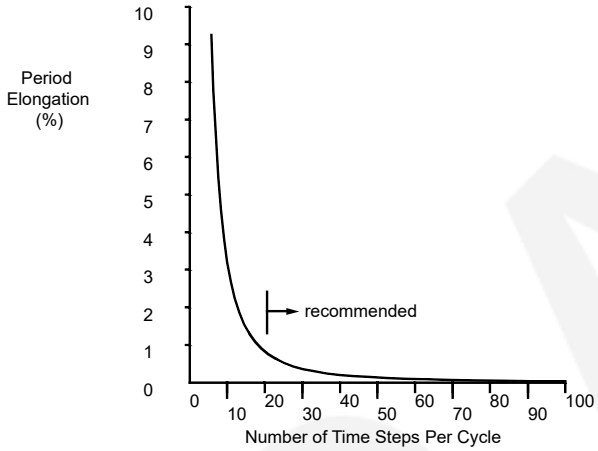
An optimum time step size can depend on several factors:

1. Response frequency: The time step should be small enough to resolve the motion (response) of the structure. Since the dynamic response of a structure can be thought of as a combination of modes, the time step should be able to resolve the highest mode that contributes to the response. The solver calculates an aggregate response frequency at every time point. A general rule of thumb it to use approximately twenty points per cycle at the response frequency. That is, if f is the frequency (in cycles/time), the integration time step (ITS) is given by:

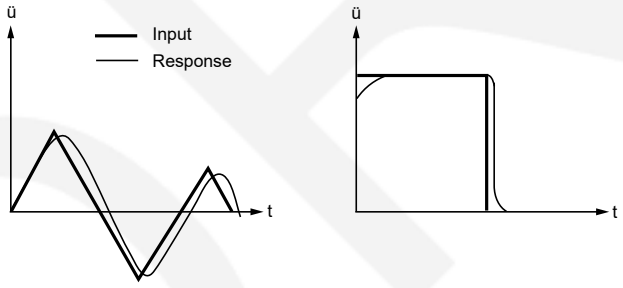
   ITS = 1/(20f)

   Smaller ITS values will be required if accurate velocity or acceleration results are needed.

   The following figure shows the effect of ITS on the period elongation of a single-DOF spring-mass system. Notice that 20 or more points per cycle result in a period elongation of less than 1 percent.



2. Resolve the applied load-versus-time curve(s). The time step should be small enough to "follow" the loading function. For example, stepped loads require a small ITS at the time of the step change so that the step change can be closely followed. ITS values as small as 1/180f may be needed to follow stepped loads.



3. Resolve the contact frequency. In problems involving contact (impact), the time step should be small enough to capture the momentum transfer between the two contacting faces. Otherwise, an apparent energy loss will occur and the impact will not be perfectly elastic. The integration time step can be determined from the contact frequency (fc) as:

   $\mathrm{ITS} = 1/\mathrm{N}\mathrm{f}_c \quad \mathrm{f}_c = (1/2\pi)\sqrt{k/m}$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where k is the gap stiffness, m is the effective mass acting at the gap, and N is the number of points per cycle. To minimize the energy loss, at least thirty points per cycle of (N = 30) are needed. Larger values of N may be required if velocity or acceleration results are needed. See the description of the **Predict for Impact** option within the **Time Step Controls** (p. 1255) contact **Advanced** settings for more information.

You can use fewer than thirty points per cycle during impact if the contact period and contact mass are much less than the overall transient time and system mass, because the effect of any energy loss on the total response would be small.

4. Resolve the nonlinearities. For most nonlinear problems, a time step that satisfies the preceding guidelines is sufficient to resolve the nonlinearities. There are a few exceptions, however: if the structure tends to stiffen under the loading (for example, large deflection problems that change from bending to membrane load-carrying behavior), the higher frequency modes that are excited will have to be resolved.

After calculating the time step sizes using the above guidelines, you need to use the minimum value for your analysis. However using this minimum time step size throughout a transient analysis can be very inefficient. For example in an impact problem you may need small time step sizes calculated as above only during and for a short duration after the impact. At other parts of the time history you may be able to get accurate results with larger time steps sizes. Use of the Automatic Time Stepping (p. 1493) procedure lets the solver decide when to increase or decrease the time step during the solution.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Setting Up Boundary Conditions

Boundary conditions are often called "loads" or "supports". They *constrain* or act upon your model by exerting forces or rotations or by fixing the model it such a way that it cannot deform.

Boundary conditions are typically applied to 2D or 3D simulations but exceptions do exist. Any exceptions are discussed in detail on the Help page for the particular boundary condition.

The boundary conditions you apply depend on the type of analysis you are performing. In addition, the geometry (body, face, edge, or vertex) or finite element selection to which a boundary condition is applied, also varies per analysis type.

Once applied, and as applicable to the boundary condition type, the loading characteristics must be considered. This includes, whether the boundary condition is defined as a constant, by using tabular entries (time history or spatially varying), or as a function (time history or spatially varying).

The following topics describe the steps involved in applying and using boundary conditions in the application.

Boundary Condition Scoping Method

Types of Boundary Conditions

Spatial Varying Loads and Displacements

Defining Boundary Condition Magnitude

Specifying Boundary Condition Loads With Tables

Specifying Model-Level Boundary Conditions

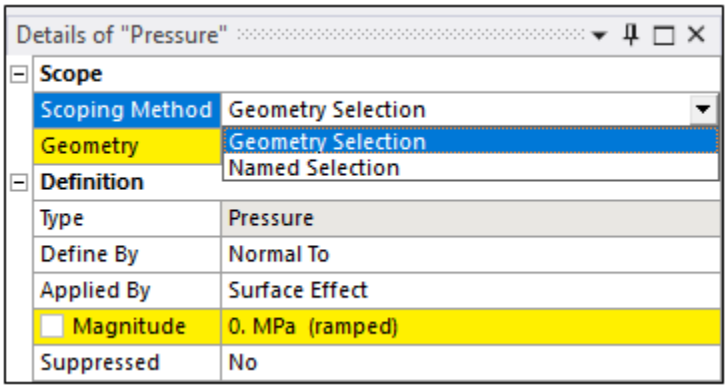
Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver)

Resolving Thermal Boundary Condition Conflicts

Boundary Condition Scoping Method

Almost every boundary condition available in the application has a Details group, **Scope**, that includes the property **Scoping Method**. A Pressure load is illustrated below. Scope refers to the geometry over which a boundary condition is applied. You can select geometry or geometries of your model using geometry selection tools (**Geometry Selection**) or through the use of the **Named Selection** (p. 1057) feature. You "scope" boundary conditions to geometric entities (bodies, faces, edges, or vertices), and depending upon the load type, mesh entities (nodes, element faces, and elements).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



For example, if you apply a force of 1000N in the X-direction to a vertex, the load is "scoped" to that vertex.

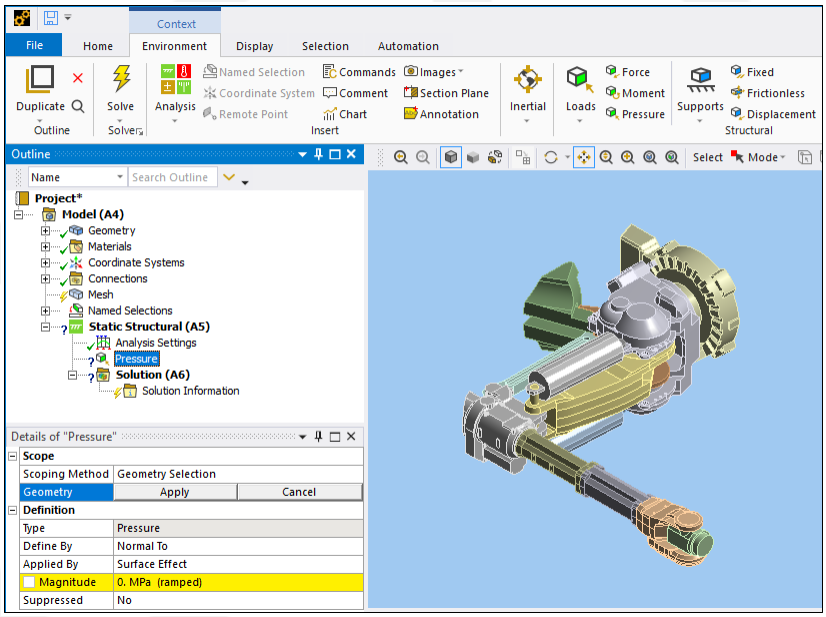
Scoping Method Techniques

The application provides several common methods for scoping boundary conditions. Regardless of the method you choose, you need to select geometry entities of the model and scope those entities with a boundary condition. The order of these selections can vary.

You can first select a boundary condition and then specify a geometry:

1. Highlight the **Environment** object.
2. Click the desired drop-down menu from the Context tab and select your boundary condition type.
3. Select the desired geometry or geometries (by pressing and holding **[Ctrl]**) on your model and then clicking the **Apply** button. Perform any additional required entries.

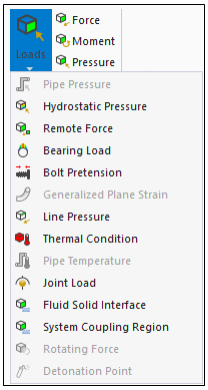
   In the example shown here, a Pressure was applied to a face. The **Magnitude** entry is undefined.



Or you can first select geometries and then apply boundary conditions:

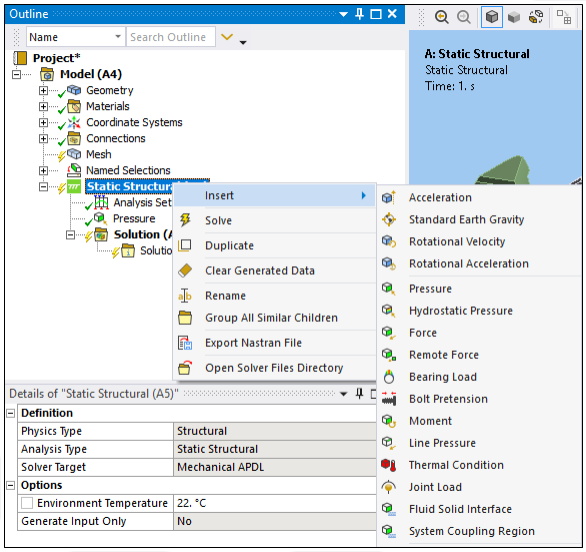
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Highlight the **Environment** object
2. Pick your geometry.
3. Apply your boundary condition by:
   a. Making a selection from the **Environment** Context tab (p. 61).



Or...

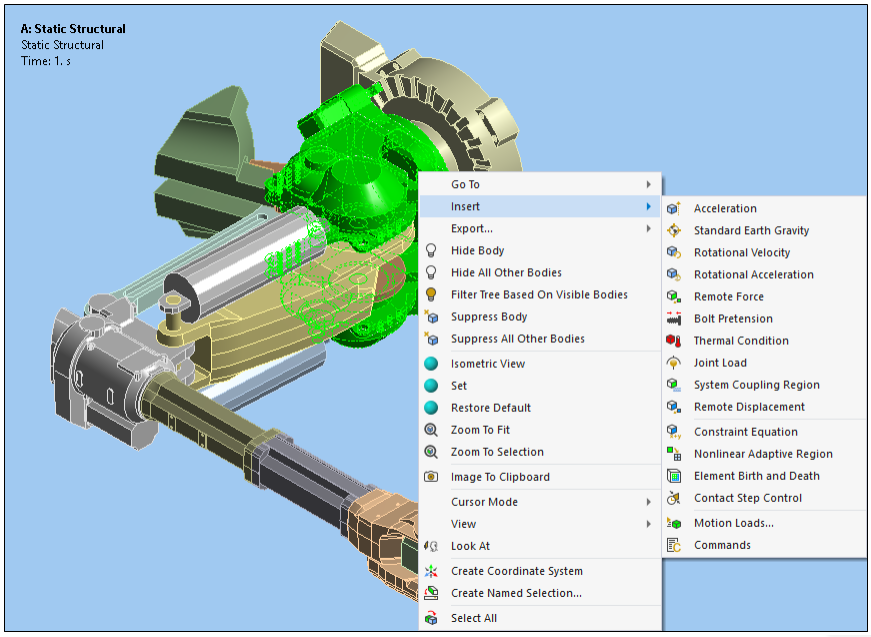
   b. Selecting the **Environment** object, right-clicking, select **Insert** and then your desired boundary condition.



Or...

   c. Right–clicking in the **Geometry** window, selecting **Insert**, and then selecting your desired boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Again, additional entries are typically required. For example, you may need to enter a **Magnitude** for the boundary condition, specify a **Coordinate System**, and/or define a **Direction**.

Types of Boundary Conditions

The Environment Context Tab (p. 61) groups most of the application's boundary conditions within the menus listed below.

- Inertial (p. 1500)
- Loads (p. 1520)
- Supports (p. 1663)
- Conditions (p. 1697)
- Direct FE (p. 1719)

See the following sections for information about how to import loading conditions as well as how to apply and scope abstract loading through the use of remote conditions.

- Remote Boundary Conditions (p. 1735)
- Imported Boundary Conditions (p. 1736)

Inertial Type Boundary Conditions

The boundary conditions contained under the **Inertial** heading are listed below.

- Acceleration (p. 1501)
- Standard Earth Gravity (p. 1509)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Rotational Velocity (p. 1511)
- Rotational Acceleration (p. 1516)

Acceleration

The global **Acceleration** boundary condition defines a linear acceleration of a structure in each of the global Cartesian axis directions.

If desired, acceleration can be used to simulate gravity (by using inertial effects) by accelerating a structure in the direction opposite of gravity (the natural phenomenon of). That is, accelerating a structure vertically upwards (+Y) at 9.80665 m/s^2 (in metric units), applies a force on the structure in the opposite direction (-Y) inducing gravity (pushing the structure back towards earth). Units are length/$time^2$.

Alternatively, you can use the Standard Earth Gravity (p. 1509) load to produce the effect of gravity. Gravity and Acceleration are essentially the same type of load except they have opposite sign conventions and gravity has a fixed magnitude. For applied gravity, a body tends to move in the direction of gravity and for applied acceleration, a body tends to move in the direction opposite of the acceleration.

Acceleration as a Base Excitation

Acceleration can also be defined as a base excitation during a Mode-Superposition Transient analysis or a Mode Superposition Harmonic Response analyses. You scope base excitations to a boundary condition. You can scope multiple base excitations to the same boundary condition, but the base excitations cannot have same direction specified (via the **Direction** property).

Important:

When duplicating an analysis within Mechanical that includes loads with the **Base Excitation** property set to **Yes** (Acceleration and/or Displacement), these loads will lose their scoping during the duplication process.

Analysis Types

Acceleration is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Harmonic Acoustics
- Modal Acoustics
- Rigid Dynamics (p. 550)
- Static Acoustics
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Harmonic Response (p. 496)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

Acceleration applies a uniform load over all bodies.

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types and Topology: By virtue of Acceleration's physical characteristics, this boundary condition is always applied to all bodies of a model.

Loading Types: The boundary condition's loading is defined using one of the following options:

- **Vector**. *Supported*.

  While loads are associative with geometry changes, load directions are not. This applies to any load that requires a vector input, such as acceleration.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Components**. *Supported*.
- **Magnitude - Phase**. *Supported* for Acceleration as a Base Excitation for Harmonic Response Analysis only.
- **Real - Imaginary**. *Supported* for Acceleration as a Base Excitation for Harmonic Response Analysis only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Time Varying)
- **Tabular** (Step Varying): *Supported* for Static Structural and Rigid Dynamics analyses only.
- **Tabular** (Frequency Varying): *Supported* for Harmonic Response Analysis only.

  By default, at least two frequency entries are required when defining a frequency dependent tabular load.

- **Function** (Time Varying): Not supported for Explicit Dynamics or LS-DYNA analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For a Rigid Dynamics analysis, if the Acceleration is defined using an expression, you cannot also have a Standard Earth Gravity object.

Boundary Condition Application

To apply **Acceleration**:

1. On the **Environment** Context tab: click **Inertial**>**Acceleration**. Or, right-click the **Environment** object in the tree or in the **Geometry** window and select **Insert**>**Acceleration**.
2. Select the method used to define the **Acceleration**: options include **Vector** (default) or **Components**.
3. Define the loading inputs: **Magnitude**, **Coordinate System**, and/or **Direction** of the Acceleration based on the above selections.

To apply **Acceleration** as a Base Excitation when the **Solver Type** property is defined as **Mode-Superposition** during a Transient (default setting for a Transient configured to a Modal solution) or a Mode-Superposition Harmonic Response analysis:

1. Set the **Base Excitation** property to **Yes**.
2. The **Boundary Condition** property provides a drop-down list of the boundary conditions that correspond to the **Acceleration**. Make a selection from this list. Valid boundary conditions for excitations include:
   - **Fixed Support**
   - **All Fixed Supports**
   - **Displacement**
   - **Remote Displacement**
   - **Nodal Displacement**
   - **Spring**: Body-to-Ground
3. The **Absolute Result** property is set to **Yes** by default. As needed, change the value to **No** if you do not want to include enforced motion.

   Note:

   If you apply more than one base excitation (either **Displacement** or **Acceleration**), the **Absolute Result** property needs to have the same setting, either **Yes** or **No**.

4. As needed, set the **Define By** property to **Real - Imaginary** from **Magnitude - Phase** (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

5. For Harmonic analyses, specify a **Phase Angle** as needed.

Note:

You can scope **Acceleration** or **Displacement** as a base excitation to the same boundary condition, but the base excitations cannot have same direction specified (via the **Direction** property).

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Geometry**: Read-only field indicating **All Bodies**. **Boundary Condition** (*Acceleration as a Base Excitation only*): drop-down list of available boundary conditions for application.
Definition	**Base Excitation** (*Acceleration as a Base Excitation only*): **No** is the default setting. Set to **Yes** to specify the **Acceleration** as a Base Excitation. **Absolute Result** (*Acceleration as a Base Excitation only*): This option allows you to include enforced motion with (**Yes** - default) or without (**No**) base motion. **Define By**: options for this property include the following. • **Vector**: Requires the specification of the following inputs: – **Magnitude** – **Phase Angle** (*Acceleration as a Base Excitation for MSUP Harmonic Response only*). – **Direction** • **Components**: Requires the specification of the following inputs. – **X Component** – **Y Component** – **Z Component** • **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. When using cyclic symmetry (p. 1134), the referenced coordinate system must match coordinate system used in the Cyclic Region (p. 1181). For a 2D axisymmetric model (p. 878) the referenced coordinate system must be the Global Coordinate System. The referenced coordinate system must be Cartesian.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	• **Magnitude - Phase**. (Acceleration as a Base Excitation for Harmonic Response Analysis only. Requires the specification of the following inputs: – **Magnitude** – **Phase Angle** • **Real - Imaginary**. (Acceleration as a Base Excitation for Harmonic Response Analysis only. Requires the specification of the following inputs: – **Magnitude - Real** – **Magnitude - Imag** **Suppressed**: Include (**No**) or exclude (**Yes**) the boundary condition.
Step Controls (Visible for Harmonic Response analysis with multiple load steps defined.)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the Remote Load is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Specify a **Step Selection** value from the values available in the **RPM Value** property of the **Analysis Settings** object to use for the Remote Load. **Step Selection**: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to **Load Step**. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the Remote Load. **Note:** Step varying displacements can only be applied in one direction for all load steps, for a selected set of nodes.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Load Vector Number: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable for this boundary condition.

- Acceleration is applied using the **ACEL** command.
- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.

Note:

- Should both an **Acceleration** and a **Standard Earth Gravity** boundary condition be specified, a composite vector addition of the two is delivered to the solver.
- In a Mode-Superposition Transient analysis, **Standard Earth Gravity** is not allowed in conjunction with **Acceleration**.

Condensed Parts

Important:

When working with substructures (p. 781), if the inertial acceleration is scoped to a **Condensed Part**, the nodes of the condensed part are not marked as the master degree of freedom. Instead, multiple super element load vectors are generated for each acceleration component via the substructure restart mechanism and scaled using **SFE ,,SELV** command. See the **ANTYPE ,SUBSTR,RESTART** command in the *Mechanical APDL Command Reference*. Also see the CMS Generation Pass section in the *Mechanical APDL Substructuring Analysis Guide* for additional information.

Note:

If your analysis includes a **Condensed Part**, the application does not support **Acceleration** when the **Step Varying** property is set to **Yes**.

Base Excitation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The following Mechanical APDL commands and considerations are applicable when Acceleration is defined as a base excitation in a Mode Superposition Transient analysis or a Mode Superposition Harmonic Response analysis.

- **Magnitude** (constant or tabular) is always represented as a table in the input file.
- Base excitation is defined using the **D** command under the Modal restart analysis (under Modal analysis in case of Standalone Harmonic Response analysis).
- Base excitation is applied using the **DVAL** command during a Mode Superposition Transient analysis or Mode Superposition Harmonic Response analysis.

Note:

Acceleration can be defined as base excitation in a Modal linked Harmonic Response and Modal linked Transient analysis only when the upstream Modal analysis **Solver Type** (p. 1428) is set to **Program Controlled** (provided program sets solver type internally to **Direct**, **Supernode**, or **Subspace**) **Direct**, **Supernode**, or **Subspace**.

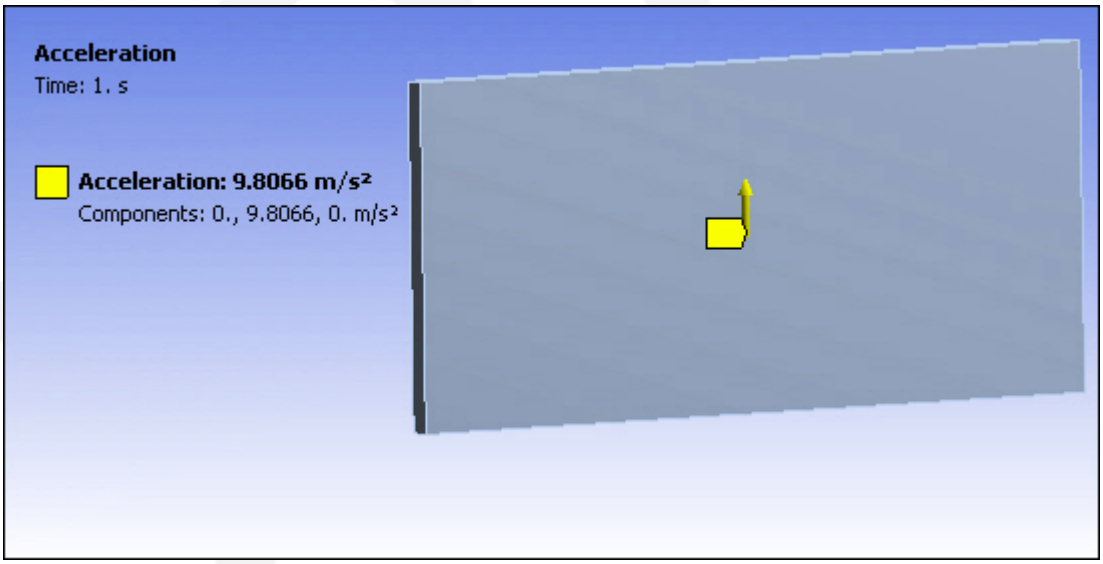
API Reference

See the Acceleration section of the ACT API Reference Guide for specific scripting information.

Acceleration Example

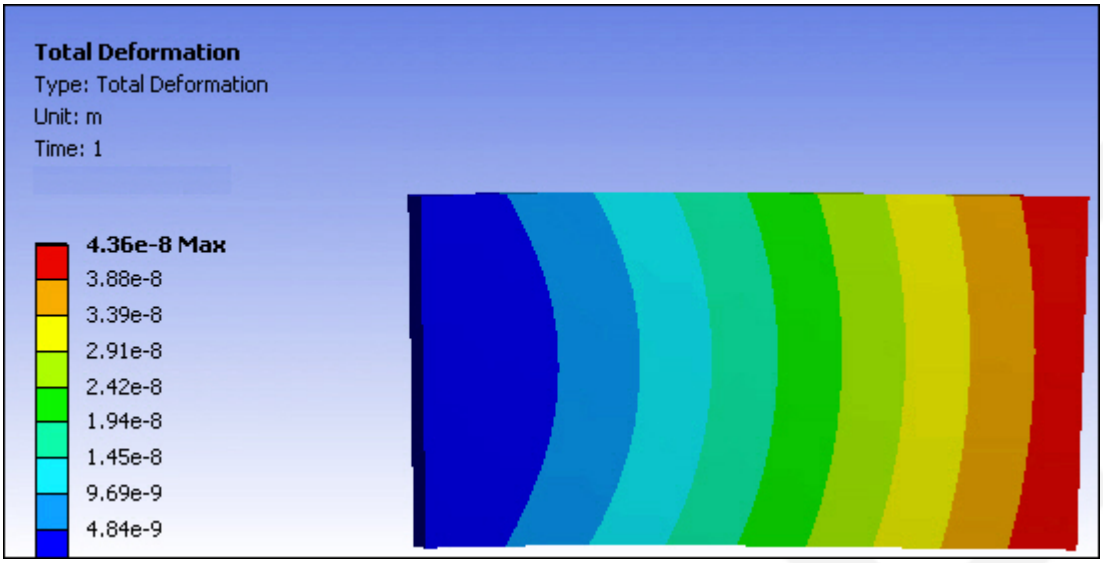
The following illustrations compare how Acceleration and Gravity can be used to specify a gravitational load with the same result.

Global Acceleration load applied in the +Y direction to simulate gravity.



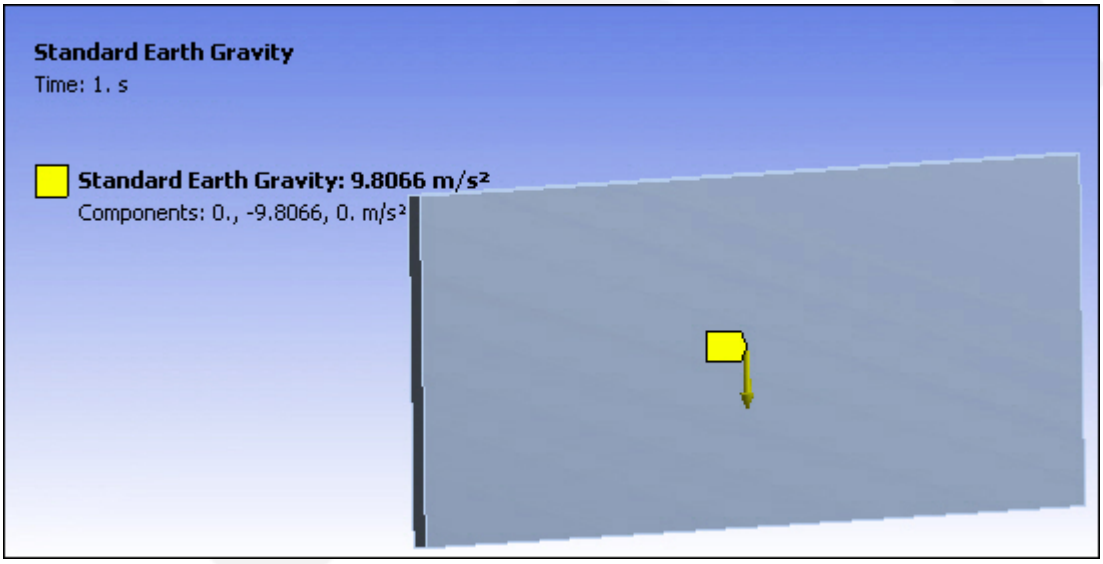
Resulting deformation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Standard Earth Gravity Example

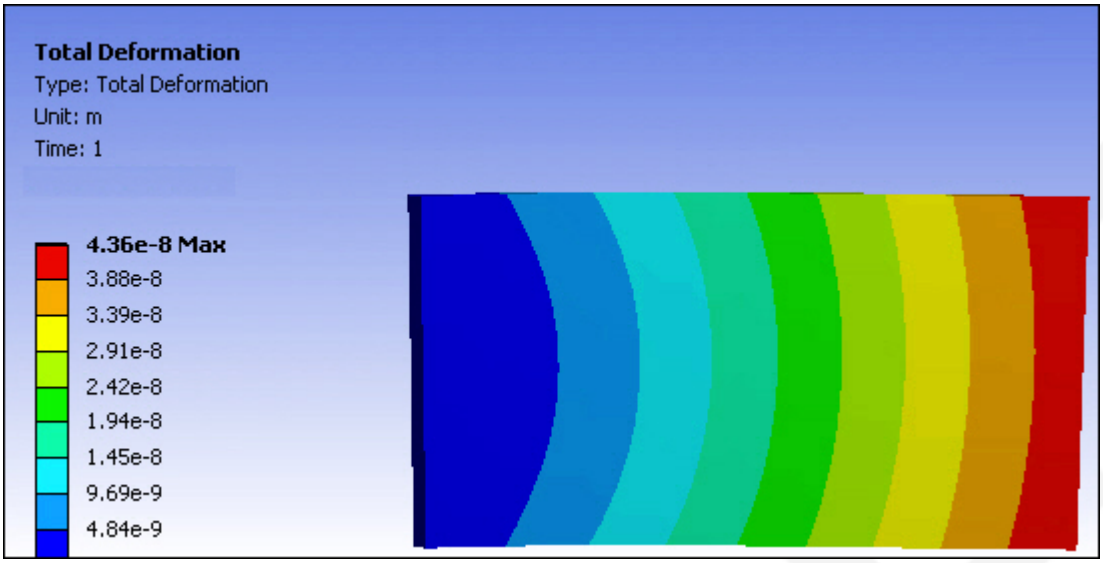
Standard Earth Gravity applied.



Resulting deformation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



API Reference

See the Acceleration section of the ACT API Reference Guide for specific scripting information.

Standard Earth Gravity

This boundary condition simulates gravitational effects on a body in the form of an external force.

Gravity is a specific example of acceleration with an opposite sign convention and a fixed magnitude. Gravity loads cause a body to move in the direction of gravity. Acceleration loads cause a body to move in the direction opposite of the acceleration. Refer to the example shown under Acceleration (p. 1501) for details.

Analysis Types

Standard Earth Gravity is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Rigid Dynamics (p. 550)
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types and Topology: By virtue of Standard Earth Gravity's physical characteristics, this boundary condition is always applied to all bodies of a model.

Loading Types: This boundary condition's loading is defined using a Coordinate System as the loading quantity.

Loading Data Definition (p. 1771): Standard Earth Gravity is constant, only the direction may be modified.

Boundary Condition Application

To apply **Standard Earth Gravity**:

1. On the **Environment** Context tab: click **Inertial**>**Standard Earth Gravity**.

   Or, right–click the **Environment** object in the tree or in the **Geometry** window and select **Insert**>**Standard Earth Gravity**.

2. Define the **Coordinate System** and/or **Direction** of the Standard Earth Gravity.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Geometry**: Read-only field indicating **All Bodies**.
Definition	**Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. When using cyclic symmetry (p. 1134) the referenced coordinate system must be the same coordinate system specified on the Cyclic Region (p. 1181). For a 2D axisymmetric model (p. 878) the referenced coordinate system must be the Global Coordinate System. The referenced coordinate system must be Cartesian. **X Component**: Read-only field with values for components based upon the **Direction** specification. **Y Component**: Read-only field with values for components based upon the **Direction** specification. **Z Component**: Read-only field with values for components based upon the **Direction** specification. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Direction: Define the vector in terms of any of the following directions: +x, -x, +y, -y, +z, -z.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property display. **Load Vector Number**: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.

Mechanical APDL References and Notes

Standard Earth Gravity is applied using the **ACEL** command.

Note:

Should both an Acceleration and a Standard Earth Gravity boundary condition be specified, a composite vector addition of the two is delivered to the solver.

API Reference

See the Standard Earth Gravity section of the ACT API Reference Guide for specific scripting information.

Rotational Velocity

Rotational velocity accounts for the structural effects of a part spinning at a constant rate.

Analysis Types

Rotational Velocity is available for the following analysis types:

- Coupled Field Modal (p. 386)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Modal Analysis (p. 515)
- Static Structural (p. 652)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Transient Structural (p. 665)
- LS-DYNA

Note:

- For a Transient Structural analysis that is linked to a Modal Analysis, Rotational Velocity is an invalid boundary condition in the Transient Structural analysis.
- For a Modal Analysis, Rotational Velocity is valid only when the following Analysis Settings (p. 1417) properties are specified:
  - **Damped** is set to **Yes** in the Solver Controls (p. 1428) group.
  - **Coriolis Effect** property is set to **On, Stationary Reference Frame** in the Rotordynamics Controls (p. 1490) group.
- If Rotational Velocity is defined in a Static Structural analysis, the spin softening effect is automatically included in rotating reference frame dynamics (**Coriolis Effect** set to **Off** in the Rotordynamics Controls (p. 1490) group). This may lead to negative or zero frequencies in a downstream perturbed Modal Analysis. This does not apply to stationary reference frame dynamics (**Coriolis Effect** set to **On, Stationary Reference Frame**).

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. A rotational velocity is applied along a user defined axis to one or more bodies.
- 2D Simulation: *Supported*. For 2D axisymmetric simulations, a Rotational Velocity load can only be applied about the y-axis.

Geometry Types: Geometry types supported for the **Rotational Velocity** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Rotational Velocity**.

Body: *Supported*. The following requirements must be met or the application will invalidate your load:

- A globally scoped (All Bodies) Rotational Velocity and a globally scoped (All Bodies) Rotational Acceleration (p. 1516) **may** coexist.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- A globally scoped Rotational Velocity **may** coexist with a partially scoped Rotational Acceleration (p. 1516).
- A partially scoped Rotational Velocity **may** coexist with a globally scoped Rotational Acceleration (p. 1516).
- Two globally scoped (All Bodies) rotational velocities **may not** coexist.
- A globally scoped (All Bodies) Rotational Velocity and a partially scoped Rotational Velocity **may not** coexist.
- A partially scoped Rotational Velocity **may not** share topology with another partially scoped Rotational Velocity.
- When using the Mechanical APDL solver target, a partially scoped Rotational Velocity **may not** share topology with partially scoped Rotational Acceleration (p. 1516).
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector**: *Supported*. While loads are associative with geometry changes, load directions are not.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Components**: *Supported*.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Time Varying)
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.

  Note:

  If you establish a step varying tabular load and you deactivate one of the steps, the application will ramp the value of this load to zero across the load step rather than immediately zeroing the value at the first substep.

- **Function** (Time Varying)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply rotational velocity to all bodies, in the Details view, accept the default **Geometry** setting of **All Bodies**.

To apply rotational velocity to selected bodies, in the Details view, set **Scoping Method** to either **Geometry Selection** or **Named Selection**, then either select the bodies in the **Geometry** window (hold down the **Ctrl** key to multiple select) or select from the list of the Named Selections available in the Details view.

To apply additional rotational velocity loads, you must have applied the original load to *selected* bodies, per above, not to **All Bodies**.

To apply a **Rotational Velocity**:

1. On the **Environment** Context tab: click **Inertial**>**Rotational Velocity**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Rotational Velocity**.
2. Select a **Scoping Method**.
3. Select the method used to define the Rotational Velocity: **Vector** (default) or **Components**.
4. Define the **Magnitude**, **Component** values, **Coordinate System**, and/or **Axis** of the Rotational Velocity based on the above selections.

> **Note:**
>
> The **Axis** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, the specified axis is unaffected by geometry updates, part transformation, or through the use of the **Configure** tool (for joints).

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body) and the number of geometric entities (for example: 1 Body, 2 Bodies) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Define By** (In a cyclic symmetry analysis, the Rotational Velocity must be defined by components.), options include: • **Vector**: A magnitude and directional axis (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Axis** • **Components**: Requires the specification of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. When using cyclic symmetry (p. 1134), the referenced coordinate system must match coordinate system used in the Cyclic Region (p. 1181). The referenced coordinate system must be Cylindrical. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Coordinate** – **Y Coordinate** – **Z Coordinate** **Note:** In a Modal analysis: • With multiple solve points (**Campbell Diagram** turned **On**), the magnitude or the resultant of the components must be in ascending order. • When specified by **Components**, only the Global Coordinate System is available (the option is read-only). **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

API Reference

See the Rotational Velocity section of the ACT API Reference Guide for specific scripting information.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable for this boundary condition.

- For global scoping, the application uses the **CGOMGA** command.
- For partially scoped bodies, the application uses the **CMOMEGA** command.

Rotational Acceleration

A Rotational Acceleration load applies a constant rotational acceleration to one or more bodies.

Analysis Types

Rotational Acceleration is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Static Structural (p. 652)
- Transient Structural (p. 665)
- LS-DYNA

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. A rotational Acceleration is applied along a user defined axis to one or more bodies.
- 2D Simulation: *Supported*. Not supported for 2D axisymmetric simulation. For 2D Plane Stress and Plane Strain simulations, a Rotational Acceleration load can only be applied about the Z-axis.

Geometry Types: Geometry types supported for the **Rotational Acceleration** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Rotational Acceleration**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Body: *Supported*. The following requirements must be met or the application will invalidate your load:

  – A globally scoped (All Bodies) Rotational Acceleration and a globally scoped (All Bodies) Rotational Velocity (p. 1511) **may** coexist.

  – A globally scoped Rotational Acceleration **may** coexist with a partially scoped Rotational Velocity (p. 1511).

  – A partially scoped Rotational Acceleration **may** coexist with a globally scoped (All Bodies) Rotational Velocity (p. 1511).

  – Two globally scoped (All Bodies) rotational accelerations **may not** coexist.

  – A globally scoped (All Bodies) Rotational Acceleration and a partially scoped Rotational Acceleration **may not** coexist.

  – A partially scoped Rotational Acceleration **may not** share topology with another partially scoped Rotational Acceleration.

  – When using the Mechanical APDL solver target, a partially scoped Rotational Acceleration **may not** share topology with partially scoped Rotational Velocity (p. 1511).

  For global scoping, the application uses the **DCGOMG** command. For partially scoped bodies, the application uses the **CMDOMEGA** command.

- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector**: *Supported*. While loads are associative with geometry changes, load directions are not.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Components**: *Supported*.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Time Varying)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.

  Note:

  If you establish a step varying tabular load and you deactivate one of the steps, the application will ramp the value of this load to zero across the load step rather than immediately zeroing the value at the first substep.

- **Function** (Time Varying)

Boundary Condition Application

To apply rotational acceleration to all bodies, in the Details view, accept the default **Geometry** setting of **All Bodies**.

To apply rotational acceleration to selected bodies, in the Details view, set **Scoping Method** to either **Geometry Selection** or **Named Selection**, then either select the bodies in the **Geometry** window (hold down the **Ctrl** key to multiple select) or select from the list of the Named Selections available in the Details view.

To apply additional rotational acceleration loads, you must have applied the original load to *selected* bodies, per above, not to **All Bodies**.

To apply a **Rotational Acceleration**:

1. On the **Environment** Context tab: click **Inertial**>**Rotational Acceleration**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Rotational Acceleration**.
2. Select a **Scoping Method**.
3. Select the method used to define the Rotational Acceleration: **Vector** (default) or **Components**.
4. Define the **Magnitude**, **Component** values, **Coordinate System**, and/or **Axis** of the Rotational Acceleration based on the above selections.

   Note:

   The **Axis** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, the specified axis is unaffected by geometry updates, part transformation, or through the use of the **Configure** tool (for joints).

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body) and the number of geometric entities (for example: 1 Body, 2 Bodies) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Define By** (In a cyclic symmetry analysis, the Rotational Acceleration must be defined by components.), options include: • **Vector**: A magnitude and directional axis (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Axis** • **Components**: Requires the specification of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. When using cyclic symmetry (p. 1134), the referenced coordinate system must match coordinate system used in the Cyclic Region (p. 1181). The referenced coordinate system must be Cylindrical. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Coordinate** – **Y Coordinate** – **Z Coordinate** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

API Reference

See the Rotational Acceleration section of the ACT API Reference Guide for specific scripting information.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable for this boundary condition.

- For global scoping, the application uses the **DCGOMG** command.
- For partially scoped bodies, the application uses the **CMDOMEGA** command.

Load Type Boundary Conditions

The boundary conditions contained under the **Loads** heading are listed below. They are separated into groups based on their physics and the applicable analysis types.

Structural Loading Conditions

Pressure (p. 1521)
Pipe Pressure (p. 1532)
Pipe Temperature (p. 1534)
Hydrostatic Pressure (p. 1537)
Force (p. 1543)
Remote Force (p. 1552)
Bearing Load (p. 1561)
Bolt Pretension (p. 1566)
Moment (p. 1574)
Generalized Plain Strain (p. 1582)
Line Pressure (p. 1585)
PSD Base Excitation (p. 1589)
RS Base Excitation (p. 1590)
Joint Load (p. 1592)
Thermal Condition (p. 1594)
Rotating Force (p. 1655)
Imported CFD Pressure (p. 1661)

Thermal Loading Conditions

Temperature (p. 1598)
Convection (p. 1602)
Radiation (p. 1609)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Heat Flow (p. 1614)
Heat Flux (p. 1617)
Internal Heat Generation (p. 1620)
Mass Flow Rate (p. 1622)

Electric Loading Conditions

Electric Charge (p. 1625)
Voltage (p. 1627)
Voltage (Ground) (p. 1633)
Current (p. 1630)
Thermal Condition (p. 1594)

Magnetostatic Loads

Electromagnetic Boundary Conditions and Excitations (p. 1635)
Magnetic Flux Boundary Conditions (p. 1636)
Conductor (p. 1637)

Interaction Loads

Motion Load (p. 1645)
Fluid Solid Interface (p. 1648)
System Coupling Region (p. 1651)

Explosive Initiation

Detonation Point

Pressure

A pressure load applies a constant pressure or a varying pressure in a single direction (x, y, or z) to one or more flat or curved faces. A positive value for pressure acts into the face, compressing the solid body.

Note:

You can add a **Pressure** during a Solution Restart (p. 1815). By default, the application sets the **Applied By** property to **Direct**.

Analysis Types

Pressure is available for the following analysis types:

- Coupled Field Harmonic (p. 380)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Note:

Eigen response (an Eigenvalue Buckling Analysis or a Modal Analysis) and Harmonic Response (Full) analyses take into account any pressure load stiffness contribution applied in a linked Static Structural analysis.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. For 3D simulations, a pressure load applies a pressure to one or more faces.
- 2D Simulation: *Supported*. For 2D simulations, a pressure load applies a pressure to one or more edges.

Geometry Types

Geometry types supported for the **Pressure** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology

The following topology selection options are supported for **Pressure**:

- Body: **Not Supported**.
- Face: *Supported - 3D Only*. If you select multiple faces when defining the pressure, the same pressure value gets applied to all selected faces. If a constant pressurized face

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

enlarges due to a change in CAD parameters, the total load applied to the face increases, but the pressure (force per unit area) value remains constant.

- Edge: *Supported - 2D*. If you select multiple edges when defining the pressure, the same pressure value gets applied to all selected edges.

  Note:

  You cannot apply a single pressure load on multiple edges of different surface bodies, when the bodies includes both a 3D surface body and a 2D surface body.

- Vertex: **Not Supported**.
- Nodes: *Supported*. When you scope to nodes, the load behaves as a Nodal Pressure (p. 1724) and follows all of the requirements of that loading condition.

  Note:

  Node-based scoping is not supported for a Harmonic Response analysis.

- Element Face: *Supported - 3D Only*.
- Element: **Not Supported**.

Loading Types

The boundary condition's loading is defined (via the **Defined By** property) using one of the following options:

- **Components**
- **Vector** (While loads are associative with geometry changes, load directions are not.)
- **Normal To** (default)
- **Normal To: Real - Imaginary** (Harmonic Response analysis only.)
- **Vector: Real - Imaginary** (Harmonic Response analysis only.)
- **Components: Real - Imaginary** (Harmonic Response analysis only.)
- **Table Assignment**

  Note:

  The Components and Vector options apply pressure in a constant direction and as a result do not contribute to any pressure load stiffness.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When using the Mechanical APDL solver, for all of the above **Defined By** property options, the **Pressure** boundary condition also displays the **Applied By** property. This property has two options: **Surface Effect** (default for non-Mode Superposition) and **Direct** (Mode Superposition default).

The **Surface Effect** option applies pressure using the surface effect elements created on the top of the scoped geometry. The **Direct** option applies pressure directly onto the faces of solid or shell elements in 3D analyses. In 2D analyses, the **Direct** option applies pressure directly onto the edges of plane elements.

Note:

- If you scope two **Pressure** objects to the same geometry, and specify the loads in the same direction, using the **Direct** option, the pressures do not produce a cumulative loading effect. The **Pressure** object that you specified last takes priority and is applied, and as a result, the application ignores the other **Pressure** object.
- If a **Nodal Pressure** and a direct **Pressure** share the same scoping, the **Nodal Pressure** always takes priority regardless of insertion order: Mechanical will ignore the direct **Pressure**.
- A pressure applied using the **Surface Effect** option and a pressure applied with the Direct option produce a resultant effect.
- When you scope a Pressure to a solid body as well as a shell body, the application does not display the annotation arrow for the loading direction (via the **Direction** property).
- During a structural analysis, you can also create a spatially varying load using the **Vector** type option. A spatially varying load allows you to define the pressure in tabular form or as a function.
- 3D Faces or 2D Edges automatically update their direction at each substep and "follow" the changing normal for large deflection analyses.
- If your analysis includes some combination of a **Pressure**, a **Force**, and a **Hydrostatic Pressure** load, and 1) all are set to the **Direct** option and 2) share the same scoping, 3) have the same **Direction**, whichever load was written to the input file last, overwrites all previous loads.
- If you have a **Nodal Force** and a **Pressure** applied using the **Direct** option and they share the same scoping, they produce a resultant loading effect.
- Applying a pressure load Normal To faces (3D) or edges (2D) could result in a pressure load stiffness (p. 353) contribution that plays a significant role in analyses

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

that support pre-stress (Pre-stressed Full Harmonic, Pre-stressed Modal, and Eigenvalue Buckling) because they use the Static Structural Solution as a starting point.

Important:

Note the following limitations when using the **Direct** option for **Applied By** property and when the **Defined By** property is set to **Vector** or **Components**:

- Only supported when the target solver type is Mechanical APDL.
- Not supported for vertices and edges of Solid bodies and Line Bodies.
- Not supported on bodies associated with Condensed parts.
- Not supported if the model has any crack defined under the fracture folder.
- Not supported if the analysis includes a **Nonlinear Adaptive Region** object and 1) the **Loaded Area** property is set to **Initial** and 2) the **Large Deflection** property (**Analysis Settings** > Solver Controls (p. 1428)) set to **On**.
- In a multiple step analysis, if you define more than one load (Pressure, Force, or Hydrostatic Pressure) using the **Direct** option and a **Nodal Pressure**, and they share the same scoping, deactivation of a particular load step in one of these loads could delete all the other loads in that load step and following steps.

Loading Data Definition (p. 1771)

Enter loading data using one of the following options:

- **Constant**: *Supported*.
- **Table Assignment**: *Supported*.
- **Tabular (Time Varying)**: *Supported*.
- **Tabular (Step Varying)**: *Supported* for Static Structural analysis only.
- **Tabular (Harmonic Index Varying)**: *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** object is defined. By default, at least two harmonic index entries are required when defining a Harmonic Index dependent tabular load.
- **Tabular (Sector Number)**: *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** is defined. By default, at least two entries are required to define a dependent tabular load.
- **Tabular (Frequency Varying)**: *Supported* for Harmonic Response Analysis only.

  By default, at least two frequency entries are required when defining a frequency dependent tabular load. The **Pressure** boundary condition in a Harmonic Response (Full, linked MSUP, or standalone) can be defined in such a way that it is fully frequency de-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

pendent. That is, the magnitude of the load as well as the **Phase Angle** of the load can be dependent upon the frequency definitions.

- **Tabular (Spatially Varying)**: *Supported*.
- **Function (Time Varying)**: *Supported*.
- **Function (Spatially Varying)**: *Supported*.

Note:

Harmonic Response Analysis Only: Spatially varying Tabular and Function data is supported for the **Normal To** and **Normal To: Real-Imaginary** loading types. The **Phase Angle** property supports Spatially varying Tabular definition but does not support Function definition.

Boundary Condition Application

To apply a **Pressure**:

1. Select the **Pressure** option from the **Environment** Context tab. Or, right-click the **Environment** object, or in the **Geometry** window, and select **Insert**>**Pressure**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Specify the **Define By** property. Options include:
   - **Components**
   - **Components: Real – Imaginary** (Harmonic Response (p. 496) Only)
   - **Vector**
   - **Vector: Real – Imaginary** (Harmonic Response Only)
   - **Normal To** (default)
   - **Normal To: Real – Imaginary** (Harmonic Response Only)
   - **Table Assignment**
4. Specify the **Applied By** property as needed: **Surface Effect** (default non-Mode Superposition) and **Direct** (Mode Superposition default).
5. Specify the **Loaded Area** property as needed: options include **Deformed** (default) and **Initial**.
6. Based on your **Defined By** property selection, specify the following additional properties as required:
   - **Magnitude** (including component magnitudes)
   - **Direction**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Coordinate System**
- **Pressure Table Assignment**

Note:

- The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).
- The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

7. For Harmonic analyses, specify a **Phase Angle** as needed.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: This property specifies how you perform geometric entity selection. Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. **Shared Reference Body**: This property is a scoping feature used to apply a Pressure to shared faces or edges between two bodies. When you have properly scoped the geometric entities, using either Geometry Selection or an appropriate Named Selection, the property provides a drop-down list of the names of the bodies that share the scoped features. By selecting a body from this list, you are specifying that the pressure be applied to its surface or edge. Once selected, the Geometry or Named Selection property displays a parenthetical of the shared face or edge, "1 Shared Face/1 Shared Edge," to indicate the condition. **Note:** • When working with the shared edges of a 2D model, the **Applied By** property must be set to **Direct**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	• Edges can be shared between a surface and line body, but the line body cannot be specified as the **Shared Reference Body** or the property becomes invalid.
Definition	**Type**: Read-only field that describes the object - **Pressure**. **Define By**, options include: • **Normal To**: Requires a **Magnitude** entry. – **Magnitude** – **Phase Angle** (Harmonic Analysis (p. 496) only) • **Normal To: Real - Imaginary** (Harmonic Analysis (p. 496) only): Real and imaginary magnitude. Requires the specification of the following inputs: – **Magnitude - Real** – **Magnitude - Imag** • **Table Assignment**: A multi-variable table (p. 1103) of numeric values that defines the pressure distribution for the selection. Requires additional input under **Pressure Table Assignment** (p. 1531). • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Direction** – **Phase Angle** (Harmonic Analysis (p. 496) only) • **Vector: Real - Imaginary** (Harmonic Analysis (p. 496) only): Real and imaginary magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude - Real** – **Magnitude - Imag** – **Direction** • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Phase Angle** (Harmonic Analysis (p. 496) only) – **Y Phase Angle** (Harmonic Analysis (p. 496) only) – **Z Phase Angle** (Harmonic Analysis (p. 496) only) **Note:** Selection of a Coordinate System rotated out of the global Cartesian X-Y plane is not supported in a 2D analysis. • **Components: Real - Imaginary** (Harmonic Analysis (p. 496) only): Option to define the loading type as real and imaginary components (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component - Real**: Defines magnitude (Real) in the X direction. – **X Component - Imaginary**: Defines magnitude (Imaginary) in the X direction. – **Y Component - Real**: Defines magnitude (Real) in the Y direction. – **Y Component - Imaginary**: Defines magnitude (Imaginary) in the Y direction. – **Z Component - Real**: Defines magnitude (Real) in the Z direction. – **Z Component - Imaginary**: Defines (Imaginary) magnitude in the Z direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Applied By: This property defines how the load is applied, either by creating surface effect elements or by direct application on the scoped geometry. Options include: • **Surface Effect**: Default option for all analyses except Mode Superposition analyses. • **Direct**: Default option for Mode Superposition analyses and when solution restart (p. 1815) points are available. **Loaded Area**: Options include **Deformed** (default) and **Initial**. The **Initial** option treats the scoped surface area as a constant throughout the analysis. For the **Deformed** option, the application incorporates the change in the surface area as a result of deformation throughout the analysis. The selection for this property can be of significance during large deflection problems. **Non-Cyclic Loading Type**: This property is available for a Full Harmonic Response analysis when a Cyclic Region is present in the model. Options include: • **No** (default). The loading is purely cyclic. That is, the load applied to the base sector is applied to each and every sector. • **Harmonic Index**. The non-cyclic loading can be specified for one or more harmonic indices. This is also known as "engine-order loading" (or traveling wave excitation). A **Harmonic Index** entry is required. • **Sector Number**: This property enables you to select the desired Sector onto which you apply the non-cyclic load. **Important:** When you specify the load as **Tabular**, the **Independent Variable** property displays and is set to **Harmonic Index** by default. The **Harmonic Index** property is hidden as their values are entered in the table. **Harmonic Index**: This property displays when the **Non-Cyclic Loading Type** property is set to **Harmonic Index**. Where NS is Number of Sectors, enter a value from: 1 to NS/2; if NS is even. 1 to (NS-1)/2; if NS is odd. **Sector Number**: This property displays when the **Non-Cyclic Loading Type** property is set to **Sector Number**. You use this property to specify the desired sector onto which you want to apply the non-cyclic load. If you choose **Tabular** for the **Magnitude** property, you can then set

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	the **Independent Variable** property to **Sector Number** and then specify your loading using the **Tabular Data** window. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays. **Load Vector Number**: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.
Pressure Table Assignment (Table Assignment Only)	Specify the following inputs to define a pressure load with a table: • **Table Assignment** • Parameterization: Check the box next to **Table Assignment** to parameterize the table. • **Spatial Coordinate System** (if applicable) • **Graphics Controls** (if applicable) See Specify Pressure Loads with Tables (p. 1781) for details.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- When you set the **Applied By** property to **Surface Effect**, the Pressure is applied as a surface load through the surface effect elements using the **SF** or **SFE** command using the SURF154 (3D) and SURF153 (2D) element types.
- When you set the **Applied By** property to **Direct**, a Pressure is applied directly on to the element faces using the **SFCONTROL** and **SFE ,,PRES** commands. Refer to **SFCONTROL** command for a list of supported solid elements, shell elements, and plane-2D elements.
- **Magnitude** (tabular and function) is always represented as a table in the input file.
- **Table** is a special type of numeric array that enables the Mechanical APDL solver to calculate (through linear interpolation) the values between entries in a multi-dimensional table. The solver

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

applies these interpolated values across the selected geometry when it computes the solution. See the discussion in Array Parameters.

API Reference

See the Pressure section of the ACT API Reference Guide for specific scripting information.

Pipe Pressure

Used in any structural analysis, Pipe Pressure is useful for pipe stress analysis and pipe design. Pipe Pressure is applied only to pipes in the form of line bodies.

Analysis Types

Pipe Pressure is available for the following analysis types:

- Harmonic Response (p. 496)
- Explicit Dynamics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. For 3D structural analyses, a pipe pressure load applies a constant, tabular, or functional variation of pressure to one or more line bodies (p. 872) which are set to be pipes.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Pipe Pressure** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: *Line Bodies Only*.

Topology: The following topology selection options are supported for **Pipe Pressure**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*.
- Vertex: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The loading type, by default, is program controlled. Internal and external pressures are input on an average basis. By default, when the pipe is subjected to internal and external pressures, the end-cap pressure effect of the pipe is included. This implies that the end caps are always in equilibrium, that is, no net forces are produced.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Time Varying)
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Spatially Varying)
- **Function** (Time Varying)
- **Function** (Spatially Varying)

Boundary Condition Application

To apply a **Pipe Pressure**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Pipe Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Pipe Pressure**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry. Pipe pressure can only be scoped to line bodies which are set to be pipes.
3. Define **Magnitude** as a constant, tabular, or functional input.
4. Select **Loading** to be **Internal** or **External** according to your problem.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Pipe Pressure**. **Magnitude**: Input field to define the magnitude of the Pipe Pressure. This value can be defined as a **Constant** or in **Tabular** form, as well as **Imported**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition. **Loading**: Specify whether the loading is **Internal** or **External**.

Mechanical APDL References and Notes

The following Mechanical APDL element types are applicable for this boundary condition. Both elements are based on Timoshenko beam theory which includes shear-deformation effects.

- PIPE288: 3D two-node pipe
- PIPE289: 3D three-node pipe.
- ELBOW290: special 3D three-node pipe used for modeling curved pipes. This element is also used when Pipe Idealization is scoped to a line body modeled as pipe and meshed with higher order elements. PIPE289 is converted ELBOW290.

Displaying Contours and Displaced Shapes on Line Bodies: The contour results line bodies are expanded to be viewed on the cross section shape, but only one actual result exists at any given node and as a result no contour variations across a beam section occur. Therefore, for Mechanical APDL plot comparison, full graphics inside **/POST1** should be used when comparing numerical values.

API Reference

See the Pipe Pressure section of the ACT API Reference Guide for specific scripting information.

Pipe Temperature

For 3D structural analyses, a pipe temperature load applies a constant, tabular, or functional variation of temperature to one or more line bodies (p. 872) which are set to be pipes. You can select it to be internal pipe temperature or external pipe temperature from the Details view.

Analysis Types

Pipe Temperature is available for the following analysis types:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Pipe Temperature** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: *Line Bodies Only*.

Topology: The following topology selection options are supported for **Pipe Temperature**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The loading type is, by default, program controlled. Internal and external temperatures are input on an average basis.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**.
- **Tabular** (Time Varying).
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Spatially Varying).
- **Function** (Time Varying).
- **Function** (Spatially Varying).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply a **Pipe Temperature**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Pipe Temperature**. Or, right-click the **Environment** object or in the **Geometry** window and select **Insert**>**Pipe Temperature**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry. Pipe Temperature can only be scoped to line bodies that are specified as pipes.
3. Define **Magnitude** as a constant, tabular, or functional input.
4. Select **Loading** to be **Internal** or **External** according to your problem.

Note:

Given elbow elements (ELBOW290) with inner pipe temperature specifications only, the application will, by default, specify this inner temperature as outer temperature as well.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Pipe Temperature**. **Magnitude**: Input field to define the magnitude of the Pipe Pressure. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Loading: Specify whether the loading is **Internal** or **External**.

Mechanical APDL References and Notes

The Mechanical APDL Solver is the only solver available for this boundary condition.

API Reference

See the Pipe Temperature section of the ACT API Reference Guide for specific scripting information.

Hydrostatic Pressure

A hydrostatic pressure load simulates pressure that occurs due to fluid weight.

Analysis Types

Hydrostatic Pressure is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types

Geometry types supported for the **Hydrostatic Pressure** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology

The following topology selection options are supported for **Hydrostatic Pressure**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Body: **Not Supported**.
- Face: *Supported (3D)*.
- Edge: *Supported (2D)*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types

The boundary condition's loading is defined (via the **Defined By** property) using one of the following options:

- **Vector** (default)

Note:

The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Components**

When using the Mechanical APDL solver, for all of the above **Defined By** property options, the **Hydrostatic Pressure** boundary condition also displays the **Applied By** property. This property has two options: **Surface Effect** (default) and **Direct**.

The **Surface Effect** option applies pressure using the surface effect elements created on the top of the scoped geometry. The **Direct** option applies pressure directly onto the faces of solid or shell elements in 3D analyses. In 2D analyses, the **Direct** option applies pressure directly onto the edges of plane elements.

Note:

- If you scope two **Hydrostatic Pressure** objects to the same geometry, and specify the loads in the same direction, using the **Direct** option, the pressures do not produce a cumulative loading effect. The **Hydrostatic Pressure** object that you specified last takes priority and is applied, and as a result, the application ignores the other **Hydrostatic Pressure** object.
- If a **Nodal Pressure** and a direct **Hydrostatic Pressure** share the same scoping, the **Nodal Pressure** always takes priority regardless of insertion order: Mechanical will ignore the direct **Hydrostatic Pressure**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- A **Hydrostatic Pressure** using the **Surface Effect** option and **Hydrostatic Pressure** using the **Direct** option produce a resultant loading effect.
- A **Nodal Force** and a **Hydrostatic Pressure** applied using the **Direct** option and they share the same scoping, produce a resultant loading effect.
- If your analysis includes some combination of a **Pressure**, a **Force**, and a **Hydrostatic Pressure** load, and 1) all are set to the **Direct** option and 2) share the same scoping, 3) have the same **Direction**, whichever load was written to the input file last, overwrites all previous loads.

Important:

For the Mechanical APDL solver, note the following limitations when using the **Direct** option for the **Defined By** property options **Vector** or **Components**:

- Not supported on bodies associated with General Axisymmetric and Condensed parts.
- Not supported if the model has any cracks defined under the Fracture folder.
- Not supported if the analysis has a **Nonlinear Adaptive Region** defined.
- In a multiple step analysis, if you define more than one load (Pressure, Force, or Hydrostatic Pressure) using the **Direct** option and a **Nodal Pressure**, and they share the same scoping, deactivation of a particular load step in one of these loads could delete all the other loads in that load step and following steps.

Loading Data Definition (p. 1771)

Hydrostatic Pressure is defined as a constant.

Note:

During a multiple step analysis, tabular data is visible for this boundary condition. This information is read-only but you can use the context menu (right-click) features of the Tabular Data (p. 193) display to activate or deactivate the loading per step.

Boundary Condition Application

To apply a **Hydrostatic Pressure**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Hydrostatic Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Hydrostatic Pressure**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection**. Hydrostatic Pressure can only be scoped to faces.
3. Specify the **Applied By** property: **Surface Effect** (default) and **Direct**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Select all of the faces that will potentially enclose the fluid.

   Or...

   If you are working with a surface body, specify the **Shell Face**, defined as the side of the shell (**Top** or **Bottom**) on which to apply the hydrostatic pressure load.

5. Enter the **Fluid Density**.

6. Specify the properties of the **Hydrostatic Acceleration** category. This is typically the acceleration due to gravity, but can be other acceleration values depending on the modeling scenario. For example, if you were modeling rocket fuel in a rocket's fuel tank, the fuel might be undergoing a combination of acceleration due to gravity and acceleration due to the rocket accelerating while flying.

   Note:

   The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).

7. Specify the properties of the **Free Surface Location** category. These properties define the location of the *top* of the fluid in the container. You can specify this location by using coordinate systems, by entering coordinate values, or by clicking a location on the model.

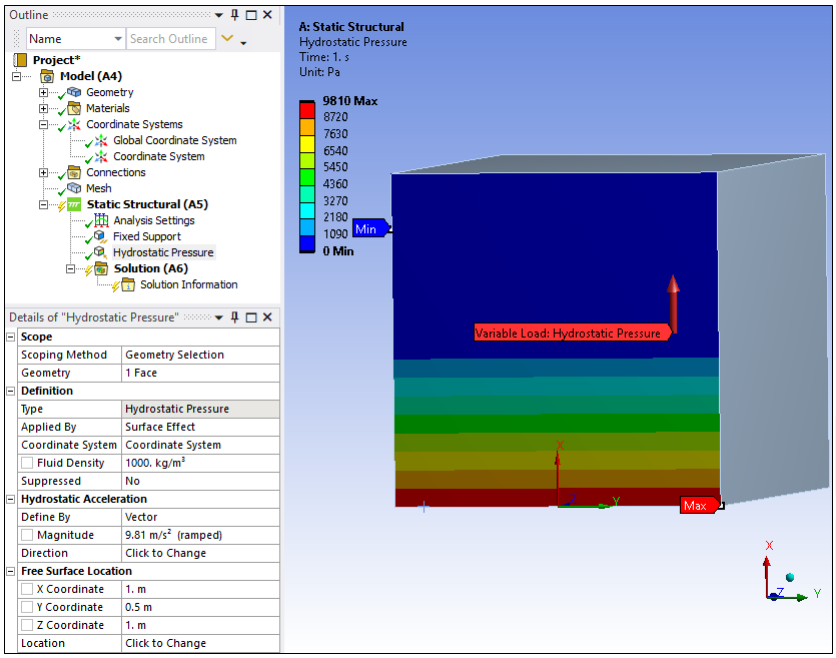
8. Mesh the model, then highlight the **Hydrostatic Pressure** object to display the pressure contours.

The following example shows the simulation of a hydrostatic pressure load on the wall of an aquarium. Here the wall is modeled as a single surface body. The load is scoped to the bottom side of the face. A fixed support is applied to the bottom edge. Acceleration due to gravity is used and the fluid density is entered as 1000 kg/m^3. Coordinates representing the top of the fluid are also entered.

The load plot shown here illustrates the hydrostatic pressure gradient.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: This property specifies how you perform geometric entity selection. Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. **Shared Reference Body**: This property is a scoping feature used to apply a Hydrostatic Pressure to shared faces or edges between two bodies. When you have properly scoped the geometric entities, using either Geometry Selection or an appropriate Named Selection, the property provides a drop-down list of the names of the bodies that share the scoped features. By selecting a body from this list, you are specifying that the pressure be applied to its surface or edge. Once selected, the Geometry or Named Selection property displays a

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	parenthetical of the shared face or edge, "1 Shared Face/1 Shared Edge," to indicate the condition. **Note:** • When working with the shared edges of a 2D model, the **Applied By** property must be set to **Direct**. • Edges can be shared between a surface and line body, but the line body cannot be specified as the **Shared Reference Body** or the property becomes invalid. **Shell Face**: This property enables you to specify the **Top** (default) or **Bottom** of the selected face.
Definition	**Type**: Read-only field that displays boundary condition type - **Hydrostatic Pressure**. **Applied By**: This property defines how the load is applied. Either by creating surface effect elements or by direct application on the scoped geometry. Options include: • **Surface Effect** (default) • **Direct** **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition. **Fluid Density**: Input field for fluid density value.
Hydrostatic Acceleration	**Define By**, options include: • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Direction** • **Components** – **X Component** – **Y Component** – **Z Component**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Free Surface Location	**X Coordinate** **Y Coordinate** **Z Coordinate** **Location**: Specify **Free Surface Location** using geometry picking tools. Valid topologies include: face, edge, vertex.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- When you set the **Applied By** property to **Surface Effect**, the Hydrostatic Pressure is applied as a surface load through the surface effect elements using the **SFE** command.
- When you set the **Applied By** property to **Direct**, a Hydrostatic Pressure is applied directly on to the element faces using the **SFCONTROL** and **SFE ,,PRES** commands. Refer to **SFCONTROL** command for a list of supported solid elements, shell elements, and plane-2D elements.
- Hydrostatic Pressure is represented as a table in the input file.

API Reference

See the Hydrostatic Pressure section of the ACT API Reference Guide for specific scripting information.

Force

Force is specified based on the following topologies:

- **Vertex**: Applies a force vector to one or more vertices.
- **Edge**: Distributes a force vector along one or more straight or curved edges, resulting in uniform line load along the edge.
- **Face**: Distributes a force vector across one or more flat or curved faces, resulting in uniform traction across the face.
- **Node**: Applies a force to an individual node or a set of nodes. This scoping is the same as using an Nodal Force (p. 1721) except that you scope the nodes directly (no Named Selection is required). As such, the force is applied using the Mechanical APDL **F** command.

  Note:

  Node-based scoping is not supported for Harmonic Response or Explicit Dynamics analyses.

- **Element Face**: Distributes a force across one or more element faces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Types

Force is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*. Force loads are not supported for 2D axisymmetric Explicit Dynamics analyses.

Geometry Types

Geometry types supported for the **Force** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*

Topology

The following topology selection options are supported for **Force**.

- Body: **Not Supported**.
- Face: *Supported*.
  - The force is applied by converting it to a pressure, based on the total area of all the selected faces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - If a face enlarges due to a change in CAD parameters, the total load magnitude applied to the face remains constant.

- Edge: *Supported*.

  - If you select multiple edges when defining the force, the magnitude of the force is distributed evenly across all selected edges.

  - If an edge enlarges due to a change in CAD parameters, the total load magnitude applied to the edge remains constant.

Note:

You cannot apply a single force load on multiple edges of different surface bodies, when the bodies includes both a 3D surface body and a 2D surface body.

- Vertex: *Supported*.

  - If you select multiple vertices when defining the force, the magnitude of the force is distributed evenly across all selected vertices.

  - A force applied to a vertex is not realistic and leads to singular stresses (that is, stresses that approach infinity near the loaded vertex). You should disregard stress and elastic strain values in the vicinity of the loaded vertex.

- Nodes: *Supported* (2D Only).

- Element Face: *Supported* (3D Only).

- Element: **Not Supported**.

Loading Types

The boundary condition's loading is defined (via the **Defined By** property) using one of the following options:

- **Vector** (Not supported for Node selection)

- **Components**

- **Vector: Real - Imaginary** (Harmonic Response analysis only.)

- **Components: Real - Imaginary** (Harmonic Response analysis only.)

When using the Mechanical APDL solver, for all of the above **Defined By** property options, the **Force** boundary condition also displays the **Applied By** property. This property has two options: **Surface Effect** (default for non-Mode Superposition) and **Direct** (Mode Superposition default).

The **Surface Effect** option applies force using the surface effect elements created on the top of the scoped geometry. The **Direct** option applies force directly onto the faces of solid or shell

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

elements in 3D analyses. In 2D analyses, the **Direct** option applies pressure directly onto the edges of plane elements.

Note:

- If you scope two **Force** objects to the same geometry, and specify the loads in the same direction, using the **Direct** option, the forces do not produce a cumulative loading effect. The **Force** object that you specified last takes priority and is applied, and as a result, the application ignores the other **Force** object.
- A **Nodal Force** and a **Force** applied using the **Direct** option and they share the same scoping, produce a resultant loading effect.
- A **Force** applied using the **Surface Effect** option and a **Force** applied using the **Direct** option produce a resultant effect.
- If your analysis includes some combination of a **Pressure**, a **Force**, and a **Hydrostatic Pressure** load, and 1) all are set to the **Direct** option and 2) share the same scoping, 3) have the same **Direction**, whichever load was written to the input file last, overwrites all previous loads.

Important:

For the Mechanical APDL solver, note the following limitations when using the **Direct** option for **Applied By** property:

- Not supported for vertices and edges of Solid bodies and Line bodies.
- Not supported on bodies associated with General Axisymmetric and Condensed parts.
- Not supported if the model has any cracks defined under the Fracture folder.
- Not supported if the analysis has a **Nonlinear Adaptive Region** defined.
- In a multiple step analysis, if you define more than one load (Pressure, Force, or Hydrostatic Pressure) using the **Direct** option, and they share the same scoping, deactivation of a particular load step in one of these loads could delete all the other loads in that load step and following steps.

Loading Data Definition (p. 1771)

Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Not supported* for Harmonic Response Analysis.
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Frequency Varying): *Supported* for Harmonic Response Analysis only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

By default, at least two frequency entries are required when defining a frequency dependent tabular load. The **Force** boundary condition in a Harmonic Response (Full, linked MSUP, or standalone) can be defined in such a way that it is fully frequency dependent. That is, the Magnitude of the load as well as the Phase Angle of the load can be dependent upon the frequency definitions.

- **Tabular** (Harmonic Index Varying): *Supported* only for a Harmonic Response (Full) analysis that includes a **Cyclic Region** object. By default, at least two harmonic index entries are required when defining a harmonic index dependent tabular load. The Force load for this analysis can be defined such that it is fully harmonic index dependent. That is, the magnitude of the load as well as the phase angle of the load can be dependent upon the harmonic index definitions.
- **Tabular (Sector Number)**: *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** is defined. By default, at least two entries are required to define a dependent tabular load.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): **Not Supported** for Harmonic Response Analysis.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply a **Force**:

1. Select the **Force** option from the **Environment** Context tab. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Force**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Select the method used to define the force: **Vector** (default), **Vector: Real - Imaginary**, **Components**, or **Components: Real - Imaginary**.
4. Specify the **Applied By** property: **Surface Effect** (non-Mode Superposition default) and **Direct** (Mode Superposition default).
5. Define the **Magnitude**, **Coordinate System** directional loading, and/or **Direction** of the load based on the above selections.

Note:

- The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).
- The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

6. For Harmonic analyses, specify a **Phase Angle** as needed.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: This property specifies how you perform geometric entity selection. Options include Geometry Selection (default) and Named Selection. **Geometry Selection**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Force**. **Define By**, options include: • **Vector**: A magnitude and direction (based on finite element selection). Requires the specification of the following inputs: **Magnitude**: Enter desired value. **Direction**: Specify using either the Node or Element Face selection option. When you select one Element Face, the Direction is normal to the selected face. When you select multiple Element Faces, the Direction is the average of the Normals of all the faces. When you select two nodes (singular selection is not supported), the Direction is the direction along a line segment that includes both points. If you select three or more Nodes, the Direction calculation equals the surface normal of the first three nodes selected. **Phase Angle** (Harmonic Analysis (p. 496) only) • **Vector: Real - Imaginary** (Harmonic Analysis (p. 496) only): Real and imaginary magnitude and direction (based on selected geometry). Requires the specification of the following inputs: **Magnitude - Real** **Magnitude - Imag** **Direction** • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Coordinate System: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **X Component**: Defines magnitude in the X direction. **Y Component**: Defines magnitude in the Y direction. **Z Component**: Defines magnitude in the Z direction. **X Phase Angle** (Harmonic Analysis (p. 496) only) **Y Phase Angle** (Harmonic Analysis (p. 496) only) **Z Phase Angle** (Harmonic Analysis (p. 496) only) **Note:** Selection of a Coordinate System rotated out of the global Cartesian X-Y plane is not supported in a 2D analysis. • **Components: Real - Imaginary** (Harmonic Analysis (p. 496) only): Option to define the loading type as real and imaginary components (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **X Component - Real**: Defines magnitude (Real) in the X direction. **X Component - Imaginary**: Defines magnitude (Imaginary) in the X direction. **Y Component - Real**: Defines magnitude (Real) in the Y direction. **Y Component - Imaginary**: Defines magnitude (Imaginary) in the Y direction. **Z Component - Real**: Defines magnitude (Real) in the Z direction. **Z Component - Imaginary**: Defines (Imaginary) magnitude in the Z direction. **Divide Load by Nodes** (visible for Node scoping only): Property options include **Yes** (default) and **No**. When set to **Yes**, the load value is normalized by dividing the **Magnitude** by number of scoped nodes. When set to **No**, the load value is applied directly to every scoped node. **Applied By**: This property defines how the load is applied. Either by creating surface effect elements or by direct application on the scoped geometry. Options include: • **Surface Effect**: default option for all analyses except Mode Superposition analyses.

Category	Fields/Options/Description
	• **Direct**: default option for Mode Superposition analyses. **Non-Cyclic Loading Type**: This property is available for Full Harmonic Analysis when Cyclic Symmetry is present in the model. Options include: • **No** (default). The loading is purely cyclic. That is, the load applied to the base sector is applied to each and every sector. • **Harmonic Index**. The non-cyclic loading can be specified for one or more harmonic indices. This is also known as "engine-order loading" (or traveling wave excitation). A **Harmonic Index** entry is required. • **Sector Number**: This property enables you to select the desired Sector onto which you apply the non-cyclic load. **Important:** When you specify the load as Tabular, the **Independent Variable** property displays and is set to **Harmonic Index** by default. The **Harmonic Index** property is hidden as their values are entered in the table. **Harmonic Index**: This property displays when the **Non-Cyclic Loading Type** property is set to **Harmonic Index**. Where NS is Number of Sectors, enter a value from: 1 to NS/2; if NS is even. 1 to (NS-1)/2; if NS is odd. **Sector Number**: This property displays when the **Non-Cyclic Loading Type** property is set to **Sector Number**. You use this property to specify the desired sector onto which you want to apply the non-cyclic load. If you choose **Tabular** for the **Magnitude** property, you can then set the **Independent Variable** property to **Sector Number** and then specify your loading using the **Tabular Data** window. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Step Controls (Visible for Harmonic Response analysis with multiple load steps defined.)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the Remote Load is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Specify a **Step Selection** value from the values available in the **RPM Value** property of the **Analysis Settings** object to use for the Remote Load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Step Selection: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to **Load Step**. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the Remote Load.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays. **Load Vector Number**: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- When you set the **Applied By** property to **Surface Effect**, a Force is applied using the using the **SFE ,,PRES** command by creating the applicable elements as listed below.
- Based on the selected topology, element types include:
  - SURF154 - 3D structural analyses for face selection.
  - SURF156 - 3D structural analyses for edge selection.
  - SURF153 - 2D structural analyses for edge selection.
  - FOLLW201 - 2D and 3D for vertex selection.
- When you set the **Applied By** property to **Direct**, a Force is applied directly on to the element faces using the **SFCONTROL** and **SFE ,,PRES** commands. Refer to **SFCONTROL** command for a list of supported solid elements, shell elements, and plane-2D elements.

API Reference

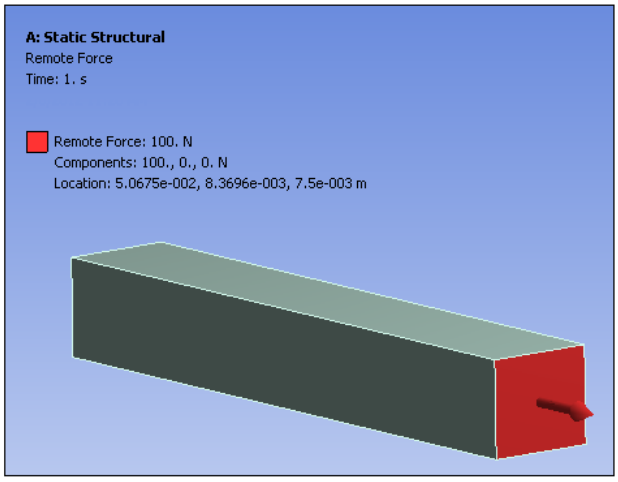
See the Force section of the ACT API Reference Guide for specific scripting information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Remote Force

A **Remote Force** is equivalent to a regular force load on a face (p. 1543) or a force load on an edge (p. 1543), plus some moment (p. 1574). A **Remote Force** can be applied to a face, edge, vertex, element face, or node of a 3D model, or to an edge, vertex, or node of a 2D model.

A **Remote Force** can be used as an alternative to building a rigid part and applying a force load to it. The advantage of using a remote force load is that you can directly specify the location in space from which the force originates.



A **Remote Force** is classified as a *remote* boundary condition. Refer to the Remote Boundary Conditions (p. 1735) section for a listing of all remote boundary conditions and their characteristics.

Analysis Types

Remote Force is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics (p. 406)
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Rigid Dynamics (p. 550)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported (not supported for Explicit Dynamics)*.

Geometry Types: Geometry types supported for the **Remote Force** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Remote Force**.

- Body: **Not Supported**.
- Face: *Supported 3D Only*.
- Edge: *Supported*.
- Vertex: *Supported*.
  - Cannot be applied to a vertex scoped to an end release (p. 1409).
  - Vertex selections do not support the Behavior option.
- Node: *Supported*.
- Element Face: *Supported* (3D Only).
- Element: **Not Supported**.

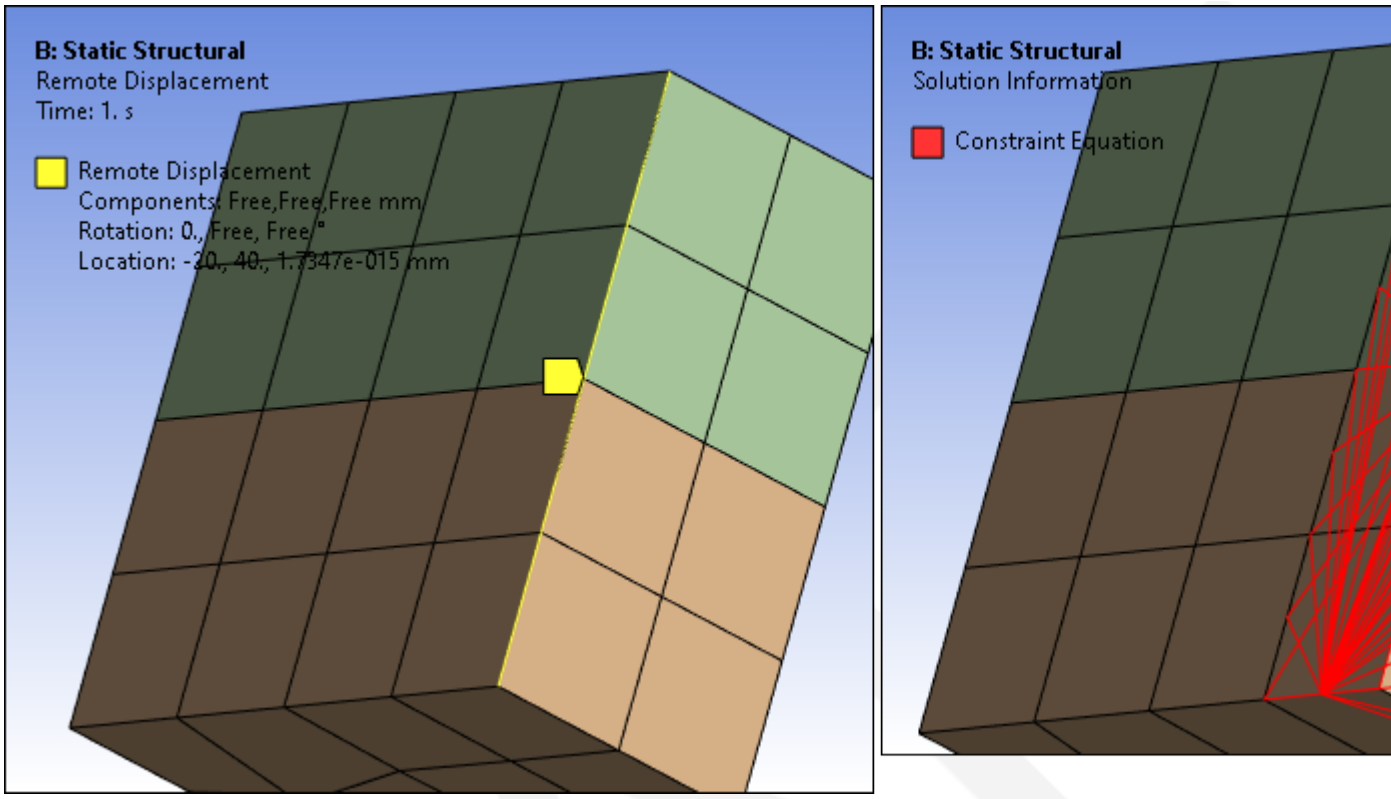
Applying Remote Loads to a Straight Edge

If you apply a remote load to a straight edge (with colinear nodes) of a deformable surface body or a solid body, the application applies the load to the edge as well as the neighboring nodes of the element faces of the deformable surface body or the solid body, respectively. This transfers the rotations of the pilot node (of the **Remote Point**) to the surface or solid body.

Exception: If you scope a remote load to a deformable beam, the application internally changes the setting of the **Behavior** property to **Rigid** and prompts you with a warning.

The following example shows how the application applies a **Remote Displacement**, scoped to a straight edge, to the neighboring element face nodes. In this example, the imbalance of constraint equations created on the blocks might result in unwanted non-zero moment reactions. You can minimize this by using either a lower order mesh or a coarser higher order mesh. See the Surface-Based Constraints section of the *Mechanical APDL Contact Technology Guide* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector**: *Supported*. While loads are associative with geometry changes, load directions are not. This applies to any load that requires a vector input.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Vector: Real - Imaginary**. *Supported* for Harmonic Response Analysis only.
- **Components**: *Supported*.
- **Components: Real - Imaginary**. *Supported* for Harmonic Response Analysis only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Frequency Varying): *Supported* for Harmonic Response Analysis only.

  By default, at least two frequency entries are required when defining a frequency dependent tabular load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Tabular** (Harmonic Index Varying): *Supported* for Harmonic Response (Full) analysis only when **Cyclic Region** is defined. By default, at least two harmonic index entries are required when defining a Harmonic Index dependent tabular load.
- **Tabular (Sector Number)**: *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** is defined. By default, at least two entries are required to define a dependent tabular load.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

You apply a **Remote Force** as you would apply a force load (p. 1543) except that the location of the load origin can be replaced anywhere in space either by picking or by entering the XYZ locations directly or by scoping a geometric entity using the **Location** property. Note that when you *first* define the properties of the Remote Force, the application automatically sets the default location of the **Location** property at the centroid of the scoped geometry selection(s). This setting is maintained even if you re-specify your geometry scoping. It is necessary to manually change the **Location** property's definition.

The location and the direction of a remote force can be defined in the Global Coordinate System or in a local coordinate system (p. 1191).

To apply a **Remote Force**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Remote Force**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Remote Force**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, or **Remote Point** (p. 1089) and then specify the scoping.

Tip:

For a project where none of the analyses have been solved, automatic Remote Point creation (promotion) is available in Harmonic Response or Transient Structural analyses that use a linked Modal system to implement the Mode Superposition (MSUP) solver method. To use this feature:

- Select a supported topology (geometric or mesh entity) on your model, and
- Insert a **Remote Force** load.

The application automatically creates a corresponding Remote Point for the specified load and scopes the load to that Remote Point. Automatic Remote Point creation improves processing performance by eliminating the need to create internal elements for the load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Specify a coordinate system as needed. The default selection is the **Global Coordinate System**. You can also specify a user-defined or local coordinate system.

4. Select the method used to define the remote force: **Vector** (default), **Vector: Real - Imaginary**, **Components**, or **Components: Real - Imaginary**.

5. Define the **Magnitude**, **Coordinate System** directional loading, and/or **Direction** of the load based on the above selections.

   Note:

   - If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction.
   - The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).

6. For Harmonic analyses, specify a **Phase Angle** as needed.

7. Select the **Behavior** of the geometry.

8. As needed, enter a **Pinball Region** value. The default value is **All**.

9. If you are performing a Harmonic MSUP analysis that is linked to upstream system, you can set the **Loading Application** property to either **Load Vector** (default) or **Table**. This property selection enables you to specify between applying the load using load vectors or tables in the harmonic analysis. The option is not available if you scope the load to a Remote Point or a vertex.

   Note:

   Since it is considered a remote boundary condition (p. 1735), a **Remote Force** can make use of remote points that were either specifically defined or created internally by the application. To do a visual check, you can display the connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Details Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. • **Remote Point**: Indicates that the geometry is defined by a Remote Point (p. 1089). – **Remote Points**: This field provides a drop-down list of available user-defined Remote Points. **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. The following properties are used to define the location of the load's origin: • **X Coordinate** • **Y Coordinate** • **Z Coordinate** **Location**: This property specifies the location of the load's origin. The default location is the centroid of your geometry selection(s). You can define this property manually using geometry entity selections as well as by making entries in the above coordinate properties.
Definition	**Type**: Read-only field that displays boundary condition type - **Remote Force**. **Define By**, options include: • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Phase Angle** (Harmonic Analysis (p. 496) only) – **Direction** • **Vector: Real - Imaginary** (Harmonic Analysis only): Real and imaginary magnitude and direction (based on selected geometry). Requires the specification of the following inputs:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	– **Magnitude - Real** – **Magnitude - Imag** – **Direction** **Note:** If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction. • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Phase Angle** (Harmonic Analysis (p. 496) only) – **Y Phase Angle** (Harmonic Analysis (p. 496) only) – **Z Phase Angle** (Harmonic Analysis (p. 496) only) • **Components: Real - Imaginary** (Harmonic Analysis (p. 496) only): Option to define the loading type as real and imaginary components (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **X Component - Real** – **X Component - Imaginary** – **Y Component - Real** – **Y Component - Imaginary** – **Z Component - Real**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	– **Z Component - Imaginary** **Note:** Selection of a Coordinate System rotated out of the global Cartesian X-Y plane is not supported in a 2D analysis. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition. **Non-Cyclic Loading Type**: This property is available for Full Harmonic Analysis when Cyclic Symmetry is present in the model. Options include: • **No** (default). The loading is purely cyclic. That is, the load applied to the base sector is applied to each and every sector. • **Harmonic Index**. The non-cyclic loading can be specified for one or more harmonic indices. This is also known as "engine-order loading" (or traveling wave excitation). A **Harmonic Index** entry is required. • **Sector Number**: This property enables you to select the desired Sector onto which you apply the non-cyclic load. **Important:** When you specify the load as Tabular, the **Independent Variable** property displays and is set to **Harmonic Index** by default. The **Harmonic Index** property is hidden as their values are entered in the table. **Harmonic Index**: This property displays when the **Non-Cyclic Loading Type** property is set to **Harmonic Index**. Where NS is Number of Sectors, enter a value from: 1 to NS/2; if NS is even. 1 to (NS-1)/2; if NS is odd. **Sector Number**: This property displays when the **Non-Cyclic Loading Type** property is set to **Sector Number**. You use this property to specify the desired sector onto which you want to apply the non-cyclic load. If you choose **Tabular** for the **Magnitude** property, you can then set the **Independent Variable** property to **Sector Number** and then specify your loading using the **Tabular Data** window. **Behavior** (p. 1095): This option dictates the behavior of the attached geometry. If the **Scope Method** property is set to **Remote Point**, the boundary condition will then assume the **Behavior** defined in the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	referenced Remote Point as well as other related properties. Options include: • **Rigid**: Does not allow the scoped geometry to deform. This is the only available option for Explicit Dynamics analyses. • **Deformable**: Allows the scoped geometry to deform. • **Coupled**: Allows the scoped geometry to have the same DOF solution on its underlying nodes as the remote point location. • **Beam**: This option specifies a connection from the remote load to the model using linear massless beam elements. It is not a valid option for a Modal Superposition Harmonic Response analysis unless a Remote Point references the load. **Material**: This property is available when the **Behavior** property is set to **Beam**. Select a material to define material properties for the beams used in the connection. Density is excluded from the material definition. **Radius**: This property is available when the **Behavior** property is set to **Beam**. Specify a radius to define the cross section dimension of the circular beam used for the connection. **Follower Load** (Rigid Dynamics (p. 550) analysis only): When set to **No** (default), the force direction doesn't change during the simulation. When set to **Yes**, the force direction is updated with the underlying body.
Step Controls (Visible for Harmonic Response analysis with multiple load steps defined.)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the Remote Load is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Specify a **Step Selection** value from the values available in the **RPM Value** property of the **Analysis Settings** object to use for the Remote Load. **Step Selection**: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to **Load Step**. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the Remote Load.
Advanced	**Pinball Region**: Modify the Pinball setting to reduce the number of elements included in the solver. **Note:** The **Pinball Region** property is not supported for the Samcef and ABAQUS solvers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Loading Application (linked MSUP Harmonic analysis only): Options include **Load Vector** (default) and **Table**. Using the **Load Vector** option, the scale factor specified through load magnitude is applied to the load vector in MSUP Harmonic analysis. Using the **Table** option, the load magnitude is specified in tabular data format and is directly applied to the selected scoping in the MSUP Harmonic analysis. **Note:** The option is not available if you scope the load to a Remote Point or a vertex.

API Reference

See the Remote Force section of the ACT API Reference Guide for specific scripting information.

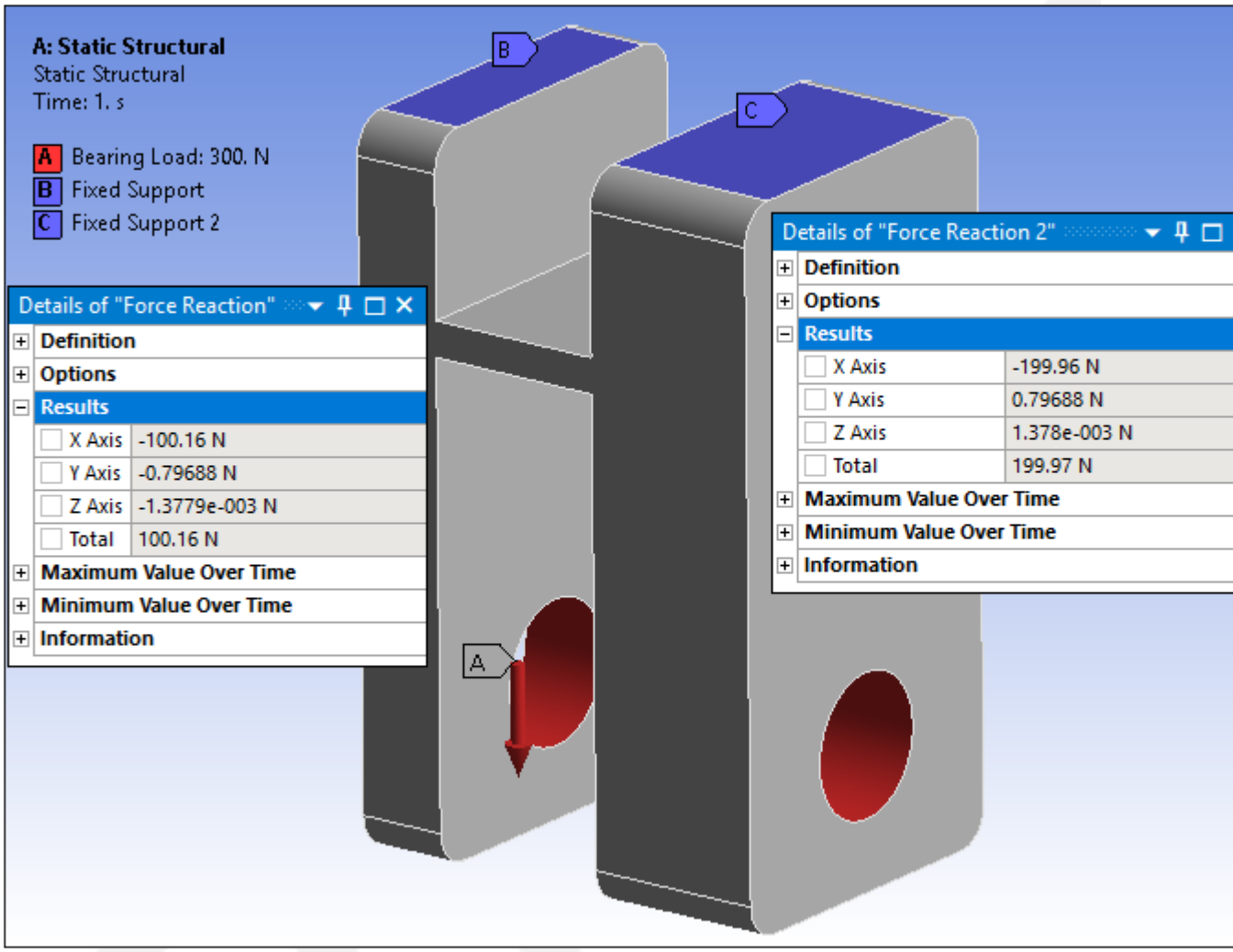
Bearing Load

The **Bearing Load** boundary condition simulates radial forces only. It is applied on the interior of a cylinder in the radial direction using a coordinate system. If the Mechanical application detects a portion of the load to be in the axial direction, the solver stops the solution and issues an appropriate error message.

Note:

- If your CAD system split the target cylinder into two or more faces, select all of the faces when defining the **Bearing Load**.
- When analyzing more than one cylinder, be sure that you scope each cylinder with its own **Bearing Load** boundary condition. Scoping a single Bearing Load to multiple cylinders, as illustrated below, divides the load among the multiple cylindrical faces by area ratio. The example shows two cylinders where the length on the right cylinders is twice the length of the left cylinder. For the single bearing load applied to the two cylinders, the reactions are proportional to each cylinder's area as a fraction of the total load area. This can be seen by the **Reaction Force** results shown below).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Although loading across multiple steps may appear as an application of tabular loading, you cannot set the magnitude of a bearing load in terms of either tabular or functional data. You must set a constant or ramped magnitude for each step such that one value corresponds to each step.

Analysis Types

Bearing Load is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. For vector-based loading on a cylindrical face or geometric axis, you define the radial direction by selecting a different piece of geometry on your model that allows you to modify the **Direction** in the desired direction.
- 2D Simulation: *Supported*. The Bearing Load boundary condition applies a variable distribution of force to a circular edge.

Geometry Types: Geometry types supported for the **Bearing Load** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Bearing Load**.

- **Body**: **Not Supported**.
- **Face**: *Supported*. If the loaded face enlarges (for example, due to a change in parameters), the total load applied to the face remains constant, but the pressure (force per unit area) decreases.
- **Edge**: *Supported - 2D Simulation Only*
- **Vertex**: **Not Supported**.
- **Nodes**: **Not Supported**.
- **Element Face**: *Supported - 3D Simulation Only*.
- **Element**: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector**: *Supported*. You define the radial direction for your vector load by selecting a piece of geometry on your model that provides the ability to specify the direction correctly.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Components**: *Supported*. While loads are associative with geometry changes, load direction are not.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Tabular** (Time Varying): *Supported*.

  Note:

  Although loading across multiple steps may appear as an application of tabular loading, you cannot set the magnitude of a bearing load in terms of either tabular or functional data. You must set a constant or ramped magnitude for each step such that one value corresponds to each step.

- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): **Not Supported**.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply a **Bearing Load**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Bearing Load**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Bearing Load**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Select the method used to define the bearing load: **Vector** (default) or **Components**.
4. Define the **Magnitude**, **Coordinate System** directional loading, and/or **Direction** of the load based on the above selections.

   Note:

   - If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction.
   - The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Face, Edge, etc.) and the number of geometric entities (for example: 1 Face, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Bearing Load**. **Define By**, options include: • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Direction** **Note:** If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction. • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Fields/Options/Description
	– **Z Component**: Defines magnitude in the Z direction. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Force is applied using the **SF**,,PRES command.
- Element types include:
  - SURF154: 3D structural analyses.
  - SURF153: 2D structural analyses.

API Reference

See the Bearing Load section of the ACT API Reference Guide for specific scripting information.

Bolt Pretension

This boundary condition applies a pretension load, typically to model a bolt under pretension. This boundary condition can be scoped to any of the following:

- Cylindrical face
- Straight edge of a line body
- Single or multiple bodies
- Beam Connection (p. 1404)
- Pretension Section (coming from External Model)
- Elements
- Element Faces
- Named Selections (specified using supported scoping)

Analysis Types

A **Bolt Pretension** load is applicable to pure structural or thermal-stress analyses. It is supported for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, loading definitions and values, and solve behavior.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported* (Body scoping only).

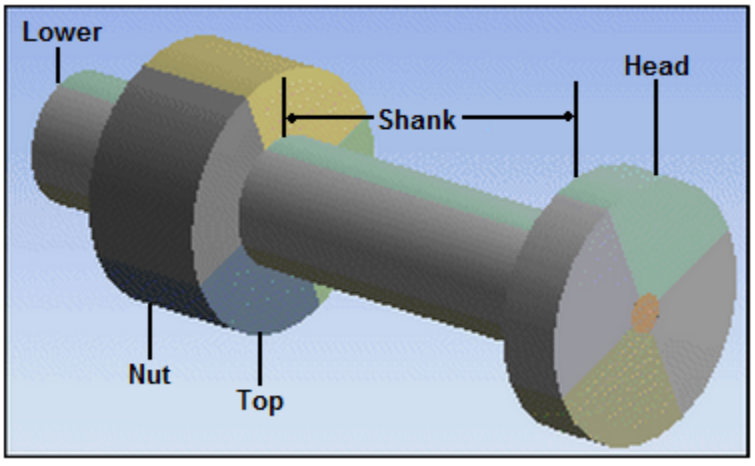
Be sure that a sufficiently fine mesh exists on a face or body that contains a **Bolt Pretension** boundary condition so that the mesh can be correctly partitioned along the axial direction (that is, at least two elements long).

Geometry Types: Geometry types supported for the **Bolt Pretension** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported - Line Body Only*.

Topology: The following topology selection options are supported for **Bolt Pretension**.

- Body: *Supported*.
  - Body scoping of a **Bolt Pretension** load can be to more than one body. In this case all the scoped bodies are cut. There is still only a single **Bolt Pretension** load created but this feature allows you to apply a bolt load to a bolt that has been cut into several bodies. This feature is illustrated in the following figure.



  - Use caution when defining bolt loads by bodies and a coordinate system because the entire body is sliced along the local cutting plane.
- Face: *Supported*.
  - If you try to apply a pre-load on the same face more than once, all definitions except the first one are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - Face selection simulates one **Bolt Pretension** load through multiple split faces of a body. When simulating Bolt Pretension using Face selection scoping on a body with multiple split faces, you need to scope/apply only one the Bolt Pretension boundary condition to only one split face. Even though you select only part of the cylinder body, the Bolt Pretension boundary condition slices though the whole cylinder body.

  - Care should be used when applying a **Bolt Pretension** boundary condition to a cylindrical face that has bonded contact. There is a possibility that if you apply a **Bolt Pretension** boundary condition to a cylinder that had a bonded contact region, the bonded contact will block the ability of the **Bolt Pretension** to deform properly.

  - The **Bolt Pretension** boundary condition should be applied to cylindrical faces that contain the model volume (that is, do not try to apply the **Bolt Pretension** load to a hole).

  - The **Bolt Pretension** boundary condition does not support scoping to a Virtual Cell (merged faces).

- Edge: *Supported*. An option for applying the boundary condition to a line body is to apply it to a single straight edge on the body. The direction of the boundary condition is inferred from the direction of the edge.

- Vertex: **Not Supported**.

- Nodes: **Not Supported**.

- Element Face: *Supported*.

  For this scoping type, the **Bolt Pretension** boundary condition slices through the elements that fall on the cutting plane and within a circular area centered at the origin of the specified coordinate system. The radius of the circular area is the maximum value of the distances calculated from the origin of the specified coordinate system to each of the scoped element faces.

  Therefore, this scoping requires coordinates. A **Coordinate System Behavior** displays automatically for this scoping method and defaults to the setting **Program Controlled**. A **Manual** option is also available to enable you to select a user-defined (p. 1192) coordinate.

- Element: *Supported*.

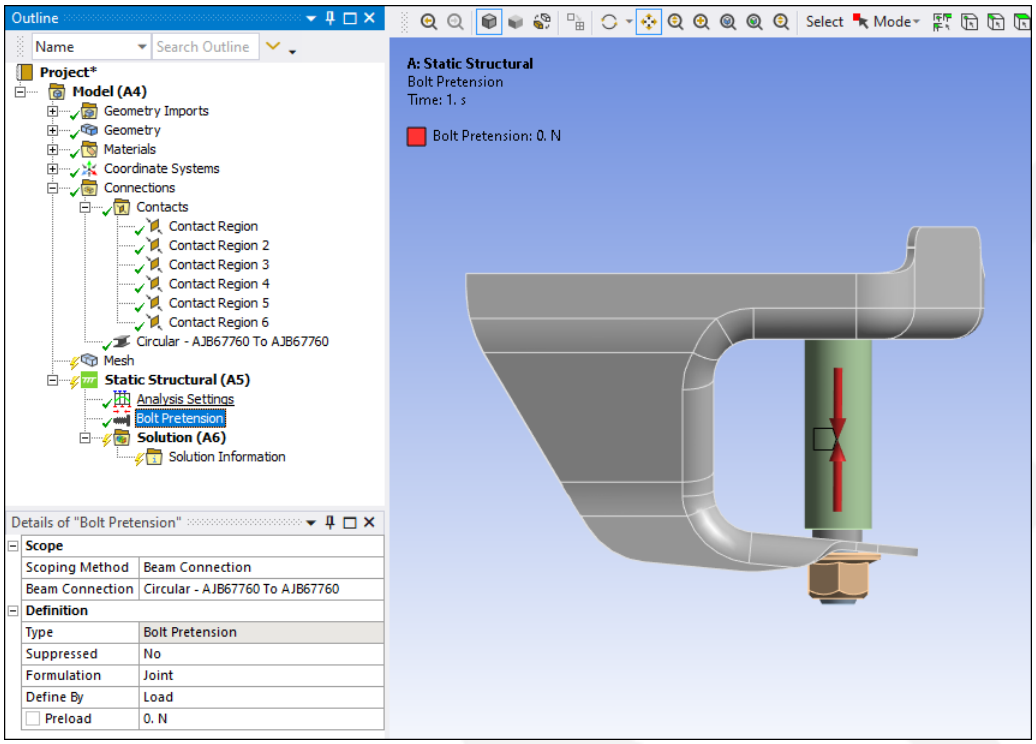
  You can scope a **Bolt Pretension** load to one or more bodies. When you select multiple bodies, all the bodies are cut along the local cutting plane. There is still only a single **Bolt Pretension** load created but this feature enables you to apply one bolt load to a bolt that has been cut into several bodies.

- Beam Connection (p. 1404): *Supported*. Note the following when using a beam connection with a pretension load:

  - When you select **Beam Connection** as your **Scoping Method**, a corresponding **Beam Connection** property displays in the Details pane. This property provides a drop-down list of available beam connections. In addition, when you specify **Beam Connection** as your **Scoping Method**, the **Coordinate System** property is unavailable in the Details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Tip:

You can drag and drop **Beam Connection** objects onto an Environment to automatically created **Bolt Pretension** objects.

– A Bolt Pretension probe (p. 2125) can be scoped to a **Bolt Pretension** defined using the **Beam Connection** option but a **Bolt Tool** cannot.

Important:

For this beam connection scoping scenario, the solver creates two beam elements. As a result, beam probes do not support the **Result Selection** property options **Shear Force** (At I/J) and **Moment** (At I/J). If selected, the application issues a warning message indicating that the results reported at location J are midspan values.

- **Pretension Section**: *Supported*. You can use this option when you have imported a pretension load through the **External Model** system.

Note:

This scoping option does not support the **Joint** setting for the **Formulation** property.

Loading Types: The boundary condition's loading is defined using one of the following options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Load**: Applies a force as a preload. A **Preload** field is displayed where you enter the value of the load in force units.
- **Adjustment**: Applies a length as a pre-adjustment (for example, to model x number of threads). A **Preadjustment** property displays when **Adjustment** is selected. Enter the value of the adjustment in length units. It applies the **Preadjustment** from the solved deformation value of the previous step to the specified **Adjustment** value of the current step.
- **Lock**: Fixes all displacements. You can set this state for any step except the first step.
- **Open**: Use this option to leave the **Bolt Pretension** load open so that the load has no effect on the applied step, effectively suppressing the load for the step. Note that in order to avoid convergence issues from having under-constrained conditions, a small load (0.01% of the maximum load across the steps) is applied. You can set this state for any step.
- **Increment**: Applies a length as an incremental adjustment. An **Increment** field is displayed where you enter the value of the **Adjustment** in length units. When applied, the specified value gets added to the solved deformation value from the previous step. You can choose this option for any step except the first step.

Note:

If a solution restart is performed from a substep of a load step that has an **Increment** specified, the increment value gets added to the solved deformation value at the beginning of the selected restart sub-step.

Loading Data Definition (p. 1771): **Bolt Pretension** is defined by constant loading data only.

Boundary Condition Application

Define **Bolt Pretension** properties:

1. On the **Environment** Context tab, open the **Loads** drop-down menu and select **Bolt Pretension**. Or, right-click the **Environment** object or within the **Geometry** window, and select **Insert** > **Bolt Pretension**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, **Beam Connection**, or **Pretension Section** and then specify the desired cylindrical body/bodies, mesh entity, connection, or section.
3. Coordinate System definition can be performed based on your geometric or mesh entity scoping:
   - The **Coordinate System Behavior** property displays when Faces, Edges, or Element Faces are specified through the scoping. The options for this property include **Program Controlled**, where the application defines the coordinates, or **Manual**, which displays a **Coordinate System** property that you use to select a user-defined coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The **Coordinate System** property displays when a solid bodies or elements are specified through the scoping. This option enables you to select a user-defined coordinate system.

Note:

When the scoping requires you to specify the **Coordinate System**, the application of the boundary condition is at the origin and along the z-axis (3D) or x-axis (2D) of the coordinate system. You can place the coordinate system anywhere in the body and reorient the required axis.

4. Specify the **Formulation** property as needed. This property is only available for 3D analyses. If the bolt specified with pretension is likely to undergo large rotation, set this property to **Joint** to instruct the application to use element MPC184. This element updates its orientation axis as the bolt undergoes rotation and helps the solution converge. For the use of the **Joint** Formulation type, review the **Considerations and Restrictions for Preload Sections** topic in the Defining Preload in a Joint Fastener Undergoing Large Rotation section of the Mechanical APDL Basic Analysis Guide for limitations and restrictions on its use.

5. Using the **Defined By** property, specify how the boundary condition is defined: by **Load**, **Adjustment**, or **Open**.

6. Specify a **Preload** or **Preadjustment** value as needed.

7. If the **Scoping Method** is set to either **Geometry Selection** or **Named Selection**, you can specify the **Solve Behavior** and **Tolerance** properties.

Details View Properties

The selections available in the Details view are described below.

Category	Properties/Descriptions
Scope	**Scoping Method**: The options for this property include **Geometry Selection** (default), **Named Selections**, **Beam Connection**, and **Pretension Section**. **Geometry**: This property displays when the **Scoping Method** is set to **Geometry Selection**. In this case, use selections filter to pick geometry or mesh entities, click in the **Geometry** field, then click **Apply**. **Named Selection**: This property displays when the **Scoping Method** is set to **Named Selection**. Select a desired **Named Selection** from the drop-down menu. **Beam Connection**: This property displays when the **Scoping Method** is set to **Beam Connection**. It enables you to scope the pretension load to a Beam Connection (p. 1404) available from the drop-down menu of user-defined beam connections. **Pretension Section**: This property displays when the **Scoping Method** is set to **Pretension Section**. It enables you to scope the load using a pretension load imported through the External Model system.

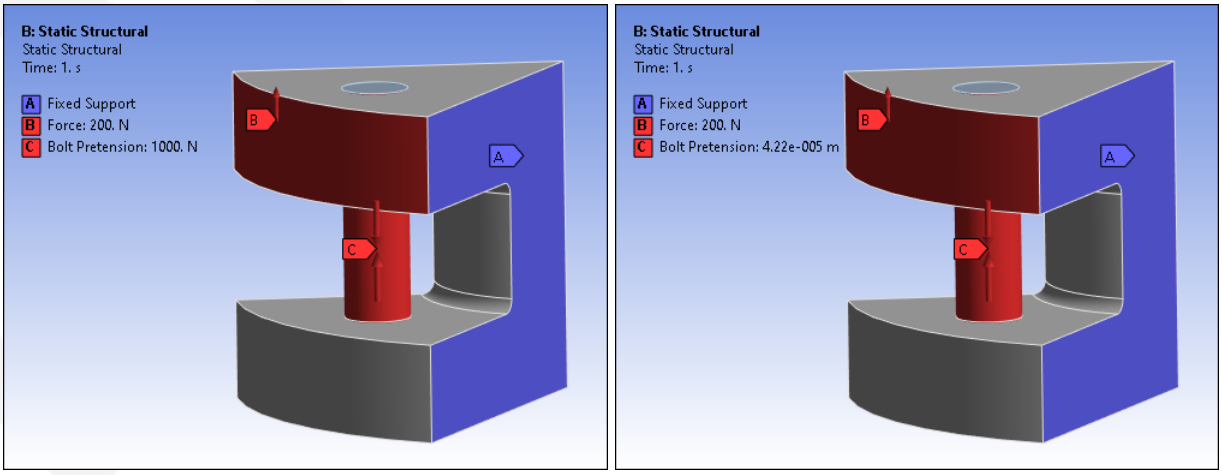
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Descriptions
	Coordinate System (Body/Element scoping only): This property provides a drop-down list of selectable coordinate systems. Only coordinate systems specified in the XY Plane are supported. The **Global Coordinate System** is the default setting. This property is not available for **Beam Connection** scoping. **Coordinate System Behavior**: This option is only visible when your scoping is made with a Face, Edge, or Element Face. Options include **Program Controlled** and **Manual**. When you select the **Manual** option, an associated **Coordinate System** property displays with the default setting of **Global Coordinate System**. You can specify a new user-defined Coordinate System as desired. **Important:** For the **Program Controlled** option, Element Face selections must be cylindrical for the application to properly calculate the corresponding Coordinate System (along Z Axis). If you do not use a cylindrical scoping, you need to set the property to **Manual** and specify the coordinate system.
Definition	**Type**: Read-only field that displays boundary condition type - **Bolt Pretension**. **Suppressed**: Options include **No** (default) and **Yes**. **Formulation**: This property is only available for 3D analyses. Options include **Program Controlled** (default), **Pretension (Small Rotation)**, and **Joint**. If a bolt specified with pretension is likely to undergo large rotation, set this property to **Joint** to instruct the application to use element MPC184. This element updates its orientation axis as the bolt undergoes rotation and helps the solution converge. The **Pretension (Small Rotation)** option indicates that little to no rotation will take place and is the option used for the **Program Controlled** default. **Define By**: The options for this property include: • **Load** • **Adjustment** • **Lock** • **Open** • **Increment** **Preload**: Visible when the **Define By** property is set to **Load**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Descriptions
	Preadjustment: Visible when the **Define By** property is set to **Adjustment**. **Increment**: Visible when the **Define By** property is set to **Increment**.
Advanced	**Solve Behavior**: This property (and category) displays when the **Scoping Method** property is set to either **Geometry Selection** (default) or **Named Selections**. This property enables you to specify how the application solves your **Bolt Pretension** boundary condition. The options for this property include: • **Combined** (default): This option treats the scoped geometric entities as one single bolt and solves the preload as a single load shared by the scoped entities. This option supports multiple body selections, multiple edge selections, from same or different line bodies, but only one face selection per **Bolt Pretension** boundary condition. • **Individual**: Using this option, the specified preload value is applied to each scoped entity and solved as an individual preload for each scoped entity. Note that this option: – Supports multiple cylindrical surface selections, however these faces cannot belong to the same body. – Does not support the use of Bolt Probes. **Tolerance**: This property displays when the **Scoping Method** property is set to either **Geometry Selection** (default) or **Named Selections**. Using this property, you can specify a tolerance that enables the application to identify nodes that occur precisely at or that are slightly below the cutting plane. This entry essentially enables you to shift the plane by a desired value. If set to 0, the application automatically calculates a default value.

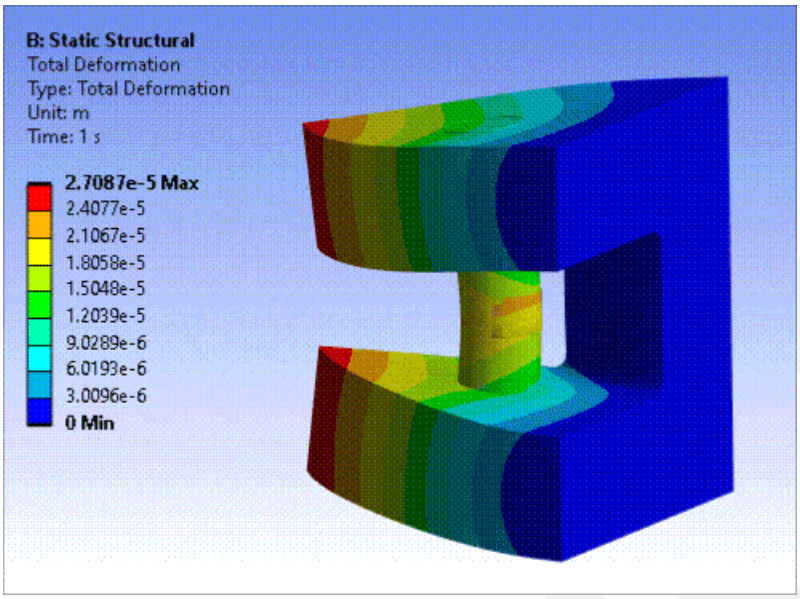
The following example shows a **Bolt Pretension** load as a preload force and as a pre-adjustment length:



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The following animation shows total deformation:

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.



API Reference

See the Bolt Pretension section of the ACT API Reference Guide for specific scripting information

Moment

This boundary condition distributes a moment "about" (the vector of) an axis across 1) one or more flat or curved faces, 2) about one or more edges or vertices, or 3) one or more element faces (3D only) or nodes. Use the right-hand rule to determine the direction of the rotation.

A **Moment** is classified as a *remote* boundary condition. Refer to the Remote Boundary Conditions (p. 1735) section for a listing of all remote boundary conditions and their characteristics.

Analysis Types

Moment is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Static Structural (p. 652)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Moment** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Note:

Face and edge selections for the moment load can span multiple parts, however, multiple vertex selections *must* be of the same part type (solid, 3D surface or line bodies) or the selection is ignored.

Topology: The following topology selection options are supported for **Moment**.

- Body: **Not Supported**.
- Face: *Supported - 3D only*. If a face enlarges (for example, due to a change in parameters), the total load applied to the face remains constant, but the load per unit area decreases.
- Edge: *Supported*.
- Vertex: *Supported*. This boundary condition cannot be applied to a vertex scoped to an end release (p. 1409).
- Nodes: **Not Supported**.
- Node: *Supported*.
- Element Face: *Supported* (3D Only).
- Element: **Not Supported**.

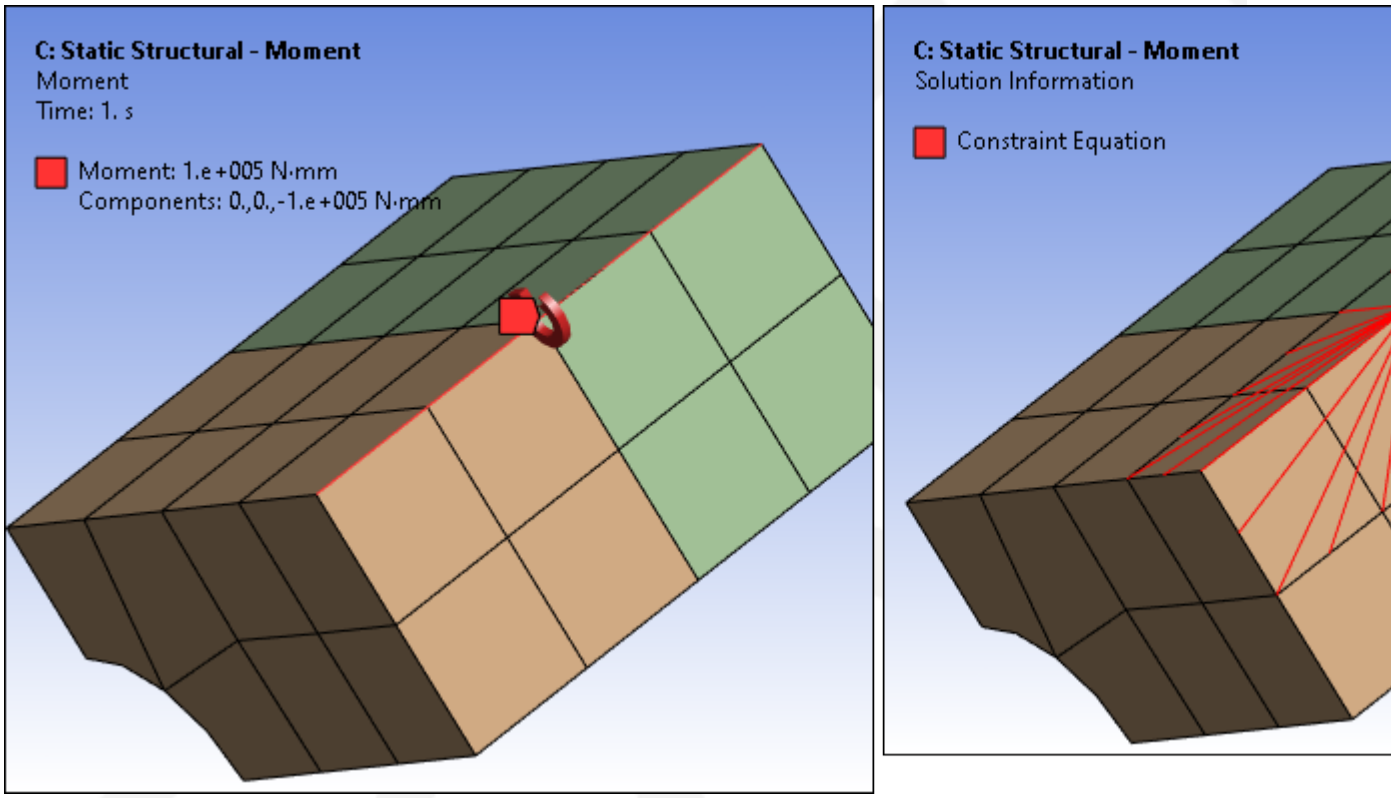
Applying Moment to a Straight Edge

If you apply a moment to a straight edge (with colinear nodes) of a deformable surface body or a solid body, the application applies the load to the edge as well as the neighboring nodes of the element faces of the deformable surface body or the solid body, respectively. This transfers the rotations of the pilot node (of the **Remote Point**) to the surface or solid body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Exception: If you scope a remote load to a deformable beam, the application internally changes the setting of the **Behavior** property to **Rigid** and prompts you with a warning.

The following example shows how the application applies a **Moment**, scoped to a straight edge, to the neighboring element face nodes. In this example, the imbalance of constraint equations created on the blocks might result in unwanted non-zero moment reactions. You can minimize this by using either a lower order mesh or a coarser higher order mesh. See the Surface-Based Constraints section of the *Mechanical APDL Contact Technology Guide* for more information.



Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector** – *Supported*. While loads are associative with geometry changes, load directions are not.

  The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Vector: Real - Imaginary**. *Supported* for Harmonic Response Analysis only.
- **Components**. *Supported*.
- **Components: Real - Imaginary**. *Supported* for Harmonic Response Analysis only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Frequency Varying): *Supported* for Harmonic Response Analysis only.

  By default, at least two frequency entries are required when defining a frequency dependent tabular load.
- **Tabular** (Harmonic Index Varying): *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** is defined. By default, at least two harmonic index entries are required when defining a Harmonic Index dependent tabular load.
- **Tabular (Sector Number)**: *Supported* for Harmonic Response (Full) analysis only when a **Cyclic Region** is defined. By default, at least two entries are required to define a dependent tabular load.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply a **Moment**:

1. Select the **Moment** option from the **Environment** Context tab. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Moment**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, or **Remote Point** and then specify the geometry.

Note:

When specifying the **Scoping Method**, faces and edges can be scoped to either the geometry where the load is to be applied (**Geometry Selection**), to a **Named Selection**, or to a **Remote Point** (p. 1089). Vertices cannot be scoped to **Remote Point**.

Tip:

For a project where none of the analyses have been solved, automatic Remote Point creation is available in Harmonic Response or Transient Structural analyses that use a linked Modal system to implement the Mode Superposition (MSUP) solver method. To use this feature:

- Select a supported topology (geometric or mesh entity) on your model, and
- Insert a **Moment** load.

The application automatically creates a corresponding Remote Point for the specified load and scopes the load to that Remote Point. Automatic Remote

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Point creation improves processing performance by eliminating the need to create internal elements for the load.

3. Select the method used to define the moment: **Vector** (default), **Vector: Real - Imaginary**, **Components**, or **Components: Real - Imaginary**.
4. Define the **Magnitude**, **Coordinate System** directional loading, and/or **Direction** of the load based on the above selections.

Note:

- If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction.
- The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).

5. For Harmonic analyses, specify a **Phase Angle** as needed.
6. Select the **Behavior** of the geometry.
7. As needed, enter a **Pinball Region** value. The default value is **All**.
8. If you are performing a Harmonic MSUP analysis that is linked to upstream system, you can set the **Loading Application** property to either **Load Vector** (default) or **Table**. This property selection enables you to specify between applying the load using load vectors or tables in the harmonic analysis. The option is not available if you scope the load to a Remote Point or a vertex.

Note:

Because a **Moment** is a remote boundary condition, it can make use of remote points that were either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. • **Remote Point** – **Remote Point**: Visible when the **Scoping Method** is set to Remote Point. This field provides a drop-down list of available user-defined Remote Points.
Definition	**Type**: Read-only field that displays boundary condition type - **Moment**. **Define By** (*3D Only*), options include: • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Phase Angle** (Harmonic Analysis (p. 496) only) – **Direction** • **Vector: Real - Imaginary** (Harmonic Analysis only): Real and imaginary magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude - Real** – **Magnitude - Imag** – **Direction** **Note:** If you scope this load using the **Element Face** or **Node** selection options, you must use either of these options to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Properties/Options/Description
	properly specify the **Direction** property. That is, select the **Direction** property field (**Click to Define**), make sure that either the **Element Face** or the **Node** selection option is active, and then define the desired direction. • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Phase Angle** (Harmonic Analysis (p. 496) only) – **Y Phase Angle** (Harmonic Analysis (p. 496) only) – **Z Phase Angle** (Harmonic Analysis (p. 496) only) • **Components: Real - Imaginary** (Harmonic Analysis (p. 496) only): Option to define the loading type as real and imaginary components (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **X Component - Real** – **X Component - Imaginary** – **Y Component - Real** – **Y Component - Imaginary** – **Z Component - Real** – **Z Component - Imaginary** **Magnitude** (*2D Only*). **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition. **Non-Cyclic Loading Type**: This property is available for Full Harmonic Analysis when Cyclic Symmetry is present in the model. Options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Description
	• **No** (default). The loading is purely cyclic. That is, the load applied to the base sector is applied to each and every sector. • **Harmonic Index**. The non-cyclic loading can be specified for one or more harmonic indices. This is also known as "engine-order loading" (or traveling wave excitation). A **Harmonic Index** entry is required. • **Sector Number**: This property enables you to select the desired Sector onto which you apply the non-cyclic load. **Important:** When you specify the load as Tabular, the **Independent Variable** property displays and is set to **Harmonic Index** by default. The **Harmonic Index** property is hidden as their values are entered in the table. **Harmonic Index**: This property displays when the **Non-Cyclic Loading Type** property is set to **Harmonic Index**. Where NS is Number of Sectors, enter a value from: 1 to NS/2; if NS is even. 1 to (NS-1)/2; if NS is odd. **Sector Number**: This property displays when the **Non-Cyclic Loading Type** property is set to **Sector Number**. You use this property to specify the desired sector onto which you want to apply the non-cyclic load. If you choose **Tabular** for the **Magnitude** property, you can then set the **Independent Variable** property to **Sector Number** and then specify your loading using the **Tabular Data** window. **Behavior** (p. 1095): This option dictates the behavior of the attached geometry. If the **Scope Method** property is set to **Remote Point**, the boundary condition will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties. Options include: • **Rigid**: Does not allow the scoped geometry to deform. • **Deformable**: Allows the scoped geometry to deform. • **Coupled**: Allows the scoped geometry to have the same DOF solution on its underlying nodes as the remote point location. • **Beam**: This option specifies a connection from the remote load to the model using linear massless beam elements. It is not a valid option for a Modal Superposition Harmonic Response analysis unless a Remote Point references the load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Properties/Options/Description
	Material: This property is available when the **Behavior** property is set to **Beam**. Select a material to define material properties for the beams used in the connection. Density is excluded from the material definition. **Radius**: This property is available when the **Behavior** property is set to **Beam**. Specify a radius to define the cross section dimension of the circular beam used for the connection.
Step Controls (Visible for Harmonic Response analysis with multiple load steps defined.)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the Remote Load is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Specify a **Step Selection** value from the values available in the **RPM Value** property of the **Analysis Settings** object to use for the Remote Load. **Step Selection**: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to **Load Step**. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the Remote Load.
Advanced	**Pinball Region** **Loading Application** (linked MSUP Harmonic analysis only): Options include **Load Vector** (default) and **Table**. Using the **Load Vector** option, the scale factor specified through load magnitude is applied to the load vector in MSUP Harmonic analysis. Using the **Table** option, the load magnitude is specified in tabular data format and is directly applied to the selected scoping in the MSUP Harmonic analysis. **Note:** The option is not available if you scope the load to a Remote Point or a vertex.

API Reference

See the Moment section of the ACT API Reference Guide for specific scripting information.

Generalized Plane Strain

This boundary condition is used during 2D simulations (p. 878) involving generalized plane strain behavior (p. 881).

Analysis Types

The **Generalized Plane Strain** boundary condition is available for the following analysis types:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Modal Analysis (p. 515)
- Eigenvalue Buckling (p. 487)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: **Not Supported**.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Generalized Plane Strain** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: *Supported - 2D Surface Only*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Generalized Plane Strain**.

- Body: *Supported - All Bodies*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types and **Loading Data Definition** (p. 1771): The **Generalized Plane Strain** boundary condition is defined as a constant.

Boundary Condition Application

To apply a **Generalized Plane Strain**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Generalized Plane Strain**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Generalized Plane Strain**.

2. The **Geometry** selection for this boundary condition is, by default, set to **All Bodies** and is a read-only property.

3. Define the **X Coordinate of Reference Point** and the **Y Coordinate of Reference Point**. These entries are distance values defining the starting point in space.

4. Define the properties for the **Condition Along Fiber Direction**, that includes options for the **Boundary Condition** property and a **Magnitude** as applicable.

   - **Free** - No magnitude.
   - **Force** - Enter magnitude.
   - **Displacement** - Enter magnitude.

5. Define the properties for the **Condition for Rotation About X-axis** and the **Condition for Rotation About Y-axis**. The options for the include **Boundary Condition** property are listed below. **Magnitude** is defined when applicable.

   - **Free** - No magnitude.
   - **Moment** - Enter magnitude.
   - **Rotation** - Enter magnitude.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Geometry**: Read-only field that displays geometry selection - **All Bodies**
Definition	**Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **X Coordinate of Reference Point** **Y Coordinate of Reference Point** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Condition for Rotation About X-axis	**Boundary Condition**, options include: • **Free**: No magnitude. • **Moment**: Enter magnitude. • **Rotation**: Enter magnitude.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Magnitude
Condition for Rotation About Y-axis	**Boundary Condition**, options include: • **Free**: No magnitude. • **Moment**: Enter magnitude. • **Rotation**: Enter magnitude. **Magnitude**

Note:

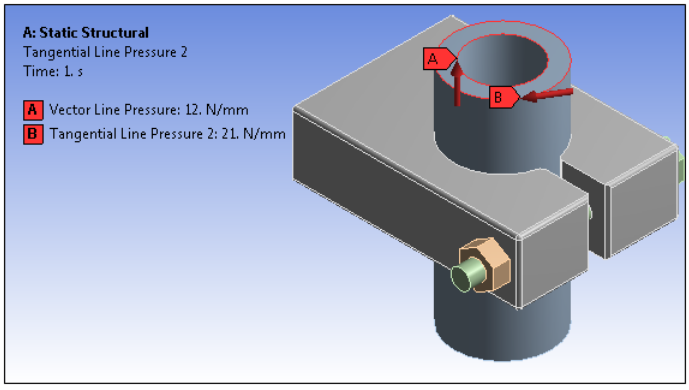
You may wish to review the Generalized Plain Strain Probes (p. 2125) section of the Help for additional information about this boundary condition.

API Reference

See the Generalized Plain Strain section of the ACT API Reference Guide for specific scripting information.

Line Pressure

For 3D simulations, a line pressure load applies a distributed force using force density loading in units of force per length. You can define force density as a vector, an axial component, or tangentially. Tangential loads can be applied to one edge only whereas vector and component loads and can be applied to one or more edges.



If a pressurized edge enlarges due to a change in CAD parameters, the total load applied to the edge increases, but the pressure (force per unit length) remains constant.

Analysis Types

Line Pressure is available for the following analysis types:

- Coupled Field Harmonic (p. 380)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Line Pressure** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Line Pressure**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Vector**: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The vector load definition displays in the Annotation legend with the label Components (p. 308). The **Magnitude** and **Direction** entries, in any combination or sequence, define these displayed values. These are the values sent to the solver.

- **Tangential**: *Supported (scoped to one edge only)*.
- **Components**: *Supported*.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.

  This is the only option supported for Explicit Dynamics.

- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Spatially Varying): *Supported for Tangential loading only*.
- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): *Supported for Tangential loading only*.

Boundary Condition Application

To apply a **Line Pressure**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Line Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Line Pressure**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Select the method used to define the Line Pressure: **Vector** (default), **Tangential**, or **Components**.
4. Define the **Magnitude**, **Coordinate System**, and/or **Direction** of the Line Pressure based on the above selections.

   Note:

   The **Direction** property is not associative and does not remain joined to the entity(s) selected for its specification. Therefore, this direction is not affected by geometry updates, part transformation, or the **Configure** tool (for joints).

5. For Harmonic analyses, specify a **Phase Angle** as needed.

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Line Pressure**. **Define By**, options include: • **Vector**: A magnitude and direction (based on selected geometry). Requires the specification of the following inputs: – **Magnitude** – **Direction** • **Tangential** (scoped to one edge only) • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. **Phase Angle** (Harmonic Analysis (p. 496) only). **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Line pressure is applied using the **SFE** command and the SURF156 element type.
- **Magnitude** (constant, tabular, and function) is always represented as one or more tables in the input file.

API Reference

See the Line Pressure section of the ACT API Reference Guide for specific scripting information.

PSD Base Excitation

PSD Base Excitation loads are used exclusively in random vibration analyses (p. 524) to provide excitation in terms of spectral value vs. frequency to your choice of the supports that were applied in the prerequisite modal analysis (p. 515). The **Boundary Condition** setting in the Details view includes a drop-down list where you can specify any of the following supports for excitation that are defined in the modal analysis: **Fixed Support**, **Displacement**, **Remote Displacement**, and **Body-to-Ground Spring**. If multiple fixed supports or multiple remote displacements are defined in the modal analysis, you can apply the excitation load to all fixed supports or all remote displacements or all of both loads using one of the following options:

- **All Fixed Supports**
- **All Remote Displacements**
- **All Fixed and Remote Displacements**
- **All Supports** (including **Fixed**, **Remote Displacement**, **Displacement**, and **Grounded Springs**)

Note:

- Only fixed degrees of freedom of the supports are valid for excitations.
- Boundary conditions defined with a local coordinate system are not supported.

You can also specify the excitation direction (**X Axis**, **Y Axis**, or **Z Axis**).

The user-defined PSD data table is created in the **Tabular Data** window. You can create a new PSD table or import one from a library that you have created, via the fly-out of the **Load Data** option in the Details view.

Note:

Only positive table values can be input when defining this load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When creating PSD loads for a Random Vibration analysis in the Mechanical application, Workbench evaluates your entries by performing a "Goodness of Fit" to ensure that your results will be dependable.

Click the fly-out of the **Load Data** option and choose **Improved Fit** after entering data points for viewing the graph and updating the table. Interpolated points are displayed if they are available from the goodness of fit approximation. Once load entries are entered, the table provides one of the following color-code indicators per segment:

- **Green**: Values are considered reliable and accurate.
- **Yellow**: This is a warning indicator. Results produced are not considered to be reliable and accurate.
- **Red**: Results produced are not considered trustworthy. If you choose to solve the analysis, the Mechanical APDL application executes the action, however the results are almost certainly incorrect. Ansys recommends that you modify your input PSD loads prior to the solution process.

Four types of base excitation are supported:

- **PSD Acceleration**
- **PSD G Acceleration**
- **PSD Velocity**
- **PSD Displacement**

The direction of the PSD base excitation is defined in the nodal coordinate of the excitation points.

Multiple PSD excitations (uncorrelated) can be applied. Typical usage is to apply three different PSDs in the X, Y, and Z directions. Correlation between PSD excitations is not supported.

RS Base Excitation

RS Base Excitation loads are used exclusively in response spectrum analyses (p. 533) to provide excitation in terms of a spectrum. For each spectrum value, there is one corresponding frequency. Use the **Boundary Condition** setting in the Details view to apply an excitation to all of the fixed supports that were applied in the prerequisite modal analysis (p. 515).

Note:

Only fixed DOFs of the supports are valid for excitations.

You can also specify the excitation in a given direction (**X Axis**, **Y Axis**, or **Z Axis**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The user-defined RS data table is created in the **Tabular Data** window. You can create a new RS table or import one from a library that you have created, via the fly-out of the **Load Data** option in the Details pane.

Note:

Only positive table values can be used when defining this load.

Three types of base excitation are supported:

- **RS Acceleration**
- **RS Velocity**
- **RS Displacement**

You should specify the direction of the RS base excitation in the global Cartesian system.

You can apply multiple response spectrum excitations as long as they are of the same type (acceleration, displacement, or velocity). Typically, you apply the same type of RS excitation in each coordinate direction (X, Y, and Z).

The following additional settings are included in the Details view of an **RS Base Excitation** load:

- **Scale Factor**: Scales the entire table of input excitation spectrum for a **Single Point** response spectrum. The factor must be greater than 0.0. The default is 1.0.
- **Missing Mass Effect**: Set to **Yes** to include the contribution of high frequency modes in the total response calculation. Including these modes is normally required for nuclear power plant design.

  The responses contributed by frequency modes higher than those of rigid responses, specifically frequency modes beyond Zero Period Acceleration (ZPA) are called residual rigid responses. The frequency modes beyond ZPA are defined as frequency modes at which the spectral acceleration returns to the Zero Period Acceleration. In some applications, especially in the nuclear power plant industry, it is critical and required to include the residual rigid responses to the total responses. Ignoring the residual rigid responses will result in an underestimation of responses in the vicinity of supports. There are two methods available to calculate residual rigid responses: the Missing Mass and Static ZPA methods. The Missing Mass method is named based on the fact that the mass associated with the frequency modes higher than that of ZPA are missing from the analysis. As a result, the residual rigid responses are sometimes referred to missing mass responses. When set to **Yes**, the **Missing Mass Effect** is used in a response spectrum analysis.

  Note:

  If the **Missing Mass Effect** property is set to **Yes** for an excitation, then all excitations must have the property set to **Yes**.

- **Rigid Response Effect**: Set to **Yes** to include rigid responses to the total response calculation. Rigid responses normally occur in the frequency range that is lower than that of missing mass responses, but higher than that of periodic responses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In many cases, it is impractical and difficult to accurately calculate all natural frequencies and mode shapes for use in the response spectrum evaluation. For high-frequency modes, rigid responses basically predominate. To compensate for the contribution of higher modes to the responses, the rigid responses are combined algebraically to the periodic responses, which occur in the low-frequency modes that are calculated using one the methods above. The most widely adopted methods to calculate the rigid responses are the Gupta and Lindley-Yow methods. These two methods are available for a response spectrum analysis under **Rigid Response Effect Type** when **Rigid Response Effect** is set to **Yes**.

Joint Load

When you are using joints in a Explicit Dynamics (p. 406), Rigid Dynamics (p. 550), Structural Optimization, Transient Structural (p. 665) analysis, you use a **Joint Load** object to apply a kinematic driving condition to a single degree of freedom on a **Joint** object. **Joint Load** objects are applicable to all joint types (p. 1298) except fixed, general, universal, and spherical joints. For translation degrees of freedom, the **Joint Load** can apply a displacement, velocity, acceleration, or force. For rotation degrees of freedom, the **Joint Load** can apply a rotation, angular velocity, angular acceleration, or moment. The directions of the degrees of freedom are based on the reference coordinate system of the joint and not on the mobile coordinate system.

A positive joint load will tend to cause the mobile body to move in the positive degree of freedom direction with respect to the reference body, assuming the mobile body is free to move. If the mobile body is not free to move then the reference body will tend to move in the negative degree of freedom direction for the **Joint Load**. One way to learn how the mechanism will behave is to use the Configure (p. 1380) feature. For the joint with the applied **Joint Load**, dragging the mouse will indicate the nature of the reference/mobile definition in terms of positive and negative motion.

To apply a Joint Load:

1. Highlight the environment object and insert a **Joint Load** from the right mouse button context menu or from the **Loads** drop-down menu in the **Environment** Context tab.

2. From the **Joint** drop-down list in the Details of the **Joint Load**, select the particular **Joint** object that you would like to apply to the **Joint Load**. You should apply a **Joint Load** to the mobile bodies of the joint. It is therefore important to carefully select the reference and mobile bodies while defining the joint.

3. Select the unconstrained degree of freedom for applying the **Joint Load**, based on the type of joint (p. 1298). You make this selection from the **DOF** drop-down list. For joint types that allow multiple unconstrained degrees of freedom, a separate **Joint Load** is necessary to drive each one. Further limitations apply as outlined under Joint Load Limitations (p. 1593) below. **Joint Load** objects that include velocity, acceleration, rotational velocity or rotational acceleration are not applicable to static structural analyses.

4. Select the type of **Joint Load** from the **Type** drop-down list. The list is filtered with choices of **Displacement**, **Velocity**, **Acceleration**, and **Force** if you selected a translational **DOF** in step

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. The choices are **Rotation**, **Rotational Velocity**, **Rotational Acceleration**, and **Moment** if you selected a rotational **DOF**.

Note:

If you are using the Mechanical APDL solver and scoping the **Joint Load** to a General joint that has the **Rotation** property set to **Free All**, then the **Moment** option is not listed in the **Type** property drop-down list.

5. Specify the magnitude of the **Joint Load** type selected in step 4 as a constant, in tabular format, or as a function of time using the same procedure as is done for most loads in the Mechanical application. Refer to Defining Boundary Condition Magnitude (p. 1771) for further information.

Tip:

On Windows platforms, an alternative and more convenient way to accomplish steps 1 and 2 above is to drag and drop the **Joint** object of interest from under the **Connections** object folder to the environment object folder. When you highlight the new **Joint Load** object, the **Joint** field is already completed and you can continue at step 3 with **DOF** selection.

6. As applicable, specify the load step at which you want to lock the joint load by entering the value of the step in the **Lock at Load Step** field. The default value for this option is zero (0) and is displayed as **Never**. This feature immobilizes movement of the joint's DOFs. For example, this option is beneficial when you want to tighten a bolt to an initial torque value (via a Moment Joint Driver on a Revolute Joint) and then lock that joint during a subsequent load step.

Note:

Mechanical APDL References:

This feature makes use of the **%_FIX%** parameter on the **DJ** command.

When a joint driver with a force or moment load is deactivated, then the lock constraint on the joint is also deleted using the **DJDELE** command. This happens if the locking occurs before the deactivation.

Joint Load Limitations

Some joint types have limitations on the unconstrained degrees of freedom that allow the application of joint loads as illustrated in the following table:

Joint Type	Unconstrained Degrees of Freedom	Allowable Degrees of Freedom for Applying Joint Loads
Fixed (p. 1298)	None	Not applicable
Revolute (p. 1299)	ROTZ	ROTZ

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Joint Type	Unconstrained Degrees of Freedom	Allowable Degrees of Freedom for Applying Joint Loads
Cylindrical (p. 1299)	UZ, ROTZ	UZ, ROTZ
Translational (p. 1300)	UX	UX
Slot (p. 1301)	UX, ROTX, ROTY, ROTZ	UX
Universal (p. 1301)	ROTX, ROTZ	None
Spherical (p. 1302)	ROTX, ROTY, ROTZ	None
Planar (p. 1302)	UX, UY, ROTZ	UX, UY, ROTZ
General (p. 1310)	UX, UY and UZ, Free X, Free Y, Free Z, and Free All	All unconstrained degrees of freedom
Bushing (p. 1303)	UX, UY, UZ, ROTX, ROTY, ROTZ	All unconstrained degrees of freedom
Point on Curve (p. 1312)	UX	UX

Note:

Where applicable, you must define all three rotations for a Joint Load before proceeding to a solve.

API Reference

See the Joint Load section of the ACT API Reference Guide for specific scripting information.

Thermal Condition

You can insert a known temperature (not from data transfer) boundary condition in an analysis by inserting a **Thermal Condition** object. Specify the value of the temperature in the Details view under the **Magnitude** property. If the load is applied to a surface body, by default the temperature is applied to both the top and bottom surface body faces. You can optionally apply different temperatures to the top and bottom faces by adjusting the **Shell Face** entry in the details view. When you apply a thermal condition load to a solid body, the Shell Face property is not available in the Details view. You can add the thermal condition load as time-dependent (p. 193) or spatially varying (p. 1769).

Note:

- When a Thermal Condition is specified on the Top or Bottom shell face of a surface body, the opposite face defaults to the environment temperature unless it is otherwise specified from another load object.
- For an assembly of bodies with different topologies (solid body, line, shell, beam), you must define a separate Thermal Condition load for each topology, that is, you must define one load scoped to line bodies, define a second load scoped to surface bodies, and so on.
- For each load step, if an Imported Body temperature load and a Thermal Condition load are applied on common geometry selections, the Imported Body temperature

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

load takes precedence. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

- If the Thermal Condition is applied to a shell face that has a Layered Section (p. 868) applied to it, you must set **Shell Face** to Both in order to solve the analysis.

Analysis Types

Thermal Condition is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Eigenvalue Buckling (p. 487)
- Electric Analysis (p. 403)
- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Thermal Condition** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Thermal Condition**.

- Body: *Supported*. The **Thermal Condition** is a body-based boundary condition.
- Face: **Not Supported**.
- Edge: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: *Supported*.

Loading Types: The **Thermal Condition** boundary condition's loading is defined by **Magnitude** only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant** (only option for Eigenvalue Buckling analyses).
- **Tabular** (Time Varying).
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Spatially Varying).
- **Function** (Time Varying).
- **Function** (Spatially Varying).
- **Table Assignment**.

Boundary Condition Application

To apply a **Thermal Condition**:

1. On the **Environment** Context tab, open the **Loads** drop-down menu and select **Thermal Condition**. Alternatively, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Thermal Condition**.
2. Define the **Scoping Method**. This property has two options: **Geometry Selection** and **Named Selection**. For either scoping type, the application supports only solid bodies, surface bodies (2D), line bodies, or elements.

   For surface bodies, in the Details view, the **Shell Face** property provides a drop-down list. Select **Top**, **Bottom**, or **Both** (default) to apply the thermal boundary condition to the selected face. For bodies that have one or more layered section objects, you need to specify **Both** for **Shell Face** or the Thermal Condition will be under-defined and an error message will be generated.
3. Specify the **Magnitude** of the temperature.
4. Depending on the load type you selected for **Magnitude**, you may need to specify additional information under **Tabular Data**, **Function**, **Graphics Controls**, or **Thermal Condition Table Assignment**.

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. • **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Displays the type of geometry (Body or Element) and the number of geometric entities (for example: 1 Body, 2 Elements) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. • **Named Selection**: Visible when the **Scoping Method** property is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. Named Selections must be either body- or element-based.
Definition	**Type**: Read-only field that displays the boundary condition type: **Thermal Condition**. **Magnitude**: Temperature load. Select one of the following: • **Import** (p. 1780) • **Export** (p. 1781) • **Constant**. The default is 22 degrees Celsius. • **Tabular** • **Function** • **Table Assignment** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Load Vector Number: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.
Function	This category displays when you set the **Magnitude** to **Function**. See Spatial Load and Displacement Function Data (p. 1777) for additional information.
Tabular Data	This category displays when you set the **Magnitude** to **Tabular**. See Spatial Load Tabular Data (p. 1776) for additional information.
Graph Controls	This category displays based upon the specifications made in the **Function** or **Tabular** categories. As applicable, see Spatial Load and Displacement Function Data (p. 1777) or Spatial Load Tabular Data (p. 1776) for additional information.
Thermal Condition Table Assignment	This category displays when you set the **Magnitude** to **Table Assignment**. See Specify Thermal Condition Loads with Tables (p. 1783) for additional information.
Graphics Controls	This category displays when you set the **Magnitude** to **Table Assignment** and select a table that contains time as an independent variable. See Specify Thermal Condition Loads with Tables (p. 1783) for additional information.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Temperatures are applied using the **BF** command. For surface bodies, with **Top** or **Bottom Shell Face** selection, temperatures are applied using the **BFE** command.
- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.

API Reference

See the Thermal Condition section of the ACT API Reference Guide for specific scripting information.

Temperature

This boundary condition simulates a uniform, time-dependent, or spatially varying temperature over the selected geometry.

A spatially varying load allows you to vary the magnitude of a temperature according to coordinates and/or time. in a single coordinate direction and as a function of time using the **Tabular Data** or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Function features. See the Defining Boundary Condition Magnitude (p. 1771) section for the specific steps to apply tabular data and function loads. Alternatively, you can specify multivariable time-dependent and spatially varying temperature loads using the **Table** feature. See Specifying Boundary Condition Loads With Tables (p. 1781) for the specific steps to apply table loads.

Note:

For each load step, if an Imported Temperature load and a Temperature load are applied on common geometry selections, the Imported Temperature load takes precedence. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

Analysis Types

Temperature is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)
- Transient Thermal Analysis (p. 680)
- Thermal-Electric Analysis (p. 661)
- LS-DYNA Analysis

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported* - ***Plane Stress*** *and* ***Axisymmetric*** *behaviors only*.

Geometry Types: Geometry types supported for the **Temperature** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Temperature**.

- Body: *Supported*. When scoping a load to a body, you need to specify whether the temperature is applied to **Exterior Faces Only** or to the **Entire Body** using the **Apply To** option.
- Face: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: *Supported*.
- Element Face: *Supported*.
- Element: *Supported*.

Note:

The same temperature value is applied when multiple faces, edges, vertices, nodes, element faces, and elements are selected.

Loading Types: The **Temperature** boundary condition's loading is defined by **Magnitude** only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**.
- **Tabular** (Time Varying).
- **Tabular** (Spatially Varying). Not supported for LS-DYNA Analyses.
- **Function** (Time Varying). Not supported for LS-DYNA Analyses.
- **Function** (Spatially Varying). Not supported for LS-DYNA Analyses.
- **Table Assignment**

Boundary Condition Application

To apply a **Temperature**:

1. Select the **Temperature** option from the **Environment** Context tab. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Temperature**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Specify the **Magnitude** of the temperature.
4. Depending on the type of temperature load you selected for **Magnitude**, you may need to specify additional information under **Tabular Data**, **Function**, or **Temperature Table Assignment**.

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. **Apply To** (*Body scoping only*), options include: • **Exterior Faces Only** • **Entire Body**
Definition	**Type**: Read-only field that displays boundary condition type - **Temperature**. **Magnitude**: Temperature load. Select one of the following: • **Import** (p. 1780) • **Export** (p. 1781) • **Constant**. The default is 22 degrees Celsius. • **Tabular** • **Function** • **Table Assignment** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Function	This category displays when you set the **Magnitude** to **Function**. See Spatial Load and Displacement Function Data (p. 1777) for additional information.
Tabular Data	This category displays when you set the **Magnitude** to **Tabular**. See Spatial Load Tabular Data (p. 1776) for additional information.
Temperature Table Assignment	This category displays when you set the **Magnitude** to **Table Assignment**. See Specify Temperature Loads with Tables (p. 1785) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Graph Controls	This category displays based upon the specifications made in the **Function** or **Tabular** categories. As applicable, see Spatial Load and Displacement Function Data (p. 1777) or Spatial Load Tabular Data (p. 1776) for additional information.
Graphics Controls	This category displays when you set the **Magnitude** to **Table Assignment** and choose a table that contains time as an independent variable. See Specify Temperature Loads with Tables (p. 1785) for additional information.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Temperatures are applied using the **D** command.
- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.
- **Table** is a special type of numeric array that enables the Mechanical APDL solver to calculate (through linear interpolation) the values between entries in a multi-dimensional table. The solver applies these interpolated values across the selected geometry when it computes the solution. See the discussion in Array Parameters.

LS-DYNA References and Notes

The following LS-DYNA keywords and considerations are applicable for this boundary condition.

- Temperature is applied using the *BOUNDARY_ TEMPERATURE keyword.

API Reference

See the Temperature section of the ACT API Reference Guide for specific scripting information.

Convection

This boundary condition causes convective heat transfer (p. 1602) to occur through one or more flat or curved faces (in contact with a fluid).

The bulk fluid temperature is measured at a distance from the face outside of the thermal boundary layer. The face temperature refers to the temperature at the face of the simulation model.

Convective Heat Transfer

Convection is related to heat flux by use of Newton's law of cooling:

$q/A = h(t_s - t_f)$

where:

- q/A is heat flux out of the face (calculated within the application)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- h is the film coefficient (you provide)
- t_s is the temperature on the face (calculated within the application)
- t_f is the bulk fluid temperature (you provide)

When the fluid temperature exceeds face temperature, energy flows into a part. When the face temperature exceeds the fluid temperature, a part loses energy.

If you select multiple faces when defining convection, the same bulk fluid temperature and film coefficient is applied to all selected faces.

Analysis Types

Convection is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)
- Thermal-Electric Analysis (p. 661)
- Transient Thermal Analysis (p. 680)
- LS-DYNA Analysis

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*: ***Plane Stress*** and ***Axisymmetric*** *behaviors only*.

Geometry Types: Geometry types supported for the **Convection** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Convection**.

- Body: *Supported (3D External Faces Only)*.
- Face: *Supported*.
- Edge: *Supported (2D Only)*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Vertex: **Not Supported**. This selection specifies the reference temperature.
- Nodes: **Not Supported**. This selection specifies the reference temperature.
- Element Face: *Supported*.
- Element: **Not Supported**.

Loading Options:

- **Film Coefficient**: The film coefficient (also called the heat transfer coefficient or unit thermal conductance) is based on the composition of the fluid in contact with the face, the geometry of the face, and the hydrodynamics of the fluid flow past the face. It is possible to have a time, temperature or spatially dependent film coefficient. Refer to heat transfer handbooks or other references to obtain appropriate values for film coefficient.
- **Coefficient Type**: This field is available when the film coefficient is temperature dependent. Its value can be evaluated at the average film temperature (average of surface and bulk temperatures), the surface temperature, the bulk temperature, or the absolute value of the difference between surface and bulk temperatures.

Note:

If you change the units from Celsius to Fahrenheit, or Fahrenheit to Celsius, when the convection coefficient type **Difference between surface and bulk** is in use, the displayed temperature values indicate a temperature difference only. The addition or subtraction of 32^{o} for each temperature in the conversion formula offset one another. In addition, switching to or from the **Difference between surface and bulk** Coefficient Type option from any other option, clears the values in the **Convection Coefficient** table. This helps to ensure that you enter correct temperature values.

- **Ambient Temperature**: The ambient temperature is the temperature of the surrounding fluid. It is possible to have a time or spatially dependent ambient temperature.
- **Convection Matrix**: Specifies whether to use a diagonal film coefficient matrix or a consistent film coefficient matrix. The default setting, **Program Controlled**, allows the solver to determine whether to use a diagonal or consistent film coefficient matrix. Not available in an LS-DYNA analysis.
- **Edit Data For**: This field allows you to select and edit **Film Coefficient** or **Ambient Temperature**. The Tabular Data window, details view, graph and graphics view will change based on the selection in the **Edit Data For** field. For example, when the **Film Coefficient** property is specified as **Tabular** or **Function** and **Edit Data For** is **Film Coefficient**, you will actively edit data for the **Film Coefficient** in the appropriate details view and Tabular Data fields. Not available in an LS-DYNA analysis.
- **Fluid Flow** (available for Steady-State Thermal and Transient Thermal analyses only): Option include **Yes** and **No** (default). Setting this property to **Yes** activates convection on one or more line bodies whose **Model Type** property is set to **Thermal Fluid**. Also, when set to **Yes**, the **Fluid Flow Scoping** and **Display Connection Lines** properties display. You use the **Fluid Flow Scoping**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

property to specify a desired edge or edges, vertex, or node using either direct selection or using a Named Selection.

When scoping to:

– Edge(s) on a fluid line body: Convection is accounted for with surface elements SURF151 and SURF152. The application maps the elements to the fluid elements using a distance-based algorithm. This option uses the two nodes from the mapped fluid element to specify the reference temperature. Note that, following mesh generation, you can use the **Display Connection Lines** property to display the connection lines between the centroid of each element face/edge of the convection surface(s)/edge(s) to the corresponding closest node on the Fluid Flow scoping. The closest node is computed using a distance-based algorithm.

– Vertex/Node on a fluid line body. This option uses the specified vertex or node to specify the reference temperature. You can specify a single node or vertex **only**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**.
- **Tabular** (Time Varying).
- **Tabular** (Spatially Varying). Not supported for LS-DYNA Analyses.
- **Tabular** (Temperature Varying): for **Film Coefficient**. Not supported for LS-DYNA Analyses.
- **Function** (Time Varying). Not supported for LS-DYNA Analyses.
- **Function** (Spatially Varying). Not supported for LS-DYNA Analyses.

You can vary the magnitude of film coefficient and ambient temperature in a single coordinate direction using either tabular data or a function. See the Defining Boundary Condition Magnitude (p. 1771) section for the specific steps to apply tabular and/or function loads.

Note:

- Scaling based on time is not supported for convection.
- The activate/deactivate (p. 1423) option is only available when the **Independent Variable** property of the **Tabular Data** category is set to **Time**.

Boundary Condition Application

To apply a **Convection**:

1. Select **Convection** from the **Environment** Context tab. Or, right-click the **Environment** object, or in the **Geometry** window, and select **Insert**>**Convection**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Film Coefficient** for the Convection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Modify the **Ambient Temperature** as needed.

5. Define the **Convection Matrix** as **Program Controlled** (default), **Diagonal**, or **Consistent**. Not Available in an LS-DYNA analysis.

6. Set the **Fluid Flow** property to **Yes** if applicable. Define your **Scoping Method** and select the desired edge/vertex/node through direct selection or by selecting an appropriately defined Named Selection from the **Fluid Flow Scoping** property.

Details View Properties

The selections available in the Details view are described below.

Category	Property/Option/Description
Scope	**Scoping Method**: This property specifies how you perform geometric entity selection. Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Note:** The Body filter selects 3D external faces only. **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. **Shared Reference Body**: This property is a scoping feature used to apply a Convection to the shared faces of two bodies, one solid and one shell. When you have properly scoped the geometric entities, using either Geometry Selection or an appropriate Named Selection, the property provides a drop-down list of the names of the bodies that share the scoped features. Select the desired body from the list. Once selected, the Geometry or Named Selection property displays a parenthetical of the shared face, "(1 Shared Face)," to indicate the condition. Not Supported in an LS-DYNA analysis. **Shell Face**: For this loading condition, this property is only available when you are working with a 3D shell body and you have activated the beta feature **Thermal Variation (Beta)** and the property is set to an option *other* than **No Variation Along Thickness**. The **Shell Face** setting enables you to apply the load to the temperature degree of freedom associated with the selected face. To activate beta option, perform the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property/Option/Description
	1. **Activate Beta Options**: On the Workbench Project page, select the Tools drop-down menu and then **Options** > **Appearance** > **Beta Options**. 2. **Activate Beta Feature Selection**: On the Workbench Project page, select the Tools drop-down menu and then **Options** > **Mechanical** > **Allow thermal variation along shell thickness**. Restart the application once complete. 3. Select the desired **Surface Body** object under the **Geometry** folder and specify the **Thermal Variation (Beta)** property. **Important:** If you specify a shell body as a **Shared Reference Body**, the solver does not support the **Linear Variation** and **Quadratic Variation** options of the **Thermal Variation (Beta)** property. In addition, you need to the **Offset Type** property to **Bottom**.
Definition	**Type**: Read-only field that displays boundary condition type - **Convection**. **Film Coefficient** **Coefficient Type** (visible only for temperature dependent Film Coefficient) **Ambient Temperature** (not visible if the **Fluid Flow** property is set to **Yes**.) **Convection Matrix**: • **Program Controlled** (default) • **Diagonal** • **Consistent** **Edit Data For**: this property displays when the **Film Coefficient** and/or **Ambient Temperature** properties are set to **Tabular** or **Function**. Not Available in an LS-DYNA analysis. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Fluid Flow Controls	**Fluid Flow** (Steady-State Thermal and Transient Thermal analyses only): **Yes** or **No** (default). **Scoping Method**: This property displays when you set the **Fluid Flow** property to **Yes**. This scoping applies to edge, vertex, or node scoping

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property/Option/Description
	of the fluid line body. Options include **Geometry Selection** and **Named Selection**. **Fluid Flow Scoping**: Based on the above **Scoping Method**, directly select desired edge/edges, vertex, or node or select a Named Selections from the available drop-down list. **Display Connection Lines**: This property requires that the mesh be generated. Options for this property include **Yes** and **No** (default). Selecting **Yes** displays connection lines between the centroid of each element face/edge of the convection surface(s)/edge(s) to the corresponding closest node on the fluid flow scoping. The closest node is computed using a distance-based algorithm. **Note:** The display of the connection lines may not be an accurate representation of the connection in the solver. The solver, by default and when applicable, uses the closes two nodes from the scoped fluid elements to specify the reference temperature.
Function	This category displays when the **Film Coefficient** and/or **Ambient Temperature** properties are set to **Function**. See the Spatial Load and Displacement Function Data (p. 1777) section of the Help for additional information. Not Available in an LS-DYNA analysis.
Tabular Data	This category displays when the **Film Coefficient** and/or **Ambient Temperature** properties are set to **Tabular**. See the Spatial Load Tabular Data (p. 1776) section of the Help for additional information. Not Available in an LS-DYNA analysis.
Graph Controls	This category displays when based upon the specifications made in the **Function** and/or **Tabular** categories. As applicable, see the Spatial Load and Displacement Function Data (p. 1777) section or the Spatial Load Tabular Data (p. 1776) section of the Help for additional information. Not Available in an LS-DYNA analysis.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Convection loading is applied using the element types SURF152 (3D thermal analyses) and SURF151 (2D thermal analyses).
- **Film Coefficient** and **Ambient Temperature** are applied using the **SF** command.
- **Film Coefficient** and **Ambient Temperature** (constant, tabular, and function) are always represented as tables in the input file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Fluid Flow** activates the Mechanical APDL element FLUID116 and specifics TEMP (Temperature) as the degree of freedom.

LS-DYNA References and Notes

The following LS-DYNA keywords and considerations are applicable for this boundary condition.

- Convection is applied using the *BOUNDARY_CONVECTION keyword.

API Reference

See the Convection section of the ACT API Reference Guide for specific scripting information.

Radiation

Applies thermal radiation to a surface of a model (an edge in a 2D model). You can define the exchange of radiation between a body and the ambient temperature, or between two surfaces.

For thermal related analyses that use the Mechanical APDL solver, the actual calculation of the radiation exchange between two surfaces is performed using the Radiosity Solver method. The Radiosity Solver method accounts for the heat exchange between radiating bodies by solving for the outgoing radiative flux for each surface, when the surface temperatures for all surfaces are known. The surface fluxes provide boundary conditions to the finite element model for the conduction process analysis in Workbench. When new surface temperatures are computed, due to either a new time step or iteration cycle, new surface flux conditions are found by repeating the process. The surface temperatures used in the computation must be uniform over each surface facet to satisfy the conditions of the radiation model.

For models that are entirely symmetrical (p. 1127), you can account for symmetry using Symmetry Regions or Cyclic Regions (p. 1128). The Radiosity Solver method respects plane or cyclic symmetries. Using a model's symmetry can significantly reduce the size of the model. The Radiosity Solver method will take symmetry into account and the Radiation Probe solution results will be valid for the full model. Settings for the Radiosity Solver method are available under the **Analysis Settings** object in the **Radiosity Controls** (p. 1446) category.

Related References

This discussion of the **Radiation** loading condition assumes that you have an understanding of the related terminology. See the Definitions section of the Mechanical APDL Thermal Analysis Guide for a listing of terminology. In addition, see the Mechanical APDL References and Notes (p. 1614) topic at the end of the section for more references.

Analysis Types

Radiation is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)
- Thermal-Electric Analysis (p. 661)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Transient Thermal Analysis (p. 680)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Radiation** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Radiation**.

- Body: **Not Supported**.
- Face: *Supported - 3D Only*.
- Edge: *Supported - 2D Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: *Supported*.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying) - for Ambient Temperature: *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Tabular** (Temperature Varying): *Supported* for Emissivity if Correlation = Surface To Surface.
- **Function** (Time Varying): *Supported*.
- **Function**: **Not Supported**.

Loading Types and Loading Definition

The unique loading characteristics and definitions for the **Radiation** boundary condition are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Ambient Temperature Radiation

When the **Correlation** property is specified as **To Ambient** in the Details view of a **Radiation** object, all of the radiation energy is assumed to be exchanged with the surroundings at ambient temperature, that is, the Form Factor is assumed to be 1.0.

You can set the following additional radiation properties in the Details view:

- **Emissivity**: The ratio of the radiation emitted by a surface to the radiation emitted by a black body at the same temperature.
- **Ambient Temperature**: The temperature of the surrounding space.

Note:

Radiation exchange between surfaces is restricted to *gray-diffuse* surfaces. *Gray* implies that emissivity and absorptivity of the surface do not depend on wavelength (either can depend on temperature). *Diffuse* signifies that emissivity and absorptivity do not depend on direction. For a gray-diffuse surface, emissivity = absorptivity and emissivity + reflectivity = 1. Note that a black body surface has a unit emissivity.

Surface to Surface Radiation

When the **Correlation** property is specified as **Surface to Surface** in the Details view of a **Radiation** object, the radiation energy is exchanged between surfaces. In this context, "surface" refers to a face of a shell or solid body in a 3D model, or an edge in a 2D model. You can then specify **Emissivity**, **Ambient Temperature** (defined above), **Enclosure**, and the **Enclosure Type**. **Emissivity** must be a positive value that is not greater than 1. Emissivity can also be defined by **Tabular Data**.

You should assign the same **Enclosure** number to surfaces radiating to each other. Specify the **Enclosure Type** as either **Open** (default) or **Perfect** as suited for a simulation of the closed radiation problems. Furthermore, closed radiation problems have no dependence on **Ambient Temperature** so that property is removed from the Details view during closed problems.

Caution:

You cannot apply a Surface to Surface Radiation load to a geometric entity that is already attached to another Radiation load.

Note:

For surfaces emitting radiation, make sure that the mesh does not penetrate into other bodies.

When using the **Surface to Surface** correlation with shell bodies, the Details also includes a **Shell Face** property that enables you apply the load to **Both** faces, to the **Top** face only, or to the **Bottom** face only.

Boundary Condition Application

To apply a **Radiation**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Select the **Radiation** option from the **Environment** Context tab. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Radiation**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Modify the **Ambient Temperature** as needed.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: This property specifies how you perform geometric entity selection. Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. **Shared Reference Body**: This property is a scoping feature used to apply a Radiation to the shared faces of two bodies, one solid and one shell. When you have properly scoped the geometric entities, using either Geometry Selection or an appropriate Named Selection, the property provides a drop-down list of the names of the bodies that share the scoped features. Select the desired body from the list. Once selected, the Geometry or Named Selection property displays a parenthetical of the shared face, "(1 Shared Face)," to indicate the condition. **Note:** The **Shared Reference Body** property is not supported when the **Correlation** property is set to **Surface to Surface**. **Shell Face**: For this loading condition, this property is available for 3D shell bodies: • When you set the **Correlation** property to **Surface to Surface** setting. The **Shell Face** setting enables you the choice of applying the load to **Both** faces, to the **Top** face only, or to the **Bottom** face only. *Or...*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	• When you 1) set the **Correlation** property to **To Ambient** and 2) set the beta feature/property **Thermal Variation (Beta)** property to an option *other* than **No Variation Along Thickness**. The **Shell Face** setting enables you to apply the load to the temperature degree of freedom associated with the selected face. To activate and define the beta option: 1. **Activate Beta Options**: On the Workbench Project page, select the Tools drop-down menu and then **Options** > **Appearance** > **Beta Options**. 2. **Activate Beta Feature Selection**: On the Workbench Project page, select the Tools drop-down menu and then **Options** > **Mechanical** > **Allow thermal variation along shell thickness**. Restart the application once complete. 3. Select the desired **Surface Body** object under the **Geometry** folder and specify the **Thermal Variation (Beta)** property. **Important:** If you specify a shell body as a **Shared Reference Body**, the solver does not support the **Linear Variation** and **Quadratic Variation** options of the **Thermal Variation (Beta)** property. In addition, you need to the **Offset Type** property to **Bottom**.
Definition	**Type**: Read-only field that displays boundary condition type - **Radiation**. **Correlation** **Emissivity** **Ambient Temperature** (not visible if **Correlation** property = **Surface to Surface** and the **Enclosure Type** property = **Perfect**) **Enclosure** (only visible if the **Correlation** property = **Surface to Surface**)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Enclosure Type (only visible if the **Correlation** property = **Surface to Surface**) **Note:** If a solver error occurs when the **Enclosure Type** is set to **Perfect**, Ansys recommends that you change the setting to **Open**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

See the following Mechanical APDL sections for additional information for this boundary condition.

- Using the Radiosity Solver Method in the Thermal Analysis Guide.
- Radiation and Radiosity Solution Method sections in the Mechanical APDL Theory Reference:
- For a perfectly closed system, the application uses the **VFSM** command. Specifically, Mechanical performs the **VFSM,,N,1** command with the perfect enclosure number N. However, note that the reciprocity relationship is not satisfied using the command. Review the Notes topic in the **VFSM** section of the *Mechanical APDL Command Reference* for additional information.

API Reference

See the Radiation section of the ACT API Reference Guide for specific scripting information.

Heat Flow

*Heat Flow is available for 3D simulations and 2D simulations for **Plane Stress** and **Axisymmetric** behaviors only*. See the 2D Analyses (p. 878) section of the Help for the required geometry settings for **Plane Stress** and **Axisymmetric** behaviors.

Heat flow simulates the transmission of heat across flat or curved surfaces or edges or across a vertex or vertices and as a result adds energy to a body over time.

Tip:

When applying this loading condition to a curved surface, use a fine mesh to capture the geometry in greater detail and improve result accuracy.

Perfectly Insulated

For a selected face or faces, Heat Flow allows you to specify a Perfectly Insulated load wherein a "no load" insulated condition is applied to the Face: that is, zero heat flow. An *insulated face* is a no load condition meant to override any thermal loads scoped to a body. The heat flow rate is 0 across this face. This load is useful in a case where most of a model is exposed to a given condition

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

(such a free air convection) and only a couple of faces do not share this condition (such as the base of a cup that is grounded). This load overrides thermal loads scoped to a body only. See Resolving Thermal Boundary Condition Conflicts (p. 1790) for a discussion on thermal load precedence.

Selecting multiple faces insulates all of the faces.

Analysis Types

Heat Flow is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)
- Thermal-Electric Analysis (p. 661)
- Transient Thermal Analysis (p. 680)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Heat Flow** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Heat Flow**.

- Body: **Not Supported**.
- Face: *Supported 3D Only*. If you select multiple faces when defining the heat flow rate, the magnitude is apportioned across all selected faces.
- Edge: *Supported*.
  - If you select multiple edges when defining the heat flow rate, the magnitude is apportioned across all selected edges.
  - During 2D analyses, the application does not enable you to apply Heat Flow to edges that share topology.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Vertex: *Supported*. If you select multiple vertices when defining the heat flow rate, the magnitude is apportioned among all selected vertices.
- Nodes: **Not Supported**.
- Element Face: *Supported*.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported for face selections in 3D and edge selections in 2D*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported for face selections in 3D and edge selections in 2D*.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply a **Heat Flow**:

1. On the **Environment** Context tab: click **Heat**>**Heat Flow**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Heat Flow**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Magnitude** for the Heat Flow.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Heat Flow**. **Define As**: **Heat Flow** (default) or **Perfectly Insulated**. Perfectly Insulated indicates zero heat flow. **Magnitude**: Loading value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Convection loading is applied using the element types SURF152 (3D thermal analyses) and SURF151 (2D thermal analyses).

- In a 3D analyses, **Heat Flow** on face selections are applied using the **SF**,,HFLUX command on SURF152 elements. **Heat Flow** is represented as a table in the input file. **Heat Flow** applied to a selected edge or vertex use the **F** command.
- In a 2D analyses **Heat Flow** on edge selections are applied using the **SF**,,HFLUX command on SURF151 elements. **Heat Flow** is represented as a table in the input file. **Heat Flow** on vertex selections are applied using the **F** command.

API Reference

See the Heat Flow section of the ACT API Reference Guide for specific scripting information.

Heat Flux

*Heat Flux is available for 3D simulations and 2D simulations for **Plane Stress** and **Axisymmetric** behaviors only*. See the 2D Analyses (p. 878) section of the Help for the required geometry settings for **Plane Stress** and **Axisymmetric** behaviors.

The Heat Flux boundary condition applies a uniform heat flux to the selected geometry. A positive heat flux acts into a face or edge, adding energy to a body. *Heat flux* is defined as energy per unit time per unit area. If you select multiple faces or edges when defining the heat flux, the same value gets applied to all selected faces.

Analysis Types

Heat Flux is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Thermal-Electric Analysis (p. 661)
- Transient Thermal Analysis (p. 680)
- LS-DYNA Analysis

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. Body and Face selections only.
- 2D Simulation: *Supported*. Edge selections only.

Geometry Types: Geometry types supported for the **Heat Flux** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Heat Flux**.

- Body: *Supported - 3D Only*.
- Face: *Supported*.
- Edge: *Supported - 2D Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: *Supported*.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*. Not supported for LS-DYNA Analyses.
- **Function** (Spatially Varying): **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Boundary Condition Application

To apply a **Heat Flux**:

1. On the **Environment** Context tab: click **Heat**>**Heat Flux**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Heat Flux**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Magnitude** for the Heat Flux.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Heat Flux**. **Magnitude**: Heat flux density value. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- **Heat Flux** is applied using the **SF** command and SURF152 (3D thermal analyses) and SURF151 (2D thermal analyses) element types.
- **Heat Flux** (constant, tabular, and function) is always represented as a table in the input file.

LS-DYNA References and Notes

The following LS-DYNA keywords and considerations are applicable for this boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Heat Flux is applied using the *BOUNDARY_FLUX keyword.

API Reference

See the Heat Flux section of the ACT API Reference Guide for specific scripting information.

Internal Heat Generation

*Available for 3D simulations, and 2D simulations for **Plane Stress** and **Axisymmetric** behaviors only*. See the 2D Analyses (p. 878) section of the Help for the required geometry settings for **Plane Stress** and **Axisymmetric** behaviors.

Applies a uniform generation rate internal to a body. A positive heat generation acts into a body, adding energy to it. Heat generation is defined as energy per unit time per unit volume.

If you select multiple bodies when defining the heat generation, the same value gets applied to all selected bodies. If a body enlarges due to a change in CAD parameters, the total load applied to the body increases, but the heat generation remains constant.

Note:

For each load step, if an Imported Heat Generation load and an Internal Heat Generation load are applied on common geometry selections, the Imported Heat Generation load takes precedence. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

Analysis Types

Internal Heat Generation is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Steady-State Thermal Analysis (p. 657)
- Thermal-Electric Analysis (p. 661)
- Transient Thermal Analysis (p. 680)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Internal Heat Generation** boundary condition include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Internal Heat Generation**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: *Supported*.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply **Internal Heat Generation**:

1. On the **Environment** Context tab: click **Heat**>**Internal Heat Generation**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Internal Heat Generation**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Magnitude** value.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Fields/Options/Description
	• **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Internal Heat Generation**. **Magnitude** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- **Heat Generation** is applied using the **BFE** command.
- **Heat Generation** (constant, tabular, and function) is always represented as a table in the input file.

API Reference

See the Internal Heat Generation section of the ACT API Reference Guide for specific scripting information.

Mass Flow Rate

*Mass Flow Rate is available for 3D simulations and 2D simulations for **Plane Stress** and **Axisymmetric** behaviors only*. See the 2D Analyses (p. 878) section of the Help for the required geometry settings for **Plane Stress** and **Axisymmetric** behaviors.

You apply the **Mass Flow Rate** boundary condition on the edges of Line Bodies (p. 872) whose **Model Type** property is specified as **Thermal Fluid**. This loading is needed to account for heat distribution which occurred due to the mass transportation of the fluid (mass/time).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Types

Mass Flow Rate is available for the following analysis types:

- Steady-State Thermal Analysis (p. 657)
- Transient Thermal Analysis (p. 680)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Mass Flow Rate** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Mass Flow Rate**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply **Mass Flow Rate**:

1. On the **Environment** Context tab: click **Mass Flow Rate**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Mass Flow Rate**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Magnitude** value.

Note:

The positive direction for **Mass Flow Rate** corresponds to the direction of the scoped edge. If you are scoping multiple edges, turn on the **Direction** option via the **Edge** group (p. 84) to display the fluid flow direction of the edges in the **Geometry** window.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. Displays the type of geometry (Edge) and the number of geometric entities (for example: 1 Edge, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** property is set to **Named Selection**. This field provides a drop-down list of edge-based Named Selections that are based on the target line body or line bodies.
Definition	**Type**: Read-only field that displays boundary condition type - **Mass Flow Rate**. **Magnitude**: speed of the flow. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- **Mass Flow Rate** is applied using the **SFE** command directly on FLUID116 elements representing Thermal Fluid line bodies.
- **Mass Flow Rate** (constant, tabular, and function) is always represented as a table in the input file.

Electric Charge

The **Electric Charge** load simulates the application of an electric charge to a body.

Analysis Types

Electric Charge is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Electric Charge** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Electric Charge**.

- Body: *Supported*.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Nodes: *Supported*.
- Element Face: *Supported*.
- Element: *Supported*.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Frequency Varying)

Boundary Condition Application

To apply **Electric Charge**:

1. Select the **Electric Charge** option from the **Electric** group on the **Environment** Context tab.. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Electric Charge**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, or **Voltage Coupling** and then specify the geometry.
3. Specify a **Magnitude**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection** (default): Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. • **Voltage Coupling**: This option is only available for Coupled Field Harmonic and Coupled Field Modal analyses. It indicates that the geometry selection is defined by a **Voltage Coupling** (p. 1700).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Voltage Coupling**: Visible when the **Scoping Method** is set to **Voltage Coupling**. This field provides a drop-down list of available user-defined **Voltage Coupling** objects.
Definition	**Type**: Read-only field that displays boundary condition type - **Electric Charge**. **Magnitude** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- **Electric Charge** is applied using the **F** command.
- **Magnitude** (constant and tabular) is always represented as a table in the input file.
- In a piezoelectric or electrostatic-structural analysis, Electric Charge loading is interpreted as negative electric charge.

API Reference

See the Electric Charge section of the ACT API Reference Guide for specific scripting information.

Voltage

A voltage load simulates the application of an electric potential to a body.

Analysis Types

Voltage is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Electric Analysis (p. 403)
- Magnetostatic Analysis (p. 546)
- Thermal-Electric Analysis (p. 661)

For each analysis type, you define the voltage by magnitude and phase angle in the **Details** view, according to the following equation.

$V = V_o \cos(\omega t + \phi)$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

V_o is the magnitude of the voltage (input value Voltage), ω is the frequency, and ϕ is the phase angle. **For a static analysis, $\omega t = 0$**.

Magnetostatic Analysis Requirements

See Voltage Excitation for Solid Source Conductors (p. 1640).

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Voltage** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Voltage**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: *Supported* for the Coupled Field Harmonic (p. 380) analysis only.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Function** (Time Varying): *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Function** (Spatially Varying): **Not Supported**.

Caution:

During an **Coupled Field Harmonic**, **Coupled Field Modal**, or **Electric/Thermal-Electric Analysis**, voltage loads cannot be applied to a face, edge, or vertex that is shared with another voltage or current load or a Coupling (p. 1698).

Boundary Condition Application

To apply **Voltage**:

1. Select the **Voltage** option from the **Environment** Context tab.. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Voltage**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, or **Voltage Coupling** and then specify the geometry.
3. Enter a **Magnitude** value.
4. Enter a **Phase Angle** value.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection** (default): Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. • **Voltage Coupling**: This option is only available for Coupled Field Harmonic and Coupled Field Modal analyses. It indicates that the geometry selection is defined by a **Voltage Coupling** (p. 1700). – **Voltage Coupling**: Visible when the **Scoping Method** is set to **Voltage Coupling**. This field provides a drop-down list of available user-defined **Voltage Coupling** objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Definition	**Type**: Read-only field that displays boundary condition type - **Voltage**. **Magnitude** **Phase Angle** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Voltages are applied using the **D** command.
- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.

Current

A current load simulates the application of an electric current to a body.

Analysis Types

Current is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Electric Analysis (p. 403)
- Thermal-Electric Analysis (p. 661)
- Magnetostatic Analysis (p. 546)

For each analysis type, you define the current by magnitude and phase angle in the **Details** view, according to the following equation.

$I = I_o \cos(\omega t + \phi)$

I_o is the magnitude of the current (input value Current), ω is the frequency, and ϕ is the phase angle. **For a static analysis, $\omega t = 0$**.

Magnetostatic Analysis Requirements

See Current Excitation for Solid Source Conductors (p. 1641) and Current Excitation for Stranded Source Conductors (p. 1643).

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Current** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Current**.

- Body: **Not Supported**.
- Face: *Supported*. An applied current assumes that the body surfaces are equipotential.
- Edge: *Supported*. An applied current assumes that the edges are equipotential.
- Vertex: *Supported*.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Note:

- Current loads assume that the scoped entities are equipotential, meaning they behave as electrodes where the voltage degrees of freedom are coupled and solve for a constant potential.
- During an Electric or Thermal Analysis, it is assumed that the material properties of the body provide conductance. A positive current applied to a face, edge, or vertex flows into the body. A negative current flows out of the body.

Caution:

Current loads cannot be applied to a face, edge, or vertex that is shared with another voltage or current load or a Coupling (p. 1698).

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Spatially Varying): **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Function** (Time Varying): *Supported*.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply **Current**:

1. Select the **Current** option from the **Environment** Context tab.. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Current**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Enter a **Magnitude** value.
4. Enter a **Phase Angle** value.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection** (default): Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Current**. **Magnitude** **Phase Angle** **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Currents are applied using a combination of **F**,,AMPS and **CP**,,**VOLT** commands when used in Magnetostatic analyses.
- **F**,,**AMPS** and **CE** commands when used in Electric or Thermal-Electric analyses.
- **Magnitude** (constant, tabular, and function) is always represented as a table in the input file.

API Reference

See the Current section of the ACT API Reference Guide for specific scripting information.

Voltage (Ground)

The **Voltage (Ground)** load simulates the application of zero electric potential to a body.

Analysis Types

Voltage (Ground) is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Modal (p. 386)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Voltage (Ground)** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Voltage (Ground)**.

- Body: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: *Supported* for the Coupled Field Harmonic (p. 380) and Coupled Field Modal (p. 386) analysis types only.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Data Definition (p. 1771): This is a grounding load. The magnitude is always zero.

Boundary Condition Application

To apply **Voltage (Ground)**:

1. Select the **Voltage (Ground)** option from the **Electric** group on the **Environment** Context tab.. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Voltage (Ground)**.
2. Define the **Scoping Method** as either **Geometry Selection**, **Named Selection**, or **Voltage Coupling** and then specify the geometry.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection** (default): Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections. • **Voltage Coupling**: This option is only available for Coupled Field Harmonic and Coupled Field Modal analyses. It indicates

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	that the geometry selection is defined by a **Voltage Coupling** (p. 1700). – **Voltage Coupling**: Visible when the **Scoping Method** is set to **Voltage Coupling**. This field provides a drop-down list of available user-defined **Voltage Coupling** objects.
Definition	**Type**: Read-only field that displays boundary condition type - **Voltage (Ground)**. **Magnitude**: This is a grounding load. The Magnitude value is always zero. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- **Voltage (Ground)** is applied using the **D** command.
- **Magnitude** is set to zero volts.

API Reference

See the Voltage (Ground) section of the ACT API Reference Guide for specific scripting information.

Electromagnetic Boundary Conditions and Excitations

You can apply electromagnetic excitations and boundary conditions when performing a Magneto-static analysis in the Mechanical application. A boundary condition is considered to be a constraint on the field domain. An excitation is considered to be a non-zero boundary condition which causes an electric or magnetic excitation to the system. Boundary conditions are applied to the field domain at exterior faces. Excitations are applied to conductors.

- Magnetic Flux Boundary Conditions (p. 1636)
- Conductor (p. 1637)
  - Solid Source Conductor Body (p. 1638)
    - → Voltage Excitation for Solid Source Conductors (p. 1640)
    - → Current Excitation for Solid Source Conductors (p. 1641)
  - Stranded Source Conductor Body (p. 1642)
    - → Current Excitation for Stranded Source Conductors (p. 1643)

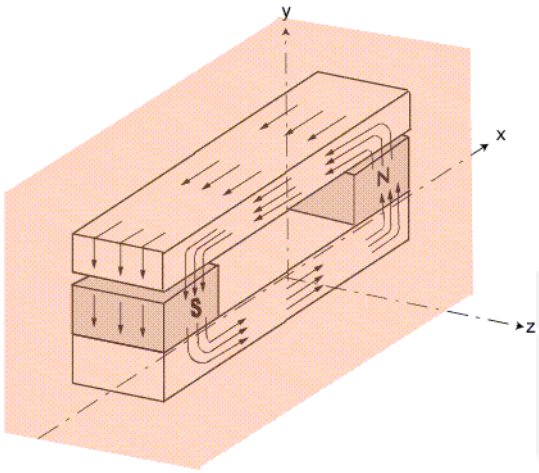
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Magnetic Flux Boundary Conditions

Available for 3D simulations only.

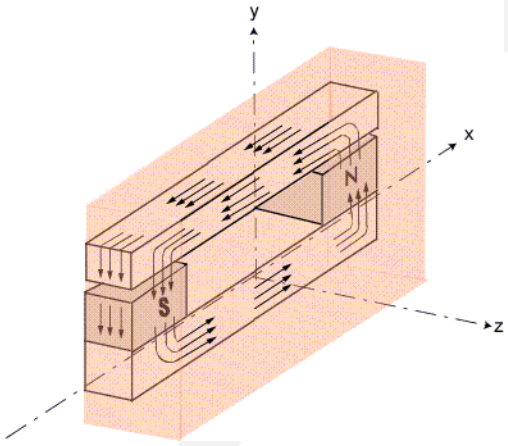
Magnetic flux boundary conditions impose constraints on the direction of the magnetic flux on a model boundary. This boundary condition may only be applied to faces. By default, this feature constrains the flux to be normal to all exterior faces.

Selecting Flux Parallel forces the magnetic flux in a model to flow parallel to the selected face. In the figure below, the arrows indicate the direction of the magnetic flux. It can be seen that the flux flows parallel to the xy plane (for any z coordinate).



A flux parallel condition is required on at least one face of the simulation model. It is typically applied on the outer faces of the air body to contain the magnetic flux inside the simulation domain or on symmetry plane faces where the flux is known to flow parallel to the face.

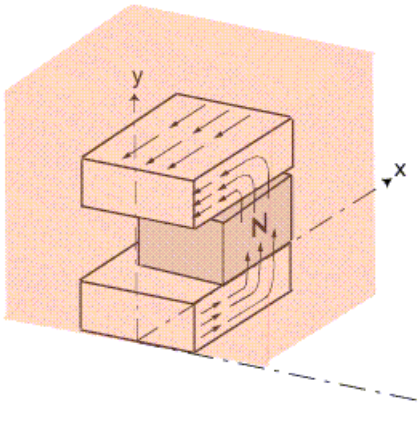
To set this feature, right-click the **Magnetostatic** environment item in the tree and select **Magnetic Flux Parallel** from the **Insert** context menu or click the **Magnetic Flux Parallel** option on the Context tab. It can only be applied to geometry faces and Named Selections (faces).



Half-symmetry model of a keepered magnet system. Note that the XY-plane is a Flux Parallel boundary. The flux arrows flow parallel to the plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Half-symmetry model of a keepered magnet system. Note that the YZ-plane is a Flux Normal boundary. The flux arrows flow normal to the plane. This is a natural boundary condition and requires no specification.

Note:

Applying the flux parallel boundary conditions to the exterior faces of the air domain may artificially capture more flux in the simulation domain than what physically occurs. This is because the simulation model truncates the open air domain. To minimize the effect, ensure the air domain extends far enough away from the physical structure. Alternatively, the exterior faces of the air domain may be left with an unspecified face boundary condition. An unspecified exposed exterior face imposes a condition whereby the flux flows normal to the face. Keep in mind that at least one face in the model must have a flux parallel boundary condition.

Conductor

Available for 3D simulations only.

A conductor body is characterized as a body that can carry current and possible excitation to the system.

Solid CAD geometry is used to model both solid source conductors and stranded source conductors. In solid conductors, such as bus bars, rotor cages, etc., the current can distribute non-uniformly due to geometry changes, hence the program performs a simulation that solves for the currents in the solid conductor prior to computing the magnetic field.

Stranded source conductors can be used to represent wound coils. Wound coils are used most often as sources of current excitation for rotating machines, actuators, sensors, etc. You may directly define a current for each stranded source conductor body.

- Solid Source Conductor Body (p. 1638)
- Stranded Source Conductor Body (p. 1642)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Solid Source Conductor Body

This feature allows you to tag a solid body as a *solid source conductor* for modeling bus bars, rotor cages, etc. When assigned as a solid source conductor, additional options are exposed for applying electrical boundary conditions and excitations to the conductor. These include applying an electrical potential (voltage) or current.

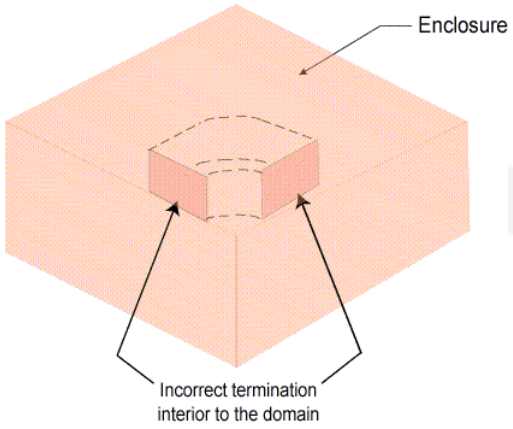
To set this condition, right-click the **Magnetostatic** environment object in the tree and select **Source Conductor** from the **Insert** drop-down menu, or click the **Source Conductor** option from the Context tab. Select the body you want to designate as a conductor body, then use the Details view to scope the body to the conductor and set **Conductor Type** to **Solid**. The default **Number of Turns** is 1, representing a true solid conductor.

A solid source conductor can be used to represent a stranded coil (p. 1642) by setting the **Number of Turns** to > 1. The conductor still computes a current distribution according to the physics of a solid conductor, but in many cases the resulting current density distribution will not significantly affect the computed magnetic field results. This "shortcut" to modeling a stranded conductor allows you to circumvent the geometry restrictions imposed by the stranded conductor bodies and still obtain acceptable results.

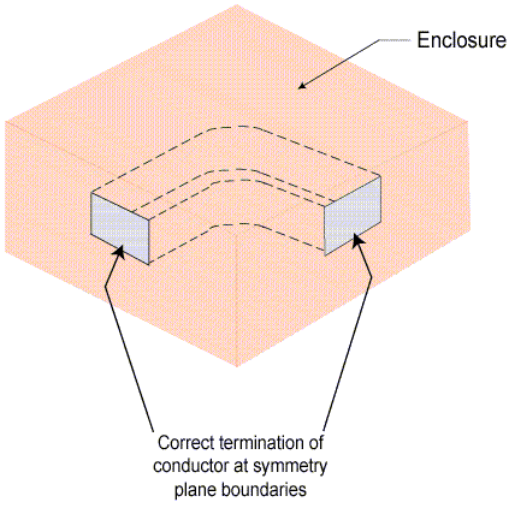
After defining the conductor body, you may apply voltage and current conditions to arrive at the desired state.

Note:

Conductors require two material properties: relative permeability and resistivity. They also must not terminate interior to the model with boundary conditions that would allow current to enter or exit the conductor. Termination points of a conductor may only exist on a plane of symmetry.



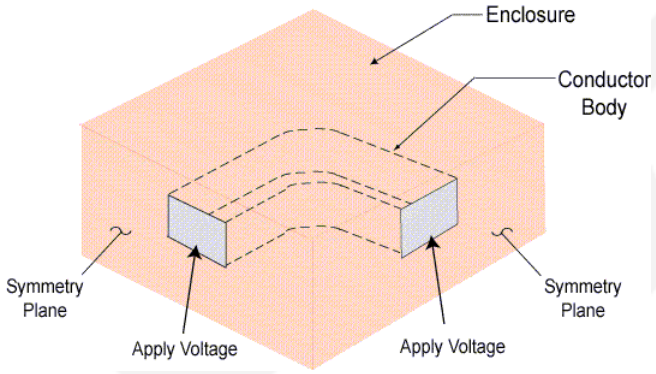
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.



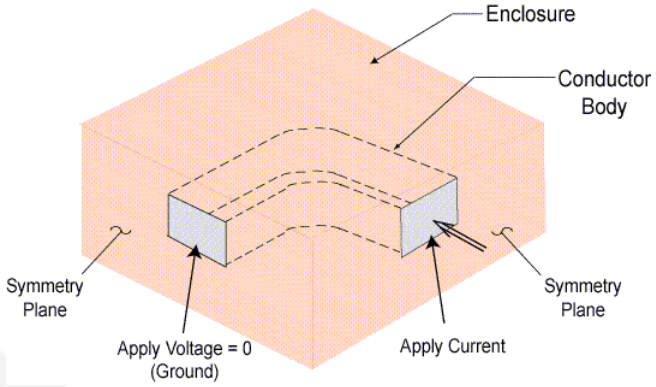
Only bodies can be scoped to a conductor. Solid conductor bodies must have at least one voltage excitation and either a second voltage excitation or a current excitation. Also, two solid conductor bodies may not 'touch' each other, that is, they must not share vertices, edges, or faces.

To establish current in the conductor, you must apply excitation to at least two locations on the conductor, typically at terminals. For example, you could:

- apply a voltage drop at two terminals of a conductor body residing at symmetry planes.



- ground one end of a conductor (set voltage to zero) and apply the net current at the terminal's other end.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Voltage Excitation for Solid Source Conductors

This feature allows you to apply an electric potential (voltage) to a solid source conductor body. A voltage excitation is required on a conductor body to establish a ground potential. You may also apply one to apply a non-zero voltage excitation at another location to initiate current flow. Voltage excitations may only be applied to faces of the solid source conductor body and can be defined as constant or time-varying.

To apply a voltage excitation to a solid source conductor body, right-click the **Conductor** object under the **Magnetostatic** environment object in the tree whose **Conductor Type** is set to **Solid**, and select **Voltage** from the **Insert** drop-down menu, or click the **Voltage** option on the Context tab.

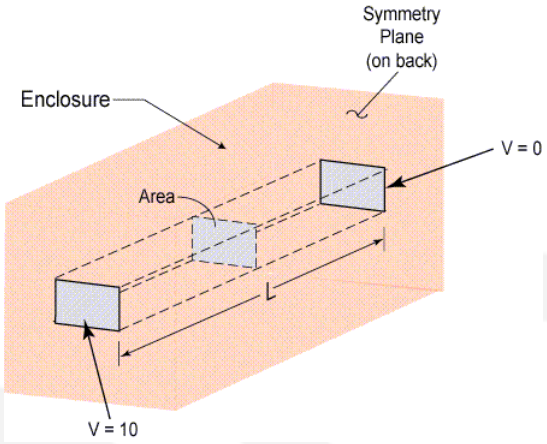
You define the voltage by magnitude and phase angle in the Details view, according to the equation below.

$V = V_o\cos(\omega t+\varphi)$

V_o is the magnitude of the voltage (input value Voltage), ω is the frequency, and φ is the phase angle. For a static analysis, $\omega t = 0$.

Note:

Voltage excitations may only be applied to solid source conductor bodies and at symmetry planes.



An applied voltage drop across the terminals of a conductor body will induce a current. In this simple example, the current in the conductor is related to the applied voltage drop, using the equations shown below. ΔV = applied voltage drop, I = current, ρ = resistivity of the conductor (material property), L = length of the conductor, and Area = cross section area of the conductor.

$\Delta V = IR$

$R = (\rho * L)/Area$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Current Excitation for Solid Source Conductors

This feature allows you to apply a current to a solid source conductor (p. 1638) or stranded source conductor (p. 1642) body. Use this feature when you know the amount of current in the conductor.

To apply a current excitation to a conductor body, right-click the **Conductor** object under the **Magnetostatic** environment object in the tree whose **Conductor Type** is set to **Solid**, and select **Current** from the **Insert** drop-down menu, or click the **Current** option on the Context tab. A positive current applied to a face flows into the conductor body. A negative current applied to a face flows out of the conductor body. For a stranded source conductor, positive current is determined by the y-direction of a local coordinate system assigned to each solid body segment that makes up the conductor.

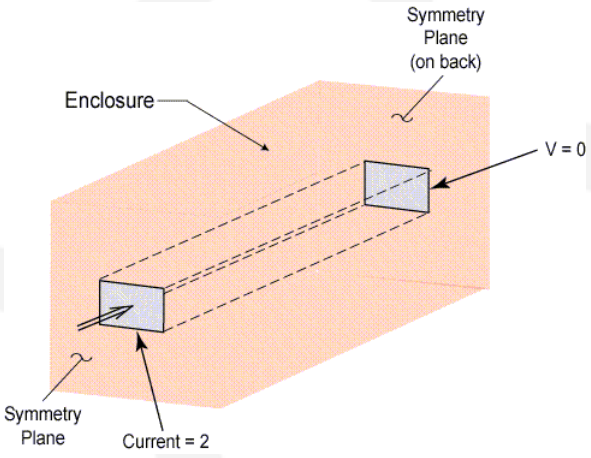
You define the current by magnitude and phase angle in the Details view, according to the equation below.

$I = I_o \cos(\omega t + \varphi)$

I_o is the magnitude of the current (input value Current), ω is the frequency, and φ is the phase angle. For a static analysis, $\omega t = 0$.

Note:

Current excitations may only be applied to a face of a solid source conductor body at symmetry planes. An excitation must be accompanied by a ground potential set at another termination point of the conductor body on another symmetry plane. No current may be applied to a conductor body face that is interior to the model domain. The symmetry plane on which the current excitation is applied must also have a magnetic flux-parallel boundary condition.



An applied current to a conductor face will calculate and distribute the current within the conductor body. A ground potential (voltage = 0) must be applied to a termination point of the conductor body.

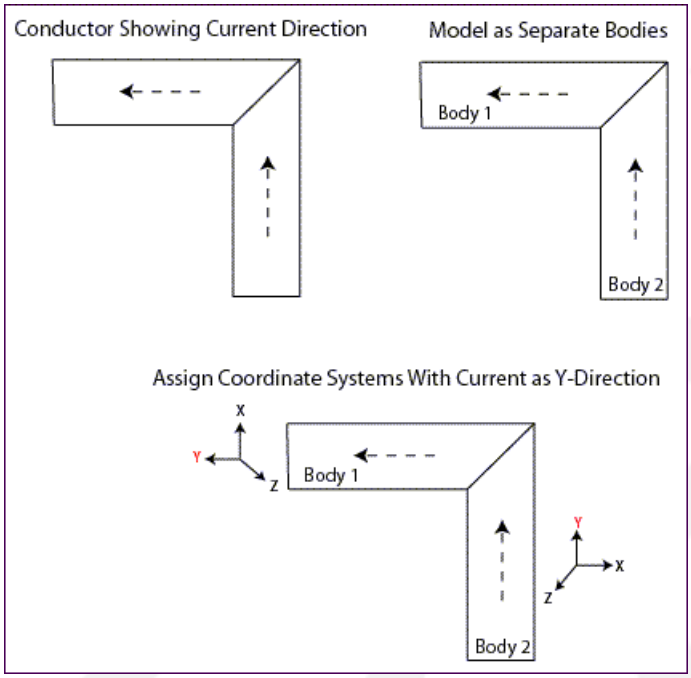
Both the applied current and voltage constraints must be applied at a symmetry plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Stranded Source Conductor Body

This feature allows you to tag solid multiple bodies as a *stranded source conductor* for modeling wound coils. When assigned as a stranded source conductor, additional options are exposed for applying electric boundary conditions and current excitation to the conductor.

Model a stranded source conductor using only isotropic materials and multiple solid bodies. Local coordinate systems assigned to these bodies (via the Details view) are the basis for determining the direction of the current that you later apply to a stranded source conductor. The model should include a separate solid body to represent each directional "turn" of the conductor. Assign a local coordinate system to each body with the positive current direction as the y-direction for each of the local coordinate systems. An illustration is shown below.



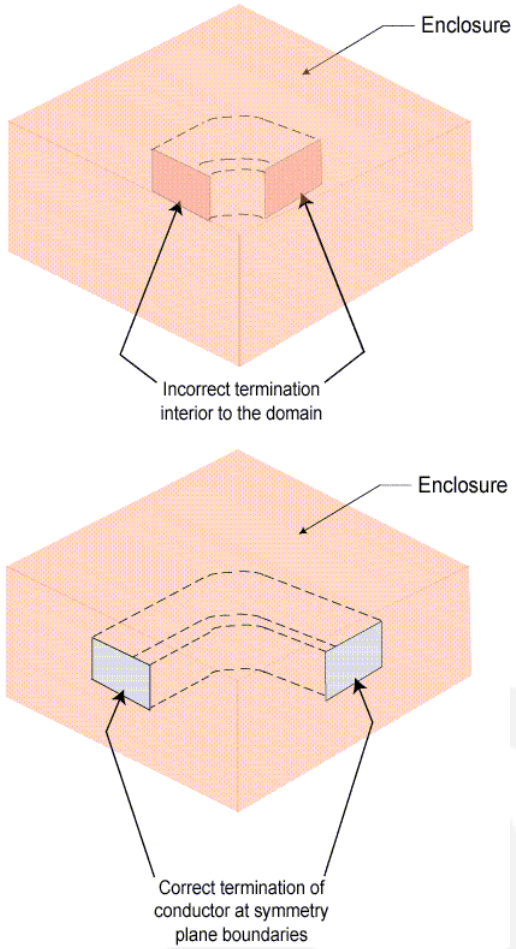
After creating the body segments and assigning coordinate systems, right-click the **Magnetostatic** environment object in the tree and select **Source Conductor** from the **Insert** drop-down menu, or click the **Source Conductor** option on the Context tab. Select all body segments, then scope the bodies to the conductor and, in the Details view, set **Conductor Type** to **Stranded**, then enter the **Number of Turns** and the **Conducting Area** (cross section area of conductor). The stranded conductor is now ready for you to apply a current. A step-by-step example is presented in the Current Excitation for Stranded Source Conductors (p. 1643) section.

Note:

Conductors require two material properties: relative permeability and resistivity. They also must not terminate interior to the model with boundary conditions that would allow current to enter or exit the conductor. Termination points of a conductor may only exist on a plane of symmetry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Current Excitation for Stranded Source Conductors

Stranded source conductor bodies are applicable to any magnetic field problem where the source of excitation comes from a coil. The coil must have a defined number of coil "turns." Stranded source body geometry is limited to straight geometry or circular arc geometry sections with constant cross-section (see below)

Source loading for a coil is by a defined current (per turn) and a phase angle according to the equation below.

$$I = I_o \cos(\omega t + \phi)$$

I_o is the magnitude of the current (input value Current), ω is the frequency, and φ is the phase angle. For a static analysis, $\omega t = 0$. The direction of the current is determined by the local coordinate systems you assign to each of the solid bodies that make up the stranded source conductor. A positive or negative assigned value of current will be with respect to that orientation.

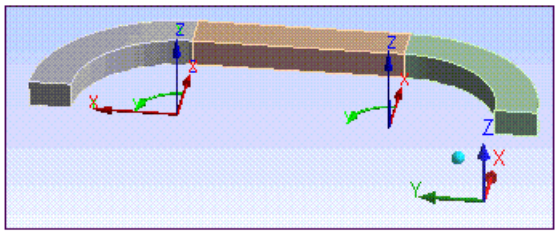
Use the following overall procedure to set up a **Stranded Source Conductor** and apply a current to the conductor:

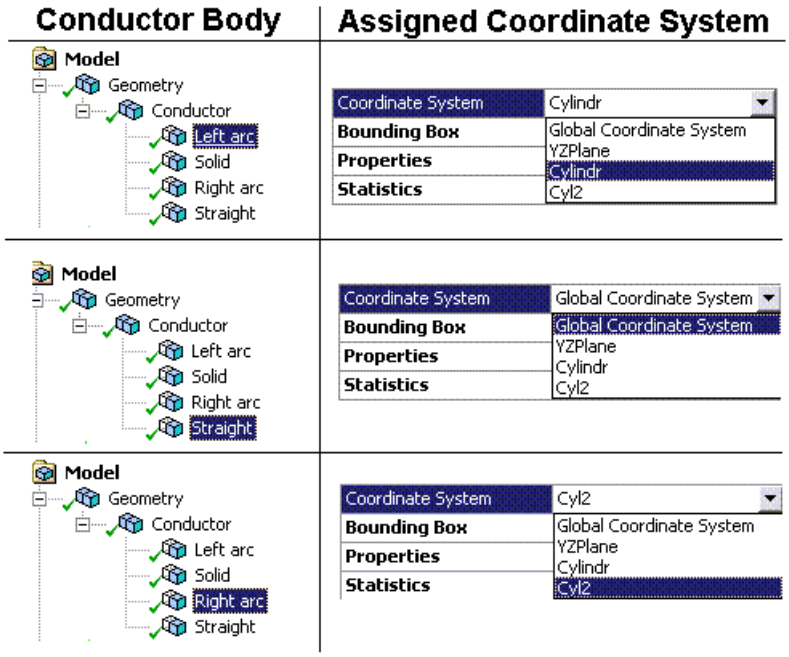
1. Define local coordinate systems that have the y-direction point in the direction of positive current flow.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Use Cartesian coordinate systems for straight geometry sections and cylindrical coordinate systems for "arc" geometry sections.

2. Assign a local coordinate system to each stranded source conductor body in the Details view of the body under the **Geometry** folder.

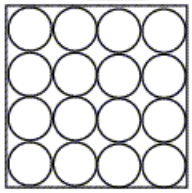




3. Right-click the **Magnetostatic** environment object in the tree and select **Source Conductor** from the **Insert** drop-down menu, or click the **Source Conductor** option on the Context tab.

- Scope the **Source Conductor** to all of the solid bodies.
- Set **Conductor Type** to **Stranded**.
- Enter the **Number of Turns** and **Conducting Area** for the conductor.

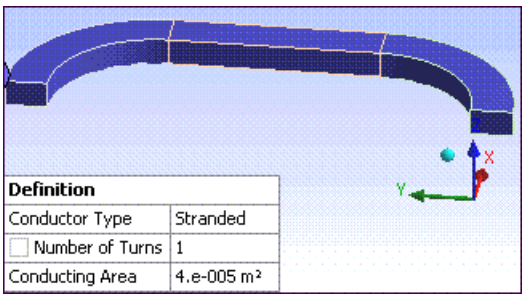
  For the **Conducting Area**, select a face that represents the conductor's cross-sectional area and read the surface area that displays in the Status Bar (p. 36) located at the bottom of the screen display.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

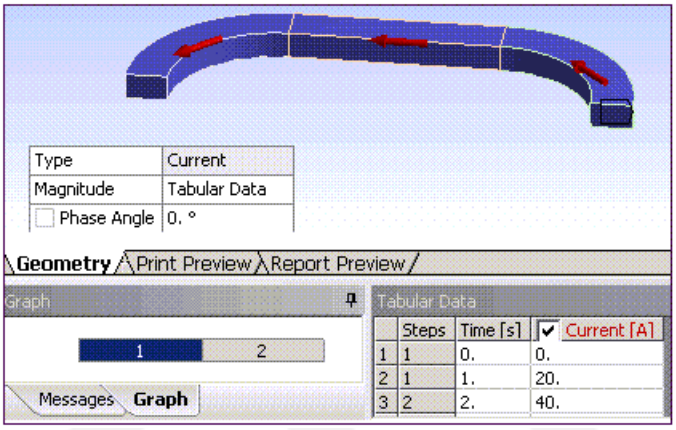
Published: 2023-10-23

The **Source Conductor** graphic and Details view listing is shown below.



4. Right-click the **Conductor** object in the tree and select **Current** from the **Insert** drop-down menu, or click the **Current** option on the Context tab.

   - Set **Magnitude** as constant or time-varying.
   - Set **Phase Angle**.



The **Current** automatically is scoped to the same bodies as the **Source Conductor**.

The displayed current arrows give you visual validation that the current direction has been properly defined by the assigned local coordinate systems for each conductor body.

Changing either the **Type** of **Source Conductor** or any coordinate system will invalidate the setup.

Motion Load

The application interacts with motion simulation software such as Dynamic Designer™ from MSC, and MotionWorks from Solid Dynamics. This is *not* the motion feature that is built into the Mechanical application. See the Rigid Dynamics Analysis (p. 550) and Transient Structural Analysis (p. 665) sections for information on the motion features built into the Mechanical application.

Motion simulation software allows you to define and analyze the motion in an assembly of bodies. One set of computed results from the motion simulation is forces and moments at the joints between the bodies in the assembly. See Inserting Motion Loads (p. 1647) for the procedure on inserting these loads. These loads are available for static structural analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Single Body Capability

Insert Motion Loads is intended to work only with a single body from an assembly. If more than one body is unsuppressed in the **Model** during **Import**, you will receive an error message stating that only one body should be unsuppressed.

Frame Loads File

The application reads a text file produced by the motion simulation software. This file contains the load information for a single frame (time step) in the motion simulation. To study multiple frames, create multiple environment objects for the **Model** and import each frame to a separate environment. The frame loads file includes joint forces and inertial forces which "balance" the joint forces and gravity.

Inertial State

If the part of interest is a moving part in the assembly, the frame loads file gives the inertial state of the body. This includes gravitational acceleration, translational velocity and acceleration, and rotational velocity and acceleration. Of these inertial "loads" only the rotational velocity is applied in the environment. The remaining loads are accounted for by solving with inertia relief (see below).

If the part of interest is grounded (not allowed to move) in the motion simulation, corresponding supports need to be added in the environment before solving.

Joint Loads

For each joint in the motion simulation, the frame loads file reports the force data - moment, force, and 3D location - for the frame. Features are also identified so that the load can be applied to the appropriate faces, edges, or vertices within the application. These features are identified by the user in the motion simulation software before exporting the frame loads file. For all non-zero moments and forces, a corresponding "Moment" and "Remote Force" are attached to the face(s), edge(s) or vertex(ices) identified in the frame loads file.

The Remote Force takes into account the moment arm of the force applied to the joint.

Solving with Inertia Relief

Inertia relief is enabled when solving an environment with motion loads. Inertia relief balances the applied forces and moments by computing the equivalent translational and rotational velocities and accelerations. Inertia relief gives a more accurate balance than simply applying the inertia loads computed in the motion simulation.

If Weak Springs are enabled, the computed reaction forces in the weak springs should be negligible.

This option will automatically be turned on if you import any motion loads.

Note:

Material properties have to be manually set to match density used in motion analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Modifying Parts with Motion Loads

If you modify a part having a motion load, you should rerun the solution in the motion simulator software (for example, Dynamic Designer) and re-export the loads to the Mechanical application. Then, in the Mechanical application, you must update the geometry, delete the load (from the **Environment** object) and re-insert the motion load.

Modifying Loads

You can modify loads that have been inserted, but you should only do so with great care. Modifying loads in the Mechanical application after importing from the motion simulation software will nullify the original loading conditions sets in the motion simulation software. Therefore, you need to examine your results in the Mechanical application carefully.

Inserting Motion Loads

You must make sure the files and data are up to date and consistent when analyzing motion loads. Use the following procedure to ensure that the correct loads are applied for a given time frame.

To insert motion loads after solving the motion simulation:

1. Advance the motion simulation to the frame of interest.
2. Export the frame loads file from the motion software.
3. Attach the desired geometry.
4. Choose any structural **New Analysis** type except **Rigid Dynamics** and **Random Vibration**.
5. Suppress all bodies except the one of interest.
6. Click the environment object in the tree, then right-click and select **Insert> Motion Loads**.
7. Select the Frame Load file that you exported from Dynamic Designer.
8. Click **Solve**. If more than one body is unsuppressed in the **Model** corresponding to the environment object, you will receive an error message at the time of solution stating that only one body should be unsuppressed.
9. View the results.

The exported loads depend on the part geometry, the part material properties, and the part's location relative to the coordinate system in the part document. When any of these factors change, you must solve the motion simulation again by repeating the full procedure. Verify that material properties such as density are consistent in the motion simulation and in the material properties.

Insert Motion Loads is intended to work with a single body only. Results with grounded bodies (bodies not in motion in the mechanism) are not currently supported.

If an assembly feature (such as a hole) is added after Dynamic Designer generates its Joint attachments for FEA, the attachments may become invalid. These attachments can be verified by opening the Properties dialog box for a **Joint** and selecting the FEA tab. An invalid attachment will have a red "X" through the icon. To correct this problem, manually redefine the joint attachments using the FEA tab in the Joint Properties dialog.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

A .log file is created when motion loads are imported. This troubleshooting file has the same name (with an .log extension) and file location as the load file. If the .log file already exists, it is overwritten by the new file.

Fluid Solid Interface

A **Fluid Solid Interface** boundary condition is used to identify the surface region where the transfer of loads to and from external fluid solvers CFX or Fluent occur.

Analysis Types

Fluid Solid Interface is available for the following analysis types:

- Coupled Field (p. 376)
- Harmonic Acoustics
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Steady-State Thermal Analysis (p. 657)
- Transient Thermal Analysis (p. 680)
- Thermal-Electric Analysis (p. 661)

Important:

If you are using System Coupling (p. 820) to perform coupled simulations, Ansys recommends that you use the System Coupling Region (p. 1651) object to define the interfaces.

Note:

- A Static Structural (p. 652) analysis coupled with other physics is intended to work with one substep (specified in the **Analysis Settings**). When a Fluid Solid Interface is present, program controlled sub-stepping will always use one substep regardless of any non-linearities present. See Steps and Step Controls for Static and Transient Analyses (p. 1491) under the Configuring Analysis Settings (p. 1417) section of the Help.
- When one or more FSI loads are present:
  - Any components defined in the Mechanical APDL input file are exported using the **CMWRITE** command to the file, `file.cm`, before the solution is completed. This aids the post-processing of results in CFD-Post.
  - The **Environment** object provides the context menu (right-click) option, **Write System Coupling Files**. For supported analysis types, this option creates an input (`.dat`) file as well as a System Coupling Participant (`.scp`) file that is

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

needed to run a System Coupling analysis (p. 820) using one of System Coupling's user interfaces.

- There is currently a limitation associated with making a name change to your **Fluid Solid Interface** object in Mechanical. The **Setup** cell on the Project Schematic for System Coupling is not automatically updated. You need to perform an action in the application in order to update the **Setup** cell so that it receives the new name.

Mechanical Structural - Fluent

Fluid-solid interfaces define the interfaces between the solid or shell elements in the Mechanical system and the fluid in the Fluent system. These interfaces are defined on faces in the Mechanical model. During the execution of the simulation, System Coupling (p. 820) orchestrates the exchange of data across these interfaces, allowing the Mechanical application and Fluent to send boundary condition results back and forth to one another (one or two-way communication is available).

Mechanical's Static Structural and Transient Structural systems can be coupled with Fluent for a fluid force and structural displacement analysis, or a fluid-thermal-structural analysis. For more information about settings and elements needed for the thermal-structural analysis, see Coupled Field Co-Simulation Using System Coupling (p. 830).

The integer **Interface Number**, found in the Details view, is incremented by default each time a new interface is added. This value can be overridden if desired.

Note:

For **Coupled Field Static** and **Coupled Field Transient** analyses, the application defines interfaces between acoustic and structural regions using the **Fluid Solid Interface** object. To define the interaction between elements in Mechanical and Fluent, use the **System Coupling Region** (p. 1651) object.

Mechanical Acoustics

Fluid-solid interfaces define the interfaces between the acoustic and the structural regions. These interfaces are defined on acoustic region faces. You can use the contextual (right-click) menu option **Create Automatic** > **FSI** on the environment object to automatically identify fluid solid interfaces.

Mechanical Thermal - Fluent

Fluid-solid interfaces define the interfaces between the thermal solid or shell elements in the Mechanical system and the fluid in the Fluent system. These interfaces are defined on faces in the Mechanical model. Data is exchanged across these interfaces during the execution of the simulation as described in Fluid-Structure Interaction (FSI) - One-Way Transfer Using System Coupling (p. 828).

For transferring temperature and heat flows from Mechanical, interfaces may only be defined on the following types of faces:

- On faces having heat fluxes.
- On faces having convections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- On faces with a temperature load.
- On faces without any loads specified (adiabatic). In this case, only temperatures are exchanged.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Fluid Solid Interface** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Fluid Solid Interface**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Fluid Solid Interface**:

1. Based on the analysis type, from the **Environment** Context tab, select the **Fluid Solid Interface** option from either the **Loads** or **Conditions** drop-down menu. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Fluid Solid Interface**.
2. Define the scoping for the object. Face scoping is supported either through geometry picking or the use of a Named Selection.

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Face) and the number of geometric entities (for example: 2 Faces) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Fluid Solid Interface**. **Interface Number**: Incremental value for each new interface. This value can be overridden if desired. **Export Results**: *Thermal analyses only*. The default value for this property is **No**. When this property is set to **Yes**, thermal data is written to **.axdt** files for use with External Data and System Coupling (p. 820), which can connect to Fluent to transfer thermal data to a CFD analysis for a one-way transfer of static data. The file format for an External Data File (.axdt) is described in the External Data File Format Help section in the Workbench User Guide. **Data to Transfer [Expert]**: The default for this property is **Program Controlled**. When set to **All System Coupling Data Transfers**, the fluid solid interface regions can participate in force, displacement, and thermal coupling through System Coupling (p. 820). You need to set **All System Coupling Data Transfers** for Mechanical to participate in a thermal-structural analysis. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Fluid Solid Interface section of the ACT API Reference Guide for specific scripting information.

System Coupling Region

The **System Coupling Region** condition is used to expose surface and volume regions for use in couplings to other products using System Coupling (p. 820).

Analysis Types

System Coupling Region is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Steady-State Thermal Analysis (p. 657)
- Transient Thermal Analysis (p. 680)
- Thermal-Electric Analysis (p. 661)

Note:

- For all Coupled Field analyses, the application supports the System Coupling Region object when the analysis includes either structural or thermal physics, or both physics types.
- When resuming legacy databases that includes a **System Coupling Region** marked as underdefined when only structural or thermal physics was included, Mechanical now automatically marks the object as fully defined. However, in Workbench, the **Setup** cell in workbench may still indicate an underdefined condition. This condition is resolved automatically when the system is solved.

Mechanical Structural - AEDT/Maxwell

System Coupling Region interfaces define the interfaces between thermal solids in the Mechanical system by making available thermal stress to the Maxwell system. These interfaces are defined on bodies alone in the Mechanical model. Data is exchanged across these interfaces during the solution process between the systems. To enable coupling steps, see Completing Participant Setups in the *System Coupling User's Guide*.

To transfer temperature and heat flow loading conditions from Mechanical, you must define the interfaces faces only and on volumes with thermal stresses. When scoped to a face, a **System Coupling Region** boundary condition is used to identify the interface where the transfer of loads to and from external fluid solvers CFX or Fluent occur. And, when scoped to a body, it enables the exchange of thermal stress information from the external Maxwell solver.

Additional System Coupling references (p. 1655) are available at the end of the section.

Common Characteristics

This section describes the characteristics of the **System Coupling Region** boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Geometry Types: Geometry types supported for the **System Coupling Region** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **System Coupling Region** condition is used to expose:

- Body: *Supported*.
- Face: *Supported*.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **System Coupling Region**:

1. On the **Environment** context tab: right-click and select **Insert** > **System Coupling Region**. You can also right-click *in* the **Geometry** window when the **Environment** object is active and select **Insert** > **System Coupling Region**.

   Note:

   This object is inserted into the **Outline** differently based on the analysis type. During structural analyses, it is inserted via the **Loads** drop-down menu and during thermal analyses it is inserted via the **Conditions** drop-down menu.

2. Define the scoping for the object. Body and Face scoping is supported either through geometry picking or the use of a Named Selection.

   The Environment object provides the context menu (right-click) option, **Write System Coupling Files**. For supported analysis types, this option creates an input (.dat) file as well as a system coupling (.scp) file that is needed to run a System Coupling (p. 820) analysis that was created in one of System Coupling 's user interfaces.

Details View Properties

The selections available in the **Details** view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body) and the number of geometric entities (1 Body) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type **System Coupling Region**. **Interface Number**: Incremental value for each new interface. This value can be overridden if desired. **Export Results**: *Thermal analyses only*. The default value for this property is **No**. When this property is set to **Yes**, thermal data is written to **.axdt** files for use with External Data and System Coupling (p. 820), which can connect to Fluent to transfer thermal data to a CFD analysis for a one-way transfer of static data. The file format for an External Data File (.axdt) is described in the External Data File Format section in the Workbench User's Guide. **Data to Transfer [Expert]**: Options include **Program Controlled** (default) and **All System Coupling Data Transfers**. When set to **All System Coupling Data Transfers**, the coupling interface regions can participate in force, displacement, and thermal coupling through System Coupling (p. 820). **Note:** The **Data to Transfer** expert setting is used only to run coupled field analyses using command snippets, which is no longer necessary because coupled field co-simulations (p. 830) are now supported for System Coupling in Workbench. **Note:** • For an analysis that does not include a **Physics Region** object (a single analysis), you must set this property to **All System Coupling Data Transfers** to enable the interface regions to participate in structural-thermal coupling.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	• For the **Coupled Field Static** and **Coupled Field Transient** analyses, this property is read-only. Based on the scoping of this object and the physics type of the corresponding **Physics Region** object, the application determines the participating interface regions. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

System Coupling Region References

Note the following additional references for the use of the System Coupling Region:

- Participant Variables and Quantity Types Supported by System Coupling
- Completing Participant Setups
- Additional Participant Setup Considerations
- Defining Geometry Transformations for Models with Different Orientations

Rotating Force

The **Rotating Force** boundary condition is available for a **Full** Harmonic Response analysis when the **Coriolis Effect** property in the **Rotodynamics Controls** category of the **Analysis Setting** is set to **On, Stationary Reference Frame**. The default setting is **Off**. If the analysis does not specify the **Solution Method** as **Full** and the **Coriolis Effect** property is not turned on, the boundary condition is not valid.

You specify a **Rotating Force** as a force rotating synchronously or asynchronously.

When you set the **Calculated From** property to **Unbalanced Mass**, the force is a specific synchronous force due to Mass Unbalance. The application calculates the actual unbalanced rotating force as show here:

$F_{Unb} = m * r * \Omega^2 = Unb * \Omega^2$

Where:

F_{Unb} = Actual Unbalanced Force due to Unbalanced Mass.

m = Magnitude of mass unbalance (input value from Mass field).

r = Distance from the mass unbalance to the spin axis (input value from Rotating Radius field).

Unb = Unbalanced Force or Mass Unbalance = m * r (computed value shown in Unbalanced Force field).

Ω = Rotational velocity of the structure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When you set the **Calculated From** property to **Direct Input**, the force is a general harmonic rotating forces and it can be applied synchronously or asynchronously based on the value of the **Synchronous Ratio** property.

See the General Asynchronous Rotating Force and Specific Synchronous Forces: Mass Unbalance topics of the Harmonic Analysis with Rotating Forces on Rotating Structures section in the *Mechanical APDL Theory Reference* for specific technical information.

Tips and Hints

Note the following:

- The axis of the **Rotating Force** should coincide with or be parallel to one of the **Global Coordinate** (X, Y, or Z) axes.
- You can create either a specific synchronous configuration or a general rotating asynchronous configuration, but not a combination of the two. A combination would require that two **SYNCHRO** commands be issued on two different components. The **Mechanical Ansys Parametric Design Language (MAPDL) Solver** does not support the use of multiple **SYNCHRO** commands. The solver overwrites any existing **SYNCHRO** definitions when the command is issued.

Important:

The **Rotating Force** boundary condition does not support multiple spool or rotor/stator problems.

Analysis Types

This boundary condition is only available for Harmonic Response (p. 496) analyses using the **Full** Solution Method.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types and Topology: This boundary condition is always applied to all bodies of a model.

Loading Data Definition (p. 1771). The **Rotating Force** boundary condition is defined as a constant.

Boundary Condition Application

To apply a **Rotating Force**:

1. Specify the **Analysis Settings** of the Harmonic Response (p. 496) analysis accordingly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. On the **Environment** Context tab, open the **Loads** drop-down menu and select **Rotating Force**. Or, right-click the **Environment** tree object or within the **Geometry** window and select **Insert**>**Rotating Force**.

   Note:

   By default, the application scopes the **Rotating Force** boundary condition to **All Bodies**. This includes connections (that is, bearings, springs, beams, etc.).

3. Select a desired geometry selection filter, Body, Face, or Edge and then select the **Click to Define** field of the **Axis** property. Select the desired entity to define as the axis of rotation, and then click **Apply**.

   Note:

   - For a Face selection, the axis of rotation is normal to the selected face.
   - For an Edge selection, the axis of rotation is along the selected edge.
   - This axis should coincide with or be parallel to one of the Global Coordinate (X, Y, or Z) axes.

4. Specify the **Defined By** property. The options specify *where* the force is applied on your model.

   Tip:

   The properties of the **Location** category define the point where the **Rotating Force** is applied on the model. Ansys recommends that you do not use a Remote Point to specify this point.

5. Specify the **Calculated From** property. Options include **Unbalanced Mass** (default) and **Direct Input**. This property defines how the force is generated, either based on the mass of the structure or a Magnitude entry.

6. Using the options of the **Hit Point Selection** property, specify whether the application selects a node or a Remote Point as the point on the axis where the Rotating Force (**F** Command) is applied. Options include **Mesh Node** (default) and **Remote Point**. Ansys recommends that you use a Remote Point for this property's definition.

Details View Properties

The properties available in the Details view are described below.

Category	Properties/Options/Description
Scope	**Define By**: A read-only field displaying how the object is scoped - **Geometry Selection**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Properties/Options/Description
	Geometry: A read-only field displaying the geometries that the object is scoped to - **All Bodies**. **Axis**: This property defines the axis of rotation of the scoped geometry and its direction of rotation.
Location	**Define By**: This property specifies *how* the force is applied on your model. The **Define By** property options, listed below, specify *where* the force is applied on your model: **Coordinates** You use this default setting to either: • Select a user-defined coordinate system from the **Coordinate System** property. For all other **Define By** options, the **Coordinate System** property is read-only. • Manually define coordinates using the **X/Y/Z Coordinate** properties. This is the only option that provides the **Location** property. **Geometry Selection** The **Geometry** property becomes visible when the **Define By** property is set to **Geometry Selection**. Select the **Geometry** property field and select the desired geometry. Displays the type of geometry and the number of geometric entities (for example: 1 Face, 2 Edges) based on specified **Location**. **Named Selection** The **Named Selection** property becomes visible when the **Define By** property is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. **Remote Point** The **Remote Points** property becomes visible when the **Define By** property is set to **Remote Point**. This field provides a drop-down list of available user-defined Remote Points. In order for this property to be an available option of the **Defined By** property, you need an existing Remote Point to be defined. **Note:** Selecting a Remote Point on the rotational axis disables the **Unbalanced Force** property because the **Rotating Radius** is zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Description
	X Coordinate/**Y Coordinate**/**Z Coordinate**: These properties are only available for as input properties when the **Define By** property is set to **Coordinates**. Otherwise these properties are read-only. **Location**: Specify a geometric entity on your model to apply the rotating force. Body, Face, and Edge selections are supported.
Definition	**Type**: Read-only field that describes the object - **Rotating Force**. **Calculated From**: Options include **Unbalanced Mass** (default) and **Direct Input**. **Mass**: Entry field when the **Calculated From** property is set to **Unbalanced Mass**. Enter a desired mass value. **Rotating Radius**: This property displays when you set the **Calculated From** property to **Unbalanced Mass**. The application automatically updates the value of this property if you make a change to the selection of the **Location** property. This field becomes read-only when the **Defined By** property is set to **Remote Point**. **Unbalanced Force**: Visible and read-only when the **Calculated From** property is set to **Unbalanced Mass**. Unbalanced Force = Mass * Rotating Radius. **Synchronous Ratio**: Entry field when the **Calculated From** property is set to **Direct Input**. The default value is **1**. **Magnitude**: Entry field when the **Calculated From** property is set to **Direct Input**. **Phase Angle**: Specify a phase shift angle that defines how the load can be out of phase with another load. Boundary conditions supported with the **Phase Angle** property enable you to specify a phase shift that defines how the loads can be out of phase with one another. You can specify the preferred unit for phase angle (in fact all angular inputs) to be degrees or radians using the **Units** menu on the **Home** tab. **Hit Point Selection**: This property specifies whether the application selects a node or a Remote Point as the point on the axis where the Rotating Force (**F** Command) is applied. Options include **Mesh Node** (default) and **Remote Point**. Ansys recommends that you use a Remote Point for this property's definition. **Remote Point for Hit Point**: Visible when the **Hit Point Selection** property is set to **Remote Point**. This field provides a drop-down list of available Remote Points. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition form the solution.
Information	The **Information** category provides read-only information for the following properties:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Properties/Options/Description
	Hit Points • **Hit Point: Node Id**: this information property displays the node identifier at the Hit Point location. It has certain requirements and conditions, including: – The **Hit Point Selection** property must be set to **Mesh Node** – A valid mesh must be generated. – A valid node needs to exist at the Hit Point location. If no node is found, then no information is displayed. – When the **Hit Point Selection** property is set to the **Remote Point** option, this property requires a valid Remote Point and then displays the Named Selection created in `ds.dat` file. • **Hit Point: Location X** • **Hit Point: Location Y** • **Hit Point: Location Z** **Axis Locations** Axis Location coordinates are always with respect to Global Coordinate System. • **Axis Location X** • **Axis Location Y** • **Axis Location Z** **Axis Components** Axis coordinates and Components gives information of where and how the axis is oriented. Axis Components are always unit vectors with respect to Global Coordinate System. • **Axis Component X** • **Axis Component Y** • **Axis Component Z**

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The **CORIOLIS** command is issued to account for gyroscopic damping effect resulted from rotational velocity defined via **CMOMEGA**.
- The **SYNCHRO** command is issued based on specific synchronous/general asynchronous definition of all of the Rotating Forces.
- The **CMOMEGA** command is issued based on axis definition of all of the Rotating Forces.
- The **F** command is applied at Hit Point location based on specific synchronous/general asynchronous definition of Rotating Force.

API Reference

See the Rotating Force section of the ACT API Reference Guide for specific scripting information.

Imported CFD Pressure

The **Imported CFD Pressure** boundary condition enables you to apply the pressure loading data contained in a Fluent-Mechanical coupling data (.cgns) file on the model. The boundary condition maps the complex pressure values contained in the file over structural entities.

Analysis Types

This boundary condition is only available for the Harmonic Acoustics analysis type when a Structural **Physics Region** is defined.

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types

Geometry types supported for the **Pressure** boundary condition include:

- Solid: *Supported*.
- Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology

The following topology selection options are supported for **Pressure**:

- Body: **Not Supported**.
- Face: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types

The boundary condition's loading is defined using the loading data of an imported Fluent-Mechanical coupling data (.cgns) file.

Boundary Condition Application

To apply a **Imported CFD Pressure**:

1. On the **Environment** Context tab open the **Loads** drop-down menu and select **Imported CFD Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert** > **Structural** > **Imported CFD Pressure**.
2. Define the **Scoping Method** as **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Using the **CFD Pressure File** property, select the desired file for import.

Note:

Only one **Imported CFD Pressure** boundary condition is allowed per Environment.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Only face selection is supported. Displays the type of geometry (Face) and the number of geometric entities (for example: 1 Face, 2 Faces) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. Only face-based Named Selections are supported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only property that displays the boundary condition type - **Imported CFD Pressure**. **CFD Pressure File**: This property opens a dialog you use to and browse to and select the desired file. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

The pressure loading data is applied using the **FLUREAD** command by creating the SURF154 element on the selected topology.

API Reference

See the Imported CFD Pressure section of the ACT API Reference Guide for specific scripting information.

Support Type Boundary Conditions

The boundary conditions contained under the **Support** heading are listed below.

Fixed Support (p. 1664)

Displacement (p. 1666)

Remote Displacement (p. 1674)

Velocity (p. 1680)

Impedance Boundary - Explicit Dynamics only

Frictionless Support (p. 1682)

Compression Only Support (p. 1685)

Cylindrical Support (p. 1688)

Simply Supported (p. 1690)

Fixed Rotation (p. 1692)

Elastic Support (p. 1695)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Fixed Support

This boundary condition prevents a selected geometric or mesh entity from moving or deforming.

Analysis Types

A **Fixed Support** is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Modal Acoustics
- Modal (p. 515)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. Fixes one or more faces, edges, or vertices.
- 2D Simulation: *Supported*. Fixes one or more edges or vertices.

Geometry Types: Geometry types supported for the **Fixed Support** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Fixed Support**.

- Body: *Supported for rigid bodies in an Explicit Analysis*.
- Face: *Supported*.
- Edge: *Supported*. A fixed edge is not realistic and leads to singular stresses (that is, stresses that approach infinity near the fixed edge). You should disregard stress and elastic strain values in the vicinity of the fixed edge.
- Vertex: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - A fixed vertex fixes both translations and rotations on faces or line bodies.
  - A fixed vertex is not realistic and leads to singular stresses (that is, stresses that approach infinity near the fixed vertex). You should disregard stress and elastic strain values in the vicinity of the fixed vertex.
  - This boundary condition cannot be applied to a vertex scoped to an end release (p. 1409).
- Node: *Supported*.
- Element Face: *Supported*.
- Element: **Not Supported**.

Note:

If you are using a surface body model, see the Simply Supported (p. 1690) boundary condition section.

Scoping Types: This boundary condition does not require a scoping type because no loading data is required.

Loading Data Definition (p. 1771): Fixed supports do not have loading data.

Boundary Condition Application

To apply a **Fixed Support**:

1. On the **Environment** Context tab: click **Supports**>**Fixed Support**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Fixed Support**.
2. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Fields/Options/Description
	– **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Fixed Support**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Fixed Support section of the ACT API Reference Guide for specific scripting information.

Displacement

Displacements are applied at the geometry level. They require that one or more flat or curved faces or edges or one or more vertices to displace relative to their original location by one or more components of a displacement vector in the *Global Coordinate System* or local coordinate system, if applied (p. 1191).

Important:

Due to an internal processing requirement, if you specify a user-defined (local) Coordinate System when defining this boundary condition, the nodal coordinate system axes may differ from the local Coordinate System axes. As needed, you can verify the actual nodal orientation in the Mechanical APDL application.

Also, see the Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) section for more information about how the application manages nodal DOF constraints.

Displacement as a Base Excitation

Displacement can also be defined as a base excitation during a Mode-Superposition (MSUP) Transient or a MSUP Harmonic Response analysis. You scope base excitations to a boundary condition. You can scope multiple base excitations to the same boundary condition, but the base excitations cannot have same direction specified (via the Direction property).

Important:

When duplicating an analysis within Mechanical that includes loads with the **Base Excitation** property set to **Yes** (Acceleration and/or Displacement), these loads will lose their scoping during the duplication process.

Analysis Types

A **Displacement** is available for the following analysis types:

- Coupled Field Analyses (p. 376)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)

> **Note:**
>
> **Full** Harmonic Response analyses support non-zero displacements for **Component/Components: Real - Imaginary** loading types.

- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. Displaces one or more faces, edges, or vertices.
- 2D Simulation: *Supported*. Displaces one or more edge or vertices.

Geometry Types: Geometry types supported for the **Displacement** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Displacement**.

- Body: *Supported for rigid bodies in an Explicit Analysis*.
- Face: *Supported*.
  - Non-zero X-, Y-, and Z-components. The face retains its original shape but moves relative to its original location by the specified displacement vector. The enforced displacement of the face causes a model to deform.
  - For Zero Y-component, no part of the face can move, rotate, or deform in the Y-direction.
  - For blank (undefined) X- and Z-components, the surface is free to move, rotate, and deform in the XZ plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Edge: *Supported*.
  - Enforced displacement of an edge is not realistic and leads to singular stresses (that is, stresses that approach infinity near the loaded edge). You should disregard stress and elastic strain values in the vicinity of the loaded edge.
  - Non-zero X-, Y-, and Z-components. The edge retains its original shape but moves relative to its original location by the specified displacement vector. The enforced displacement of the edge causes a model to deform.
  - For Zero Y-component, no part of the edge can move, rotate, or deform in the Y-direction.
  - For blank (undefined) X- and Z-components, the edge is free to move, rotate, and deform in the XZ plane.
- Vertex: *Supported*.
  - Non-zero X-, Y-, and Z-components. The vertex moves relative to its original location by the specified displacement vector. The enforced displacement of the vertex causes a model to deform.
  - For Zero Y-component, the vertex cannot move in the Y-direction.
  - For blank (undefined) X- and Z-components, the vertex is free to move in the XZ plane.
  - This boundary condition cannot be applied to a vertex scoped to an End Release (p. 1409).
- Nodes: *Supported*.
- Element Face: *Supported*.
- Element: **Not Supported**.

Note:

- Multiple surfaces, edges, or vertices can be selected.
- Avoid using multiple Displacements on the same face/edge/vertex and on faces/edges/vertices having shared faces/edges/vertices.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Components**: *Supported*.
  - Entering a zero for a component prevents deformation in that direction.
  - Entering a blank for a component allows free deformation in that direction.
  - In a cylindrical coordinate system X, Y, and Z are used for R, Θ, and Z directions. When using a cylindrical coordinate system, non-zero Y displacements are interpreted as translational displacement quantities, $\Delta Y = R\Delta\Theta$. Since they are treated as linear displacements it is a reasonable approximation only, for small values of angular motion $\Delta\Theta$.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - For Explicit Dynamics analyses, when using a cylindrical coordinate system, the Y component (that is, Θ direction) of a displacement constraint is defined as a rotation.

- **Components: Real - Imaginary**. *Supported* for Harmonic Response Analysis only. Define direct loading without **Phase Angle**.
- **Normal To**: *Supported (3D Faces Only)*.
- **Normal To: Real - Imaginary**. *Supported* for Harmonic Response Analysis only. Define direct loading without **Phase Angle**.
- **Magnitude - Phase**. *Supported* for Displacement as a Base Excitation during Harmonic Response analysis only.
- **Real - Imaginary**. *Supported* for Displacement as a Base Excitation during Harmonic Response analysis only.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*. (**Not supported for Harmonic Response analysis**).
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Tabular** (Frequency Varying): *Supported* for Harmonic Response analysis only.
- **Tabular** (Spatially Varying): *Supported* (**Not Supported for Explicit Dynamics and Harmonic Response** analyses).
- **Function** (Time Varying): *Supported* (**Not Supported for Harmonic Response** analyses).
- **Function** (Spatially Varying): *Supported* (**Not Supported for Explicit Dynamics and Harmonic Response** analyses).
- **Free**: *Supported*. (**Not Supported for Displacement as a Base Excitation**).

Note:

Solution Restarts (p. 1815) are only supported for Tabular data modifications.

Boundary Condition Application

To apply a **Displacement**:

1. On the **Environment** Context tab, select the **Displacement** option from the **Structural** group. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Displacement**.
2. Define the **Scoping Method** and select a geometric or mesh entity.
3. Select the method used to define the **Displacement**: **Components** (default), **Components: Real - Imaginary**, **Normal To**, or **Normal To: Real - Imaginary**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. Define the **Coordinate System** and axial displacements or the **Distance**, of the Displacement based on the above selections.

5. As needed, set the **Rev Dir for Inv Steps** property to **Yes**. See the description below for requirements.

6. For Harmonic analyses, specify a **Phase Angle** as needed.

To apply **Displacement** as a **Base Excitation** when the **Solver Type** property is defined as **Mode-Superposition** during a Transient (default setting for a Transient configured to a Modal solution) or a MSUP Harmonic Response analysis:

1. In the **Definition** category of the Details view, set the **Base Excitation** property to **Yes**.

2. The **Boundary Condition** property provides a drop-down list of the boundary conditions that correspond to the **Displacement**. Make a selection from this list. Valid boundary condition option for excitations include:

   - **Fixed Support**
   - **All Fixed Supports**
   - **Displacement**
   - **Remote Displacement**
   - **Nodal Displacement**
   - **Spring**: Body-to-Ground

3. The **Absolute Result** property is set to **Yes** by default. As needed, change the value to **No** if you do not want to include enforced motion.

   Note:

   If you apply more than one base excitation (either **Displacement** or **Acceleration**), the **Absolute Result** property needs to have the same setting, either **Yes** or **No**.

4. To use complex definition entries, change the **Define By** property setting to **Real - Imaginary** from **Magnitude - Phase** (default).

5. Define the loading inputs: **Magnitude**, **Phase Angle** (only in MSUP Harmonic Response), and **Direction**.

   Note:

   You can scope **Acceleration** or **Displacement** as a base excitation to the same boundary condition, but the base excitations cannot have same direction specified (via the **Direction** property).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: options include: • **Geometry Selection**: default setting, indicating that the boundary condition is applied to a geometry/geometries or mesh entity that are selected using a graphical selection tools. – **Geometry**: visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) or mesh entity (Element Face) and the number of geometric/mesh entities (for example: 1 Body, 11 Nodes) to which the boundary has been applied using the selection tools. • **Named Selection**: indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. **Boundary Condition** (Displacement as a Base Excitation only): drop-down list of available boundary condition options for application.
Definition	**Type**: read-only field that describes the object - **Displacement**. **Base Excitation** (**Displacement** as a Base Excitation only): **No** is the default setting. Set to **Yes** to specify the **Displacement** as a **Base Excitation**. **Absolute Result** (**Displacement** as a Base Excitation only): This option allows you to include enforced motion with (**Yes** - default) or without (**No**) base motion. **Define By**. options include: • **Normal To**: Requires entries for the following: – **Distance**. This is the distance of displacement, that is, a magnitude. – **Phase Angle** (Harmonic Analysis (p. 496) only) • **Normal To: Real - Imaginary** (Harmonic Analysis (p. 496) only): real and imaginary distance. Requires the specification of the following inputs: – **Distance - Real** – **Distance - Imag** • **Components**: option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	– **Coordinate System**: drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. – **X Phase Angle** (Harmonic Analysis (p. 496) only) – **Y Phase Angle** (Harmonic Analysis (p. 496) only) – **Z Phase Angle** (Harmonic Analysis (p. 496) only) **Note:** Selection of a Coordinate System rotated out of the global Cartesian X-Y plane is not supported in a 2D analysis. • **Components: Real - Imaginary** (Harmonic Analysis (p. 496) only): option to define the loading type as real and imaginary components (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component - Real**: Defines magnitude (Real) in the X direction. – **X Component - Imaginary**: Defines magnitude (Imaginary) in the X direction. – **Y Component - Real**: Defines magnitude (Real) in the Y direction. – **Y Component - Imaginary**: Defines magnitude (Imaginary) in the Y direction. – **Z Component - Real**: Defines magnitude (Real) in the Z direction. – **Z Component - Imaginary**: Defines (Imaginary) magnitude in the Z direction. • **Magnitude - Phase** (**Displacement** as a Base Excitation for Harmonic Response only): Requires entries for the following: – **Magnitude** – **Phase Angle** (Harmonic Analysis only)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	• **Real - Imaginary** (**Displacement** as a Base Excitation for Harmonic Response only): real and imaginary magnitude. Requires the specification of the following inputs: – **Magnitude - Real** – **Magnitude - Imag** **Rev Dir for Inv Steps**: This property is only available when the following Advanced Analysis Settings (p. 1461) properties are defined: • **Inverse Options** property is set to **Yes**. • **End Step** equals the setting of the **Number of Steps** property. Options include **No** (default) and **Yes**. Setting this property to **Yes** inverts the direction of your specified Displacement and is displayed by the change in direction of the displacement annotation in the Geometry window. **Suppressed**: include (**No** - default) or exclude (**Yes**) the boundary condition.
Step Controls (Visible for Harmonic Response analysis with multiple load steps defined.)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the Remote Load is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Specify a **Step Selection** value from the values available in the **RPM Value** property of the **Analysis Settings** object to use for the Remote Load. **Step Selection**: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to **Load Step**. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the Remote Load.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable when Displacement is defined as a base excitation in a Mode Superposition Transient analysis or a Mode Superposition Harmonic Response analysis.

- **Magnitude** (constant or tabular) is always represented as a table in the input file.
- Base excitation is defined using the **D** command under the Modal restart analysis (under Modal analysis in case of standalone Harmonic Response analysis).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Base excitation is applied using the **DVAL** command during a Mode Superposition Transient analysis or Mode Superposition Harmonic Response analysis.

Note:

Displacement can be defined as base excitation in a Modal linked Harmonic Response and Modal linked Transient analysis only when the upstream Modal analysis **Solver Type** (p. 1428) is set to **Program Controlled** (provided program sets solver type internally to **Direct** or **Supernode**) or **Direct** or **Supernode**.

API Reference

See the Displacement section of the ACT API Reference Guide for specific scripting information.

Remote Displacement

The **Remote Displacement** boundary condition enables you to apply both displacements and rotations at an arbitrary remote location in space. You specify the origin of the remote location under **Scope** category in the Details by picking or by entering the XYZ coordinates directly. The default location of the origin is at the centroid of the model. You specify the displacement and rotation under **Definition**.

Analysis Types

A **Remote Displacement** is available for the following analysis types:

A **Remote Displacement** is classified as a *remote* boundary condition. See the Remote Boundary Conditions (p. 1735) section for a listing of all remote boundary conditions and their characteristics.

- Coupled Field Analyses (p. 376)
- Explicit Dynamics
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Modal (p. 515)[a]
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Rigid Dynamics (p. 550)

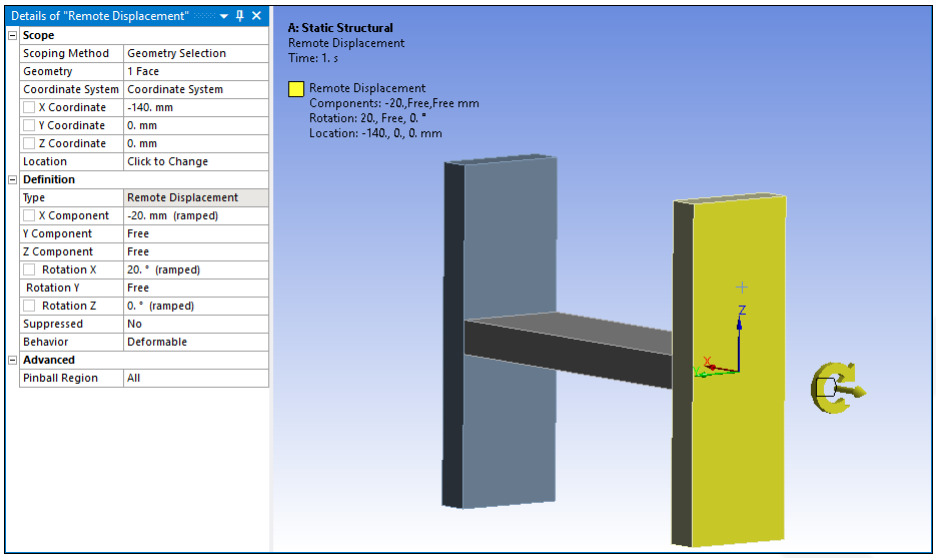
[a] For a Modal analysis, only zero magnitude **Remote Displacement** values are valid. These function as supports. If non-zero magnitude remote displacements are needed for a Pre-Stress Modal analysis, apply the **Remote Displacement** in the static structural environment.

Example Applications

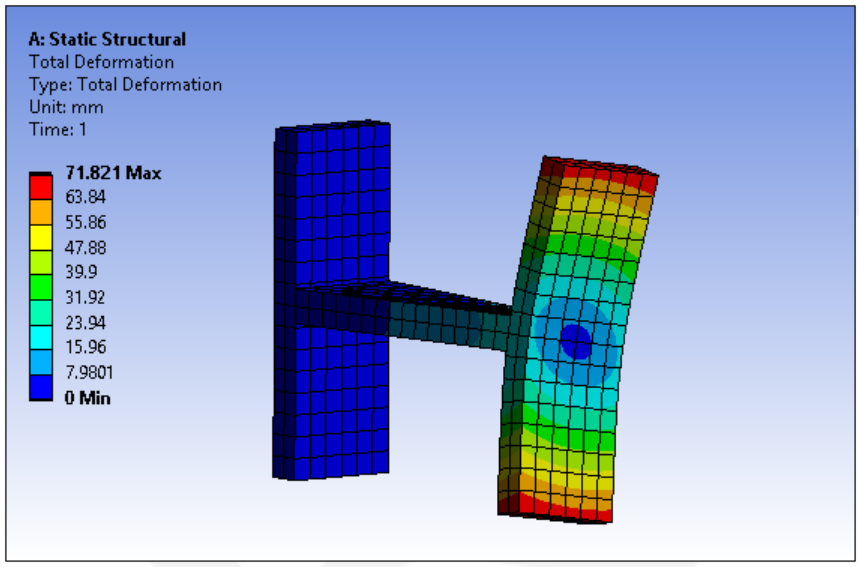
A common application is to apply a rotation on a model at a local coordinate system. An example is shown below along with a plot of the resulting Total Deformation (p. 2018).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Total Deformation Result Example



Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Remote Displacement** boundary condition include:

- Solid: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Remote Displacement**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*. This boundary condition cannot be applied to a vertex scoped to an end release (p. 1409).
- Nodes: *Supported*
- Element Face: *Supported*.
- Element: **Not Supported**.

Applying Remote Loads to a Straight Edge

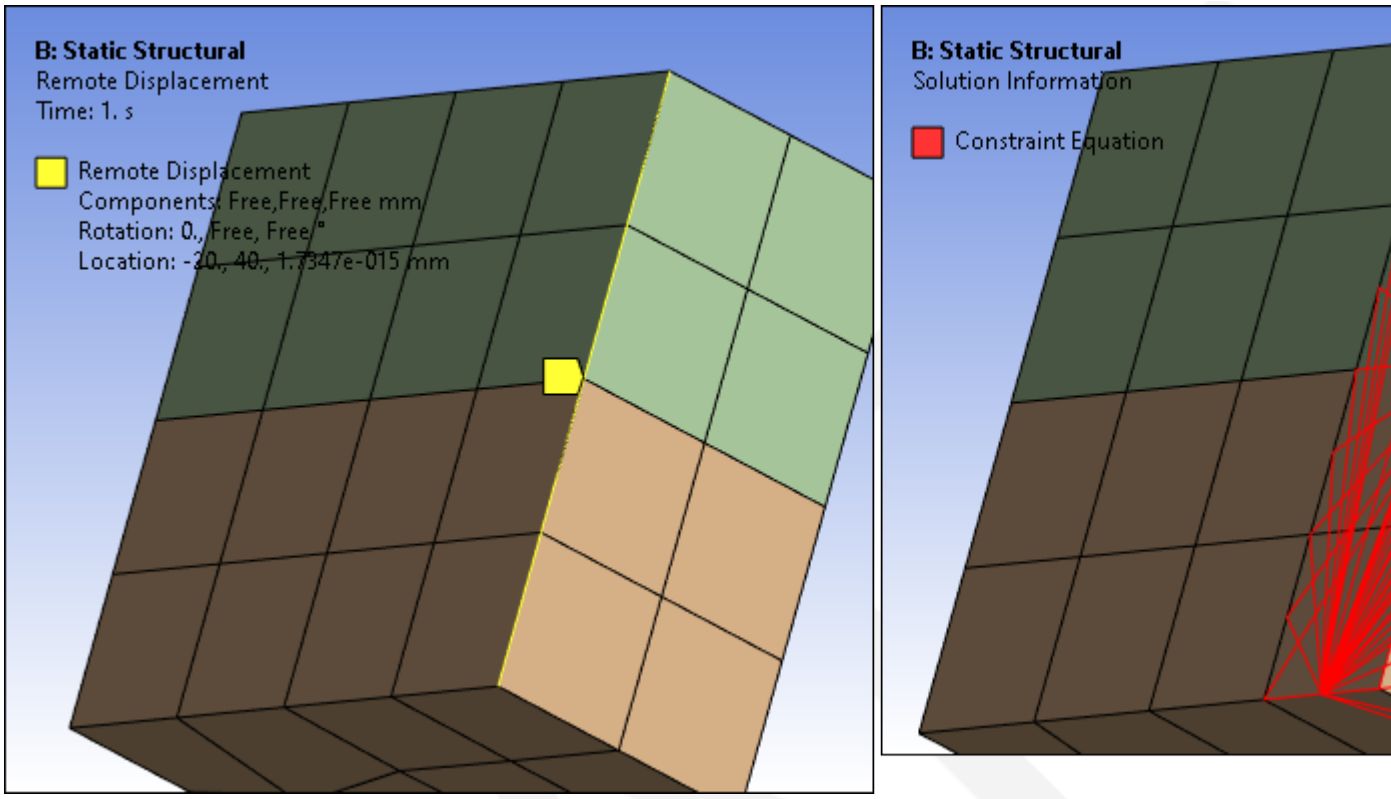
If you apply a remote load to a straight edge (with colinear nodes) of a deformable surface body or a solid body, the application applies the load to the edge as well as the neighboring nodes of the element faces of the deformable surface body or the solid body, respectively. This transfers the rotations of the pilot node (of the **Remote Point**) to the surface or solid body.

Exception: If you scope a remote load to a deformable beam, the application internally changes the setting of the **Behavior** property to **Rigid** and prompts you with a warning.

The following example shows how the application applies a **Remote Displacement**, scoped to a straight edge, to the neighboring element face nodes. In this example, the imbalance of constraint equations created on the blocks might result in unwanted non-zero moment reactions. You can minimize this by using either a lower order mesh or a coarser higher order mesh. See the Surface-Based Constraints section of the *Mechanical APDL Contact Technology Guide* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Loading: This boundary condition's loading in defined in one or more of the following directions.

- **X Component**
- **Y Component**
- **Z Component**
- **X Rotation**
- **Y Rotation**
- **Z Rotation**

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*.
- **Tabular** (Time Varying): *Supported*.
- **Tabular** (Step Varying): *Supported* for Static Structural analysis only.
- **Function** (Time Varying): *Supported*.
- **Free**: *Supported*.

Note:

Solution Restarts (p. 1815) are only supported for Tabular data modifications.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply a **Remote Displacement**:

1. On the **Environment** Context tab: click **Supports**>**Remote Displacement**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Remote Displacement**.
2. Define the **Scoping Method**.
3. Specify the origin of the remote location or enter the XYZ coordinates. The default location is at the centroid of the geometry.
4. As needed, set the **Rev Dir for Inv Steps** property to **Yes**. See the description below for requirements.
5. Specify the translational and rotational displacement components.

Note:

Because a **Remote Displacement** is a remote boundary condition, it can make use of remote points that were either specifically defined or created internally by the application. As a visual check, you can display connection lines between your scoping and the remote point by selecting the **Remote Point Connections** option of the Style (p. 80) group (**Display** tab).

Details Properties

The selections available in the Details are described below.

Category	Property/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections. • **Remote Point**: Indicates that the geometry selection is defined by a **Remote Point**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property/Options/Description
	– **Remote Points**: Visible when the **Scoping Method** is set to **Remote Point**. This field provides a drop-down list of available user-defined Remote Points. **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **Z Coordinate** **Y Coordinate** **X Coordinate** **Note:** Selection of a Coordinate System rotated out of the global Cartesian X-Y plane is not supported in a 2D analysis. **Location**: The location of a Remote Displacement can be defined in the Global Coordinate System or in a local Cartesian coordinate system (p. 1191). It is by default at the centroid of selected geometry.
Definition	**Type**: Read-only field that describes the object - **Remote Displacement**. **X Component**: Defines distance (+/-) in the X direction. **Y Component**: Defines distance (+/-) in the Y direction. **Z Component**: Defines distance (+/-) in the Z direction. **X Rotation**: Defines rotational distance (+/-) in the X direction. **Y Rotation**: Defines rotational distance (+/-) in the Y direction. **Z Rotation**: Defines rotational distance (+/-) in the Z direction. **Rev Dir for Inv Steps**: This property is only available when the following Advanced Analysis Settings (p. 1461) properties are defined: • **Inverse Options** property is set to **Yes**. • **End Step** equals the setting of the **Number of Steps** property. Options include **No** (default) and **Yes**. Setting this property to **Yes** inverts the direction of your specified Remote Displacement and is displayed by the change in direction of the displacement annotation in the Geometry window. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition. **Behavior** (p. 1095): **Rigid**, **Deformable**, **Coupled**, or **Beam**. The **Beam** option specifies a connection from the remote load to the model using linear massless beam elements. If the **Scope Method** property is set to **Remote Point**, the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property/Options/Description
	boundary condition will then assume the **Behavior** defined in the referenced Remote Point as well as other related properties. **Material**: This property is available when the **Behavior** property is set to **Beam**. Select a material to define material properties for the beams used in the connection. Density is excluded from the material definition. **Radius**: This property is available when the **Behavior** property is set to **Beam**. Specify a radius to define the cross section dimension of the circular beam used for the connection.
Advanced	**Pinball Region**: Specify the radius of the sphere (length unit). The displacement is applied to the elements that are within the pinball region.

API Reference

See the Remote Displacement section of the ACT API Reference Guide for specific scripting information.

Velocity

Analysis Types

Velocity is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Explicit Dynamics - For Explicit Dynamics analyses, the Y Component (that is, Θ direction) of a velocity constraint defined with a cylindrical coordinate system has units of angular velocity.
- Transient Structural (p. 665)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Velocity** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Topology: The following topology selection options are supported for **Velocity**.

- Body: *Supported*.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.
  - Avoid using multiple velocities on the same vertex.
  - This boundary condition cannot be applied to a vertex scoped to an end release (p. 1409).
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Loading Types: The boundary condition's loading is defined using one of the following options.

- **Normal To**: *Supported (3D Faces Only)*.
- **Components**: *Supported*.
  - Entering a zero for a component sets the velocity to zero.
  - Entering a blank for a component allows free velocity in that direction.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**
- **Tabular** (Time Varying)
- **Function** (Time Varying)
- **Free**

Boundary Condition Application

To apply a **Velocity**:

1. On the **Environment** Context tab: click **Supports**>**Velocity**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Velocity**.
2. Define the **Scoping Method**.
3. Select the method used to define the Velocity: **Components** (default) or **Normal To**.
4. Define the loading data based on the above selections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Velocity**. **Define By**: Options include: • **Normal To**: Requires a **Magnitude** entry. • **Components**: Option to define the loading type as **Components** (in the Global Coordinate System or local coordinate system, if applied). Requires the specification of at least one of the following inputs: – **Coordinate System**: Drop-down list of available coordinate systems. **Global Coordinate System** is the default. – **X Component**: Defines magnitude in the X direction. – **Y Component**: Defines magnitude in the Y direction. – **Z Component**: Defines magnitude in the Z direction. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Velocity section of the ACT API Reference Guide for specific scripting information.

Frictionless Support

You use this boundary condition to prevent one or more flat or curved faces from moving or deforming in the normal direction. The normal direction is relative to the selected geometry face. No portion of the surface body can move, rotate, or deform normal to the face.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For tangential directions, the surface body is free to move, rotate, and deform tangential to the face.

For a flat surface body, the frictionless support is equivalent to a symmetry condition.

Important:

Due to an internal processing requirement, if you specify a user-defined (local) Coordinate System when defining this boundary condition, the nodal coordinate system axes may differ from the local Coordinate System axes. As needed, you can verify the actual nodal orientation in the Mechanical APDL application.

Also, see the Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) section for more information about how the application manages nodal DOF constraints.

Analysis Types

A **Frictionless Support** is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Frictionless Support** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Topology: The following topology selection options are supported for **Frictionless Support**.

- Body: **Not Supported**.
- Face: *Supported - 3D Only*.
- Edge: *Supported - 2D Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Frictionless Support**:

1. Select the **Frictionless** option from the **Structural** group on the **Environment** Context tab. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Frictionless Support**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Frictionless Support**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

API Reference

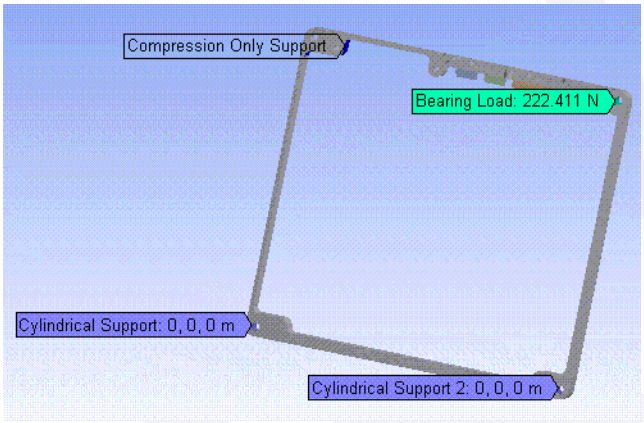
See the Frictionless Support section of the ACT API Reference Guide for specific scripting information.

Compression Only Support

Applies a compression only constraint normal to one or more faces. It is modeled internally using Asymmetric rigid-flexible contact. A rigid target surface is constructed and/or mirrored from the scoped faces/edges of the Compression Only Support. Therefore, the following points should be kept in mind:

- The underlying technology is using penalty-based formulations. As a result, normal contact stiffness can be an important parameter if nonlinear convergence issues arise. Control normal contact stiffness using the Normal Stiffness property of the Compression Only Support object.
- Because source and target topologies are perfect mirrors of one another, be careful during non-linear analyses to make that contact doesn't "fall off" the target face. Be sure that the contacting area on the rigid body is large enough to accommodate any potential sliding taking place during the analysis. To avoid this, consider using a fully fixed rigid body and a nonlinear contact to replace the compression only support.

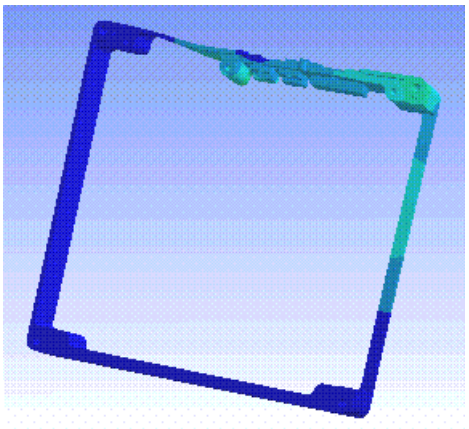
Consider the following model with a bearing load and supports as shown.



Note the effect of the compression only support in the animation of total deformation.

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Since the region of the face in compression is not initially known, a nonlinear solution is required and may involve a substantial increase in solution time.

Analysis Types

A **Compression Only Support** is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Compression Only Support** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Compression Only Support**.

- Body: **Not Supported**.
- Face: *Supported - 3D Only*.
- Edge: *Supported - 2D Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Compression Only Support**:

1. On the **Environment** Context tab: click **Supports**>**Compression Only Support**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Compression Only Support**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.
3. Specify **Normal Stiffness** property. If set to **Manual**, enter a **Normal Stiffness Factor** value.
4. Specify **Update Stiffness** property.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Definition	**Type**: Read-only field that describes the object - **Compression Only Support**. **Normal Stiffness**: Defines a contact Normal Stiffness factor. Options include: • **Program Controlled**: This is the default setting. The Normal Stiffness Factor is calculated by the program. • **Manual**: The **Normal Stiffness Factor** is input directly by the user. The **Normal Stiffness Factor** property displays for this setting. **Update Stiffness**: Specify if the program should update (change) the contact stiffness during the solution. Options include: • **Never**: This is the default setting. Turns off the program's automatic **Update Stiffness** feature. • **Each Iteration**: Sets the program to update stiffness at the end of each equilibrium iteration. • **Each Iteration, Aggressive**: Sets the program to update stiffness at the end of each equilibrium iteration, but compared to the option, **Each Iteration**, this option allows for a more aggressive changing of the value range. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Compression Only Support section of the ACT API Reference Guide for specific scripting information.

Cylindrical Support

For 3D simulations, this boundary condition prevents one or more cylindrical faces from moving or deforming in combinations of radial, axial, or tangential directions. Any combination of fixed and free radial, axial, and tangential settings are allowed.

Important:

Due to an internal processing requirement, if you specify a user-defined (local) Coordinate System when defining this boundary condition, the nodal coordinate system axes may differ from the local Coordinate System axes. As needed, you can verify the actual nodal orientation in the Mechanical APDL application.

Also, see the Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) (p. 1789) section for more information about how the application manages nodal DOF constraints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Types

A **Cylindrical Support** is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Harmonic Acoustics
- Harmonic Response (p. 496)
- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*. Cylindrical supports can only be applied to circular edges

Geometry Types: Geometry types supported for the **Cylindrical Support** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Cylindrical Support**.

- Body: **Not Supported**.
- Face: *Supported - 3D Cylindrical Face Only*.
- Edge: *Supported - 2D Circular Edge Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply a **Cylindrical Support**:

1. On the **Environment** Context tab: click **Supports**>**Cylindrical Support**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Cylindrical Support**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Cylindrical Support**. **Radial**: **Fixed** (default) or **Free**. **Axial** (3D Only): **Fixed** (default) or **Free**. **Tangential**: **Fixed** (default) or **Free**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Cylindrical Support section of the ACT API Reference Guide for specific scripting information.

Simply Supported

Available for 3D simulations only.

This boundary condition prevents one or more straight or curved edges or a vertex or vertices from moving or deforming. However, rotations are allowed. If you want to fix the rotations as well, use

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the Fixed Support (p. 1664) boundary condition. It is applicable for surface body models or line models only.

Analysis Types

A **Simply Supported** is available for the following analysis types:

- Harmonic Response (p. 496)
- Modal (p. 515)
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Explicit Dynamics

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Simply Supported** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: *Supported* - Surface Only.
- Wire Body/Line Body/Beam: *Supported* - Line Only.

Topology: The following topology selection options are supported for **Simply Supported**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*.
- Vertex: *Supported*. This boundary condition cannot be applied to a vertex scoped to an End Release (p. 1409). In addition, a simply supported vertex is not realistic and leads to singular stresses (that is, stresses that approach infinity near the simply supported vertex). You should disregard stress and elastic strain values in the vicinity of the simply supported vertex.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply a **Simply Supported**:

1. On the **Environment** Context tab: click **Supports**>**Simply Supported**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Simply Supported**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type**: Read-only field that describes the object - **Simply Supported**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

API Reference

See the Simply Supported section of the ACT API Reference Guide for specific scripting information.

Fixed Rotation

You can apply a **Fixed Rotation** boundary condition to faces, edges, and vertices of a *surface body*. When you only apply a fixed rotation support to a surface body, the geometry is free in all translational directions. However, by default, the rotation of the geometry is fixed about the axes of the corresponding coordinate system.

Note:

- Rotation constraints are combined with other constraints that produce rotational DOF assignments to determine which values to apply. They are combined with

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

all other constraints to determine the Nodal Coordinate System orientation (frictionless supports, cylindrical supports, given displacements, etc.).

- There may be circumstances in which the rotational support and other constraints cannot resolve a discrepancy for preference of a particular node's coordinate system.

Analysis Types

A **Fixed Rotation** is available for the following analysis types:

- Harmonic Response (p. 496)
- Explicit Dynamics
- Modal (p. 515)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Fixed Rotation** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: *Supported - Surface Body only*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Fixed Rotation**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: *Supported*.
- Vertex: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - A fixed vertex rotation support is not realistic and leads to singular stresses (that is, stresses that approach infinity near the fixed vertex rotation support). You should disregard stress and elastic strain values in the vicinity of the fixed vertex rotation support.
  - This boundary condition cannot be applied to a vertex scoped to an end release (p. 1409).
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Fixed Rotation**:

- In the Details view, select **Free** or **Fixed** for **Rotation X**, **Rotation Y**, and **Rotation Z** to define the fixed rotation support.

1. On the **Environment** Context tab: click **Supports**>**Fixed Rotation**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Fixed Rotation**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.
3. As needed, specify the coordinate system for the corresponding rotational constraint.
4. Define the rotational axes as **Fixed** (default) or **Free**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection** - Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** - Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection** - Indicates that the geometry selection is defined by a Named Selection. – **Named Selection** - Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Type** - Read-only field that describes the object - **Fixed Rotation**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Coordinate System - Drop-down list of available coordinate systems. **Global Coordinate System** is the default. **Rotation X** - **Fixed** (default) or **Free**. **Rotation Y** - **Fixed** (default) or **Free**. **Rotation Z** - **Fixed** (default) or **Free**. **Suppressed** - Include (**No** - default) or exclude (**Yes**) the boundary condition. **Caution:** When parameterizing this boundary condition, a **Free** axis of rotation is represented by a one (**1**) and **Fixed** with a value of zero (**0**) inside the Parameter tab in Ansys Workbench (outside of Mechanical). Entering any value other than 0 or 1 causes the application to produce unexpected results.

API Reference

See the Fixed Rotation section of the ACT API Reference Guide for specific scripting information.

Elastic Support

Allows one or more faces (3D) or edges (2D) to move or deform according to a spring behavior. The **Elastic Support** is based on a **Foundation Stiffness** set in the Details view, which is defined as the pressure required to produce a unit normal deflection of the foundation.

Analysis Types

An **Elastic Support** is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Harmonic Acoustics
- Harmonic Acoustics
- Modal (p. 515)
- Modal Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Elastic Support** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**

Topology: The following topology selection options are supported for **Elastic Support**.

- Body: **Not Supported**.
- Face: *Supported*.
- Edge: *Supported - 2D Only*.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply an **Elastic Support**:

1. On the **Environment** Context tab: click **Supports**>**Elastic Support**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Elastic Support**.
2. Specify **Scoping Method** and **Geometry** or **Named Selection**.

Details View Properties

The selections available in the Details view are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection** - Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** - Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection** - Indicates that the geometry selection is defined by a Named Selection. – **Named Selection** - Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Type** - Read-only field that describes the object - **Elastic Support**. **Suppressed** - Include (**No** - default) or exclude (**Yes**) the boundary condition. **Foundation Stiffness**

API Reference

See the Elastic Support section of the ACT API Reference Guide for specific scripting information.

Conditions Type Boundary Conditions

The boundary conditions contained under the **Conditions** heading are listed below.

- Coupling (p. 1698)
- Voltage Coupling (p. 1700)
- Constraint Equation (p. 1702)
- Pipe Idealization (p. 1704)
- Nonlinear Adaptive Region (p. 1707)
- Geometry Based Adaptivity (p. 1707)
- Element Birth and Death (p. 1707)
- Contact Step Control (p. 1711)
- Plastic Heating (p. 1715)
- Viscoelastic Heating (p. 1717)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Coupling

While setting up a model for analysis, you can establish relationships among the different degrees of freedom of the model by physically modeling the part or a contact condition. However, sometimes there is a need to be able to model distinctive features of a geometry (for example, models that have equipotential surfaces) which cannot be adequately described with the physical part or contact. In this instance, you can create a set of surfaces/edges/vertices which have a coupled degree of freedom by using the **Coupling** boundary condition.

Coupling the degrees of freedom of a set of geometric entity constrains the results calculated for one member of the set to be the same for all members of the set.

Analysis Types

Coupling is available for the following analysis types:

- Electric Analysis (p. 403)
- Steady-State Thermal Analysis (p. 657)
- Transient Thermal Analysis (p. 680)
- Thermal-Electric Analysis (p. 661)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. Apply to one or more faces or edges or at least two vertices.
- 2D Simulation: *Supported*. Apply to one or more edges or at least two vertices.

Geometry Types: Geometry types supported for the **Coupling** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Coupling**.

- Body: **Not Supported**.
- Face: *Supported 3D Only*.
- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Coupling** boundary condition:

1. On the **Environment** Context tab: click **Conditions**>**Coupling**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Coupling**.
2. Define the **Scoping Method**.

Restrictions

Make sure that you meet the following restrictions when scoping **Coupling**.

- You cannot specify more than one Coupling (the same DOF) on the same geometric entity, such as two edges sharing a common vertex or two faces sharing a common edge.
- Coupling should not be applied to a geometric entity that also has a constraint applied to it.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection** - Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** - Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection** - Indicates that the geometry selection is defined by a Named Selection. – **Named Selection** - Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**DOF Selection** - For a Thermal-Electric analysis, select either **Temperature** or **Voltage**, otherwise this is a read-only field displaying the DOF selection type. **Suppressed** - Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Coupling is achieved through the use of the **CP** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

API Reference

See the Coupling section of the ACT API Reference Guide for specific scripting information.

Voltage Coupling

While setting up a model for analysis, you can establish relationships among the volt degrees of freedom of the model by physically modeling the part or a contact condition. However, sometimes there is a need to be able to model distinctive features of a geometry (for example, models that have equipotential surfaces) which cannot be adequately described with the physical part or contact. In this instance, you can create a set of surfaces/edges/vertices which have a coupled degree of freedom by using the Voltage Coupling boundary condition.

Coupling the degrees of freedom of a set of geometric entity constrains the results calculated for one member of the set to be the same for all members of the set.

Analysis Types

Coupling is available for the following analysis types:

- Coupled Field Harmonic (p. 380)
- Coupled Field Modal (p. 386)
- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*. Apply to one or more faces or edges or at least two vertices.
- 2D Simulation: *Supported*. Apply to one or more edges or at least two vertices.

Geometry Types: Geometry types supported for the **Coupling** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Coupling**.

- Body: **Not Supported**.
- Face: *Supported 3D Only*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Edge: *Supported*.
- Vertex: *Supported*.
- Nodes: *Supported* (Coupled field (p. 376) analyses only.).
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Coupling** boundary condition:

1. On the **Environment** Context tab: click **Conditions**>**Voltage Coupling**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Voltage Coupling**.
2. Define the **Scoping Method**.

Restrictions

Make sure that you meet the following restrictions when scoping **Coupling**.

- You cannot specify more than one Voltage Coupling on the same geometric entity, such as two edges sharing a common vertex or two faces sharing a common edge.
- Voltage Coupling should not be applied to a geometric entity that also has a constraint applied to it.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**. Options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Face, Edge, etc.) and the number of geometric entities (for example: 1 Face, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Definition	**DOF Selection**: This is a read-only field displaying the DOF selection type - **Voltage (VOLT)**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

Coupling is achieved through the use of the **CE** command.

Constraint Equation

This feature allows you to relate the motion of different portions of a model through the use of an equation. The equation relates the degrees of freedom (DOF) of one or more Remote Points (p. 1089) for Coupled Field Analyses, Harmonic Response, Harmonic Acoustics, Modal, Modal (Samcef), Static Structural, Static Structural (Samcef), or Transient Structural systems, or one or more joints for the Ansys Rigid Dynamics solver.

For example, the motion along the X direction of one remote point (Remote Point A) could be made to follow the motion of another remote point (Remote Point B) along the Z direction by:

0 = [1/mm · Remote Point A (X Displacement)] - [1/mm · Remote Point B (Z Displacement)]

The equation is a linear combination of the DOF values. Thus, each term in the equation is defined by a coefficient followed by a node (Remote Point) and a degree of freedom label. Summation of the linear combination may be set to a non-zero value. For example:

7 = [4.1/mm · Remote Point A (X Displacement)] + [1/rad · Remote Vertex(Rotation Z)]

Similarly, for the **Ansys Rigid Dynamics** solver, to make the rotational velocity of gear A (Revolute A) to follow the rotational velocity of gear B (Revolute B), in the Z direction, the following constraint equation should be written:

0 = [1/rad · Revolute A (Omega Z)] - [1/rad · Revolute B (Omega Z)]

This equation is a linear combination of the Joints DOF values. Thus, each term in the equation is defined by a coefficient followed by a joint and a degree of freedom label. Summation of the linear combination may be set to a non-zero value. For example:

7 = [4.1/mm · Joint A (X Velocity)] + [1/rad · Joint B (Omega Z)]

Note that the Joints DOF can be expressed in terms of velocities or accelerations. However, all terms in the equation will be based on the same nature of degrees of freedom, that is, all velocities or all accelerations.

To apply a constraint equation support:

1. Insert a **Constraint Equation** object by:

   a. Selecting **Constraint Equation** from the **Conditions** drop-down menu on the Context tab.

   Or...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

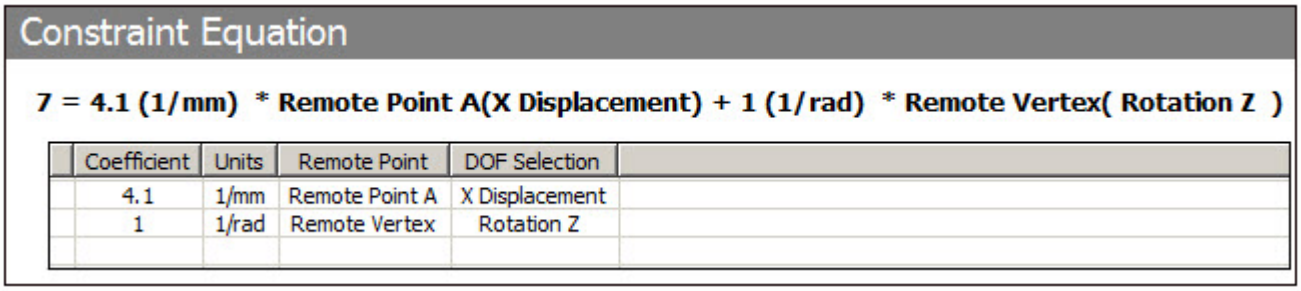
Published: 2023-10-23

b. Right-clicking on the environment object and selecting **Insert> Constraint Equation**.

2. In the **Details** view, enter a constant value that will represent one side of the constraint equation. The default constant value is zero.

3. In the **Worksheet**, right-click in the first row and choose **Add**, then enter data to represent the opposite side of the equation. For the first term of the equation, enter a value for the **Coefficient**, then select entries for **Remote Point** or **Joint** and **DOF Selection**. Add a row and enter similar data for each subsequent term of the equation. The resulting equation displays as you enter the data.

Using the example presented above, a constant value of 7 is entered into the **Details** view, and the data shown in the table is entered in the **Worksheet**.



Note:

For **Harmonic**, **Modal**, **Static Structural**, and **Transient Structural** systems, the first unique degree of freedom in the equation is eliminated in terms of all other degrees of freedom in the equation. A unique degree of freedom is one which is not specified in any other constraint equation, coupled node set, specified displacement set, or master degree of freedom set. You should make the first term of the equation be the degree of freedom to be eliminated. Although you may, in theory, specify the same degree of freedom in more than one equation, you must be careful to avoid over-specification.

Constraint Equation Characteristics

- In the **Worksheet**, you can insert rows, modify an existing row, or delete a row.
- A local coordinate system is defined in each remote point that is used.
- The constant term is treated as a value with no unit of measure.
- Coefficients for X Displacement, Y Displacement, Z Displacement, X Velocity, Y Velocity, Z Velocity, X Acceleration, Y Acceleration, and Z Acceleration have a unit of 1/length.
- Coefficients for Rotation X, Rotation Y, Rotation Z, Omega X, Omega Y, Omega Z, Omega Dot X, Omega Dot Y, and Omega Dot Z have a unit of 1/angle. Note that in a velocity based constraint equation, coefficients use angle units and not rotational velocity units.
- If you change a DOF such that the unit type of a coefficient also changes (for example, rotation to displacement, or vice versa), then the coefficient resets to 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- You can parameterize the constant value entered in the **Details** view.
- The state for the **Constraint Equation** object will be under-defined (**?** in the tree) under the following circumstances:
  - There are no rows with valid selections.
  - Remote Points being used are underdefined or suppressed.
  - Joints being used are underdefined or suppressed.
  - The analysis type does not support this feature.
  - The selected DOFs are invalid for the analysis (2D versus 3D, or remote point versus joints DOFs).
- The graphic user interface does not check for overconstraint.

API Reference

See the Constraint Equation section of the ACT API Reference Guide for specific scripting information.

Pipe Idealization

Pipe Idealization is a (boundary) condition used to model pipes that have cross-section distortion. This is common for curved pipe structures under loading. It is related to the mesh and acts much like a mesh control. Pipe elements are created by meshing lines or curves.

Prerequisites

1. In the Line Body's (Geometry Object) Details view Definition category, the **Model Type** option must be set to **Pipe**.
2. The scoped line-body must be meshed with higher order elements. This means that the **Element Order** option under the Defaults group of the Mesh Object must be set to **Quadratic**. If not, the solver reports an error.

Extend Elbow Elements

You can extend the elbow elements to adjacent edges within the same body in order to reduce the boundary effects caused by the incompatible section deformation between edges modeled as straight pipes and high deformation pipes (elbows). If you do not want to extend the elements, under the **Extend to Adjacent Elements** section of the Details panel set **Extend** to No. To extend the elements, set **Extend** to **Factor**. You can then enter a **Factor** value, which will extend the elements to the adjacent edge up to a length of factor times selected pipe diameter. If the length calculated by factor times pipe diameter is less than the length of one element, it will still be extended by one element.

Support Limitations

Note the following limitations for this condition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- If one or more of the elbow elements has a subtended angle of more than 45 degrees, a warning is reported. The solution can proceed, or you may want to use a finer mesh for better results.
- Pipe Idealization cannot be use with symmetry.
- Although the solution will account for cross section distortions, the graphics rendering for the results will display the cross sections in their original shape.
- Given elbow elements (ELBOW290) with inner pipe temperature specifications only, the application will, by default, specify this inner temperature as outer temperature as well.

Analysis Types

Pipe Idealization is available for the following analysis types:

- Modal (p. 515)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported - Line Bodies Only*. Apply to one or more edges or at least two vertices.
- 2D Simulation: **Not Supported**.

Geometry Types: Geometry types supported for the **Pipe Idealization** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: **Not Supported**.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Pipe Idealization**.

- Body: **Not Supported**.
- Face: **Not Supported**.
- Edge: *Supported*. It can only be scoped to edges that have been modeled as pipes. It can be scoped directly to the geometry or to a Named Selection containing edges that are modeled as pipes.
- Vertex: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Nodes: *Supported* - for node-based Named Selections
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Pipe Idealization**:

1. On the **Environment** Context tab: click **Conditions**>**Pipe Idealization**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Pipe Idealization**.
2. Verify that in the Details panel for the Mesh object, **Element Order** in the Defaults group is set to **Quadratic**.
3. If you choose to **Extend to Adjacent Elements**, enter a **Factor**.
4. Define the **Scoping Method**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method** - Options include: • **Geometry Selection** - Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry** - Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection** - Indicates that the geometry selection is defined by a Named Selection. – **Named Selection** - Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user–defined Named Selections.
Definition	**Suppressed** - Include (**No** - default) or exclude (**Yes**) the boundary condition. **Formulation** - Read-only field defined as **Curved/Deformed**.
Extend to Adjacent Elements	**Extend** - Do not extend to adjacent elements (**No**) or specify as **Factor** (default). **Factor**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Pipe element types include PIPE288 (3D two-node pipe) and PIPE289 (3D three-node pipe).
- If a pipe idealization is scoped to a pipe, the underneath PIPE289 elements of the pipe are modified to ELBOW290 elements.

Nonlinear Adaptive Region

See the Nonlinear Adaptive Region (p. 360) section of the Adaptivity Analyses (p. 359) documentation for information about this feature.

API Reference

See the Nonlinear Adaptive Region section of the ACT API Reference Guide for specific scripting information.

Geometry Based Adaptivity

See the Geometry Based Adaptivity (p. 369) section of the Adaptivity Analyses (p. 359) documentation for information about this feature.

API Reference

See the Geometry Based Adaptivity section of the ACT API Reference Guide for specific scripting information.

Element Birth and Death

The **Element Birth and Death** object enables you to activate and/or deactivate the element status for a specific load step in your analysis.

When you deactivate an element or elements (**Death**), the solver does not remove elements. Instead, the solver deactivates the elements by multiplying their stiffness (or conductivity, or other analogous quantity) by a reduction factor so severe that the application renders the elements inactive.

Any load associated with deactivated elements:

- Zero-out of the loading vector. This includes, mass, damping, specific heat, and other such effects.
- Does not include the mass and energy of deactivated elements in the summations over the model.

The application also sets strains to zero as soon as that element is deactivated.

Likewise, for activated elements (**Alive**), the application does not add the elements to the model. They are simply reactivated. You need to first deactivate an element in order to reactivate the element for the desired load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When you reactivate elements, stiffness, mass, element loads, etc., return to their original values. The application reactivates elements with no record of strain history (heat storage, etc.); the application considers reactivated elements to be generally strain-free. However, thermal strains are computed for newly-activated elements based on the current step temperature and the reference temperature. Therefore, newborn elements with thermal loads may not be stress-free as intended.

This feature is useful for analyzing excavation (as in mining and tunneling), staged construction (as in shored bridge erection), sequential assembly (as in fabrication of layered computer chips), and many other applications in which you can easily identify activated or deactivated elements by their known locations.

Element Birth and Death Requirements

Review the following guidelines when applying the **Element Birth and Death** object:

- You cannot apply Constraint Equations to inactive DOFs. Inactive DOFs occur when a node has no active (**Alive**) elements attached to it.
- You can model stress-relieving operations (such as annealing) by deactivating and then reactivating elements.
- During nonlinear analyses, do not deactivate or reactivate elements in such a way as to create singularities (such as sharp re-entrant corners in a structural analysis) or sudden large changes in stiffness. Convergence difficulties may result.
- The **Program Controlled** setting for the **Newton-Raphson Option** property (**Analysis Settings**> **Nonlinear Controls** (p. 1468)) automatically specifies the **Full** setting if a Dead element status is detected.
- The **Nonlinear Solution** read-only property (**Analysis Settings** > **Analysis Data Management**) is automatically set to the **Yes** setting.
- In order to obtain a good result, Ansys recommends that you set the **Large Deflection** property (**Analysis Settings**> **Solver Controls** (p. 1435)) to **On**.
- If you deactivate a **Contact Region** using the **Contact Step Control** (p. 1711) feature or if you deactivate the underlying elements of a **Contact Region** using the **Element Birth and Death** feature, the application changes the status of the contact pair to far field contact (open and not near contact). This includes bonded contact. To reestablish the pre-death contact status when reactivating the **Contact Region**, you may need to deactivate both the **Contact Region** and their underlying elements.
- If an element in a Static Structural analysis is killed during the first load step and then activated in the subsequent load step, any downstream Linear Perturbation analyses restarted from the activated load step will activate the element during the first phase of the analysis. However, if an element in a Static Structural analysis is active during the first load step and then deactivated in the subsequent load step, any downstream Linear Perturbation analyses restarted from the deactivated load step will deactivate the element during the first phase of the analysis. For additional information, see the First Phase of the Linear Perturbation Analysis in the *Mechanical APDL Structural Analysis Guide*.

For additional information, see the Element Birth and Death topic in the *Mechanical APDL Advanced Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Analysis Types

Element Controls is available for the following analysis types:

- Coupled Field Static and Transient
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal
- Thermal-Electric
- Electric

Common Characteristics

The following section outlines the common condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Element Birth and Death** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The following topology selection options are supported for **Element Birth and Death**.

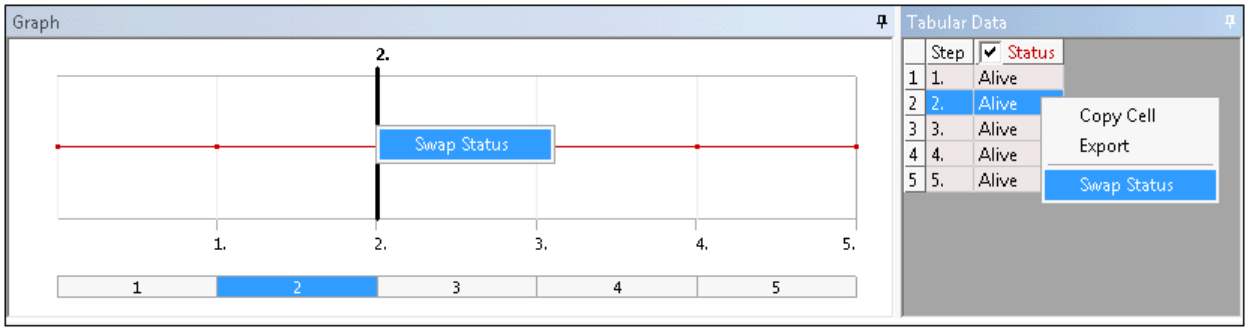
- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Elements: *Supported*.
- Element Face: **Not Supported**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Boundary Condition Application

To apply a **Element Birth and Death** condition:

1. On the **Environment** Context tab: click **Conditions**>**Element Birth and Death**. Or, right-click the **Environment** tree object or within the **Geometry** window and select **Insert**>**Element Birth and Death**.
2. Define your **Scoping Method** and select geometry.
3. Define the desired number of steps in the **Step Controls** (p. 1418) category of the **Analysis Settings**.
4. Select a desired step in the **Current Step** property of the **Element Birth and Death** object.
5. Specify the **Status** of each load step as desired, **Alive** or **Dead**. The default setting for each step is **Alive**. The **Graph** and **Tabular Data** windows each provide the right-click option **Swap Status**. This option also changes the **Status** of a load step.



Details View Properties

The selections available in the Details view are described below.

Category	Properties/Options/Descriptions
Scope	**Scoping Method** - Options include: • **Geometry Selection**: Default setting, indicating that the condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body or element only) and the number of geometric entities (for example: 1 Body, 2 Bodies) to which the condition has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available body- or element-based Named Selections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Descriptions
Definition	**Type**: Read-only field that describes the object - **Element Birth and Death**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the condition.
Step Controls	**Current Step**: This property displays the currently selected Step. This step is also highlighted in the **Graph** window. **Status**. The options for this property include: • **Alive**: Scoped bodies/elements for the current step are active. • **Dead**: Scoped bodies/elements for the current step are deactivated.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable for this condition.

- In order to create the Element Birth and Death component, the application uses the **CM** command for body-based scoping and the **CMBLOCK** command for element-based scoping.
- When the application completes the creation of the Element Birth and Death components, it issues the **ALLSEL ,ALL** command.
- The application issues the **EKILL** command when the **Status** is changed from **Alive** to **Dead** or **Dead** at the first step.
- The application issues the **EALIVE** command when the **Status** is changed from **Dead** to **Alive**.

API Reference

See the Element Birth and Death section of the ACT API Reference Guide for specific scripting information.

Contact Step Control

The **Contact Step Control** condition enables you to activate or deactivate specific contact regions on a load-step basis during your analysis. It also enables you to specify the Normal Stiffness for a specific load step. This condition is similar to the Element Birth and Death (p. 1707) condition that activates/deactivates element status for a specific load step.

Automatic Property Specification

Note the following application controlled actions that occur when applying the **Contact Step Control** object:

- The **Program Controlled** setting for the **Newton-Raphson Option** property (**Analysis Settings** > **Nonlinear Controls**) automatically specifies the **Full** setting if a Dead element status is detected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The **Nonlinear Solution** read-only property (**Analysis Settings** > **Analysis Data Management**) is automatically set to the **Yes** setting.

Requirement

If an element in a Static Structural analysis is killed during the first load step and then activated in the subsequent load step, any downstream Linear Perturbation analyses restarted from the activated load step will activate the element during the first phase of the analysis. However, if an element in a Static Structural analysis is active during the first load step and then deactivated in the subsequent load step, any downstream Linear Perturbation analyses restarted from the deactivated load step will deactivate the element during the first phase of the analysis. For additional information, see the First Phase of the Linear Perturbation Analysis in the *Mechanical APDL Structural Analysis Guide*.

Recommendation

In order to obtain a good result, Ansys recommends that you set the **Large Deflection** property (**Analysis Settings** > **Solver Controls** (p. 1435)) to **On** setting.

If you deactivate a **Contact Region** using the **Contact Step Control** feature or if you deactivate the underlying elements of a **Contact Region** using the **Element Birth and Death** feature, the application changes the status of the contact pair to far field contact (open and not near contact). This includes bonded contact. To reestablish the pre-death contact status when reactivating the **Contact Region**, you may need to deactivate both the **Contact Region** and their underlying elements.

Limitations

The **Contact Step Control** feature does not support Convergence.

Analysis Types

Contact Step Control is available for the following analysis types:

- Coupled Field Static and Transient
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal
- Thermal-Electric
- Electric

Common Characteristics

The following section outlines the common condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Contact Step Control** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: Topology selection is not required for this condition. Scoping is based on contact regions.

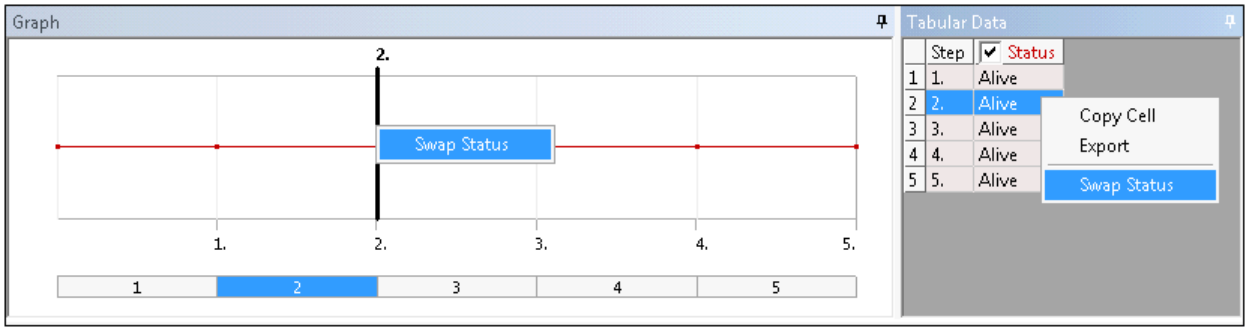
Boundary Condition Application

To apply a **Contact Step Control** condition:

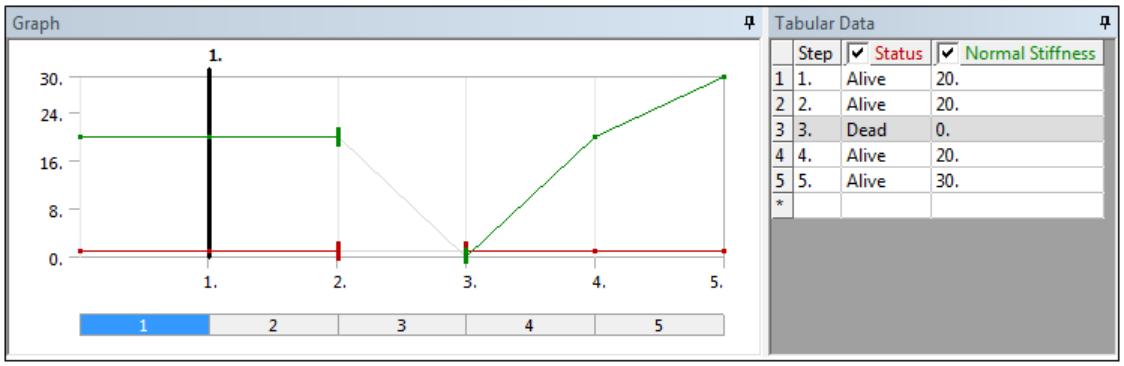
1. On the **Environment** Context tab: click **Conditions**>**Contact Step Control**. Or, right-click the **Environment** tree object or within the **Geometry** window and select **Insert**>**Contact Step Control**.
2. Select the desired **Contact Region**.

> **Note:**
>
> In addition to the above actions, you can drag and drop a Contact Region onto the Environment (for example, Static Structural) object to automatically create a **Contact Step Control** condition scoped to the given Contact Region.

3. Specify the **Normal Stiffness** property. Options include **From Contact Region**, **Factor**, or **Absolute Value**.
4. Select a desired step in the **Current Step** property.
5. Specify the **Status** of each load step as desired, **Alive** or **Dead**. The default setting for each step is **Alive**. The **Graph** and **Tabular Data** windows each provide the right-click option **Swap Status**. This option also changes the **Status** of a load step.



6. If you specify the **Normal Stiffness** property as **Factor** or **Absolute Value**, specify the **Normal Stiffness Factor** or **Normal Stiffness Value** properties for each load step. The values can also be specified in the **Tabular Data** window.



Details View Properties

The selections available in the **Details** view are described below.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: This is a read-only property set to **Contact Region**. **Contact Region**: Select the desired Contact Region from the drop-down list.
Definition	**Type**: Read-only field that describes the object - **Contact Step Control**. **Normal Stiffness**: The options for this property include the following. • **From Contact Region** (default): This setting uses the Contact Region's definition for **Normal Stiffness**. The application makes no modifications for contact normal stiffness due to the **Contact Step Control**. The Normal Stiffness column is hidden in the Tabular Data window. • **Factor**: This setting defines **Normal Stiffness** as a factor for each load step. • **Absolute Value**: This setting defines the **Normal Stiffness** as a value for each load step. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the condition.
Step Controls	**Current Step**: This property displays the currently selected Step. This step is also highlighted in the **Graph** window. **Status**. The options for this property include: • **Alive**: Scoped Contact Region for the current step is active. • **Dead**: Scoped Contact Region for the current step is deactivated. **Normal Stiffness Factor**: This property becomes available when you set the **Normal Stiffness** property as **Factor**. Enter a **Normal Stiffness Factor** for the current step. This is a unit-less entry and only non-zero positive values are

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Properties/Options/Descriptions
	supported. The usual factor range is from **0.01** - **10**. See the Normal Stiffness (p. 1249) topic in the *Advanced Settings* section of the contact documentation for more information. **Normal Stiffness Value**: This property becomes available when you set the **Normal Stiffness** property as **Absolute Value**. Enter the **Normal Stiffness Value** for the current step. The unit of measure for this property is [Force]/[Length]^3 for a traction-based model and [Force]/[Length] for a force-based model. Only non-zero positive values are supported. See the Normal Stiffness (p. 1249) topic in the *Advanced Settings* section of the contact documentation for more information.

Mechanical APDL References and Notes

The following Mechanical APDL commands and considerations are applicable for this condition.

- The application issues the **ESEL** command to select the contact elements by type number.
- The application issues the **EKILL** command when the **Status** property is changed from **Alive** to **Dead** or **Dead** at the first step.
- The application issues the **EALIVE** command when the **Status** property is changed from **Dead** to **Alive**.
- The application issues the **RMODIF** command to modify the **Normal Stiffness Factor** property or the **Normal Stiffness Value** property.

API Reference

See the Contact Step Control section of the ACT API Reference Guide for specific scripting information.

Plastic Heating

The **Plastic Heating** boundary condition enables the thermoplastic effect, that manifests itself as an increase in temperature, during plastic deformation due to the conversion of some of the plastic work into heat.

Analysis Types

Plastic Heating is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Viscoelastic Heating** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported (2D Only)*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Viscoelastic Heating**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Plastic Heating**:

1. On the **Environment** Context tab, select **Plastic Heating** from the **Conditions** drop-down menu of the **Structural** group. Or, right-click the **Environment** object or within the **Geometry** window and select **Insert**>**Plastic Heating**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Specify the **Plastic Work Fraction**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	– **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Plastic Work Fraction**: This value defines the fraction of work that is converted to heat. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Thermoplasticity

API Reference

See the Plastic Heating section of the ACT API Reference Guide for specific scripting information.

Viscoelastic Heating

This condition converts Viscoelastic loss to heat. The **Viscoelastic Heating** object must be scoped to a body whose material assignment includes the Viscoelastic material properties **Prony Shear Relaxation** and **Prony Volumetric Relaxation**, as defined in Engineering Data.

Analysis Types

Viscoelastic Heating is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)

Common Characteristics

This section describes the characteristics of the boundary condition, including the application requirements, support limitations, and loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Viscoelastic Heating** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported (2D Only)*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The following topology selection options are supported for **Viscoelastic Heating**.

- Body: *Supported*.
- Face: **Not Supported**.
- Edge: **Not Supported**.
- Vertex: **Not Supported**.
- Nodes: **Not Supported**.
- Element Face: **Not Supported**.
- Element: **Not Supported**.

Boundary Condition Application

To apply a **Viscoelastic Heating**:

1. On the **Environment** Context tab, select **Viscoelastic Heating** from the **Conditions** drop-down menu of the **Structural** group. Or, right-click the **Environment** object or within the **Geometry** window and select **Insert**>**Viscoelastic Heating**.
2. Define the **Scoping Method** as either **Geometry Selection** or **Named Selection** and then specify the geometry.
3. Specify the **Viscoelastic Loss Fraction**.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**, options include: • **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools. – **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. • **Named Selection**: Indicates that the geometry selection is defined by a Named Selection. – **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.
Definition	**Viscoelastic Loss Fraction**: This value defines the fraction of work that is converted to heat. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Mechanical APDL References and Notes

The following Mechanical APDL commands, element types, and considerations are applicable for this boundary condition.

- Thermoviscoelasticity

Direct FE Type Boundary Conditions

The Direct Finite Element (FE) menu contains options that allow you to apply boundary conditions directly to the nodes on the finite element mesh of a model. These boundary conditions are scoped via node-based Named Selections (p. 1057). They differ from geometry based boundary conditions in the fact that they are applied directly to the nodes during solution calculations whereas geometry-based boundary conditions are applied through special loading elements such as SURF, CONTAC, or FOLLW201 elements.

These boundary conditions are applied in the Nodal Coordinate System (except **Nodal Pressure**). Direct FE boundary conditions **cannot** be applied to nodes that are already scoped with geometry-based constraints which may modify the Nodal Coordinate system.

The boundary conditions contained under the **Direct FE** heading are listed below.

Nodal Orientation

Nodal Force

Nodal Pressure

Nodal Displacement

Nodal Rotation

EM (Electro-Mechanical) Transducer

Nodal Orientation

Nodal Orientation objects are meant to rotate the nodes to a given coordinate system that you select in the GUI. By inserting a **Nodal Orientation** object and scoping it to a subset of nodes, you can create a Nodal Coordinate System and apply nodal rotations to the scoped nodes. Later, other

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

node based boundary conditions (Nodal Force, Nodal Displacements, and Nodal Rotations) can use these Nodal Coordinate Systems.

When two or more Nodal Orientations prescribe different Nodal Coordinate Systems at a single node, the object that is added last (in the tree) is applied.

Analysis Types

Nodal Orientation is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Modal (p. 515)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Nodal Orientation** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The **Nodal Orientation** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Boundary Condition Application

To define **Nodal Orientation** and apply it to nodes:

1. On the **Environment** Context tab, click **Direct FE** > **Nodal Orientation**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Nodal Orientation**.
2. Click the **Named Selection** drop-down list and then select the node-based **Named Selection** to prescribe the scope of the boundary conditions.
3. Select the coordinate system that you want to use to define nodal orientation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Details View Properties

The **Details View** selections are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: read-only field that displays scoping method – **Named Selection** (p. 1057). **Named Selection**: drop-down list of available node-based Named Selections. **Coordinate System**: drop-down list of available coordinate systems. The selected system is used to orientate the nodes in the Named Selection.
Definition	**Suppressed**: includes or excludes the boundary condition in the analysis.

API Reference

See the Nodal Orientation section of the ACT API Reference Guide for specific scripting information.

Nodal Force

Using a **Nodal Force**, you can apply a force to an individual node or a set of nodes. You must create a node-based Named Selection before you can apply a **Nodal Force**. The **Nodal Force** that you apply in Mechanical is represented as an **F** Command in the Mechanical APDL application.

Note:

A **Nodal Force** object may be added during Solution Restart (p. 1815) without losing the restart points.

Analysis Types

Nodal Force is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Eigenvalue Buckling (p. 487) (for Nonlinear-based Eigenvalue Buckling Analysis only)
- Explicit Dynamics
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Nodal Force** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The **Nodal Force** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Note:

The **Nodal Force** boundary condition supports spatially varying loading on the scoped nodes for Static and Transient analyses only. For Harmonic Response and Eigenvalue Buckling analyses, only constant loading conditions are supported.

Boundary Condition Application

To apply a **Nodal Force**:

1. On the **Environment** Context tab, click **Direct FE** > **Nodal Force**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Nodal Force**.
2. Click the **Named Selection** drop-down list and then select the node-based Named Section to prescribe the scope of the Nodal Force.
3. Enter a magnitude for the **X**, **Y**, and **Z** component to define the load.

Tip:

Define a Nodal Orientation for the Named Selection to control the Nodal Coordinate System.

Details View Properties

The **Details View** selections are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**: a read-only field that displays scoping method - **Named Selection** (p. 1057). **Named Selection**: a drop-down list of available node-based **Named Selection**.
Definition	**Type**: a read-only field that describes the node-based object - **Force**. **Coordinate System**: a read-only field that displays the coordinate system - **Nodal Coordinate System**. The Nodal Coordinate System can be modified by applying Nodal Orientation (p. 1719) objects. **X Component**: defines force in the **X** direction **Y Component**: defines force in the **Y** direction **Z Component**: defines force in the **Z** direction You can define the Component values as a **Constant**, in **Tabular** form as a function of varying Time or varying Step (Static Structural only), or as a **Function**. **Divide Load by Nodes**: Property options include **Yes** (default) and **No**. When set to **Yes**, the load value is normalized by dividing the **Magnitude** by number of scoped nodes. When set to **No**, the load value is applied directly to every scoped node. **Non-Cyclic Loading Type**: This property is available for Full Harmonic Analysis when Cyclic Symmetry is present in the model. Options include: • **No** (default). The loading is purely cyclic. That is, the load applied to the base sector is applied to each and every sector. • **Harmonic Index**. The non-cyclic loading can be specified for one or more harmonic indices. This is also known as "engine-order loading" (or traveling wave excitation). A **Harmonic Index** entry is required. • **Sector Number**: This property enables you to select the desired Sector onto which you apply the non-cyclic load. **Harmonic Index**: This property displays when the **Non-Cyclic Loading Type** property is set to **Harmonic Index**. Where NS is Number of Sectors, enter a value from: 1 to NS/2; if NS is even. 1 to (NS-1)/2; if NS is odd. **Sector Number**: This property displays when the **Non-Cyclic Loading Type** property is set to **Sector Number**. You use this property to specify the desired sector onto which you want to apply the non-cyclic load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Suppressed: includes or excludes the boundary condition in the analysis.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays. **Load Vector Number**: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.

Note:

- If you have a **Nodal Force** and a **Pressure**, and/or **Force**, and/or **Hydrostatic Pressure** load that are 1) all are set to the **Direct** option and 2) share the same scoping, they will create a resultant loading effect.
- When **Divide Load by Nodes** is set to **Yes**, the forces are evenly distributed across the nodes and do not result in a constant traction.
- Two **Nodal Force** objects that have same scoping do not produce a cumulative loading effect. The Nodal Force that was specified last takes priority and is applied, and as a result, the other Nodal Force is ignored. For Explicit Dynamics analyses, a resultant effect is always calculated if multiple loads are applied to a node (either by geometric entity or as a nodal force).
- A load applied to a geometric entity and a **Nodal Force** produce a resultant effect.

API Reference

See the Nodal Force section of the ACT API Reference Guide for specific scripting information.

Nodal Pressure

Using **Nodal Pressure**, you can apply pressure on element faces. You must create a node based named selection before you can apply a **Nodal Pressure**. It is applicable for solid and surface bodies only. Specifically, an elemental face pressure is created only if all of the nodes of a given

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

element face (including midside) are included. If all nodes defining a face are shared by an adjacent face of another selected element, the face is not free and will not have a load applied.

Warning:

For application to surface bodies, the Mechanical APDL solver logic for this load is such that if all of the nodes of a shell element are specified, then the load is applied to the whole element face. However, if only some nodes are specified on an element and those nodes constitute a complete external edge, then an edge pressure is created. Therefore, it is critical that you make sure that you have not selected nodes that constitute only a free shell edge. This is because shell edge pressures are input on a per-unit-length basis, and Mechanical treats this load always as a per-unit-area quantity. See the SHELL181 Element Description for more information.

Nodal Pressures applied to shell bodies act in the opposite direction of geometry-based pressures.

Note:

A **Nodal Pressure** may be added during Solution Restart (p. 1815) without losing the restart points.

Analysis Types

Nodal Pressure is available for the following analysis types:

- Coupled Field Static (p. 391)
- Coupled Field Transient (p. 395)
- Eigenvalue Buckling (p. 487) (for Nonlinear-based Eigenvalue Buckling Analysis only)
- Harmonic Response (Full) Analysis Using Pre-Stressed Structural System (p. 506)
- Static Structural (p. 652)
- Substructure Generation (p. 540)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Geometry Types: Geometry types supported for the **Nodal Pressure** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The **Nodal Pressure** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Note:

The **Nodal Pressure** boundary condition supports spatially varying loading on the scoped nodes for Static and Transient analyses only. For Eigenvalue Buckling and Harmonic Response analyses, only constant loading conditions are supported.

Boundary Condition Application

To apply a **Nodal Pressure**:

1. On the **Environment** Context tab, click **Direct FE** > **Nodal Pressure**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Nodal Pressure**.
2. Click the **Named Selection** drop-down list, and then select the node-based Named Selection to prescribe the scope of the **Nodal Pressure**.
3. Enter a magnitude for the load.

Details View Properties

The **Details View** selections are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Read-only field that displays scoping method - **Named Selection** (p. 1057). **Named Selection**: Drop-down list of available node-based Named Selections.
Definition	**Type**: Read-only field that displays boundary condition type - **Pressure**. **Define By**: Read-only field that displays that the boundary condition is acting **Normal To** the surface to which it is attached.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
	Magnitude: Input field to define the magnitude of the boundary condition. This value can be defined as a **Constant**, in **Tabular** form as a function of Time or Step (Static Structural only), or as a **Function**. **Note:** Spatially varying loading (**Tabular**/**Function**) is supported for Static and Transient analyses only. For Eigenvalue Buckling and Harmonic Response analyses, only constant loading conditions are supported. **Suppressed**: Includes or excludes the boundary condition in the analysis.
Load Vector Controls (**Substructure Generation** Analysis Only)	**Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. When set to **Manual**, the **Load Vector Number** property displays. **Load Vector Number**: Specify a **Load Vector Number** using any value greater than **1**. A setting of **1** is reserved for a pre-stress Substructure Generation analysis. **Note:** If multiple loads have the same **Load Vector Number**, the application groups these loads during the solution process.

Note:

- To apply a node-based pressure, the named selections that you create must include nodes such that they define an element face.
- Two **Nodal Pressure** objects that have same scoping do not produce a cumulative loading effect. The **Nodal Pressure** object that was specified last takes priority and is applied, and as a result, the other **Nodal Pressure** object is ignored.
- A load applied to a geometric entity and a **Nodal Pressure** produce a resultant effect.
- You can apply a spatially varying **Nodal Pressure** to scoped nodes.
- If a **Nodal Pressure** and a **Direct Pressure**, **Direct Force**, or **Direct Hydrostatic Pressure** share the same scoping, the **Nodal Pressure** always takes priority regardless of insertion order: Mechanical will ignore the **Direct Pressure**, **Direct Force**, and **Direct Hydrostatic Pressure**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mechanical APDL References and Notes

For more information on the solver representation of this load, reference the **SF** command in the *Mechanical APDL Command Reference*.

API Reference

See the Nodal Pressure section of the ACT API Reference Guide for specific scripting information.

Nodal Displacement

Using **Nodal Displacement**, you can apply a displacement to an individual node or a set of nodes. You must create a node based named selection before you can apply a **Nodal Displacement**.

Analysis Types

Nodal Displacement is available for the following analysis types:

- Coupled Field Analyses (p. 376)
- Eigenvalue Buckling (p. 487) (for Nonlinear-based Eigenvalue Buckling Analysis only)
- Explicit Dynamics
- Modal (p. 515)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Nodal Displacement** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Topology: The **Nodal Displacement** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Note:

The **Nodal Displacement** boundary condition supports spatially varying loading on the scoped nodes for Static and Transient analyses only. For Eigenvalue Buckling, Harmonic Response, and Modal analyses, only constant loading conditions are supported.

Boundary Condition Application

To apply a **Nodal Displacement**:

1. On the **Environment** Context tab, click **Direct FE**>**Nodal Displacement**. Or, right-click the **Environment** tree object or right-click within the **Geometry** window and select **Insert**>**Nodal Displacement**.
2. Click the **Named Selection** drop-down list and then select the node-based Named Section to prescribe the scope of the Nodal Displacement.
3. Define loads in the X, Y, and/or Z directions.
4. As needed, set the **Rev Dir for Inv Steps** property to **Yes**. See the description below for requirements.

Tip:

Define a Nodal Orientation for the Named Selection to control the Nodal Coordinate System.

Details View Properties

The **Details View** selections are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: read-only field that displays scoping method - **Named Selection** (p. 1057). **Named Selection**: drop-down list of available node-based Named Selections.
Definition	**Type**: read-only field that displays boundary condition type - **Displacement**. **Coordinate System**: read-only field that displays the coordinate system - **Nodal Coordinate System**. **X Component**: specify a displacement value in the **X** direction. The default value is **Free** (no Displacement constraint applied).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Fields/Options/Description
	Y Component: specify a displacement value in the **Y** direction. The default value is **Free** (no Displacement constraint applied). **Z Component**: specify a displacement value in the **Z** direction. The default value is **Free** (no Displacement constraint applied). You can define the Component values as a **Constant**, in **Tabular** form as a function of varying Time or varying Step (Static Structural only), or as a **Function**. **Rev Dir for Inv Steps**: This property is only available when the following Advanced Analysis Settings (p. 1461) properties are defined: • **Inverse Options** property is set to **Yes**. • **End Step** equals the setting of the **Number of Steps** property. Options include **No** (default) and **Yes**. Setting this property to **Yes** inverts the direction of your specified Nodal Displacement. **Suppressed**: includes or excludes the boundary condition in the analysis.

Note:

- Solution Restarts (p. 1815) are only supported for Tabular data modifications.
- If a **Component** property is set to **Function**, all other **Components** properties automatically default to the **Free** setting and become read-only.
- Two **Nodal Displacement** objects that have same scoping do not produce a cumulative loading effect. The **Nodal Displacement** object that was specified last takes priority and is applied, and as a result, the other **Nodal Displacement** object is ignored. For Explicit Dynamics analyses, the compatibility of multiple Nodal Displacements applied to a node must be respected. The solver will attempt to combine the constraints, but if this is not possible, the solve will fail with an appropriate error message.

API Reference

See the Nodal Displacement section of the ACT API Reference Guide for specific scripting information.

Nodal Rotation

Using **Nodal Rotation**, you can apply a fixed rotation to an individual node or a set of nodes that have rotational degrees of freedom (DOFs).

Analysis Types

Nodal Rotation is available for the following analysis types:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Modal (p. 515)
- Harmonic Response (p. 496)
- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

- 3D Simulation: *Supported*.
- 2D Simulation: *Supported*.

Geometry Types: Geometry types supported for the **Nodal Rotation** boundary condition include:

- Solid: **Not Supported**.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: *Supported*.

Topology: The **Nodal Rotation** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Boundary Condition Application

To apply a **Nodal Rotation**:

1. On the **Environment** Context tab, click **Direct FE**>**Nodal Rotation**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert**>**Nodal Rotation**.
2. Click the **Named Selection** drop-down list and then select the node-based Named Section to prescribe the scope of the Nodal Rotation.
3. Define the X, Y, and/or Z axis as **Fixed** or **Free**. At least one Component must be defined as **Fixed**.

Tip:

Define a Nodal Orientation for the Named Selection to control the Nodal Coordinate System.

Details View Properties

The **Details View** selections are described below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
Scope	**Scoping Method**: read-only field that displays scoping method - **Named Selection** (p. 1057). **Named Selection**: drop-down list of available node-based Named Selections.
Definition	**Type**: read-only field that displays boundary condition type - **Fixed Rotation**. **Coordinate System**: read-only field that displays the coordinate system - **Nodal Coordinate System**. **X Component**: define the x-axis of rotation as **Fixed** (default) or **Free**. **Y Component**: define the y-axis of rotation as **Fixed** (default) or **Free**. **Z Component**: define the z-axis of rotation as **Fixed** (default) or **Free**. **Suppressed**: includes or excludes the boundary condition in the analysis.

Note:

When parameterizing this boundary condition, a **Free** axis of rotation is represented by a zero (**0**) and **Fixed** with a value of one (**1**) inside the Parameter tab in Ansys Workbench (outside of Mechanical).

API Reference

See the Nodal Rotation section of the ACT API Reference Guide for specific scripting information.

EM (Electro-Mechanical) Transducer

Using the **EM Transducer** boundary condition, you can model simple Micro-Electro-Mechanical Systems (MEMS) devices.

Analysis Types

EM Transducer is available for the following analysis types:

- Static Structural (p. 652)
- Transient Structural (p. 665)

Common Characteristics

The following section outlines the common boundary condition characteristics that include application requirements of the boundary condition, support limitations, as well as loading definitions and values.

Dimensional Types

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- 3D Simulation: *Supported*. Node-based Named Selections only support face node selection.
- 2D Simulation: *Supported*. Node-based Named Selection only support edge node selection.

Geometry Types: Geometry types supported for the **EM Transducer** boundary condition include:

- Solid: *Supported*.
- Surface/Shell: *Supported*.
- Wire Body/Line Body/Beam: **Not Supported**.

Topology: The **EM Transducer** boundary condition is scoped via node-based **Named Selections** only. See the Specifying Named Selections by Direct Node Selection (p. 275) Help section for more information.

Loading Types: The loading for this boundary condition is always defined as a **Voltage Difference**.

Loading Data Definition (p. 1771): Enter loading data using one of the following options.

- **Constant**: *Supported*. This value can be Parameterized.
- **Tabular** (Time Varying): *Supported*. This value **cannot** be Parameterized.
- **Tabular** (Spatially Varying): **Not Supported**.
- **Tabular** (Step Varying): *Supported* (Static Structural only). This value **cannot** be Parameterized.
- **Function** (Time Varying): *Supported*. This value **cannot** be Parameterized.
- **Function** (Spatially Varying): **Not Supported**.

Boundary Condition Application

To apply **EM Transducer**:

1. On the **Environment** Context tab, click **Direct FE** > **EM Transducer**. Or, right-click the **Environment** tree object or in the **Geometry** window and select **Insert** > **EM Transducer**.
2. Enter a **Voltage Difference** value.
3. Specify a **GAP Direction**, either **X**, **Y**, or **Z** based on the default Nodal Coordinate System or a user-defined nodal coordinate system.
4. Enter **Initial Gap** and **Minimal Gap** values.

Details View Properties

The selections available in the Details view are described below.

Category	Fields/Options/Description
Scope	**Scoping Method**: Read-only field that displays scoping method - **Named Selection** (p. 1057).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Fields/Options/Description
	Named Selection: Drop-down list of available node-based Named Selections.
Definition	**Type**: Read-only field that describes the node-based object - **EM Transducer**. **Voltage Difference**: Input field for Voltage value. **Stiffness Method**: Property options include **Program Controlled** (default), **Augmented**, and **Full**. The application determines which setting, **Full** or **Augmented**, the **Program Controlled** option uses based upon the configuration of your system or systems. The **Program Controlled** option selects the **Full** setting when your Static Structural analysis includes a downstream pre-stressed Harmonic Response or Modal analysis and the **Augmented** setting for all other cases. The **Full** setting is required in order to obtain accurate results when your Static Structural analysis includes a downstream pre-stressed Harmonic Response system. These settings are based on a system's stiffness in an electro-mechanical scenario. The stiffness of such systems consist of structural stiffness and electrostatic stiffness. Because forces increase when a spring is stretched, structural stiffness has a positive value. However, given a parallel plate capacitor, the attraction force between the plates decreases with an increasing gap, resulting in a negative electrostatic stiffness. As a result, you can be presented with a negative system stiffness and the analysis can experience convergence problems near unstable solutions. When the **Augmented** option is specified for the **Stiffness Method**, the application sets the electrostatic stiffness to zero, ensuring that the analysis has a positive system stiffness, in order to address convergence issues. In addition, once convergence is reached, the application automatically reestablishes electrostatic stiffness for subsequent analyses. **Note:** For additional technical information about the operation of the **Stiffness Method**, see the TRANS126 Element section in the *Mechanical APDL Element Reference* and the **EMTGEN** command in the *Mechanical APDL Command Reference*. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.
Voltage Surface Location	**Coordinate System**: Read-only field that displays the coordinate system - **Nodal Coordinate System**. **GAP Direction**: Specify the structural DOF used, **X**, **Y**, or **Z** based on the Nodal Coordinate System. This is used with the Volt DOF. **Initial Gap**: Input field for initial range of motion (in GAP Direction). Can be Parameterized. **Minimal Gap**: Input field for minimal range of motion (in GAP Direction). Can be Parameterized.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Fields/Options/Description
Function [1] (p. 1735)	**Unit System**: Read-only field displaying the unit of measure associated with the Voltage. **Angular Measure**: Read-only field displaying the unit of measure for the voltage's angle.
Graph Controls [1] (p. 1735)	**Number of Segments**: The function is graphed with a default value of 200 line segments. You can change this value to better visualize the function.

1. *This category displays only when **Voltage Difference** is specified as a **Function**.*

Mechanical APDL References and Notes

This boundary condition uses the **EMTGEN** command to generate a set of TRANS126 elements between the surface nodes of a moveable structure and a plane of nodes, typically representing a ground plane.

Note:

The newly created (by **EMTGEN** command) ground plane nodes (of TRANS126 elements) are assumed to be fixed.

API Reference

See the EM (Electro-Mechanical) Transducer section of the ACT API Reference Guide for specific scripting information.

Remote Boundary Conditions

The boundary conditions listed here can make use of the **Remote Point** (p. 1089) feature (object) provided by Mechanical. The Remote Point associated with one of the given objects is either created and defined by you (you create a **Remote Point** object that the remote boundary condition references) or it is automatically generated by the system (you can think of it as an "internal" remote point - no **Remote Point** object exists in the object tree).

When defined with a remote point, these objects are considered remote boundary conditions. The remote point gives the object an "abstract" quality because it is not directly applied to the nodes or vertices of a model.

However, you can directly scope a single node or vertex of your model to some of the boundary conditions listed below, specifically **Point Masses**, **Springs**, and **Joints**. Using the Details view property, **Applied By**, for these objects you can switch between the settings **Remote Attachment** and **Direct Attachment**. When directly applied, they are not considered remote boundary conditions and as a result do not provide certain properties, such as **Pinball** or **Formulation**.

- **Point Mass** (p. 901)
- **Thermal Point Mass** (p. 905)
- **Joints** (p. 1292)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Spring** (p. 1395)
- **Bearing** (p. 1413)
- **Beam Connection** (p. 1404)
- **Remote Displacement** (p. 1674)
- **Remote Force** (p. 1552)
- **Moment** (p. 1574)

Remote boundary conditions have the following characteristics:

- All remote boundary conditions make use of MPC contact used in the Mechanical APDL application. See the Geometry Behaviors and Support Specifications (p. 1095) section in the Mechanical Help as well as the Surface-Based Constraints section in the *Contact Technology Guide* - part of the Mechanical APDL Help, for more information.
- You are advised to check reaction forces to ensure that a remote boundary condition has been fully applied, especially if the boundary condition shares geometry with other remote boundary conditions, any type of constraint, or even MPC contact.
- Once a remote boundary condition is created, you can generate an external Remote Point based on the scoping of the remote boundary condition using the **Promote Remote Point** (p. 1099) feature (RMB menu).

Annotations are available for point masses, springs, beam connections, and bearings. You can toggle the visibility of these annotations in the **Annotation Preferences** dialog box. For more information, see Specifying Annotation Preferences (p. 310).

Imported Boundary Conditions

Ansys Workbench and Mechanical analysis systems enable you to import results and apply them as boundary conditions. Mechanical enables you to import data from:

- Mechanical analysis systems as Imported Loads or through Submodeling (p. 807).
- Mechanical Result Files as Imported Loads.
- Fluids Result File as Imported Loads.
- External Data (p. 736) systems as Imported Loads.
- FLUENT CFD analyses using System Coupling (p. 820).
- HFSS, Maxwell, or Q3D Extractor applications as Imported Loads.

Additional information on Thermal-Stress (p. 836), Fluid-Structure Interaction (FSI) (p. 756), Ansoft - Mechanical Data Transfer (p. 706), Icepak to Mechanical Data Transfer (p. 764), Submodeling (p. 807), and External Data Import (p. 736) can be found in the Special Analysis Topics (p. 685) section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

This section outlines the systems that you can link for importing boundary conditions as well as the supported boundary conditions, and it presents the steps to import and apply imported boundary conditions. Select a link below to navigate to a desired topic.

Note:

You can work with imported loads only when you perform an analysis with data transfer.

Supported Analysis Types and Systems
Supported Boundary Conditions
Applying Imported Boundary Conditions
Load Mapping Workflow Specification

Supported Analysis Types and Systems

The following table lists the systems that interact with one another using data transfer. The associated imported boundary conditions (as a data type) are show in parenthesis.

Source Analysis/System (Transfer Data Type)	**Target Analysis**
CFD (Convection)	Steady-State Thermal, Transient Thermal, Thermal-Electric
CFD (Pressure)	Static Structural, Transient Structural [a] [1] (p. ?)
CFD (Temperature)	Steady-State Thermal, Transient Thermal, Thermal-Electric, Static Structural, Transient Structural[a]
CFD (Body Temperature)	Static Structural, Transient Structural, LS-DYNA
Coupled Field Harmonic (Joule Heat)	Steady-State Thermal and Transient Thermal
Coupled Field Harmonic (Velocity)	Stand-alone Acoustics Coupled Field Harmonic
Electric (Joule Heat)	Steady-State Thermal, Transient Thermal
Electromagnetic (Force Density)	Static Structural, Transient Structural[a]
Electromagnetic (Power Loss Density)	Steady-State Thermal, Transient Thermal
Electromagnetic (Force and Moment)	Harmonic Response[b]
External Files (Force, Temperature, Stress, Strain, Body Force Density)	Static Structural, Transient Structural[a]
External Files (Displacement)	Static Structural, Transient Structural[a], LS-DYNA
External Files (Temperature, Convection, Heat Flux, Heat Generation)	Steady-State Thermal, Transient Thermal, Thermal-Electric
External Files (Pressure)	Harmonic Response, Explicit Dynamics, LS-DYNA, Static Structural, Transient Structural[a], and Substructure Generation
External Files (Velocity)	Acoustic Analysis
External Files (Imported Heat Flux, Imported Temperature, Imported	Coupled Field Static, Coupled Field Transient

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Source Analysis/System (Transfer Data Type)	Target Analysis
(Thermal Condition) Body Temperature)	
External Files (Imported Body Force Density)	Harmonic Response
Harmonic Response (Velocity)	Harmonic Acoustic Analysis
Motion	Harmonic Acoustic Analysis, Coupled Field Harmonic
Polyflow (Temperature)	Steady-State Thermal, Transient Thermal, Thermal-Electric, Static Structural, Transient Structural[a]
Static Structural, Transient Structural (Displacement, Temperature, Force[c], and Moment[c])	Static Structural, Transient Structural[a]
Steady-State Thermal, Transient Thermal (Temperature)	Static Structural, Transient Structural[a], Electric
System Coupling (Force)	Static Structural, Transient Structural
System Coupling (Heat Flow, Convection, Temperature)	Steady-State Thermal, Transient Thermal, Thermal-Electric
Thermal-Electric (Temperature)	Static Structural, Transient Structural[a]

[a] Rigid dynamics solver is not supported.

[b] See the Importing Data into a Harmonic Analysis (p. 710) section for the specific steps to perform the analysis.

[c] Supported for Beam to Shell or Solid Submodeling only

Supported Boundary Conditions

Mechanical supports the boundary conditions listed below. The availability of a boundary condition in an analysis is dependent upon the types of systems that you have joined.

- Imported Body Force Density
- Imported Body Temperature
- Imported Boundary Remote Constraint
- Imported Convection Coefficient
- Imported Cut Boundary Constraint
- Imported Cut Boundary Remote Force
- Imported Displacement
- Imported Force
- Imported Heat Flux
- Imported Heat Generation
- Imported Initial Strain
- Imported Initial Stress
- Imported Pressure
- Imported Remote Loads

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Imported Surface Force Density

Imported Temperature

Imported Velocity

Imported Body Force Density

When electromagnetic body forces are transferred to a structural environment, an **Imported Body Force Density** object can be inserted to represent the transfer. See the Applying Imported Boundary Conditions (p. 1757) section for steps to transfer data.

Note:

- For a particular load step, an active **Imported Body Force Density** load will overwrite other **Imported Body Force Density** loads that exist higher (previously added) in the tree, on common geometry selections. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.
- For large-deflection analyses, the application applies this load to the initial size of the element, not the current size.

Go to a section topic:

- Comparing Imported Body Force Density Total Forces (Source and Target) (p. 1739)
- Limitations (p. 1740)

Comparing Imported Body Force Density Total Forces (Source and Target)

When your imported body force density load, imported from the Workbench External Data system, is scoped to solid bodies/elements and each source element includes volume data, a table comparing the total force components and magnitudes for both the source and target mesh is displayed in the Comment pane of the Imported Load Transfer Summary object, as shown below.

Row	Source				Target			
	Force X (N)	Force Y (N)	Force Z (N)	Total Force (N)	Force X (N)	Force Y (N)	Force Z (N)	T
1	5.75	4.25	4.875	8.6539	5.8309	4.2351	4.938	

To display this information, you must first specify **Volume** in the **Data Type** column in the External Data **Setup** page as well as a unit system and **Volume1** as the **Data Identifier** for the desired volume data. The example below illustrates these settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	A	B	C	D	E
1	Column	Data Type	Data Unit	Data Identifier	Combined Identifier
2	A	X Coordinate	m		File1
3	B	Y Coordinate	m		File1
4	C	Z Coordinate	m		File1
5	D	Body Force Density	N m^-3	Body Force Density1	File1:Body Force Density1
6	E	Body Force Density	N m^-3	Body Force Density2	File1:Body Force Density2
7	F	Body Force Density	N m^-3	Body Force Density3	File1:Body Force Density3
8	G	Volume	m^3	Volume1	File1:Volume1

The application calculates each component of the total force of the source file by summing force components multiplied by the volume of each source data row.

Important:

- This table does not display if you specify an Analytical Transformation to a coordinate in the upstream External Data system.
- The **Gallon** unit in the External Data system is an Imperial (UK) Gallon and equals 4.5461e-3 cubic meter.

Limitations

Static and Transient Structural Analyses

The application does not support sending the **Imported Body Force Density** load to the solver as a table. For this imported load, the **Tabular Loading** property controls the creation of the data table. The values are then applied from the data table at each load step. However, the **KBC** command value (*Key*) specified for the analysis controls the ramping or stepping within a load step. Therefore, if the analysis time specified by the Imported Load definition matches the load step end time, then there is no difference in solution between the **Stepped** and the **Ramped** option (of **Tabular Loading** property) for the load step.

Harmonic Response Mode Superposition (MSUP) Analyses

For a Harmonic Response MSUP analyses, the **Element Based Volumetric Harmonic Force** load is not supported:

- As tabular data (frequency varying).
- For analyses that include Multiple RPMs.

Imported Body Temperature

When temperatures are transferred to a structural, electric, or LS-DYNA analysis, an **Imported Body Temperature** object is automatically inserted to represent the transfer.

If the load is applied to one or more surface bodies, the **Shell Face** option in the details view enables you to apply the temperatures to **Both** faces, to the **Top** face(s) only, or to the **Bottom**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

face(s) only. By default, the temperatures are applied to both the top and bottom faces of the selection. See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Be sure to review the following information about the requirements and considerations when employing this loading condition.

- Operational Rules for Imported Thermal Loads (p. 1741)
- Tabular Loading Processing Specifications (p. 1741)
- General Notes (p. 1743)
- Load Vector Controls for Substructure Generation Analyses (p. 1744)

Operational Rules for Imported Thermal Loads

Observe the following operational conditions when you apply Imported Body Temperature loads and Thermal Condition on the same geometry. For each load step:

- An active or reactivated Imported Body Temperature load overwrites any Thermal Condition loads on common geometry selections.
- The most recently added Imported Body Temperature load, whether active or reactivated overwrites any other Imported Body Temperature loads that exist in the tree and that share common geometry or node selections.
- Imported Body Temperature loads for which no commands are sent to the solver, for a particular load step, are overwritten by a reactivated or deactivated:
  - Thermal Condition load on common geometry selections.
  - Imported Body Temperature load on common geometry selections.

  The Imported Body Temperature will remain deleted until it gets reactivated during one of the subsequent steps (status change from deactivated to active).

See the Activating and Deactivating Loads (p. 1423) Help for additional rules about when multiple load objects of the same type exist on common geometry selections.

Note:

For an active, reactivated, or deactivated loading condition, the application sends a command to the solver. The Tabular Loading topic below describes when during the solution process the command is sent to the solver.

Tabular Loading Processing Specifications

Processing Characteristics

Note the following tabular loading characteristics for Imported Body Temperature loads:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- When Imported Body Temperature loads are not sent using tables, the commands are sent to the solver at load steps specified in the Data View worksheet or when the load is reactivated or deactivated.
- For Imported Body Temperature loads specified in Tabular Data, commands are sent to the solver only at the first active or reactivated load step.

Defining the Program Controlled Option

The **Tabular Loading** property has the following options:

- **Program Controlled**: The application selects either **Ramped** or **Off**. The selection depends upon whether the **Analysis Time (s)** values in the **Data View** pane match the **End Time** values of the **Tabular Data** window. If the value match, the application selects the **Off** option. If they do not match, then the **Ramped** option is used.
- **Ramped**: Temperature values are Ramped based on the **Analysis Time (s)** table.
- **Stepped**: Temperature values are Stepped based on the **Analysis Time (s)** table.
- **Off**: Temperature values are applied as a constant at the end of each load step and ramped during the load step.

Tip:

The application processes the **Ramped** and **Off** options the same except that when using the **Off** option, **Analysis Time** values must match the **End Times** exactly. Otherwise, the application generates an error message. The **Ramped** option is not affected buy this limitation.

Specifying Source Time

The **Source Time** property includes the following options:

- **All**: All time points from the upstream Thermal analysis populate the **Source Time (s)** column in the Data View pane. And, by default, the **Analysis Time (s)** column contains the same values. When selected, a **Graphics Controls** category displays in the Details pane enabling you to specify the display of the load data by **Active Row** or **Time**.
- **Range**: This option filters the data points from the upstream Thermal analysis based on the minimum and maximum input values and populates the **Source Time (s)** column accordingly. When selected, the additional properties, **Minimum** and **Maximum** display. These properties enables you to enter minimum and maximum time values for which to display results. When you enter minimum and maximum time values, a **Graphics Controls** category displays in the Details pane enabling you to specify the display of the load data by **Active Row** or **Time**.
- **Worksheet** (default): Manually enter desired data points to obtain load data from the upstream Thermal analysis in order to connect them to **Analysis Time** of the down-stream Structural analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Based on the option that you select for the **Source Time** property, the **Tabular Loading** property is set depending upon on whether the values of **Analysis Time** column match the values of the Load Step **End Times** values of the Structural analysis.

General Notes

Review to the following to make sure that you properly configure your imported load.

- If you have a Nonlinear Adaptive Region (p. 1707) object present in a Static Structural analysis, the **Imported Body Temperature** includes the property: **Apply To Initial Mesh**. You use this property to specify whether the boundary conditions are applied directly to the initial mesh or to the newly generated mesh.
- **Adaptive Convergence** objects inserted under an environment that is referenced by an **Imported Body Temperature** object will invalidate the **Imported Body Temperature** object, and not allow a solution to progress.
- For a particular load step, an active **Imported Body Temperature** load will overwrite any **Thermal Condition** loads on common geometry selections.
- When a Thermal Condition is specified on the Top or Bottom shell face of a surface body, the opposite face defaults to the environment temperature unless it is otherwise specified from another load object.
- For an assembly of bodies with different topologies, you must define a separate **Imported Body Temperature** load for surface bodies.
- The values used in the solution are calculated by first converting the imported load values into the solver unit system and then multiplying the scale value.
- For each load step, if an **Imported Body Temperature** load and a **Thermal Condition** load are applied on common geometry or node selections, the **Imported Body Temperature** load takes precedence. An active **Imported Body Temperature** load will also overwrite other Imported Body Temperature loads that exist higher (previously added) in the tree, on common geometry or node selections. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.
- If a scale factor is specified, the values used in the solution are calculated by first converting the imported load values into the solver unit system and then multiplying the scale value.
- For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data system, the **Shell Thickness Factor** property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.
- For an LS-DYNA analysis, Mechanical-Based Mapping and Source Time are not available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Load Vector Controls for Substructure Generation Analyses

You can insert **Imported Body Temperature** objects into a Substructure Generation (p. 540) analysis when Beta Options are active. The Details pane will display the **Load Vector Controls** category. This category includes the following properties.

- **Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. If you select **Manual**, a **Load Vector Number** property displays, as described below, enabling you to define the value.
- **Load Vector Number (Real)**: Use this property to manually specify a **Load Vector Number (Real)** value. This value can be any number greater than 1 – the application reserves this value for pre-stressed Substructure Generation analyses. If multiple loads have the same load vector number, the application groups these loads during the solution process.

Imported Boundary Remote Constraint

The **Imported Cut Boundary Remote Constraint** object applies displacements and rotational forces from beam-based coarse models to shell or solid submodels (p. 813). These loading conditions are generated from the results defined on the nearest beam node of the coarse model. Using this method, the application applies each displacement or rotation using remote displacement objects that by default, are rigid.

The remote loads generated for this option are, by default, read-only. You can modify the properties of boundary condition as required by setting the **Read Only** property to **Off**.

See the Imported Boundary Conditions (p. 1736) section for additional information about the supported source (coarse) and target (submodel) systems as well as the specific steps to transfer data.

Imported Convection Coefficient

When CFD convection coefficients are transferred to a thermal analysis, an **Imported Convection Coefficient** object can be inserted to represent the transfer.

To define a convection, you need a film coefficient value and a bulk temperature value. For the **Imported Convection Coefficient** object, film coefficient values need to be imported from an external source, such as External Data. Bulk temperature values can be imported from an external source or defined internally.

Note:

A warning message will appear if negative mapped HTC values are present. Insert a validation object and use the **Source Value** option to determine source nodes with values less than zero.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Application Options

If your model includes two bodies that share faces, one solid and one shell, the **Shared Reference Body** property displays automatically. The property provides a drop-down list of the names of the bodies that share the scoped features. See the Convection (p. 1602) section for the steps to use this property.

Imported convection loads from **External Data** can be mapped and applied to either the centroids (**Elements**) or **Corner Nodes** of the selected element faces (3D) or element edges (2D) using the **Applied To** property.

Specify Reference Temperature from Line Bodies for Fluid Flow

In addition, if your model includes one or more Line Bodies (p. 872), with the **Model Type** property set to **Thermal Fluid**, you can obtain reference temperature values from these specified Thermal Fluid bodies.

To specify reference temperature using the Thermal Fluid bodies, you set the **Fluid Flow** property (of the **Fluid Flow Controls** category), of the **Imported Convection Coefficient** object, to **Yes**. When you activate the **Fluid Flow** property, the application automatically displays a **Scoping Method** property and a **Fluid Flow Scoping** property in the Details view category.

Using these scoping properties, you specify a desired edge or edges, a vertex, or a node using either direct selection or using a Named Selection. Note the following processing methods based on the defined scoping:

- Edge(s) on a fluid line body: Convection is accounted for with surface elements SURF151 and SURF152. The application maps the elements to the fluid elements using a distance-based algorithm. This option uses the two nodes from the mapped fluid element to specify the reference temperature. Note that you can use the **Display Connection Lines** property to display the connection lines between the centroid of each element face/edge of the convection surface(s)/edge(s) to the corresponding closest node on the fluid flow scoping. The closest node is computed using a distance-based algorithm.

  Note:

  The display of the connection lines may not be an accurate representation of the connection in the solver. The solver, by default and when applicable, uses the closes two nodes from the scoped fluid elements to specify the reference temperature.

- Vertex/Node on a fluid line body. This option uses the specified vertex or node to specify the reference temperature. You can specify a single node or vertex only.

Imported Cut Boundary Constraint

The **Imported Cut Boundary Constraint** (p. 1745) object applies displacements for Solid- or Shell-Solid submodeling and displacements and rotations for Shell-Solid submodeling.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

See the Imported Boundary Conditions (p. 1736) section for additional information about the supported source (coarse) and target (submodel) systems as well as the specific steps to transfer data.

Imported Cut Boundary Remote Force

The **Imported Cut Boundary Remote Force** object applies forces (p. 1543) and moments (p. 1574) from beam-based coarse models to shell or solid submodels (p. 813). Each remote force and moment pair share a remote point that is by default, deformable.

The remote loads generated for this option are, by default, read-only. You can modify the properties of boundary condition by setting the **Read Only** property to **Off**.

Requirements

Note the following requirements:

- In order to add beam forces to the results file and make them available for your submodel, you need to set the **Nodal Forces** property (Analysis Settings>Output Controls (p. 1473)) to **On** in the coarse model.
- If your submodel includes only forces and moments and has no other constraints, you must set the **Weak Springs** (p. 1433) property (Analysis Settings>Solver Settings (p. 1428)) to **On**.

See the Imported Boundary Conditions (p. 1736) section for additional information about the supported source (coarse) and target (submodel) systems as well as the specific steps to transfer data.

Imported Displacement

When displacements are transferred to a structural analysis, an **Imported Displacement** object can be inserted to represent the transfer. See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

- If one or more nodes with imported displacements have nodal rotations specified on them, Mechanical attempts to negotiate and apply the imported displacements. The imported displacements are transformed to the nodal coordinate system and then applied on the node(s). However, there may be cases when a suitable transformation cannot be obtained (for example, [x,y,z] > [fixed, free, free] in the Global Coordinate System becomes [fixed, fixed, free] in the nodal coordinate system if the coordinate system is rotated about the z-axis). For such situations, Mechanical will report a conflict.
- If an Imported Displacement and other support constraints are applied on common geometry or node selections, you can choose to override the specified constraints by using the **Override Constraints** property in the details of the **Imported Displacement** object. By default, the specified constraints are respected and Imported Displacement is applied only to the free degrees of freedom of a node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you have multiple **Imported Displacements** scoped to common geometry or node selections, the application could ignore some constraints and apply the **Imported Displacements** to only the free degrees of freedom of a node. Use the **Override Constraints** property to override the specified constraints.
- If you have a Nonlinear Adaptive Region (p. 1707) object present in a Static Structural analysis, the **Imported Displacement** includes the property: **Apply To Initial Mesh**. You use this property to specify whether the boundary conditions are applied directly to the initial mesh or to the newly generated mesh.
- This imported loading condition supports the Vector Display (p. 74) options.

Imported Force

When forces are transferred to a structural analysis, an **Imported Force** object can be inserted to represent the transfer.

Force is a conservative quantity, which requires the total force (sum total over source points) applied on the source to match the target. To achieve this, conservative mapping algorithms are available, when importing force data.

For uniform source and target meshes, profile preserving algorithms along with an appropriate scale factor, can also be employed to conserve the total force.

In addition to mapping data across the meshes, the import process also calculates and reports the total force and the individual X/Y/Z components on both the source and target regions. When mapping data using the conservative algorithm, you can validate that the total force is conserved between source and target. For profile preserving algorithms, this information may be used to appropriately apply a scaling factor.

Note:

When you apply total force in the Cylindrical Coordinate System, the application does not calculate the Force Components in Radial (X) and Tangential (Y) directions. The application reports these components as N/A in the summary.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or External Data (p. 736) for specific steps to transfer data.

Note:

- Both conservative and profile preserving algorithms are available for importing force data. Conservative algorithms are the default and automatically ensure that the total force is conserved between source and target.
- Profile preserving algorithms may also be used to import force loads. When using profile preserving algorithms, the total force on the source and target may not match. Use the scaling factor reported in the Transfer Summary to appropriately scale the load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Profile preserving algorithms are not recommended to be used with non-uniform source or target meshes.

- This imported loading condition supports the Vector Display (p. 74) options.

Imported Heat Flux

When thermal heat is transferred to a thermal environment, an **Imported Heat Flux** object can be inserted to represent the transfer.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data system, the **Shell Thickness Factor** property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.

Imported Heat Generation

An **Imported Heat Generation** object is created in the **Imported Load** folder when you transfer Joule heating from a Coupled Field Harmonic (p. 380) or an Electric (p. 403) analysis to a thermal analysis. See the Imported Boundary Conditions (p. 1736) section for additional information about the supported source and target systems (p. 1737) as well as the specific steps to transfer data.

Note:

- You can scope an **Imported Heat Generation** object to bodies, body-based Named Selections, Elements, and element-based Named Selections only.
- For each time step, the total heat mapped to the target system is available in the **Imported Load Transfer Summary**.
- The Joule heating, from an Electric analysis, resulting from limited contact electric conductance is ignored during this data transfer.
- For each load step, if an Imported Heat Generation load and an Internal Heat Generation load are applied on common geometry selections, the Imported Heat Generation load takes precedence. An active Imported Heat Generation load will also overwrite other Imported Heat Generation loads that exist higher (previously added) in the tree, on common geometry selections. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.
- For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data system, the **Shell Thickness Factor**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.

- When you import Joule heat from a Coupled Field Harmonic analysis, the application only considers the heating contributed from the bodies that are scoped to the **Physics Region** object with the **Structural** and **Electric** properties set to **Yes**.

Imported Initial Strain

When strains are transferred to define the state of a structure at the beginning of a structural analysis, an **Imported Initial Strain** object can be inserted to represent the transfer. The following supported strain types can be chosen using **Sub Type** property in the details of the **Imported Initial Strain** object:

- Elastic Strain
- Plastic Strain
- Equivalent Plastic Strain

You can import values for all six components of the symmetric strain tensor (XX, YY, ZZ, XY, YZ and ZX). See the External Data Import (p. 736) section for more information.

This imported load provides the **Apply To** property that enables you to map and apply the load to either the centroids of the **Elements** or to the **Corner Nodes** of the elements for the selected bodies. When the **Apply To** property is set to **Corner Nodes**, the imported initial strain can also be scoped to node-based Named Selections.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

- Imported Initial Strain can only be applied at the start of the first step.

  Activation/Deactivation of loads (p. 1423) is not available for Imported Strain load.

- Imported elastic strain values are not supported for bodies which have the following material types assigned:
  - Materials with kinematic hardening properties
  - Gasket materials
  - Hyperelastic materials
- Imported plastic strain values are not supported for bodies which have the following material types assigned:
  - Porous media

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - Rate-dependent plasticity
  - Viscoplasticity

- For shell bodies, you have the option to import strain on **All**, **Top**, **Middle**, or **Bottom** shell face(s).
- For shells with layered sections, **All** is the only supported option for importing strain on shell faces.
- Initial strain can only be applied to a shell body with a default coordinate system. If a coordinate system is specified either directly through the **Coordinate System** property on the body or indirectly through the **Coordinate System** property on **Layered Section**, then the object becomes invalid and strain cannot be imported.
- This imported loading condition supports the Vector Display (p. 74) options.

Important:

Mechanical maps every individual tensor by direct interpolation of individual components. This is numerically the simplest method but is physically inconsistent especially in nonlinear solid mechanics applications. See the Recommendations and Guidelines for Mapping of Initial Stress and Strain Data (p. 1751) section for more information.

Imported Initial Stress

When stresses are transferred to define the state of a structure at the beginning of a structural analysis, an **Imported Initial Stress** object can be inserted to represent the transfer.

You can import values for all six components of the symmetric stress tensor (XX, YY, ZZ, XY, YZ and ZX). See External Data Import (p. 736) for additional information.

This imported load provides the **Apply To** property that enables you to map and apply the load to either the centroids of the **Elements** or to the **Corner Nodes** of the elements for the selected bodies. When the **Apply To** property is set to **Corner Nodes**, the imported initial stress can also be scoped to node-based Named Selections.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

- Imported Initial Stress can only be applied at the start of the first step.

  Activation/Deactivation of loads (p. 1423) is not available for Imported Initial Stress load.
- Imported Initial Stress load is not supported for bodies which have the following material types assigned:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Materials with kinematic hardening properties
  - Gasket materials

- For shell bodies, you have the option to import stress on **All**, **Top**, **Middle**, or **Bottom** shell face(s).
- For shells with layered sections specified, **All** is the only supported option for importing stress on shell faces.
- Initial stress can only be applied to a shell body with a default coordinate system. If a coordinate system is specified either directly through the **Coordinate System** property on the body or indirectly through the **Coordinate System** property on **Layered Section**, then the object becomes invalid and stress cannot be imported.
- This imported loading condition supports the Vector Display (p. 74) options.

Important:

Mechanical maps every individual tensor by direct interpolation of individual components. This is numerically the simplest method but is physically inconsistent especially in nonlinear solid mechanics applications. See the Recommendations and Guidelines for Mapping of Initial Stress and Strain Data (p. 1751) section for more information.

Recommendations and Guidelines for Mapping of Initial Stress and Strain Data

Mechanical maps initial stress and strain data by direct interpolation of individual components. Numerically, this is the simplest method but it is physically inconsistent, especially in nonlinear solid mechanics applications.

Tensor fields associated with solid mechanics applications – such as stress, strains, and plastic strains – are not independent of each other. The strains are related to the displacements through the compatibility equations and the stresses are related to strains through the constitutive equations. In addition, for plasticity, other equations like the flow rule also relate the plastic strain tensors to the stress tensors. Hence independent interpolation of these tensors will violate these equations which in turn will create a globally un-equilibrated state of stress in the mapped domain. So, using these mapped quantities in nonlinear solid mechanics applications is not recommended. However, irrespective of these limitations, if the user wants to use these mapped fields, it is strongly recommended that he uses a dummy load step in the solver with the imported initial stress/strain results and only apply new loads and/or boundary conditions if and only if the dummy load step converges and the resulting deformation is physically consistent with the problem. Generally, the analysis with the dummy load step will not converge with loads generated via incorrectly mapped stress/strain fields. Even with a chance convergence in the dummy load step, no guarantee can be given with respect to the correctness of the results.

Mechanical provides an option to view contours of **Equivalent (von-Mises)** stress/strain, as well as individual components (**XX, YY, ZZ, XY, YZ** and **ZX**) using **Component** property in Details pane of Imported Initial Stress/Strain. User can insert a Mapping Validation (p. 2412) object under the Imported Load, perform **Source Value** validation, and set the **Display In Parent**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

property to **On** to view overlapping contours of interpolated data with source data and compare the equivalent stress/strain from the interpolated data with the source data.

The equivalent stress and strain are calculated using the von Mises equation:

Figure 32: Equivalent (von-Mises) stress

$$\sigma_{eq}=\frac{1}{\sqrt{2}}\sqrt{\left(\sigma_x-\sigma_y\right)^2+\left(\sigma_y-\sigma_z\right)^2+\left(\sigma_z-\sigma_x\right)^2+6\left(\sigma_{xy}{}^2+\sigma_{yz}{}^2+\sigma_{zx}{}^2\right)}$$

Figure 33: Equivalent (von-Mises) strain (elastic/plastic/equivalent plastic)

$$\varepsilon_{eq}=\frac{1}{\sqrt{2}}\sqrt{\left(\varepsilon_x-\varepsilon_y\right)^2+\left(\varepsilon_y-\varepsilon_z\right)^2+\left(\varepsilon_z-\varepsilon_x\right)^2+\frac{3}{2}\left(\gamma_{xy}{}^2+\gamma_{yz}{}^2+\gamma_{zx}{}^2\right)}$$

Imported Pressure

When pressures are transferred to a structural or harmonic analysis, an **Imported Pressure** object can be inserted to represent the transfer. See the Imported Boundary Conditions (p. 1736) section for more procedural steps for applicable transfers. Be sure to review the Additional Mapping Options (p. 1754) outlined below.

Pressure Application Options

The **Imported Pressure** load provides the following properties that enable you to define how to map and apply the load:

- **Shared Reference Body**: This property is a scoping feature used to apply the Imported Pressure to shared faces or edges between two bodies. When you have properly scoped the geometric entities, using either Geometry Selection or an appropriate Named Selection, the property provides a drop-down list of the names of the bodies that share the scoped features. By selecting a body from this list, you are specifying that the pressure be applied to its surface or edge. Once selected, the Geometry or Named Selection property displays a parenthetical of the shared face or edge, such as "(1 Shared Face/1 Shared Edge)" to indicate the condition.

  Note:

  – When working with the shared edges of a 2D model, the **Applied By** property must be set to **Direct**.

  – Edges can be shared between a surface and line body, but the line body cannot be specified as the Shared Reference Body or the property becomes invalid.

- **Apply To**: This property that enables you to map and apply the load to either the centroids of the **Elements** (default) or to the **Corner Nodes** of the elements included in the scoping. Also see the Additional Mapping Options (p. 1754) topic below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Applied By**: You use this property to specify the imported load as either **Surface Effect** (default) or **Direct**. The **Surface Effect** option applies the pressure using the surface effect elements created on the top of the scoped geometry. The **Direct** option applies the pressure directly onto the faces (3D) or edges (2D) of the solid/shell elements.

- **Apply To Initial Mesh**: If you have a Nonlinear Adaptive Region (p. 1707) object present in a Static Structural analysis, this property is displayed. You use this property to specify whether the boundary condition is applied directly to the initial mesh or to the newly generated mesh. The **Yes** setting for this property is not supported when the **Applied By** property is set to **Surface Effect**. See the Initial-Mesh Loading and Constraint section of the *Mechanical APDL Nonlinear Adaptivity Analysis Guide* for more information.

- For Substructure Generation analyses, the Details pane displays the **Load Vector Controls** category. This category includes the following properties.

  – **Load Vector Assignment**: Options include **Program Controlled** (default) and **Manual**. If you select **Manual**, a **Load Vector Number** property displays, as described below, enabling you to define the value.

  – **Load Vector Number (Real)**: Use this property to manually specify a **Load Vector Number (Real)** value. This value can be any number greater than 1 – the application reserves this value for pre-stressed Substructure Generation analyses. If multiple loads have the same load vector number, the application groups these loads during the solution process.

  – **Load Vector Number (Imaginary)**: This property is only visible if the imported data includes imaginary values. This value can be any number greater than 1 – the application reserves this value for pre-stressed Substructure Generation analyses. If multiple loads have the same load vector number, the application groups these loads during the solution process.

Note:

- For large-deflection analyses, this load is applied to the original loaded area by default, not the new area. When the **Applied By** property is set to:

  Surface Effect: KEYOPT(7) = 1 is set for the surface effect elements SURF154 and SURF153.

  Direct: The **SFCONTROL** command is sent with associated kArea(1) value.

- For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data (p. 736) system, the **Shell Thickness Factor** property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.

- If you scope two **Pressure** objects to the same geometry, and specify the loads in the same direction, using the **Direct** option, the pressures do not produce a cumulative loading effect. The **Imported Pressure** object that you specified last takes priority and is applied, and as a result, the application ignores the other **Imported Pressure** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- An **Imported Pressure** applied using the **Surface Effect** option and a **Imported Pressure** applied with the **Direct** option produce a resultant effect.
- If your analysis includes some combination of a **Imported Pressure**, **Pressure**, a **Force**, and a **Hydrostatic Pressure** load, and 1) all are set to the **Direct** option and 2) share the same scoping, 3) have the same **Direction**, whichever load was written to the input file last, overwrites all previous loads.
- This imported loading condition supports the Vector Display (p. 74) options.

Important:

Note the following limitations when using the **Direct** option for **Applied By** property and when the **Defined By** property is set to **Components**:

- Only supported when the target solver type is Mechanical APDL.
- Not supported for vertices and edges of Solid bodies and Line Bodies.
- Not supported on bodies associated with General Axisymmetric and Condensed parts.
- Not supported if the model has any crack defined under the fracture folder.
- Not supported if the analysis includes a **Nonlinear Adaptive Region** object and 1) the **Loaded Area** property is set to **Initial** and 2) the **Large Deflection** property (**Analysis Settings** > Solver Controls (p. 1428)) set to **On**.
- In a multiple step analysis, if you define more than one load (Imported Pressure, Pressure, Force, or Hydrostatic Pressure) using the **Direct** option and a **Nodal Pressure**, and they share the same scoping, deactivation of a particular load step in one of these loads could delete all the other loads in that load step and following steps.

Additional Mapping Options

Imported pressure loads from External Data can be mapped and applied to either the centroids (**Elements**) or **Corner Nodes** of the selected 3D element faces or element edges (2D) using the **Applied to** property.

When the Applied To property is set to **Corner Nodes**, the **Filter** property under the **Scope**category enables you to select a subset of the scoped element faces/edges and imports the load only on the specified subset. To filter a subset of element faces/edges, follow the following steps:

1. Create a nodal Named Selection to select all the nodes in the region of interest.
2. Select the created named selection in the **Filter** property. You may also choose any pre-existing nodal Named Selection.

The filtered subset of element edges/faces is then determined by the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

1. The element faces/edges which have all their corner nodes defined in the filter will be included in the mapping

2. For the element edges/faces whose corner nodes are only partially defined the filter, that is, the faces/edges which have some corner nodes included in the filter, but not all the **Include Partial Faces/Edges** property can be used to include or exclude the element faces/edges from the scoping.

Imported Remote Loads

An **Imported Remote Loads** object houses force and moment boundary conditions created by an external analysis, such as Maxwell. The application automatically creates this object when your harmonic system is linked to an upstream Maxwell system, or you can create the object manually and import an external Maxwell file. For linked systems, you can only use these loading conditions during a Harmonic Response analysis. When importing a Maxwell file, the application supports Coupled Field Harmonic and Harmonic Acoustics analyses in addition to a Harmonic Response analysis. The **Environment** tab contains an option to insert this object and then specify an external Maxwell file.

See the Importing Forces and Moments (p. 711) section for the specific steps.

Imported Surface Force Density

When electromagnetic surface forces are transferred to a structural environment, an **Imported Surface Force Density** object can be inserted to represent the transfer.

Important:

When making common geometry selections during an MSUP Harmonic Response analysis (**Solution Method** set to **Mode Superposition**) with the **Load Application Type** property set to either **Program Controlled** (default) or **Nodal**, an active Imported Surface Force Density load will overwrite other Imported Surface Force Density loads that were inserted into the Outline prior to the active load.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Imported Temperature

When temperatures are transferred to a thermal analysis, an **Imported Temperature** object can be inserted to represent the transfer.

See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

- For each load step, if an Imported Temperature load and Temperature load are applied on common geometry or node selections, the Imported Temperature load takes precedence. An active Imported Temperature load will also overwrite other Imported Temperature loads that exist higher (previously added) in the tree, on common geometry or node selections. See Activating and Deactivating Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.
- If a scale factor is specified, the values used in the solution are calculated by first converting the imported load values into the solver unit system and then multiplying the scale value.
- For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data system, the **Shell Thickness Factor** property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.

Imported Velocity

You can import velocities from an upstream Motion, Harmonic Response or a Harmonic Acoustics analysis to a downstream Harmonic Acoustics or Coupled Field Harmonic analysis. The application automatically inserts an Imported Velocity object is into the downstream system to represent the transfer. See the Imported Boundary Conditions (p. 1736) section for applicable transfers or for specific steps to transfer data.

Note:

- You can use the **Source Body** property to specify a Named Selection as source data. This option can improve performance by reducing the number of transferred source nodes.
- For surface bodies, the thickness of each target node is ignored when data is mapped. When importing data from an External Data system, the **Shell Thickness Factor** property enables you to account for the thickness at each target node, and consequently modify the location used for each target node during the mapping process. See External Data (p. 736) for additional information.
- This imported loading condition supports the Vector Display (p. 74) options.

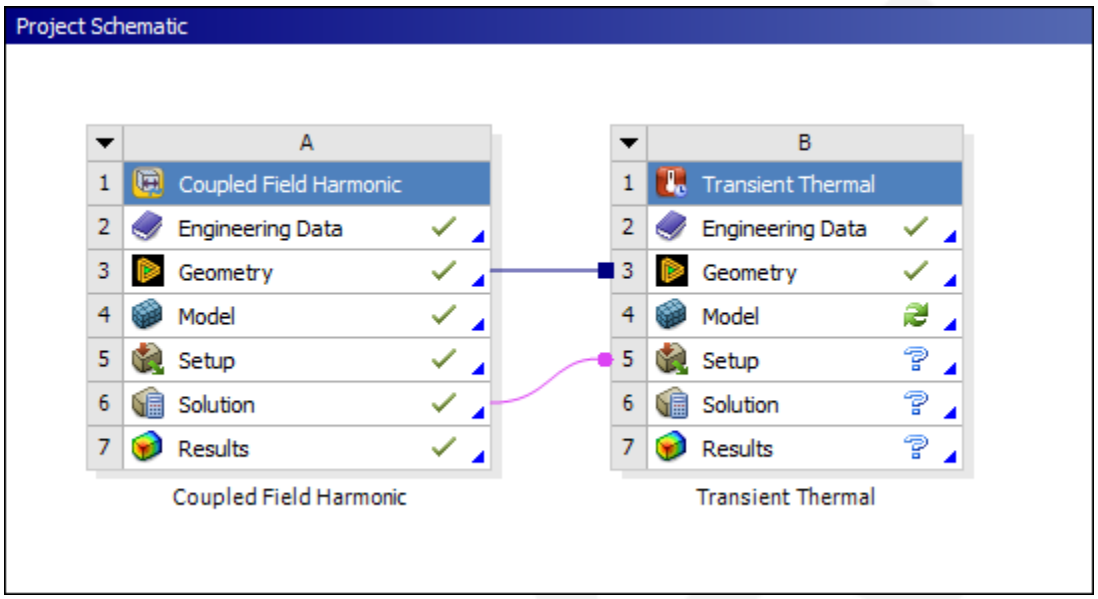
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Applying Imported Boundary Conditions

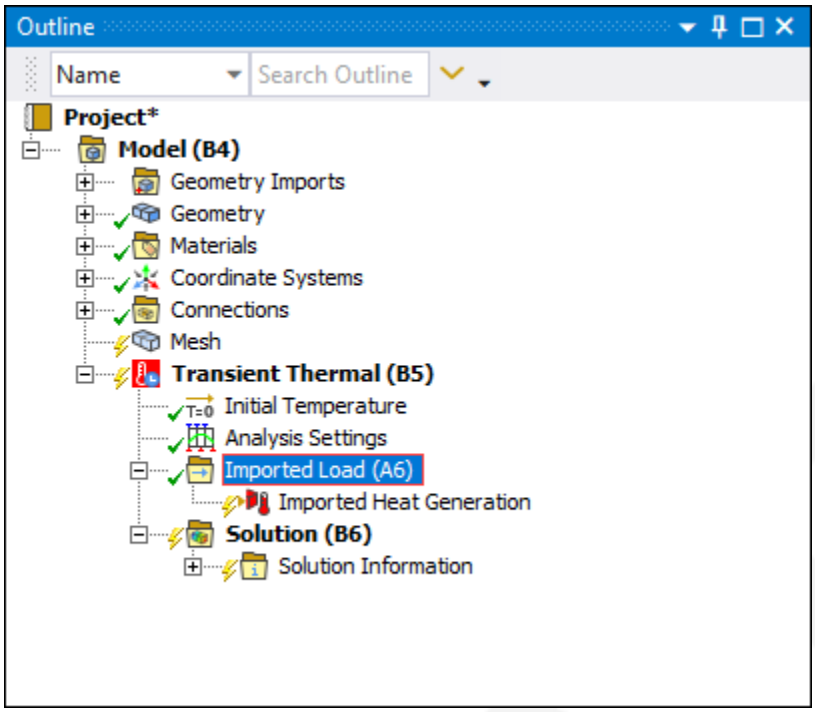
To import loads for an analysis:

1. On the Workbench **Project** page, add the desired analysis that supports data transfer. Link the **Solution** cell of the upstream system to the **Setup** cell of the downstream system, as illustrated below. As required/desired, you can also link the **Engineering Data** and **Geometry** cells between the systems.



2. Double-click **Setup** cell of the downstream system to open the analysis in Mechanical. As illustrated in the following example, an **Imported Load** folder is automatically added under the environment folder. And, for this example, an Imported Heat Generation (p. 1748) object was also automatically included in the folder.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. To add an imported load, select the **Imported Load** folder and make a selection from the **Environment** Context tab. You can also right-click the folder, select **Insert**, and add the desired load.

Note:

Imported loads can be created using the **Duplicate** option of the right-click context menu.

---

4. Based on the load definition, whether it has been mapped or if it is a newly inserted load, scope the loading condition to a geometric entity, mesh type, or to a geometry- or mesh-based **Named Selection**. Click **Apply** when complete.

Note:

Supported scoping options (geometric and/or mesh) vary based on the imported load type. For example, the following imported loads can be scoped to node-based **Named Selections**.

- **Imported Body Temperature** (from External Data, for Submodeling [**Shell-Shell** not supported], or for Thermal-Stress)
- **Imported Displacement** (from External Data or for Submodeling)
- **Imported Force** (from External Data)
- **Imported Temperature** (from External Data or for Submodeling)
- **Imported Velocity** (from External Data)

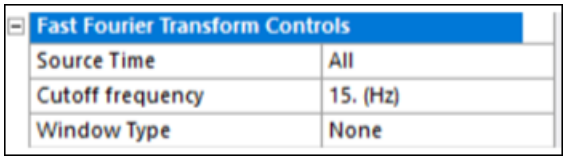
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Imported Initial Stress** and **Imported Initial Strain** (from External Data), when the **Apply To** property is set to **Corner Nodes**

5. Set the appropriate options in the **Details** pane.

6. If you are transferring from Motion to a Harmonic Acoustics or Coupled Field Harmonic analysis, **Fast Fourier Transform Controls** properties will appear. These properties handle the parameters for the conversion of time-dependent source velocities to frequency-dependent velocities.



Fast Fourier Transform Controls	
Source Time	All
Cutoff frequency	15. (Hz)
Window Type	None

- **Source Time**: Select the time domain from Motion to process. Options include **All** or **Range**.
- **Cutoff Frequency**: Select the highest frequency that can be obtained in the output of the conversion.

Note:

Based on these two properties, the application will internally perform resampling on the time domain data to ensure evenly spaced data, optimize FFT performance and output frequency data up to the **Cutoff frequency**.

First, the minimum number of points is calculated using the maximum between (*source_duration x cutoff_frequency x 2*) and the initial number of points. The number of sampling points is then calculated by taking the next 2^n value after the minimum number of points.

- **Window Type**: Use this control to apply windowing to attenuate spectral leakage. Available options are as follows:
  - **None** does not use a window for FFT.
  - **Rectangular** uses windows of $w(n)=1$
  - **Triangular** uses windows of $w(n)=1-\left|\frac{n-\frac{N-1}{2}}{\frac{L}{2}}\right|$ with $L=N-1$
  - **Bartlett** uses windows of $w(n)=1-\left|\frac{n-\frac{N-1}{2}}{\frac{L}{2}}\right|$ with $L=N$
  - **Hanning** uses windows of $w(n)=0.5\left(1-\cos\left(\frac{2\pi n}{N-1}\right)\right)$

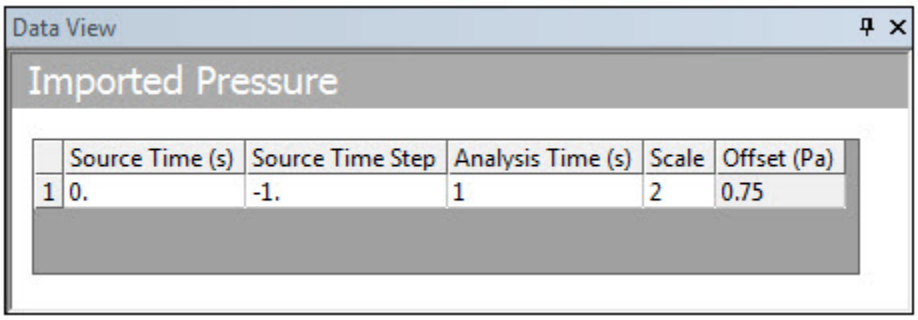
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

– **Hamming** uses windows of $w(n)=\alpha-\beta\cos\left(\frac{2\pi n}{N-1}\right)$ with $\alpha=0.53836, \beta=0.46164$

– **Welch** uses windows of $w(n)=1-\left(\frac{n-\frac{N-1}{2}}{\frac{N-1}{2}}\right)^2$

– **Blackman** uses windows of $w(n)=a_0-a_1\cos\left(\frac{2\pi n}{N-1}\right)+a_2\cos\left(\frac{4\pi n}{N-1}\right)$

7. The **Data View** can be used to control the load data that is imported. Each data transfer incorporates some or all of the column types shown below.



- **Source Time/Frequency**: Time at which the load will be imported.
- **Source Time Step**: Time Step at which the load will be imported.
- **Analysis Time/Frequency**: Time at which the load will be applied when the analysis is solved.
- **Scale**: The amount by which the imported load values are scaled before they are sent to the solver. The scale value is applied to the imported load values in the solver unit system.

  For Imported Temperature and Imported Body Temperature loads:

  – The values used in the solution are calculated by first converting the imported load values into the solver unit system and then multiplying the scale value.
- **Offset**: An offset that is added to the imported load values before they are sent to the solver. The offset value is applied to the imported load values in the solver unit system.

Specific transfer details can be found in the Special Analysis Topics (p. 685) section.

8. If you are using the Mechanical APDL solver, loads can be applied using tables or they can be applied at each analysis time/frequency specified in the imported load using the **Tabular Loading** property. When sending as tables, the loads can either be ramped or step changed (stepped) between the specified **Analysis Times/Frequencies**.

   a. When ramped, the load value at step/sub-step is calculated using linear interpolation in the range where solve step/sub-step falls.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

b. When stepped, the load value specified at t2 is applied in the range (t1, t2], where (t1, t2] is the range greater than t1 and less than or equal to t2.

Note:

- When program controlled, the loads are sent as tables when **Analysis Time(s)/Frequency(ies)** not matching any step end times/maximum frequency are present in the load definition. The loads are ramped for static/steady state and harmonic analyses and step applied for transient analyses.
- The loads are always sent as tables when **Ramped** or **Stepped** is chosen.

Important:

Note that these options do not change the **KBC** command value (*Key*) which controls whether all of the loads within a load step are linearly interpolated or step changed. In addition, certain limitations apply to loads that do not support tabular loading, such as Imported Body Force Density (p. 1739). The limitations are described on the Help page for the respective loads (p. 1736).

- Extrapolation is not performed when stepping/ramping the loads. If the solve time for a step/sub-step falls outside the specified **Analysis Time/Frequency**, then the load value at the nearest specified analysis time is used.
- For temperature loads, the values are ramped from reference temperature for the first time step. For all other loads, the values are ramped from zero.
- You can choose not to send the loads as tables using the **Off** option. The analysis times/frequencies specified in the load definition must match the step end times/maximum frequency in this case for the solution to succeed.
- In a LS-DYNA analysis, the **Off** option is equivalent to the ramped option.

9. Import the load by right-clicking the **Imported Load** object and selecting **Import Load**.
10. When the load has been imported successfully, a contour or vector plot will be displayed in the **Geometry** window.
    - For vector loads types, contours plots of the magnitude (**Total**) or X/Y/Z component can be viewed by changing the **Component** property in the Details pane. Defaults to a vector plot (**All**).
    - For tensor loads types, contours plots of **Equivalent (von-Mises)** or XX/YY/ZZ/XY/YZ/ZX component can be viewed by changing the **Component** property in the Details pane. Defaults to a Vector Principal plot (**All**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For **Imported Convection** loads, contours plots of film coefficient or ambient temperature can be viewed by changing the **Component** property in the Details pane.
- For complex load types, such as pressure and velocity in Harmonic Response analyses, the real/imaginary component of the data can be viewed by changing the **Complex Data Component** option in the details pane.
- The **Legend controls** options enable you to control the range of data displayed in the graphics window. By default, it is set to **Program control**, which allows for complete data to be displayed. If you are interested in a particular range of data, you can select the **Manual** option, and then set the minimum/maximum for the range.
- Many imported loading conditions support Vector Display (p. 74) options.

Note:

- When you scope imported loading conditions to elements, you may see graphic artifacts on your model in the form of color "bleeding". Selecting the **Wireframe** option (p. 80) corrects the display.
- The isoline option is drawn based on nodal values. When drawing isolines for imported loads that store element values (**Imported Body Force Density**, **Imported Convection**, **Imported Heat Generation**, **Imported Heat Flux**, **Imported Pressure**, **Imported Surface Force Density**, **Imported Initial Stress** and **Imported Initial Strain**), the program automatically calculates nodal values by averaging values of the elements to which a node is attached.
- The minimum and maximum values of source data are also available in **Legend Controls** for External Data Import (p. 736), Thermal-Stress (p. 836), Submodeling (p. 807), and Acoustic Coupling analyses.

---

11. To preview the imported load contour that applies to a given row in the **Data View**, use the **Active Row** option in the Details view.
12. To activate or deactivate the load at a step, highlight the specific step in the **Graph** or **Tabular Data** window, and choose **Activate**/**Deactivate** *at this step!* See Activation/Deactivation of Loads (p. 1423) for additional rules when multiple load objects of the same type exist on common geometry selections.

To export data, right-click the child load object and select **Export** > **Export Text File**.

Load Mapping Workflow Specification

When importing load data, you can select how to process the data, either by writing load data into the input file, or using the data processing framework (DPF) to generate a binary file from which

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

the application processes data. Using a DPF binary file improves processing times and requirements while reducing the size of the required files.

Important:

Currently, this capability only supports Imported Velocity (p. 1756) loads imported from an upstream:

- **Harmonic Response** to downstream **Harmonic Acoustics**.
- **Harmonic Acoustics** to downstream **Harmonic Acoustics**.
- **Coupled Field Harmonic** to downstream **Coupled Filed Harmonic**.

Go to a section topic:

- Requirements and Limitations (p. 1763)
- Application (p. 1763)
- Background (p. 1768)

Requirements and Limitations

This load mapping specification supports:

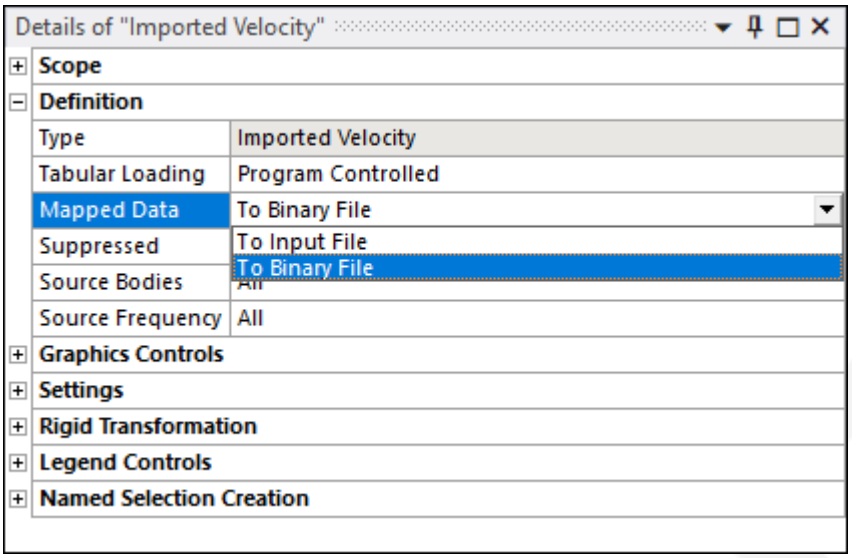
- Harmonic analyses specified with **Multiple RPM** and **On Demand Expansion**.
- The use of the Ansys Remote Solve Manager (RSM) or Ansys Distributed Compute Services (DCS).
- The **Mapped Data** property **only** when the parent **Imported Load (Group)** object is created by an upstream analysis system with shared a **Model** cell.

Application

To import loads for analysis:

1. Import an Imported Velocity (p. 1756) load from an existing upstream system in Workbench.
2. Select the imported velocity load object (contained under the Imported Load object) and specify the **Mapped Data** property. Options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



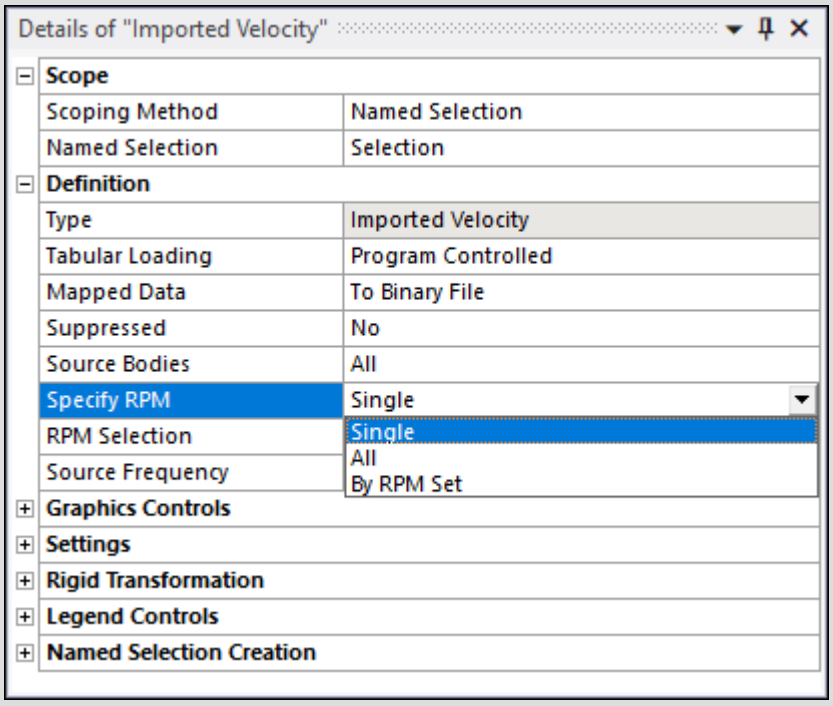
- **To Input File**: The application maps data during the import process and writes the mapped data into load tables in the input file. During the solution, the application reads the data tables and creates load tables that are stored in memory.
- **To Binary File**: The application maps data when it generates the input file and transfers mapped data using a binary file. This option creates a smaller input file and replaces load tables with DPF load commands.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Includes Multiple RPMs?

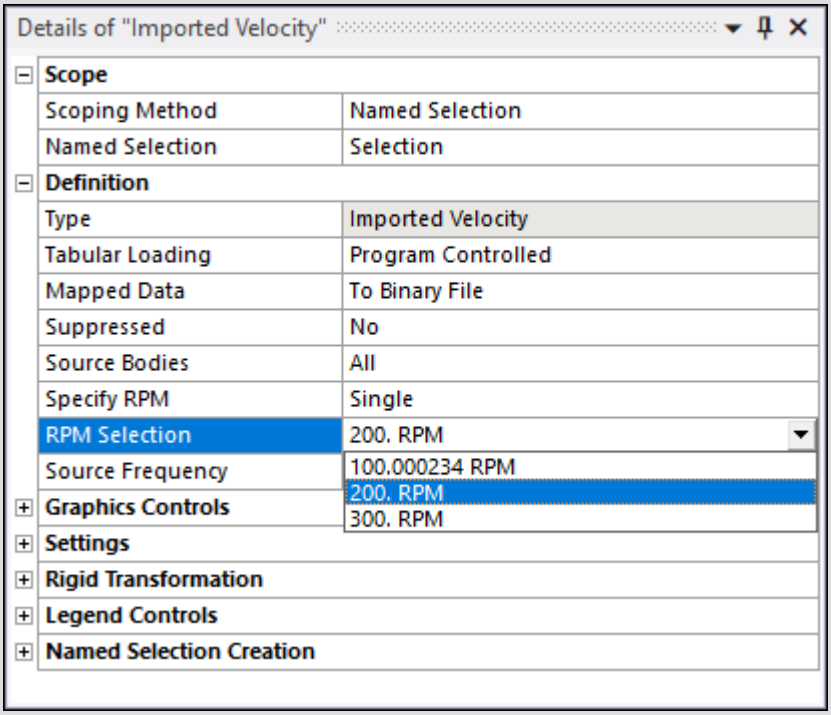
If your analysis includes multiple RPMs, you can use the **Specify RPM** property to define the load. Note that this property is only available when the **Mapped Data** property is set to **To Binary File**.



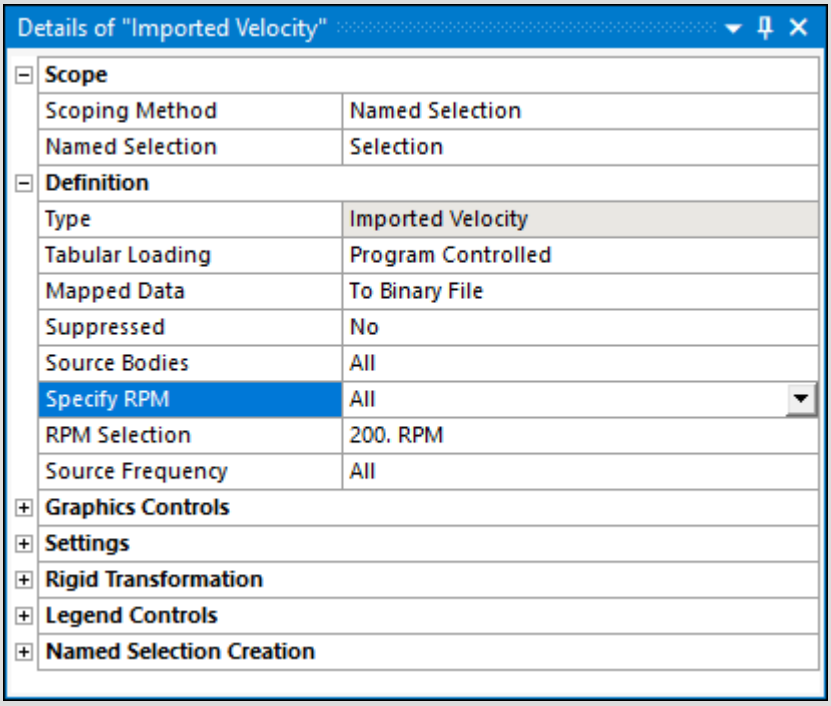
Specify RPM property options include:

– **Single** (default): This option applies the **Imported Velocity** on single RPM load step. For this setting, you also need to specify a value in the **RPM Selection** property. This property provides a drop-down list of available RPM values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



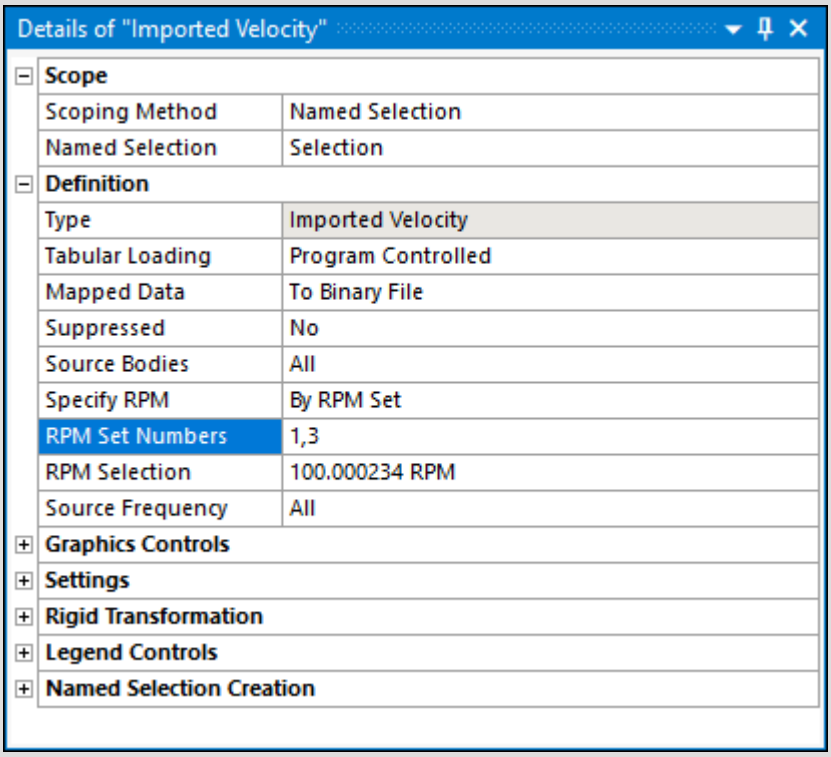
– **All**: Applies the **Imported Velocity** on all RPM load steps.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **By RPM Set**: Selecting this option displays the **RPM Set Numbers** property displays. Use the **RPM Set Numbers** property to specify the desired RPM set or sets in the supported format (1, 2, 5-10). These values correspond to the number of **Number of Steps** property defined in the Step Controls (p. 1426) category of the **Analysis Settings** object.

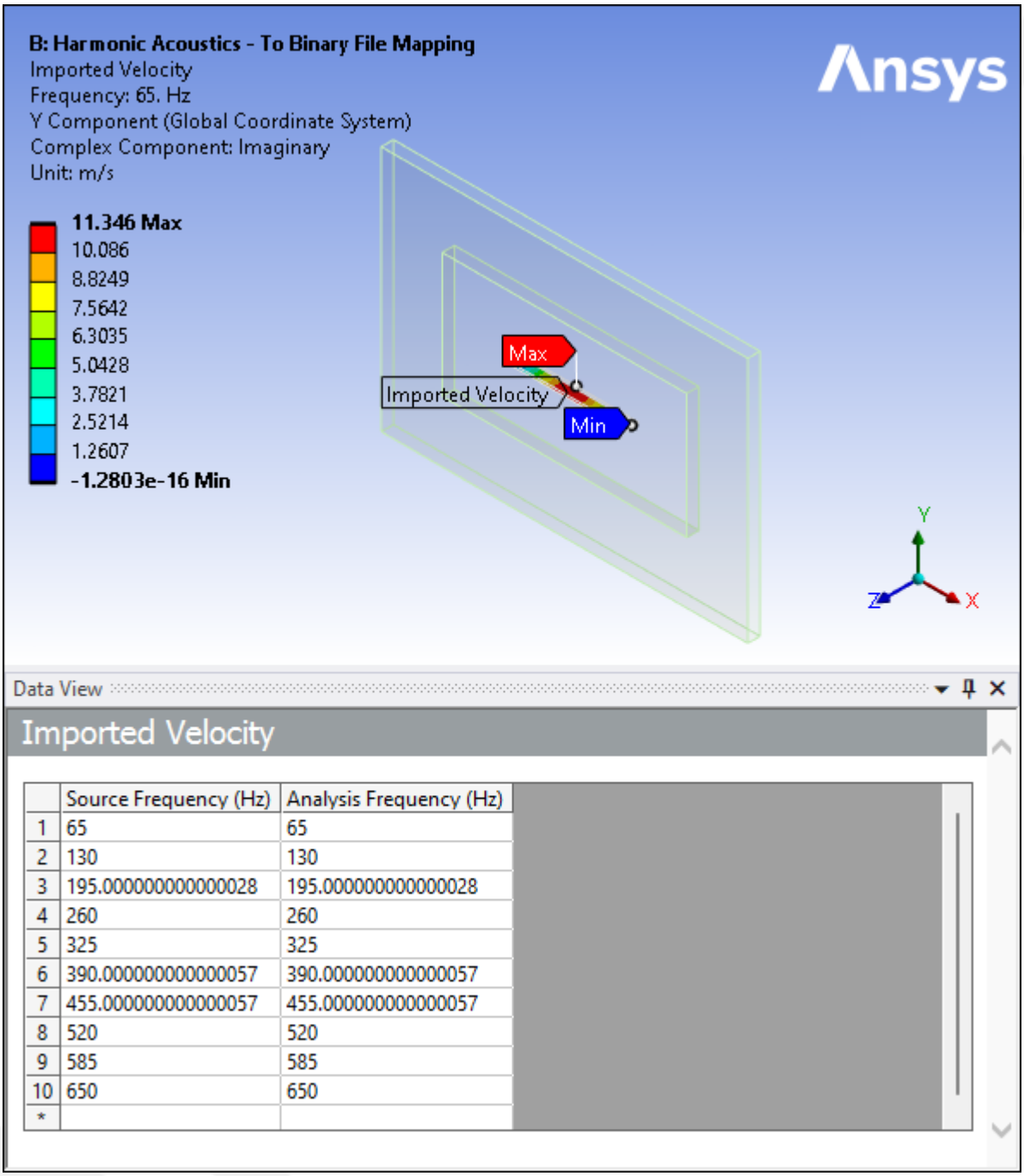


Note:

Based on your analysis requirements, use the **All** or **By RPM Set** options for the greatest efficiency.

3. Specify additional properties of the **Imported Velocity** object as needed. See the Imported Load (Group) object reference page for descriptions of all properties.
4. Right-click the velocity load object, or its parent object, and select **Import Load**.
5. Load data is generated and displayed in corresponding Details pane properties and the **Data View** worksheet. In addition, and as illustrated here, mapped contours are displayed on the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



	Source Frequency (Hz)	Analysis Frequency (Hz)
1	65	65
2	130	130
3	195.000000000000028	195.000000000000028
4	260	260
5	325	325
6	390.000000000000057	390.000000000000057
7	455.000000000000057	455.000000000000057
8	520	520
9	585	585
10	650	650
*		

Note:

When you use either the **All** or **By RPM Set** options, the content in the **Data View** table (**Source Frequency**/**Analysis Frequency**) is read-only. You can change the value of the **RPM Selection** property to view different **Source Frequency** and **Analysis Frequency** data.

Background

The following table describes the current mapping process and its drawbacks.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Currently, the application…	This action…
Maps data during import process.	
Stores the data in Mechanical (object) memory.	• Increases the size of the database. • Decreases performance during save and resume.
Writes the load tables to the input file.	• Increases the size of the input file. • Decreases performance when writing the input file.
Reads the load tables from the input file and creates the load tables during solution.	• Increases memory requirements for the load tables data. • Increases the processing requirements to create the load table.

The load mapping specification addresses the drawbacks of writing the load table data into the input file and then reading and creating the load tables during the solution. Specifically, you can control:

- How and when the application performs mapping.
- How the application transfers the mapped data to the solver.

Spatial Varying Loads and Displacements

A spatially varying load or displacement has a variable magnitude in a single coordinate direction (x, y, or z). The following load and displacement types qualify as varying loads and varying displacements, and can also be a function of time.

- **Pressure** (p. 1521) (In a Normal direction only during a Static Structural analysis)
- **Line Pressure** (p. 1585) (In a Tangential direction only during Static Structural analysis)
- **Pipe Pressure** (p. 1532) (Static Structural analysis only)
- **Pipe Temperature** (p. 1532) (Static Structural analysis only)
- **Convection** (p. 1602) (Thermal analysis only)
- **Heat Flux** (p. 1617) (Thermal analysis only)
- **Temperature** (p. 1598) (Thermal analysis only)
- **Thermal Condition** (p. 1594) (Static Structural analysis only)
- **Displacement** (p. 1666) (For **Faces**, **Edges**, or **Vertices** during a Static Structural analysis.)
- **Nodal Displacement** (p. 1728)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Nodal Force** (p. 1721)
- **Nodal Pressure** (p. 1724)

Note:

If you select multiple **Convection** load objects that include variable data, the application displays only one solid color for the scoped entities.

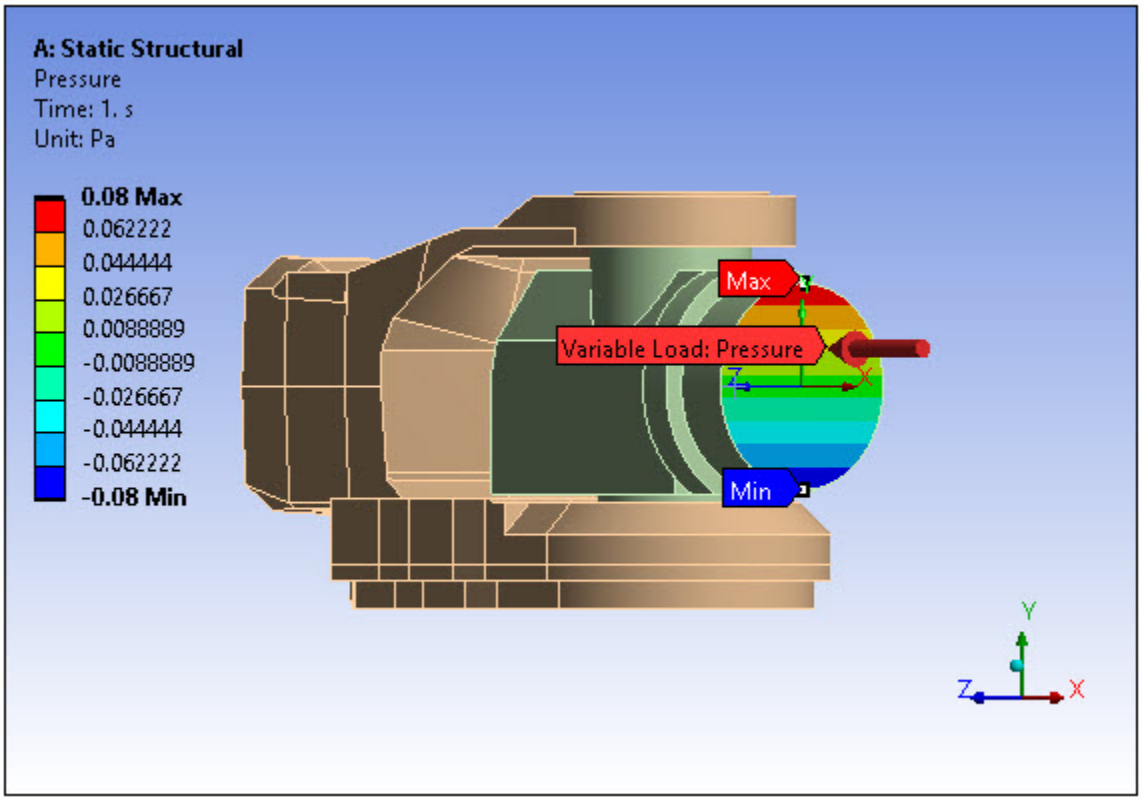
For spatial varying loads and displacements, the spatial **Independent Variable** property uses the origin of the coordinate system for its calculations and therefore it does not affect the direction of the load or displacement. To apply a spatial varying load or displacement, you must first generate the mesh, and then specify the **Magnitude** property of the variable load, or the load input, as **Tabular** (p. 1776) or as a **Function** (p. 1777) and then specify the desired coordinate (X, Y, or Z) for the **Independent Variable** property.

Once specified, and given that you have **generated the mesh** on the model, variable loading conditions display as contours. For convection loads and loads specified using the General Axisymmetric (p. 1167) feature, varying loads are displayed in purple and there is an annotation in the **Geometry** window that indicates the load as a "Variable Load."

And, the **Display** group (shown below) becomes visible on the Environment Context Tab (p. 61). From this group, the **Variable Data** drop-down menu provides the display options: **Smooth Contours**, **Contour Bands**, and **Isolines**. When you select the **Isolines** display option, the **Isoline Thickness** drop-down menu enables you to change the thickness of the displayed lines. Options include **Single** (default), **Double**, or **Triple**. The toolbar also contains options to display the **Maximum** and **Minimum** values of the spatial varying load or displacement. You can toggle these min/max options on (default) and off.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note:

You can manually customize (p. 2286) the legend for a variable load.

Important:

If you have a model with a large number of parts/bodies, there is a performance-based display preference available for displaying variable load contours. In the Options (p. 217) dialog, under the Graphics (p. 228) category, the **Varying Loads (Optimization Options)** control provides the options **Accuracy** (default) and **Performance**. The **Performance** option displays discrete points on the model, based on legend colors, of the load variation. This option provides significantly faster redrawing times that is beneficial for large models. The **Accuracy** setting displays variable load contours normally.

Defining Boundary Condition Magnitude

This section describes the methods you can use to define the magnitude of a boundary condition. Mechanical enables you to define a load value or magnitude as a:

- **Constant** (p. 1772): defined by a static value or through an Expression (p. 1772)
- **Tabular Load** (p. 1774): defined by varying time/frequency/space as well as varying per load step (for Static Structural only).
- **Function Load** (p. 1777): defined by varying time or space.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In addition, you can import (p. 1780) and export (p. 1781) load histories.

Note:

Changing the method of how a multiple-step load value is specified (such as **Tabular** to **Constant**), the Activation/Deactivation state of all steps resets to the default, Active.

Support Limitations

- Tabular Heat Flow loads applied to an edge in a 3D analysis are not supported.
- Function Heat Flow loads applied to an edge in a 3D analysis are not supported.

Constant Magnitude Values

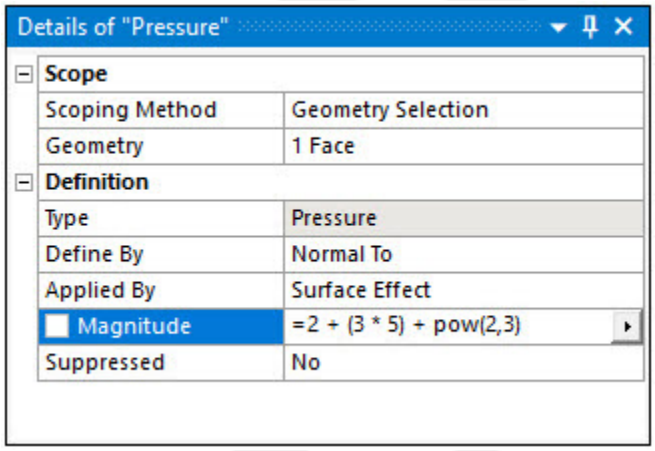
Once you have scoped the geometry for your boundary condition, generally, the **Magnitude** option defaults to the **Constant** setting and you can simply enter your desired magnitude value in the field.

As discussed below, you can also define constant values as expressions.

Constant Magnitude Expressions

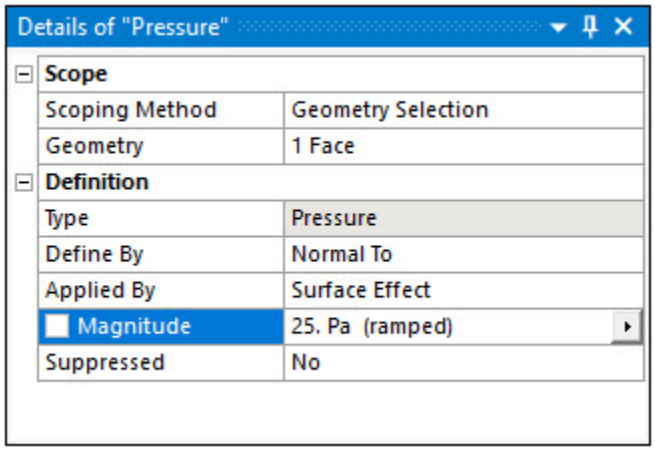
The **Magnitude** field defaults to the option **Constant**. Expressions are simply typed into the field. The expression is evaluated and applied.

For example and as illustrated, entering the expression =2 + (3 * 5) + pow(2,3) in English in the numeric field is evaluated as a Magnitude of 25.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The equal sign (=) **must** be used to begin an expression. Additional operators include: +, -, *, /, ^ (for power) and % (integer Modulus). Operator examples are shown here:

2+3	2^10
10.5-2.5	10%3
3.5*3.3	2*(3+5)
10.12/1.89	

The order of operator precedence is:

1. Parentheses
2. Intrinsic functions (like sin or cos)
3. Power (^)
4. Multiplication (*), division (/) and integer modulus (%)
5. Addition (+) and subtraction (-)

Note:

If the decimal separator (p. 125) in the current language is a comma (,) as it is in German, then the separator for the list of parameters of a function is a semicolon (;). For example, if an English expression is =2.5 + pow (1.3, 6), the equivalent German expression is =2,5 + pow (1.3; 6).

The supported intrinsic functions include the following:

Supported Intrinsic Function	Sample Usage	Usage (Angles use current Units setting)
sin(x)	sin(3.1415926535/2)	Calculate sines and hyperbolic sines.
sinh(x)	sinh(3.1415926535/2)	
cos(x)	cos(3.1415926535/2)	Calculate the cosine (cos) or hyperbolic cosine (cosh).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Supported Intrinsic Function	Sample Usage	Usage (Angles use current Units setting)
cosh(x)	cosh(3.1415926535/2)	
tan(x)	tan(3.1415926535/4)	Calculate the tangent (tan) or hyperbolic tangent (tanh).
tanh	tanh(1.000000)	
asin(x)	asin(0.326960)	Calculates the arcsine. (x - Value whose arcsine is to be calculated).
acos(x)	acos(0.326960)	Calculates the arccosine. (x - Value between –1 and 1 whose arccosine is to be calculated).
atan(x)	atan(-862.42)	Calculates the arctangent of x (atan) or the arctangent of y/x (atan2). (x,y Any numbers).
atan2(y,x)	atan2(-862.420000,78.514900)	
pow(x,y)	pow(2.0,3.0)	Calculates x raised to the power of y. (x – Base y - Exponent).
sqrt(x)	sqrt(45.35)	Calculates the square root. (x should be a Nonnegative value).
exp(x)	exp(2.302585093)	Calculates the exponential. (x - Floating-point value).
log(x)	log(9000.00)	Calculates the natural logarithm. (x - Value whose logarithm is to be found).
log10(x)	log10(9000.00)	Calculates the common logarithm. (x - Value whose logarithm is to be found).
rand()	rand()	Generates a pseudorandom number.
ceil(x)	ceil(2.8)	Calculates the ceiling of a value. It returns a floating-point value representing the smallest integer that is greater than or equal to x. (x - Floating-point value).
	ceil(-2.8)	
floor(x)	floor(2.8)	Calculates the floor of a value. It returns a floating-point value representing the largest integer that is less than or equal to x. (x - Floating-point value).
	floor(-2.8)	
fmod(x,y)	fmod(-10.0, 3.0)	Calculates the floating-point remainder. The fmod function calculates the floating-point remainder f of x / y such that x = i * y + f, where i is an integer, f has the same sign as x, and the absolute value of f is less than the absolute value of y. (x,y - Floating-point values).

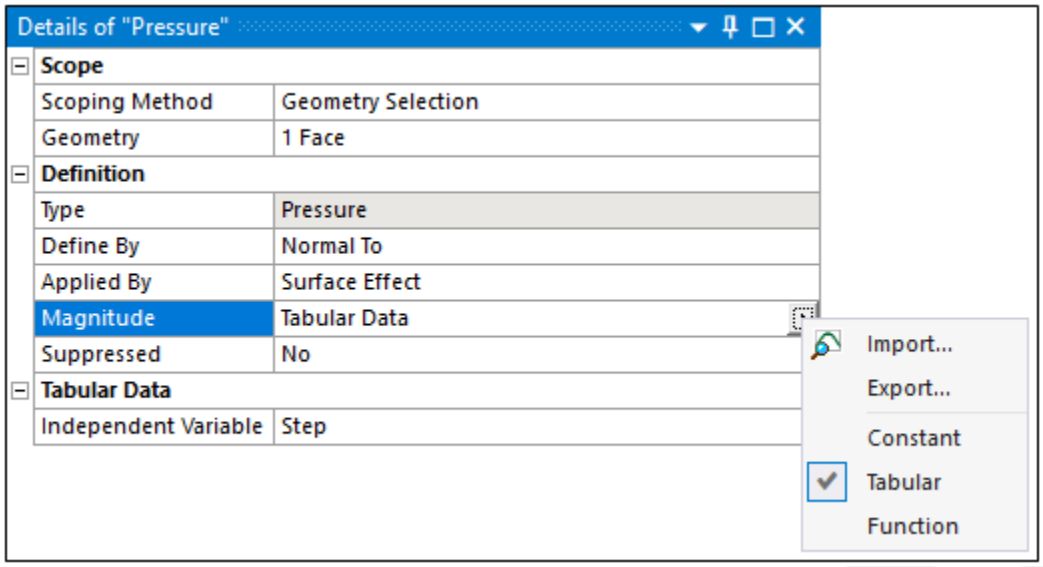
You can also enter hexadecimal (starting with 0x) and octal (starting with &) numbers, for example 0x12 and &12.

Tabular Loads

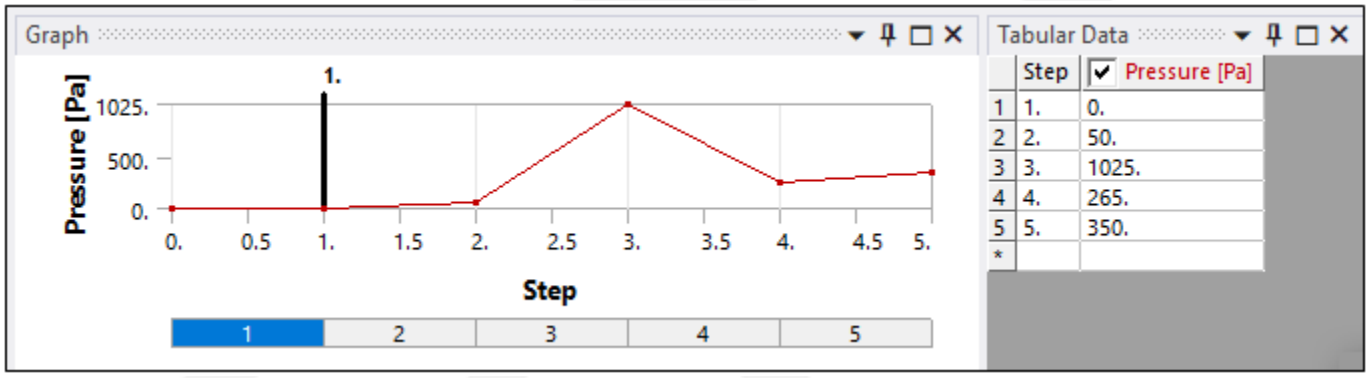
Tabular data is most often entered using the **Magnitude** property of the boundary condition. The **Distance** property is also an option for **Displacements**. As illustrated below, from the appropriate property in the Details view, open the fly-out menu and select **Tabular**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Based on your analysis system, enter the appropriate data in the **Tabular Data** window (pressure, force, frequency, etc). The **Graph** window displays the variation of the load with time for Static and Transient analysis systems or frequency for Harmonic analysis systems.



For varying loads, annotations in the **Geometry** window display the current time in the **Graph** window along with the load value at that time. Tabular Loads allow up to 100,000 entries. For frequency varying loads, annotations in the **Geometry** window displays the minimum range of harmonic frequency sweep and load value of first frequency entry.

Note:

- All new data is entered into the row that begins with an asterisk (*) regardless of whether the time or frequency point is higher or lower than the last defined point in the table. The application automatically sorts the content of the table into ascending order.
- Any **Tabular Data** values preceded by an equal sign (=) are not defined table values. These values are application interpolated values shown for reference.
- To specify more complex loads with multiple independent and dependent variables or to import this type of load from an external data source, use tables (p. 1103). See Specifying Boundary Condition Loads With Tables (p. 1781) for the types of analyses where you can use tables to define boundary condition loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Spatial Load Tabular Data

When using spatial varying loads (p. 1769), selecting **Tabular** as the input option displays the **Tabular Data** category and based on the entry of the **Tabular Data** category, the **Graph Controls** category may also display.

The **Tabular Data** category provides the following options:

Independent Variable

The **Independent Variable** property specifies how the load varies with either **Time** (default), load **Step** (Static Structural only), or in the **X**, **Y**, or **Z** spatial direction. For a Harmonic Response analysis the default setting is **Frequency**, unless you set the **Non-Cyclic Loading Type** property to either **Harmonic Index** or **Sector Number**, in which case, for the supported loading types, the default setting is **Harmonic Index** or **Sector Number**. And, for certain temperature-based loads, you can select **Temperature** as the **Independent Variable**.

Note:

- The application typically writes loading values to the input file as a table of values. When you set the **Independent Variable** property to **Step**, the application instead writes a constant load value for each load step.
- For a **Pressure** (p. 1521) load, the **Define By** property must be set to **Normal To**.
- The option **Normalized S** becomes available for **Line Pressure** (p. 1585) loads in a 3D analysis when the **Define By** property is set to **Tangential** or **Pressure** (p. 1521) loads in a 2D analysis when the **Define By** property is set to **Normal To**.

  The **Normalized S** option enables you to define pressure as a function of the distance along a path whose length is denoted by **S**. When you select the **Normalized S** variable, the Tabular Data window accepts input data in the form of normalized values of path length (**Normalized S**) and corresponding Pressure values. A path length of 0 denotes the start of the path and a 1 denotes the end of the path. Any intermediate values between 0 and 1 are acceptable in the table. Load values are sent to the solver for each element on the defined path based on a first-order approximation.

Coordinate System

The **Coordinate System** property displays if you specify the **Independent Variable** in a spatial direction (X, Y, or Z). Use this property to specify a coordinate system.

Graph Controls

The **Graph Controls** category displays when you define the **Independent Variable** as a spatial direction (X, Y, or Z), as **Normalized S**, or as **Temperature**. This category provides the property **X-Axis** which you use to change the Graph window's display. The options of the **X-Axis** property vary based upon analysis type and the selection made for the **Independent Variable** property. Options may include **Time**, or the spatial direction specified, or **Temperature**.

- When the **X-Axis** property is defined as **Time**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **Tabular Data** content can be Scaled against time.

– You can Activate and/or Deactivate (p. 1423) the load at a solution load step.

– If **Time** is not an available option of the **X-Axis** option, then scaling or activation/deactivation are not possible for the boundary condition.

Function Loads

For entering a mathematical function, click the flyout arrow in the input field (for example, **Magnitude**), choose **Function**, then enter a function, such as **=1000*sin(10*time)**. Note that the values you are evaluating can exceed the final time value by as much as one time step. The **Graph** window displays the variation of the load with time. Annotations in the Geometry window display the current time in the **Graph** window along with the load value at that time. The following functions are supported:

sin/cos/tan	exp
asin/acos/atan/atan2	pow
log/log10,	sqrt

Note:

When you specify a **Magnitude** as a **Function**, the integer Modulus (%) operator is not supported.

Spatial Load and Displacement Function Data

When using spatial varying loads or displacements (p. 1769), selecting **Function** as the input option in the Details view presents an editable function field. Enter a mathematical expression in this field. Expressions have the following requirements:

- For a **Pressure** load, the **Define By** option must be set to **Normal To**.
- For a **Line Pressure** load, the **Define By** option must be set to **Tangential**.
- You can use the spatial variation independent variables x, y, or z, and time (entered in lowercase) in the definition of the function.
- For Line Pressure loads in a 3D analysis or Pressure loads in a 2–D analysis, you can also use the variable s, which allows you to define pressure as a function of the distance along a path whose length is denoted by s. When defining a path length, valid primary variables you can enter are s alone or s combined with time, for example, s*time, or s*sin(time/s). Load values are sent to the solver for each element on the defined path based on a first-order approximation.
- Define only one direction, x, y, or z, or path length, s. After entering a direction or path length, the **Graph Controls** category (see above) displays.

When the Details view property **Magnitude** is set to **Function**, the following categories automatically display.

- **Function** - properties include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - **Unit System**: The unit system that defined when you select the **Function** option. As a read-only property, the unit system does not change.
  - **Angular Measure**: The angular measure that is used to evaluate trigonometric functions. As a read-only property, the angular unit does not change.

Note:

The application uses the currently selected **Unit System** and **Angle** when you specify the **Function** option for the **Magnitude**. This setting does not change even if you manually change the **Unit System** and/or **Angle**.

- **Graph Controls** - based of the defined function, properties include:
  - **X-Axis**: This provides options to display time or the spatial independent variable in the graph. When set to **Time** you can activate and deactivate the load at a solution step.
  - **Alternate Value**: If the function combines time and a spatial independent variable, one of these values (alternate) must be fixed to evaluate the function for the two dimensional graph.
  - **Range Minimum**: If the **X-Axis** property is set to a spatial independent variable, this is the minimum range of the graph. For time, this value defaults to 0.0 and cannot be modified.
  - **Range Maximum**: If the **X-Axis** property is set to a spatial independent variable, this is the maximum range of the graph. For time this defaults to the analysis end time and can't be modified.
  - **Number of Segments**: The function is graphed with a default value of two hundred line segments. This value may be changed to better visualize the function. The function can be graphed with up to 100,000 segments.

Caution:

Specifying a large number of points may slow the response time of Mechanical.

Spatial Varying Displacements

You can also apply spatial varying displacements (p. 1663), which have the following additional or unique characteristics:

- Edge scoping is available.
- Displacements are shown as vectors instead of contours except if you choose **Normal To** the surface. Vectors are only displayed if the model has been meshed. The vector arrows are color-coded to indicate their value. A contour band is included for interpretation of the values. The contour band is the vector sum of the possible three vector components and therefore will only display positive values.
- For one **Displacement** object, you can select up to three displacement components that can all vary using the same direction. If an additional direction is required, you can use an additional **Displacement** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

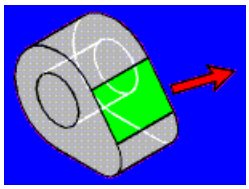
- A constant value and a table cannot be used in different components. A table will be forced in any component having a constant value if another component has a table.

Direction

There are four types of Direction:

Planar Face (p. 1779)
Edge (p. 1779)
Cylindrical Face or Geometric Axis (p. 1780)
Two Vertices (p. 1780)

Planar Face



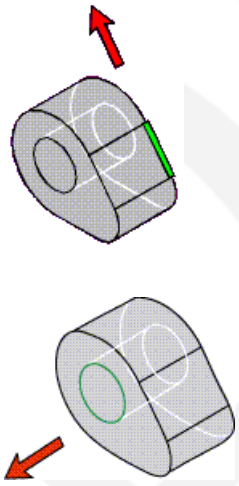
- Selected planar face. The load is directed normal to the face.

Note:

Not applicable to Rotational Velocity or Rotational Acceleration. These objects are aligned along the normal to a planar face and along the axis of a cylindrical face (p. 1780).

Edge

Straight	Colinear to the edge
Circular or Elliptical	Normal to the plane containing the edge

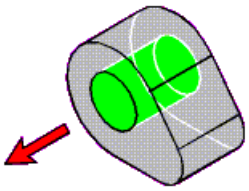


- Selected straight edge

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

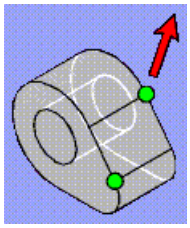
Cylindrical Face or Geometric Axis

Applies to cylinders, cones, tori, and cylindrical or conical fillets. For vector-based loading on a cylindrical face or geometric axis, you define the radial direction by selecting a different piece of geometry on your model that allows you to modify the **Direction** in the desired direction.



- Selected cylinder

Two Vertices



- 2 selected vertices

Note:

Hold the **Ctrl** key to select the second vertex.

Loads that require you to define an associated direction include the **Define By** Details view control. Setting **Define By** to **Vector** allows you to define the direction graphically, based on the selected geometry. Setting **Define By** to **Components** allows you to define the direction by specifying the x, y, and z magnitude components of the load.

Note:

If you switch the load direction setting in the **Define By** field, the data is lost.

Importing Load History

To import a load history from a library:

1. Select geometric or mesh entities and specify your desired load.
2. Go to the Details view and in the input field, such as the **Magnitude** field, click the flyout field and choose **Import**. Note that the **Import** feature can present different dependencies, such as time and temperature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Choose the desired load history if it is listed, then click **OK**. If it is not listed, click the **Add** button, choose a load history or **Browse** to one that is stored, then click **OK** in both dialog boxes.

Exporting Load History

To export a load history:

By default, any load history that you create in the application remains in the application. To save the load history for future use:

1. Create a load history using the **Graph** or **Tabular Data** windows.
2. Go to the Details view and in the input field, such as the **Magnitude** field, click the flyout field, choose **Export**, and save the file to a specific location.

Specifying Boundary Condition Loads With Tables

You can assign tables to specify pressure (p. 1521), thermal condition (p. 1594), and temperature (p. 1598) loads. Tables define loads that vary over multiple coordinate dimensions, time, and/or temperature. The numeric data in the table determine how the load varies across these dimensions.

Pressure and temperature values are specified as dependent variables in tables. These physical quantities depend on a table's independent variables: measurable quantities such as time, temperature, and/or one or more coordinates (X, Y, Z, theta). Temperature can be either a dependent or independent variable depending on what kind of analysis you are performing. Both dependent and independent variables are represented by columns of data, with their values entered as rows in the table.

You can import tables from external data files, enter and edit tables directly in Mechanical, and visualize table data via the Tables feature. Specifying Tables (p. 1103) describes how to create and work with tables.

Table assignments can also be parameterized to perform design point studies. Each design point can be assigned a different table to model different load profiles for a boundary condition.

See the following sections for instructions on how to specify boundary condition loads with tables.

Specify Pressure Loads with Tables

Specify Thermal Condition Loads with Tables

Specify Temperature Loads with Tables

Matching the Spatial Coordinate System and Table Coordinate System

Specify Pressure Loads with Tables

To specify a pressure (p. 1521) boundary condition with a table:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Create a table of pressure loads. **Pressure** or pressure components (**Pressure_i**, **Pressure_j**, and/or **Pressure_k**) must be dependent variables. See Create a Table (p. 1105).

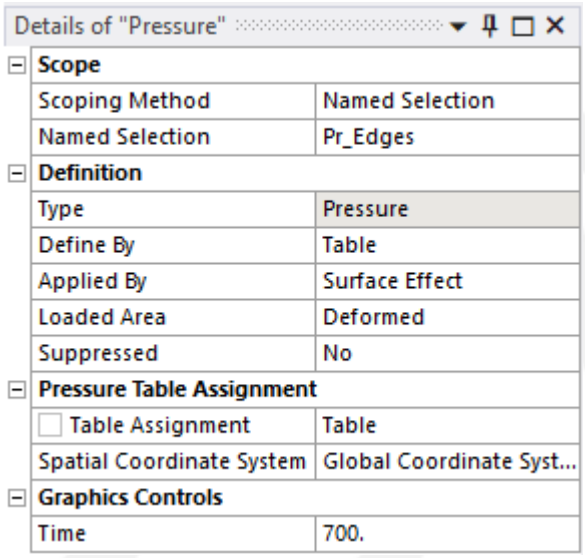
   Note:

   If you are performing a two-dimensional analysis, Mechanical ignores the independent variable **Z Coordinate**. If you specify pressure components as dependent variables, the application ignores **Pressure_k**. However, both **Pressure_i** and **Pressure_j** must be included in the table.

2. Select **Pressure** as the **Static Structural (AS)** analysis type.

3. Select the desired **Scoping Method** for the boundary condition.

4. For **Define By**, select **Table Assignment** (as shown in the screen shot below).



5. Under **Pressure Table Assignment**, specify the following:

   - **Table Assignment**: Select a table from the drop-down menu. The menu only lists tables that include pressure or pressure components as dependent variables.

     If you did not already create a table of pressure values, you can manually enter a new table by selecting **Add Table** from the drop-down menu. See Manually Enter a Table (p. 1109) for instructions. The new table is then listed on the drop-down menu and can be assigned to the boundary condition.

   - Check the box next to **Table Assignment** to parameterize the table. This allows you to perform design point studies with different tables of pressure data. See Table Parameters and Design Points in the *Workbench User's Guide* for details.

   - **Spatial Coordinate System**: Select the coordinate system used by the independent variables in the table. It must be the same type of coordinate system (cartesian or spherical) as the coordinate data. See Matching the Spatial Coordinate System and Table Coordinate System (p. 1786).

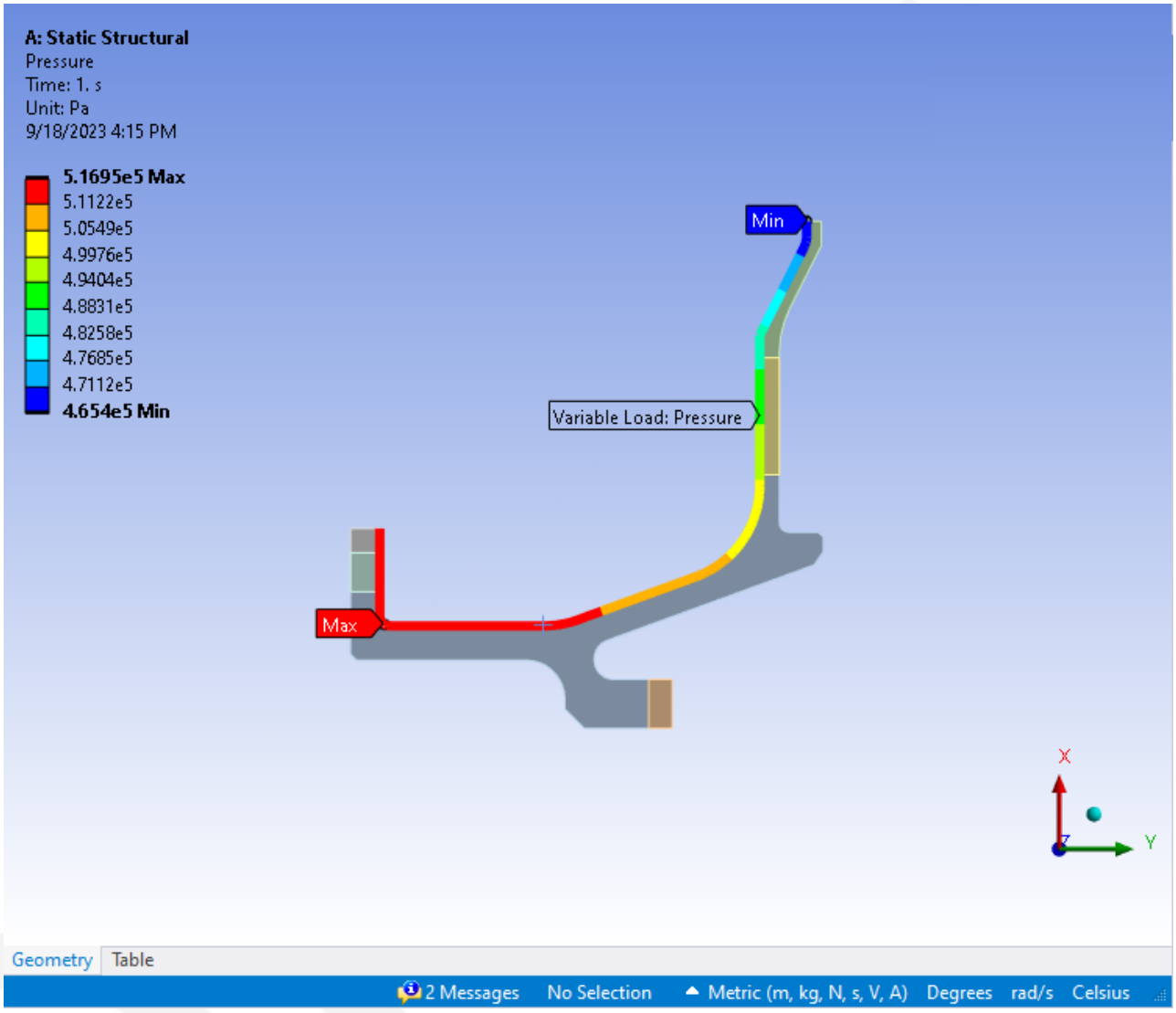
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

This field only displays if the selected table includes an x, y, z, or theta coordinate as an independent variable.

6. Under **Graphics Controls**, specify the start time for displaying the load data. This category only displays if the selected table includes time as an independent variable.

Mechanical plots the pressure values in the table on the selected geometry.



Specify Thermal Condition Loads with Tables

To specify a thermal condition (p. 1594) load with a table:

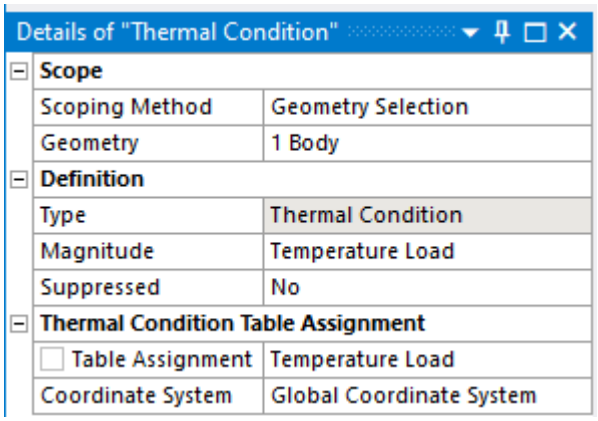
1. Create a table of temperature loads. **Temperature** must be a dependent variable. See Create a Table (p. 1105).

2. Select **Thermal Condition** as the **Static Structural (AS)** analysis type.

3. Select the desired **Scoping Method** for the boundary condition. Note that you must choose a three-dimensional body for a thermal condition analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. For **Magnitude**, select **Table Assignment** (as shown in the screen shot of the **Details** pane below).



5. Under **Thermal Condition Table Assignment**, specify the following:

   - **Table Assignment**: Select a table from the drop-down menu. The menu only lists tables that include temperature as a dependent variable.

     If you did not already create a table of temperature values, you can manually enter a new table by selecting **Add Table** from the drop-down menu. See Manually Enter a Table (p. 1109) for instructions. The new table is then listed on the drop-down menu and can be assigned to the boundary condition.

   - Check the box next to **Table Assignment** to parameterize the table. This allows you to perform design point studies with different tables of temperature data. See Working with Parameters and Design Points in the *Workbench User's Guide* for details.

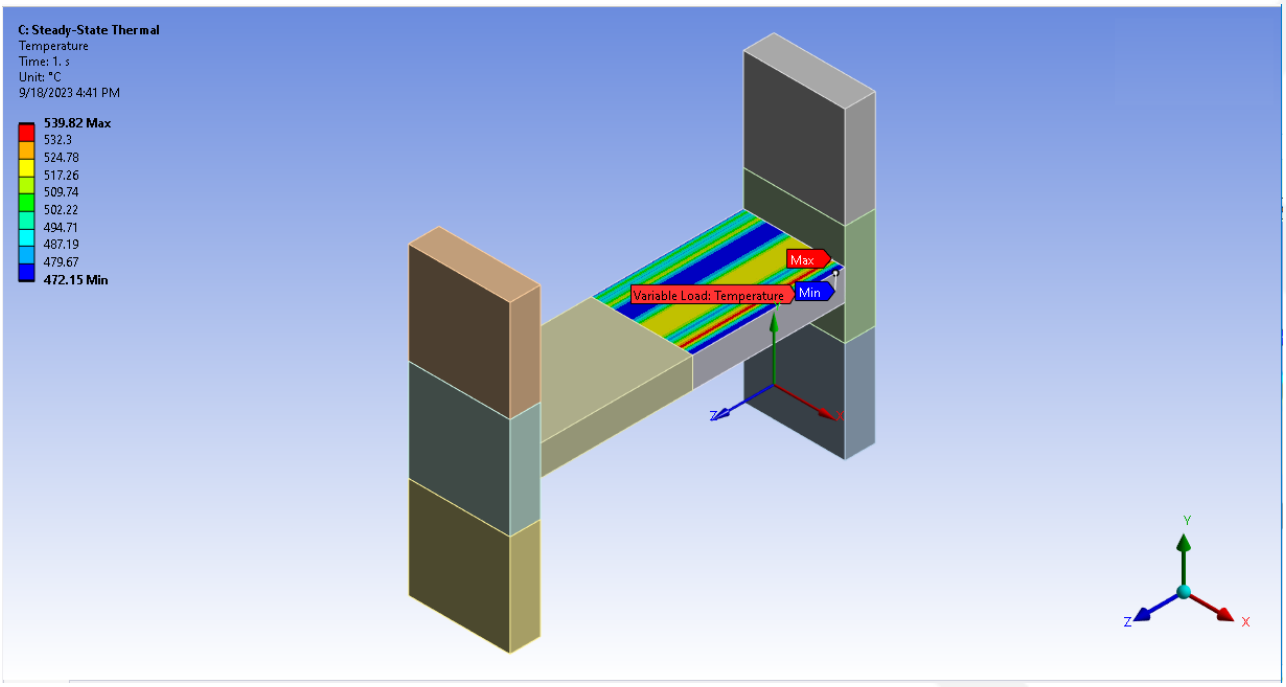
   - **Spatial Coordinate System**: Select the coordinate system used by the independent variables in the table. It must be the same type of coordinate system (cartesian or spherical) as the coordinate data. See Matching the Spatial Coordinate System and Table Coordinate System (p. 1786). This field only displays if the selected table includes an x, y, z, or theta coordinate as an independent variable.

6. Under **Graphics Controls**, specify the start time for displaying the load data. This category only displays if the selected table includes time as an independent variable.

Mechanical plots the temperature values in the table on the selected object(s).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Specify Temperature Loads with Tables

To specify a temperature (p. 1598) boundary condition with a table:

1. Create a table of temperature loads. **Temperature** must be a dependent variable. See Create a Table (p. 1105).
2. Select **Temperature** as the **Steady-State Thermal (CS)** analysis type.
3. Select the desired **Scoping Method** for the boundary condition.
4. For **Define By**, select **Table** (as shown in the screen shot of the **Details** pane below).

Details of "Temperature"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	0
Magnitude	Table Assignment
Suppressed	No
Temperature Table Assignment	
Table Assignment	T-table-2
Coordinate System	Global Coordinate System

5. Under **Temperature Table Assignment**, specify the following:
   - **Table Assignment**: Select a table from the drop-down menu. The menu only lists tables that include temperature as a dependent variable.

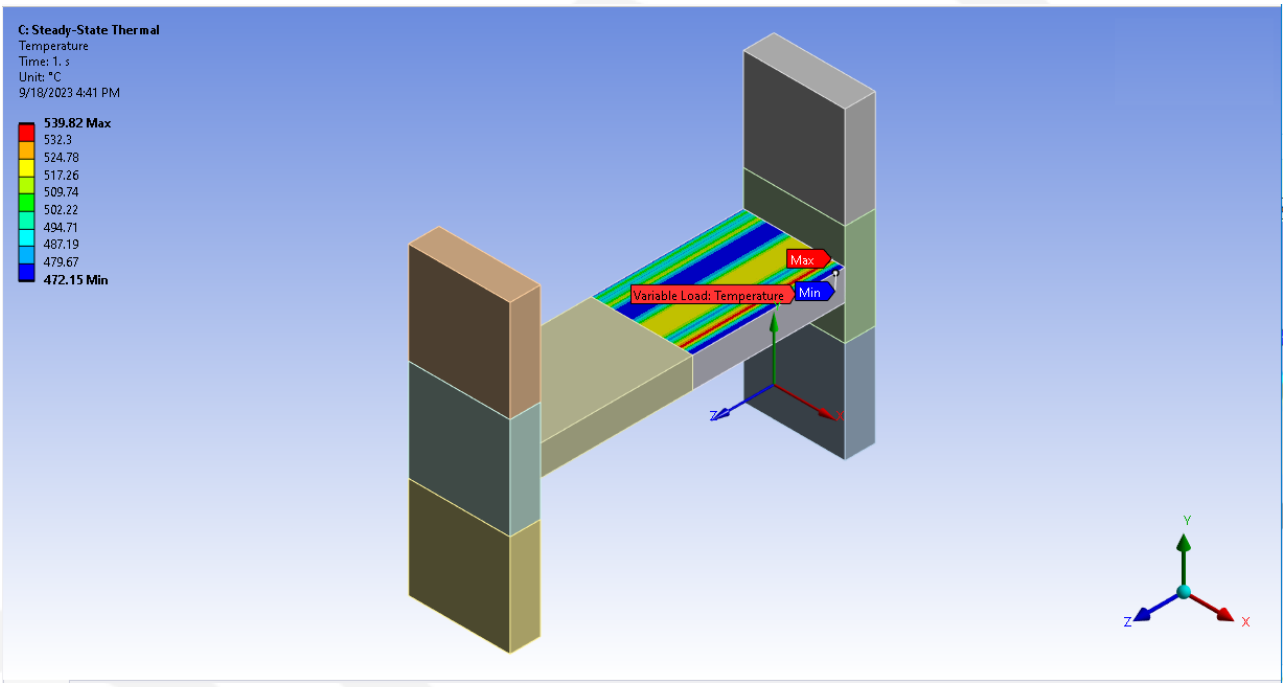
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

If you did not already create a table of temperature values, you can manually enter a new table by selecting **Add Table** from the drop-down menu. See Manually Enter a Table (p. 1109) for instructions. The new table is then listed on the drop-down menu and can be assigned to the boundary condition.

- Check the box next to **Table Assignment** to parameterize the table. This allows you to perform design point studies with different tables of temperature data. See Working with Parameters and Design Points in the *Workbench User's Guide* for details.
- **Spatial Coordinate System**: Select the coordinate system used by the independent variables in the table. It must be the same type of coordinate system (cartesian or spherical) as the coordinate data. See Matching the Spatial Coordinate System and Table Coordinate System (p. 1786). This field only displays if the selected table includes an x, y, z, or theta coordinate as an independent variable.

6. Under **Graphics Controls**, specify the start time for displaying the load data. This category only displays if the selected table includes time as an independent variable.

Mechanical plots the temperature values in the table on the selected geometry.



Matching the Spatial Coordinate System and Table Coordinate System

If your table includes an x, y, z, or theta coordinate as an independent variable, you must select a **Spatial Coordinate System**. This defines the type and location of the coordinate system that is used

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

by the table. Select either **Global Coordinate System** (the default) or a local coordinate system that you created as described in Setting Up Coordinate Systems (p. 1191).

Note:

The spatial coordinate system must match the type of coordinate system that is used by the table (cartesian or cylindrical).

- You cannot assign a cartesian spatial coordinate system to a table that includes **Theta Coordinate** as an independent variable. Theta coordinates are only used in cylindrical coordinate systems.
- You cannot assign a cylindrical spatial coordinate system to a table that includes **Y Coordinate** as an independent variable. Y coordinates are only used in cartesian coordinate systems.

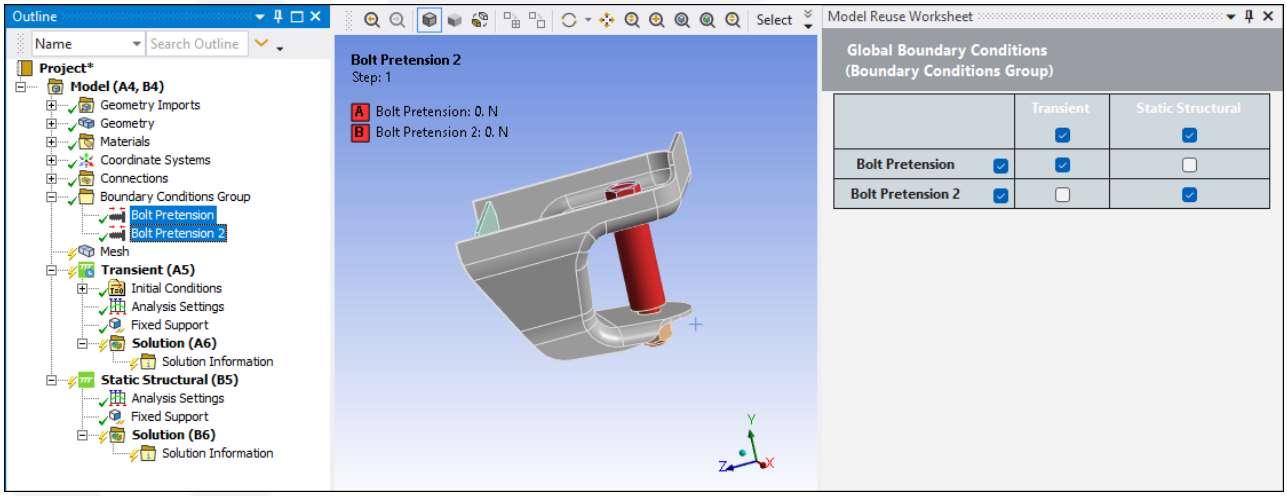
Specifying Model-Level Boundary Conditions

You can insert boundary conditions from the Model object of your simulation. This enables you to define a boundary condition once and apply it to all of the analyses included in your simulation. That is, if your simulation includes multiple Static Structural analyses, you can apply the boundary condition to each analysis separately. Currently, Bolt Pretension (p. 1566) is the only boundary condition you can add at the model-level of a simulation.

Note:

The **Pressure** load is a supported beta option for this feature.

Here is an example of multiple boundary conditions created at the model object level of a simulation. The feature works in combination with a worksheet pane that you use to select the available boundary conditions and the analyses to which you can apply them.



Go to a section topic:

- Application (p. 1788)
- Model Reuse Worksheet (p. 1788)

Application

1. You can insert the boundary condition directly at the model-level or you can you can create it under the analysis environment and promote the object.

   From the Model Object

   1. Select the **Model** object.
   2. From the **Model** Context tab, open the **Boundary Conditions Group** option and select **Bolt Pretension**. Or, right-click the **Model** object or within the **Geometry** window, and select **Insert** > **Boundary Conditions Group** > **Bolt Pretension**.
   3. Specify the properties of the boundary condition.

   From the Environment Object

   1. Select the Environment object.
   2. From the Environment Context tab: open the Loads drop-down menu and select **Bolt Pretension**. Or, right-click the Environment object or within the **Geometry** window, and select **Insert** > **Bolt Pretension**.
   3. Specify the properties of the boundary condition.
   4. Right-click the **Bolt Pretension** object and select **Move Items to the Boundary Conditions Group**. This automatically inserts the boundary condition under the **Model** object.

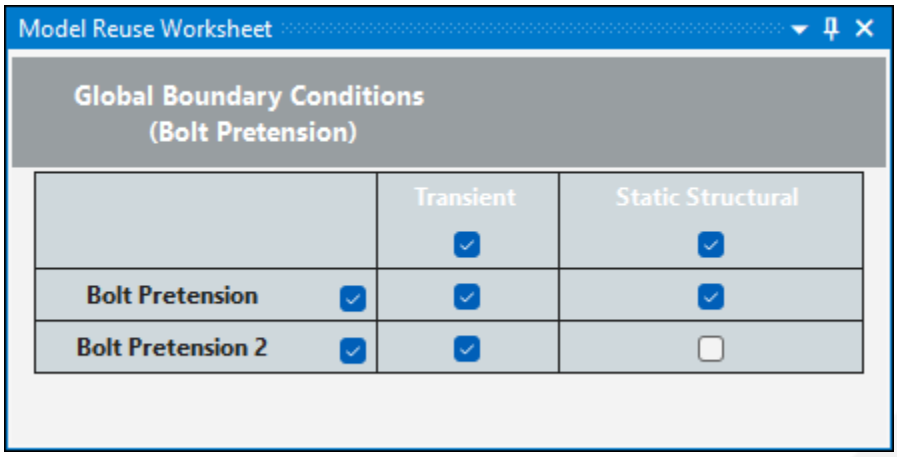
2. Highlight the **Boundary Conditions Group** object and select the **Model Reuse Worksheet** option from the **Home** tab.
3. The worksheet displays all of the environments and model-level boundary conditions available in the simulation. Select the check box beside each analysis system to apply the boundary condition.

Model Reuse Worksheet

Use the **Model Reuse Worksheet** to select the loads to include in each analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver)

This section discusses converting structural boundary conditions on the geometry to constraints on the mesh for analyses targeting the Mechanical APDL solver.

In the Mechanical APDL application, structural degree-of-freedom constraints can be defined at individual nodes. Specifically, you can choose to constrain each node along any of the three axis directions (x, y, z) of its local coordinate system to simulate the kinds of supports your model requires. In the Mechanical application, however, you specify boundary conditions on the geometry, so the program must automatically convert them into nodal constraints prior to solution. Ordinarily, this process is straightforward and the boundary conditions can be transcribed directly onto the nodes. In certain cases, however, the Mechanical application may be confronted with combinations of boundary conditions that require negotiation to produce an equivalent rendition of the effective constraints acting on the nodes. A common case occurs in structural analyses where two or more boundary conditions are applied to neighboring topologies, for example, Frictionless Supports applied to neighboring faces that meet at an angle: the nodes on the edge are subject to two separate combinations of DOF constraints, one from each Frictionless Support. The Mechanical application attempts to identify a suitable orientation to the nodal coordinate system that accommodates both frictionless supports and, if successful, constrain its axes accordingly. Should this attempt ever fail, the solution will be prevented and an error will be issued to the Message Window (See The Solver Has Found Conflicting DOF Constraints (p. 2379) in the Troubleshooting section.)

Among the boundary conditions that participate in this conversion, there are:

Fixed Supports (p. 1664) (Fixed Face, Fixed Edge, Fixed Vertex)

Simply Supported (p. 1690) (Edge or Vertex)

Fixed Rotation (p. 1692)

Displacements (Displacements for Faces (p. 1666), Displacement for Edges (p. 1666), Displacements For Vertices (p. 1666))

Frictionless Support (p. 1682)

Cylindrical Support (p. 1688)

Symmetry Regions (p. 1128)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The calculations that convert the boundary conditions into nodal constraints involve:

- the identification of the linear span contributed by each of the boundary conditions
- the combination of the individual spans into a final nodal constraint choice.

Angular tolerances are involved in distinguishing and combining the spans. A program controlled tolerance of 0.01 degrees will be used.

Note:

The calculations have a built in preference for producing nodal coordinate systems that are closest in orientation to the global coordinate system.

Resolving Thermal Boundary Condition Conflicts

Conflicts between boundary conditions scoped to parts and individual faces

Boundary conditions applied to individual geometry faces always override those that are scoped to a part(s). For conflicts associated with various boundary conditions, the order of precedence is as follows:

1. Applied temperatures (Highest).
2. Convection, heat fluxes, and flows (Cumulative, but overridden by applied temperatures).
3. Insulated (Lowest. Overridden by all of the above).

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Understanding Solving

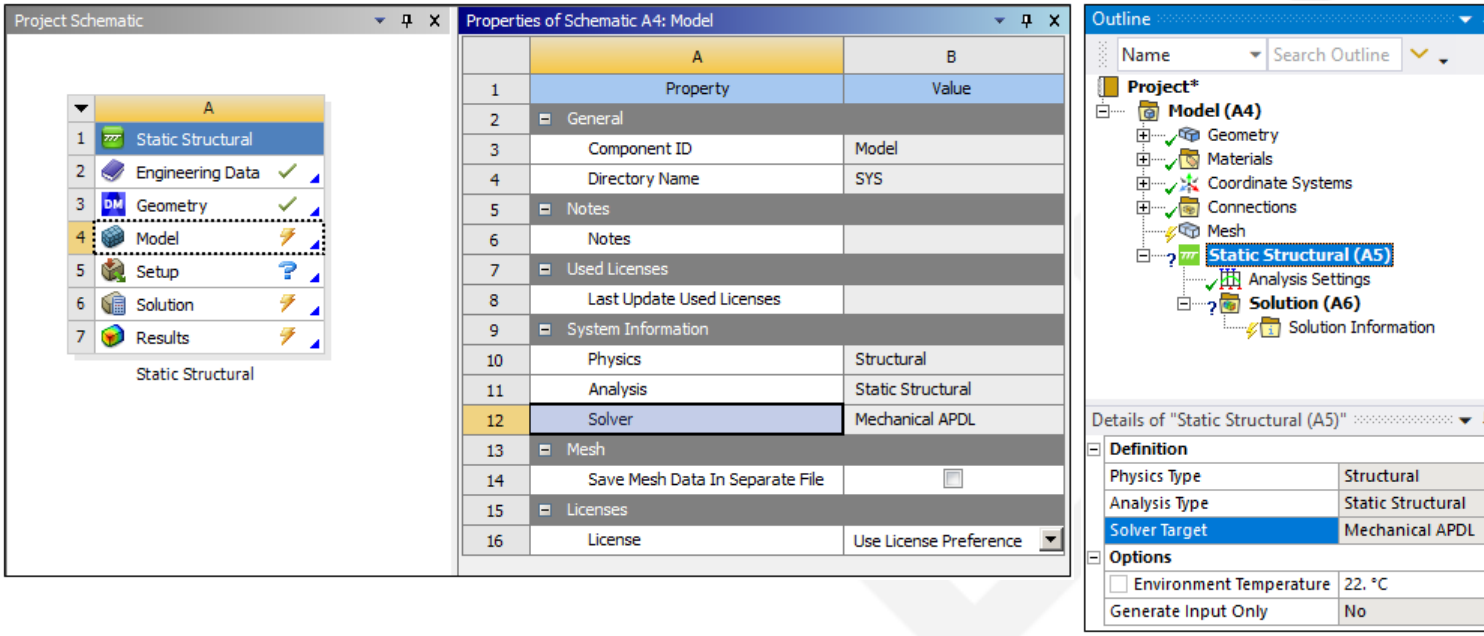
Go to a solution section topic:

Understanding Interface Options
Performing the Solution
Selecting Solve Modes for an Analysis
Using Solve Process Settings
Using Solution Restarts
Restart Analysis Extension
Understanding Solve Scenarios
Specifying Solution Information
Postprocessing During Solve
Using Result Trackers
Using Adaptive Convergence
Saving and Managing Results
Writing and Reading Solution Data
Understanding Solving Units

Understanding Interface Options

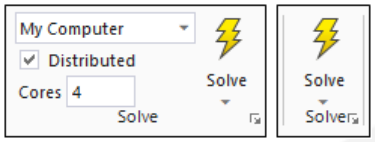
When you initiate a simulation in Workbench, the application automatically specifies the solver (p. 355). You can view the specified solver in Workbench by displaying the properties (of the **Model** cell) of the analysis. The **System Information** category displays a **Solver** property. Once you open Mechanical, the Details for the Environment object displays a **Solver Target** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Solve Options

Home Tab **Context Tab**



The **Solve** group provides options that enable you to specify some basic solution configurations and to solve your analysis.

- The **Solver Handler** drop-down menu enables you to specify the desired target machine on which to perform your solution. It contains the options **My Computer** (default) and **My Computer, Background**. When you create remote solution options, they are also displayed in the menu.
- The **Solve** button/option also provides a drop-down menu that contains the same options as the **Solver Handler** menu. These options initiate the solution when selected.
- The **Distributed** option, active by default, indicates the use of distributed-memory parallel (DMP) processing. For more information, see the Using DMP Processing section of the Mechanical APDL Parallel Processing Guide.
- The **Cores** field enables you to change the number of CPU cores to use during the solution. The default is **4**. If your computer does not have four cores, the application automatically scales to the lower number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The **Resource Prediction** (p. 170) option opens a window that enables you to select an available analysis and produce a prediction the computing resources needed to perform a solution for the environment.

Important:

In the lower right-hand corner of the **Solve** group is an option that launches the Solve Process Settings (p. 1804) dialog. This dialog enables you to specify solution settings for remote computers as well as specific process settings to be used during the solution process.

Solution Progress Display

Once you start a solution, its progress is displayed in the Status Bar (p. 135). The status bar also displays solution Interrupt and Stop options. If you would like the solver to halt immediately and forego writing any outstanding restart points, select the Stop option. Using the Interrupt option, you instruct the solver to complete its current iteration and record outstanding restart points (available for Static Structural, Transient Structural, and Structural Optimization analyses). Neither of these options affects previous restart points.

In addition, you can select the Progress pane to display the **Solution Status** window. Like the status bar display, this window displays the progress for synchronous solutions. The window also includes a **Stop Solution** button and an **Interrupt Solution** button. Using the **UI Options** (p. 244) preference setting **Progress**, you can choose to always display the progress window.

Note:

- Clicking the **Interrupt Solution** button places a file named `file.abt` in the working directory. This may be useful for those who are familiar with Mechanical APDL functionality.
- The Ansys HPC Platform Services (HPS) application does not support the Interrupt option.

Performing the Solution

This section examines the options to perform a solution, either locally or on a remote machine. Go to a desired topic:

- Solving: My Computer (p. 1793)
- Solving: My Computer, Background and Remote (p. 1794)
- Solution Options While Processing (p. 1795)

Solving: My Computer

To perform a solution from your computer, select the **My Computer** option (default) and then select the **Solve** option (or select **[F5]**) to use the default solve process settings. When you run a local solution, a temporary file directory is created to hold the solution files. It is not until you save the project that

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

solution data and folders are saved to your project folder. The context menu (right-click) option **Open Solver File Directories** of the **Solution** object enables you display the files of the temporary directory.

Once your solution is complete and successful, green check marks display on the **Solution** and result objects. A red lightning bolt indicates a failed or unconverged solution. And an overlaid pause icon indicates the solution could resume with the use of restart points.

Note how the following affect the solution process. When you select:

- Any object (Model, Coordinate Systems, Mesh, etc.) in the tree that is above your environment(s), the application solves all specified environments in sequence.
- A specific environment, or any child object of that environment, the application solves only that analysis.

Important:

Make sure that the project file path you specify for the solve does include Asian language characters. This can result in the solution using a temporary folder rather than the Scratch Solver Files Directory (p. 1487).

Solving: My Computer, Background and Remote

The background and remote processing capabilities of Mechanical require you to have the Ansys Remote Solve Manager (RSM) application and/or the Ansys HPC Platform Services (HPS) application installed.

My Computer, Background

The **My Computer, Background** option enables you to solve on your local machine with the ability to close the Workbench session. This option prompts you to save your project before it begins the solution process. This enables you to close the application while the solution completes.

Remote

Using the options of the solve process settings (p. 1804) dialog, you specify a desired remote machine. Options include:

- **Ansys Remote Solve Manager (RSM)**. Using the RSM Configuration, you can define configurations that enable you submit a solve to an established compute cluster or Cloud portal. This step may be done for you by a person designated as the RSM administrator to make RSM queues that map to HPC queues available.
- **Ansys HPC Platform Services (HPS)**. Ansys HPC Platform Services (HPS) enables you to submit solutions and store solver files using multiple heterogeneous compute resources. Also note that when you have a simulation with linked systems, HPS does not require you to download the solution data from the upstream systems. Review the Using Ansys HPC Platform Services (HPS) to Solve Linked Analyses (p. 1801) as well as the Understanding Ansys HPC Platform Services (HPS) Limitations (p. 1802) sections for more information.

Review the recommendations in the Solve Modes and Recommended Usage (p. 1803) section for configuring your solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

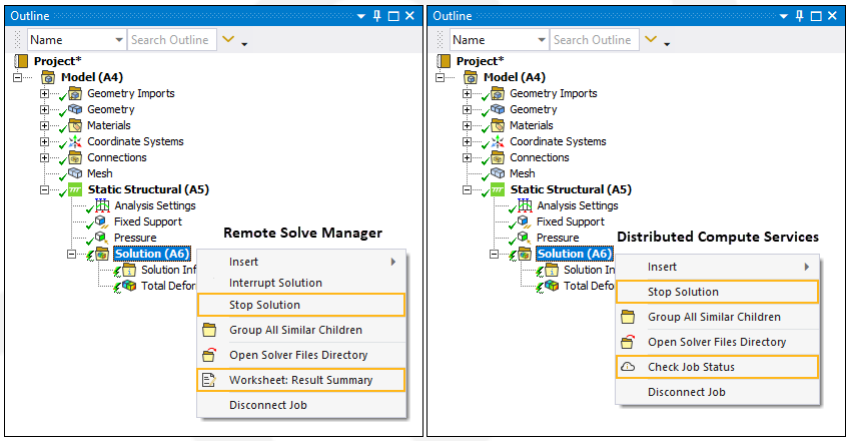
Published: 2023-10-23

Solution Options While Processing

Review the following characteristics for a remote solution as well as certain options/actions that are available while a remote solution is being performed. In addition, note that there are specific actions that only apply to the RSM application (p. 1797) as well as the HPS application (p. 1800):

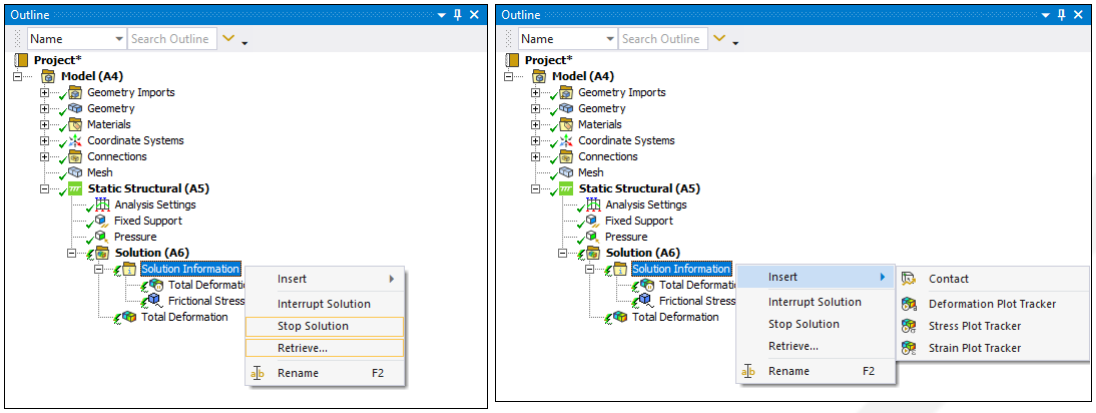
Active solution options:

- While a background solution is in progress, the system branch is in a read-only state with the exception that result objects can be deleted during this time. Other branches can be edited freely.
- As illustrated below, there are several context (right-click) menu options available from the **Solution** object *while* the solution is processing.
  - **Stop Solution**: Use this option to cancel a running job and reset the state of the environment. This action immediately stops the solution and the application does not attempt to retrieve any solution files (if solving on a remote server).
  - **Worksheet: Result Summary**: This option is a general option for Mechanical solutions but is also available for the Remote Solve Manager. It displays the Solution Summary Worksheet (p. 1961) for the **Result Summary** option.
  - **Check Job Status**: This option is available for Distributed Compute Services and opens the URL path of the current job in a browser window to display its status as well as the list of all associated files as they are generated. Note that your solution files are stored on the web server. You can manage the files using the HPS web interface. For example, you can choose to manually download or delete specific files for a completed solution from the HPS server as shown. If you are performing a solution on a linked analysis, make sure that you keep the files on the web server until all the analyses have been solved.



- The Solution Information (p. 1833) object also offers the **Stop Solution** option as well as the option to **Retrieve** result data. When running background and remote solutions, you can check the status of the solution by using the **Retrieve** option. The object also enables you to insert result trackers (p. 1844) during the remote solution process.

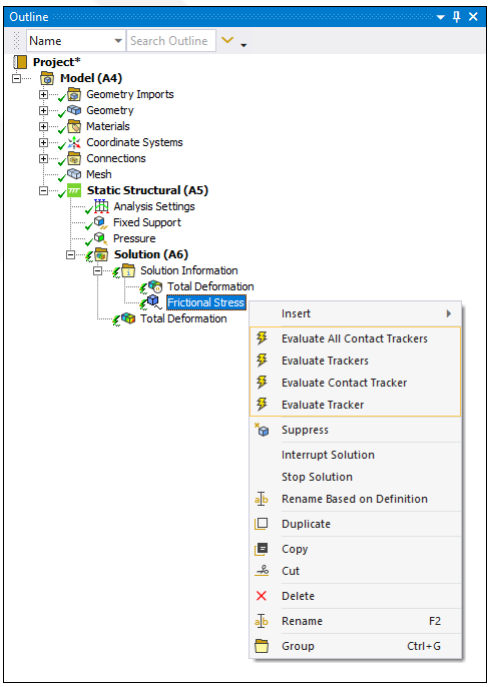
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

In rare instances, the **Retrieve** feature could fail if the necessary files do not become available at a particular time. Simply choosing the option again will likely solve the issue.

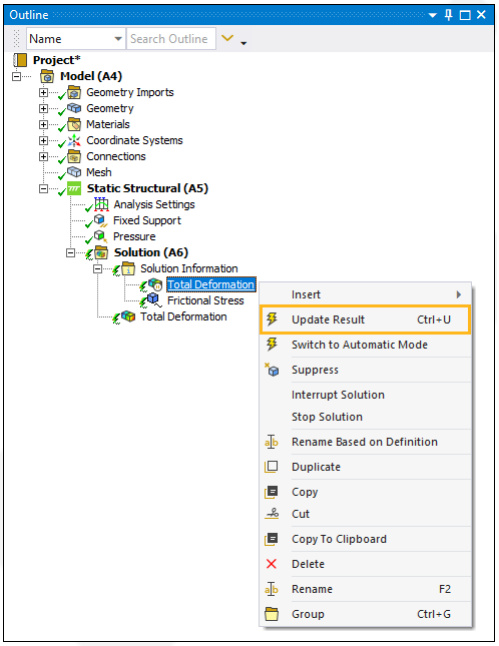
For Result Trackers, when you use the **Retrieve** option (required), the application provides the following highlighted options to evaluate (obtain result data for) result trackers at the given moment in the solution process.



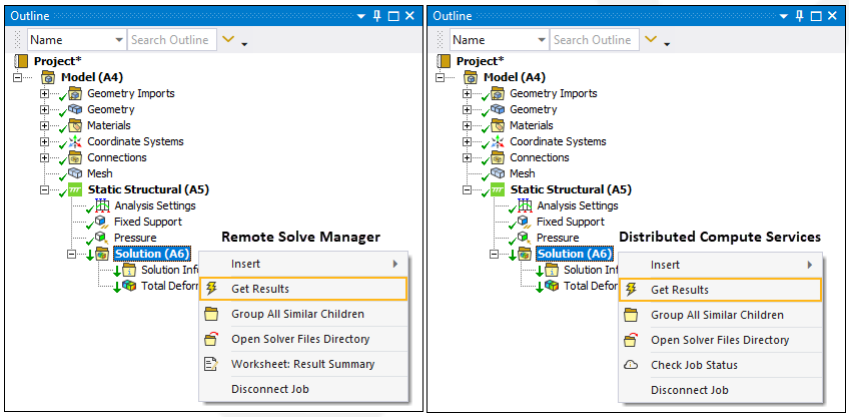
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For Result Plot Trackers, when you use the **Retrieve** option (required), the application provides the **Update Result** option to evaluate (obtain result data for) the result plot tracker at the given moment in the solution process.



- Once your solution is complete, a green down arrow status symbol (p. 109) indicates that the results are ready for download. From the context (right-click) menu of the **Solution** object, select the **Get Results** option.



- The context menu (right-click) option **Disconnect Job** is available from the **Solution** object. The option is available during the solution process as well as once the solution completed. This option disconnects Mechanical from the remote handler and the application returns to the beginning of the solution process.

- When using a **Local** solve process setting and a solve is in progress, do not reboot or log off the Windows client machine. If you reboot or log off, the connection to the Linux job will be lost and results will not be retrievable. If the Linux job has completed, then rebooting or logging off is safe.

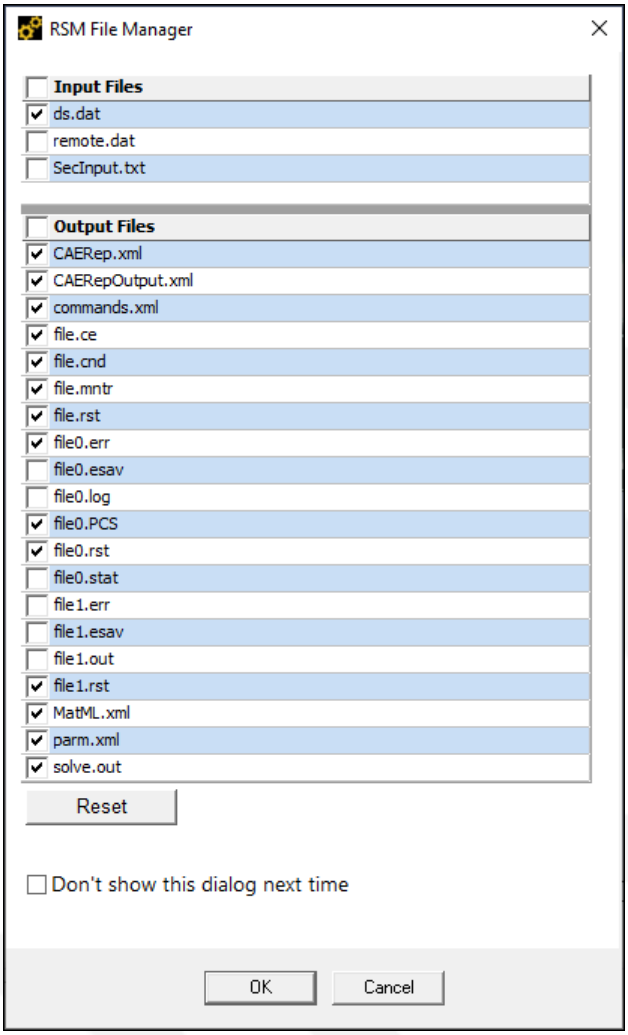
RSM Application Only

Note the following actions that can only be performed when you specify the RSM application as your remote target:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If you have the **RSM Output Files Download** preference set to **Show**, the **RSM File Manager** dialog displays as illustrated below, and enables you to select or clear the input/output files you wish to download. See the **Results** (p. 242) category of the **Options** dialog to change this preference. The default setting is to hide this dialog.

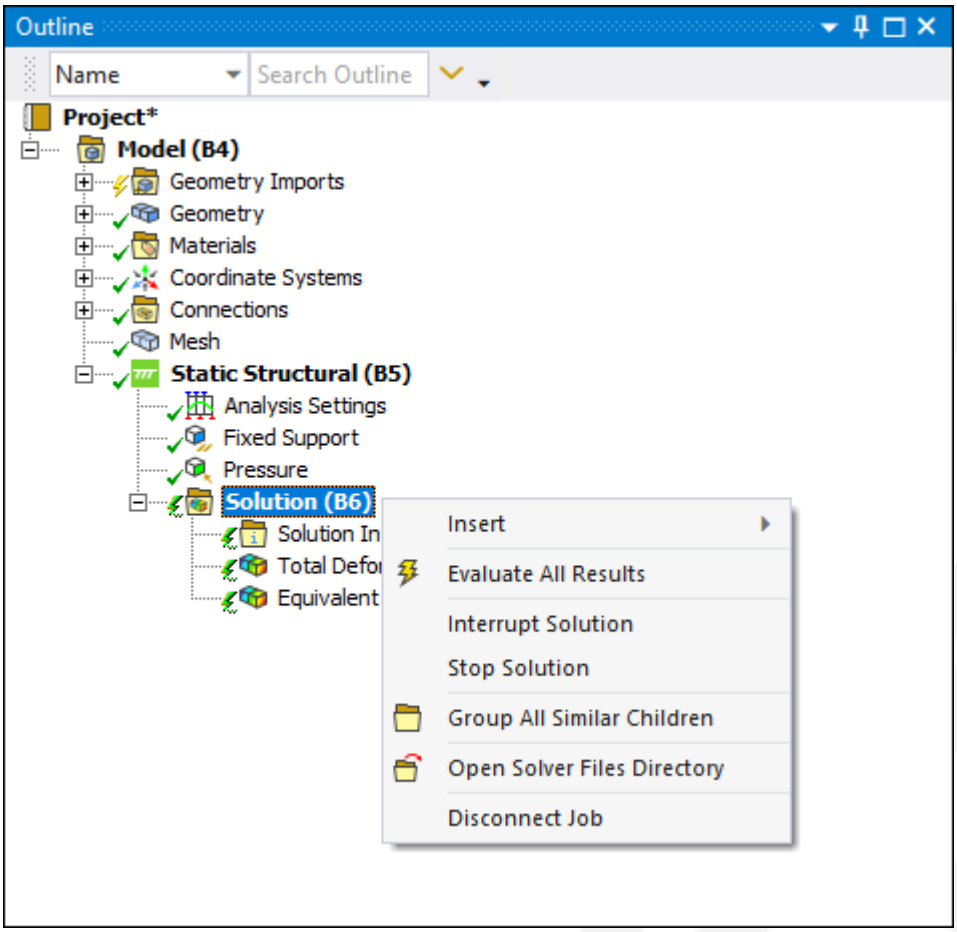
  The dialog displays with certain files selected by default (illustrated below). As shown, there is an option to turn off the display of the dialog: doing so instructs the application to download all available files. Additional options include the check boxes beside the **Input Files** and **Output Files** labels that automatically select all options beneath the heading as well as a **Reset** button that restores default checkbox selections.



- An alternative to canceling a job is to choose **Interrupt Solution** in the context menu. As in a synchronous solution, this will allow the solver to complete its current iteration and record outstanding restart points. This option creates a file named `file.abt` in the working directory.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- In the event of a network connection loss to the **Remote Solve Manager**, the **Get Results** function prompts you with a warning message to address the connection issue. You can perform the **Get Results** operation and retrieve your results information once you re-establish a connection.

Note:

- By default, the **Options** dialog preference, **RSM Polling Delay**, specifies how often the processing from the remote solution is polled or queried for result data. The default setting is 5 seconds. The supported range is 1 - 6000 seconds. This control is available from the Analysis Settings and Solution (p. 238) category of the dialog. This feature does not prolong solution processing. Solution completion ends the polling process.

- When the compute server is a remote machine, the application transfers all necessary data to the remote machine for evaluation. Currently there is a beta option available from the context (right-click) menu of the **Solution** object: **Fetch Remote Results**. This option enables you to display result content for the most recently completed load step while the solution is running. See the Fetching an RSM Result section for more information.

- When performing the solution on a remote machine or cluster, the **Evaluate All Results** option may not automatically download the `file.rst` and as a result may not eval-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

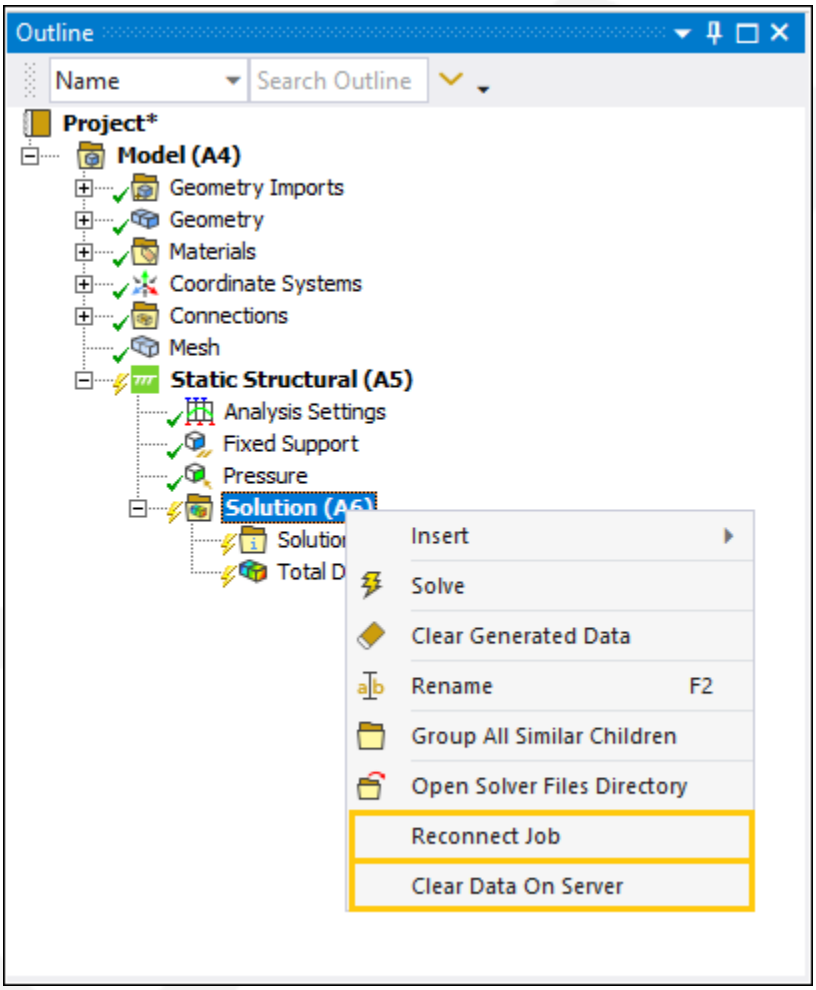
uate the results while the solution is running. If this is the case, you need to manually download the file to the scratch directory to evaluate the results using this option.

HPS Application Only

Note the following **Solution** object context menu options that you can only perform when you specify the HPS application as your remote target:

- **Reconnect Job**: This option enables you to connect to the last submitted job on HPS for the current system. You will be able to restore the connection to the HPS job using this option when you have reset the state of the system by either disconnecting or clearing the results after download. This option is available until you perform a new solve.
- **Clear Data on Server**: This option enables you to delete all the output files of last run HPS job on the server. This helps you manage the disc space of the remote server. This option is available once you results download and remains available until you perform another solution.

  This option cannot be used when either the current system or its linked downstream system is running on the HPS server or if the current results have not yet downloaded.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using HPC Platform Services to Solve Linked Analyses

Although HPC Platform Services (HPS) enables you to solve any Mechanical linked analyses, for the analysis combination listed below, HPS solves without having to download the results of the upstream systems. If your simulation includes linked systems, and all the systems are solved on the same HPS web server, the output files of the upstream system are stored on the HPS server but are not downloaded before the start of the downstream analysis. HPS enables referencing of the required files on the server between the upstream and the downstream analyses. This file referencing can save a significant amount of solution processing time.

Without needing to download the results of the upstream system, HPS currently supports the following linked analyses:

- Mode Superposition analyses: Mode Superposition Harmonic Response, Transient, Random Vibration, and Response Spectrum (including Prestress)
- Prestress Modal
- Prestress Full Harmonic Response
- Prestress Eigenvalue Buckling
- Prestress Coupled Field Modal and Coupled Field Harmonic
- Static Acoustics and Modal Acoustics
- Static Acoustics and Harmonic Acoustics

Synchronous Submissions

When you initiate a solve on a downstream system or at the **Model** level of a system, the application submits all the jobs for the linked systems synchronously, even if the upstream system has not completed its solution. All jobs in the HPS queue process in order. Because the application does not wait to submit all jobs for linked systems, you can close the application once all jobs have been submitted to conserve your license for other purposes.

Furthermore, the application automatically determines whether it needs to wait for upstream systems to complete. This eliminates the need for you to monitor the process.

Limitations of Synchronous Submissions

The synchronous submission process is not supported if an analysis includes any of the following:

- Spot Welds
- Bolt Pretension
- Weak Springs
- EM Transducer loading condition
- Command (APDL) objects

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Contact Splitting

These features and capabilities may require the application to create additional elements or nodes during the solution process. As a result, Mechanical would need to wait for upstream jobs to complete in order to receive the necessary data for the downstream jobs.

Downloading Upstream Results

HPS also supports the following types of linked analyses for which the upstream results are required to be downloaded. In such cases, the upstream results are automatically downloaded before submitting the downstream analysis to HPS.

- Analyses involving imported loads from the upstream system such as Steady-State Thermal and Static Structural systems and Electric Conduction and Steady-State Thermal systems.
- Steady-State Thermal and Transient Thermal systems.

Note:

- If you are performing the upstream and downstream analyses on different HPS servers, make sure to download the results of the upstream system before submitting the downstream system. However, we recommend submitting the linked systems to the same server to take advantage of the file referencing capabilities of HPS.
- When you initiate a solve on a downstream system or at the **Model** level of a system, and you abort the job or the job fails, all jobs currently running or that are pending stop automatically.

Note:

As stated above, you can close the application after all systems have been submitted. However, if your simulation includes systems that must wait for upstream processes to complete, and/or the results to be downloaded, it may take some time for the last system to be submitted. In this instance, you may want to submit the jobs individually if the application needs to be closed during the solution process.

Understanding HPC Platform Services (HPS) Limitations

HPC Platform Services (HPS) is not supported for the following analysis types and solver selection:

Analysis Types	Solvers
Additive Manufacturing Structural Optimization Condensed Geometry (Generation Pass) Rigid Dynamics Motion	Autodyn LS-DYNA Restart

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Selecting Solve Modes for an Analysis

Mechanical includes capabilities for efficiently solving various kinds of analyses taking CPU usage and solving time into consideration. The following table defines the various solve "modes" available and includes references to recommended usages and associated solve process settings. Further details are discussed in the various other sections under Understanding Solving (p. 1791).

Solve Start Mode	Solve Monitor Mode	Recommended Usage	Solve Process Settings
In Process - The solve starts and finishes on your computer in the directory where your project resides.	**Synchronous** - The solve runs and finalizes within the same Workbench session.	Analyses that are not expected to be extremely CPU intensive.	**My Computer**
Out of Process - The solve starts and finishes either on another computer, or on your computer but in a directory that is separate from the one where your project resides.	**Asynchronous** - The solve is not restricted to run and finalize during any particular Workbench session.[a]	Analyses involving large models or a large amount of processing time and machine resources, **excluding** linked analyses and analyses that involve multiple convergence loops.[b]	**My Computer, Background** or remote solve handler (RSM or DSC)
	Synchronous - The solve runs and finalizes within the same Workbench session.	Analyses involving large models or a large amount of processing time and machine resources, **including** linked analyses and analyses that involve multiple convergence loops.	**My Computer, Background** or remote solve handler (RSM Only), then click **Advanced...** button and check **Solve in synchronous mode (Ansys only)**.

[a] When solving in asynchronous mode, you are free to continue working independently of the solve job, or close the Workbench session and retrieve the solution results at a later time. You can even shut down your computer if the job has been submitted to a remote cluster (See Using Solve Process Settings (p. 1804)). An asynchronous solution is queued with other solutions and can run either on your local machine or on a more powerful remote machine. Background solutions are recommended for large models or simulations that require a large amount of processing time and machine resources. Sending the Solve to a remote cluster can increase productivity when a high-end server is available on your network.

[b] Though not recommended for a linked analysis using this solve mode combination, you can solve a linked analysis or an analysis involving multiple convergence loops provided you solve each analysis separately, that is, you must obtain the first solution, then choose **Get Results** from the context menu in the first analysis before obtaining the solution in the second analysis. The Out of Process and Synchronous mode combination is recommended for these types of analyses because the solve can occur from a single user action. Also, asynchronous solutions involving linked analyses that are initiated from the **Project Schematic** by choosing **Update** will automatically achieve the same effect as choosing **Get Results**, thus providing another method for solving linked analyses from a single user action. See the Understanding Solving (p. 1791) help section for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

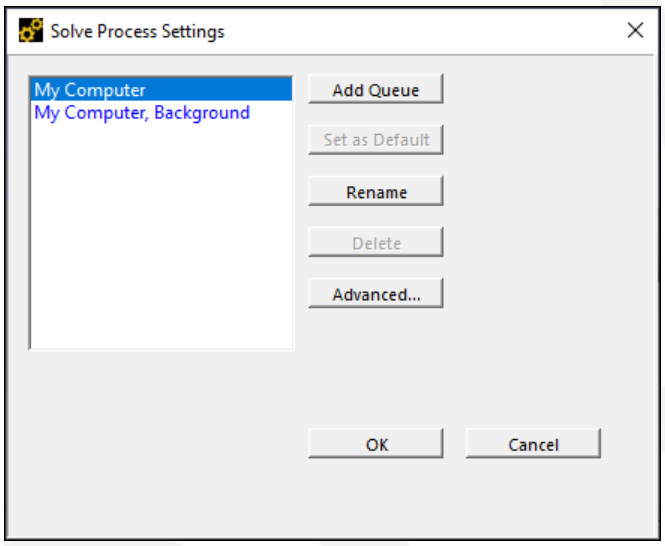
Using Solve Process Settings

Solve process settings are individual solving configurations (p. 1803) that you set up prior to initiating solves. Settings include specifying a *synchronous* or *background* solve, as well as solve manager machine and queue designations for background configurations. Using the **Solve Process Settings** dialog box, you can:

- Add a remote solve process
- Specify a default solve process
- Modify existing solve process settings
- Delete an existing solve process
- Rename a solve process

Solve Process Settings

You access the **Solve Process Settings** dialog by selecting the **Solve Process Settings** option from the **File** tab or the option on the bottom right corner of the **Solve** group of **Model** Context tab. A dialog similar to the displays as illustrated below based. The left pane displays the available solver options.



Based on your selected solver option (**My Computer**, etc.), you can use the following options:

- **Add Queue**: Specify a remote handler. This option displays an entry pane wherein you enter a name for your remote solution handler.
- **Set As Default**: Specifies the solve process as default across Workbench sessions.
- **Rename**: Renames the selected solve handler. You can rename any handler.
- **Delete**: Deletes the selected solve process. Only handlers that you create can be deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Advanced**: Displays the **Advanced Properties** dialog box associated with either **My Computer**, **My Computer, Background**, or specified **Remote** handler (see below).

Note:

Each solve handle must have a unique name.

Specifying Solve Handlers

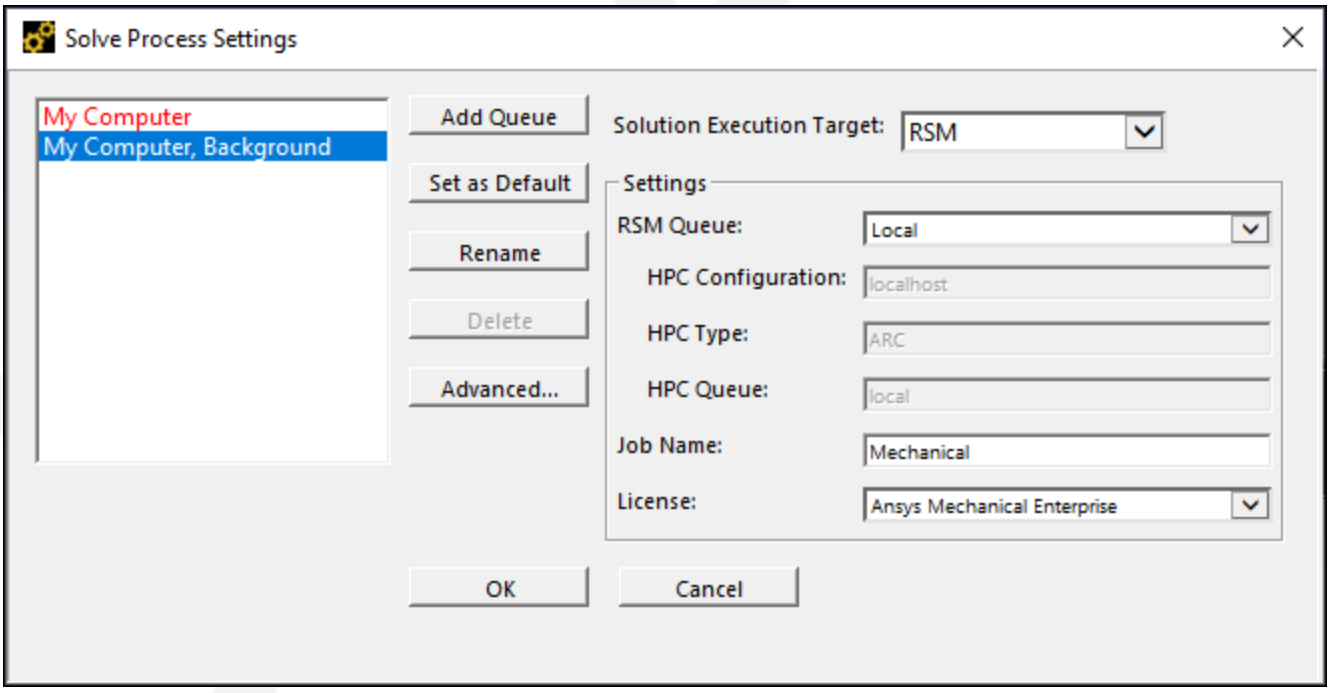
The left pane of the dialog displays the available solver handlers. These are the options available in the Solve Handler drop-down menu from the **Home** tab. This portion of the dialog contains the options **My Computer** (default) and **My Computer, Background**. When you create remote solution option, it is also displayed in the menu, as illustrated below. Solver handler is described below.

My Computer

The setting **My Computer** is the default setting. When using this setting, the application solves and finalizes the solution on the local computer in the current Workbench session.

My Computer, Background

Selecting this setting, solves on the local machine but is not restricted to finalizing in a particular Workbench session. *You need more than one solver license to use this setting*. However, you can perform Rigid Dynamics and Explicit Dynamics analyses with one solver license by selecting the **Use Shared License, if possible** option on the **Advanced Properties** dialog box. Although solved locally, the **Solution Execution Target** menu is set to **RSM**.

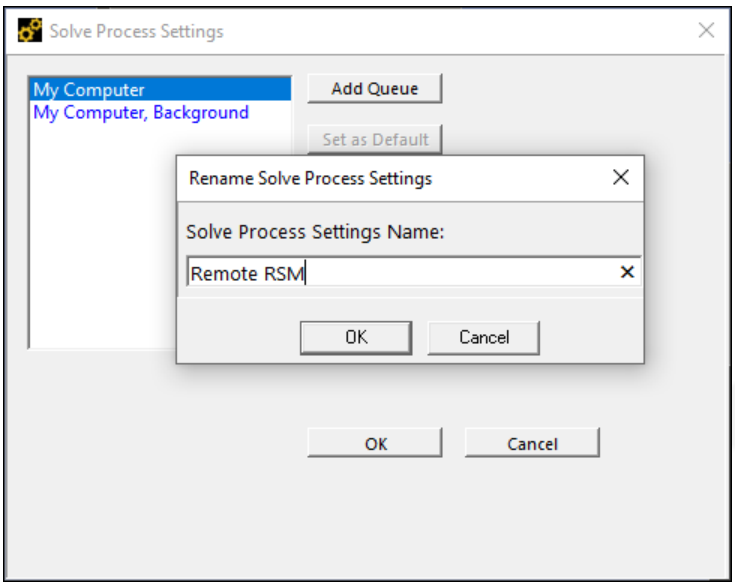


Remote

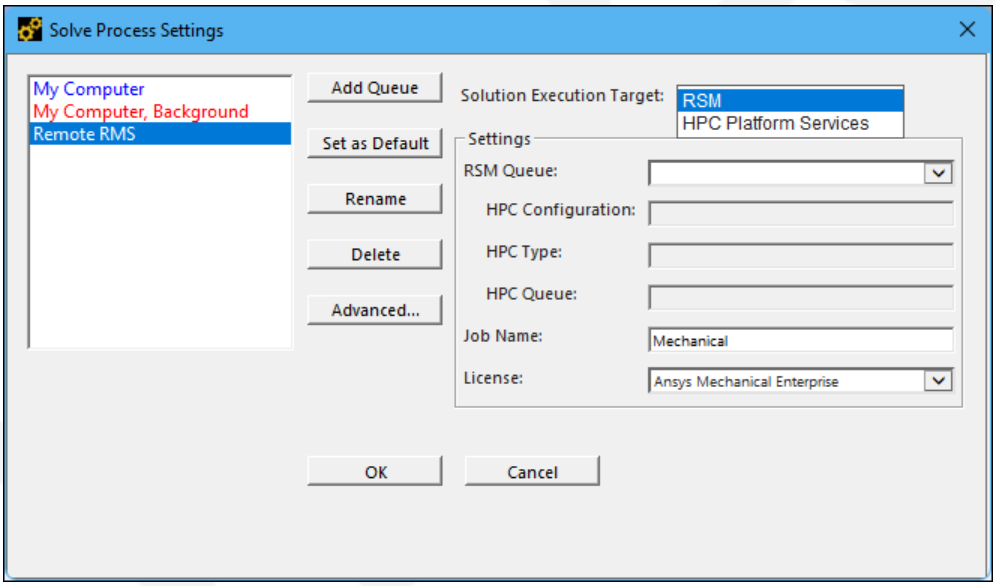
To create a solve handler for a remote location, you need to create a new handler and specify the required properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. Select the **Add Queue** option, you can create and name a remote solve handler. For this example, the handler is named to be used with RSM.



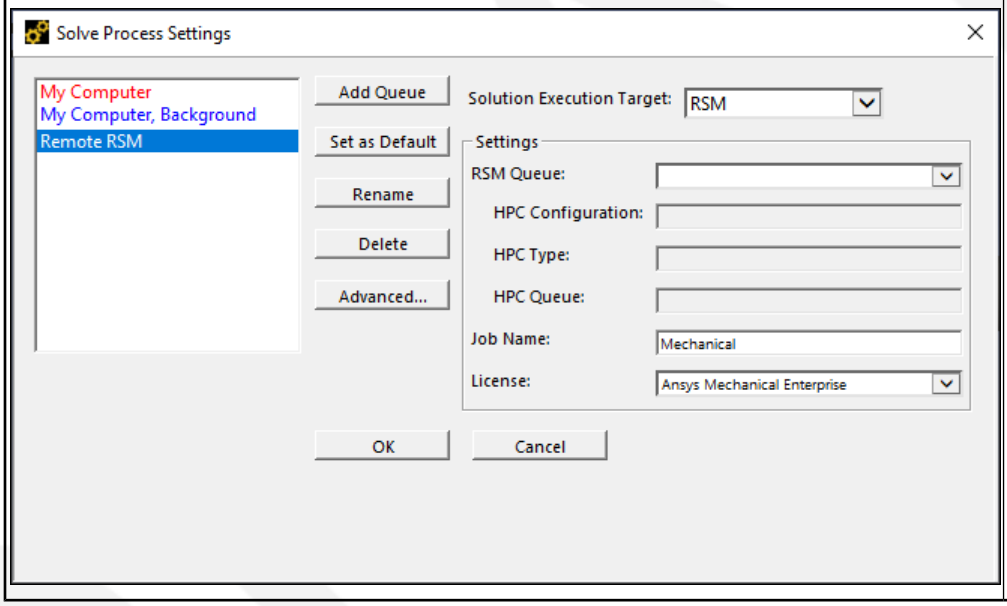
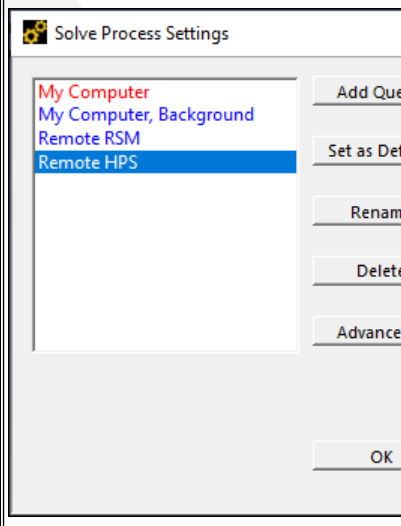
2. Provided by the **Solution Execution Target** menu, shown below, there are two types of remote solve handlers that are supported: **RSM** (Ansys Remote Solve Manager) and **HPS** (Ansys HPC Platform Services). For this example, **RSM** is specified for the "Remote RSM" handler created in the last step.



3. Now, based on the target you select, you specify the settings, including:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Ansys Remote Solve Manager	Ansys Distributed Compute S
To create a RSM handler, define the following: • **RSM Queue**: This field provides a drop-down menu of all available queues configured using the RSM Configuration tool. If RSM queues are not available, use the **RSM Configuration** tool to check your RSM configurations. • **HPC Configuration**: Read-only field that reports the name of the RSM configuration with which the selected RSM queue is associated. • **HPC Type**: Read-only field that reports the type of HPC resource to which the job will be submitted. • **HPC Queue**: Read-only field that displays the name of the HPC-side queue with which the RSM queue is associated. • **Job Name**: Specify the name to be used to identify the job. For RSM, this is the job on the HPC resource. See the Remote Solve Manager documentation for more details.	To create a **HPS** handler, you ne on a remote machine. You can using the **Test Connection** opt As necessary, specifies the nam used for the solution on the se This handler requires you to sp **Name**.
Interface Example 	**Inter** 

Note:

Your **Job Name** must adhere to the following naming requirements:

- Acceptable naming characters include alphanumeric characters (letter or numbers) as well as the following characters: ~#%^&,'()-_+=/<>.
- The job name must begin with a letter or a number (i.e. an alphanumeric character).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The maximum length of a job name is 16 characters.

Note:

- You must specify a valid Ansys product license because a separate instance of an Ansys application is being used.
- The license from your current Ansys Workbench client session cannot be accessed from the remote Ansys application executable.

Note:

- Settings are not available when you select the built-in **My Computer** solve process.
- **RSM Queue** fields are required for all Remote *background* configurations.
- When you submit a job to a Remote Queue and you have not defined corresponding credentials in the Accounts section of the RSM Configuration, you are prompted with the **Remote Solve Cache Credentials** dialog box. Enter your User Name and Password to continue.
- See the Submitting Solutions to Remote Solve Manager section of the *Workbench User's Guide* for additional information for using the Remote Solve Manager during your solution processes.

Note:

The solve process in red indicates that the process is selected as the default solve process and persists across Workbench sessions.

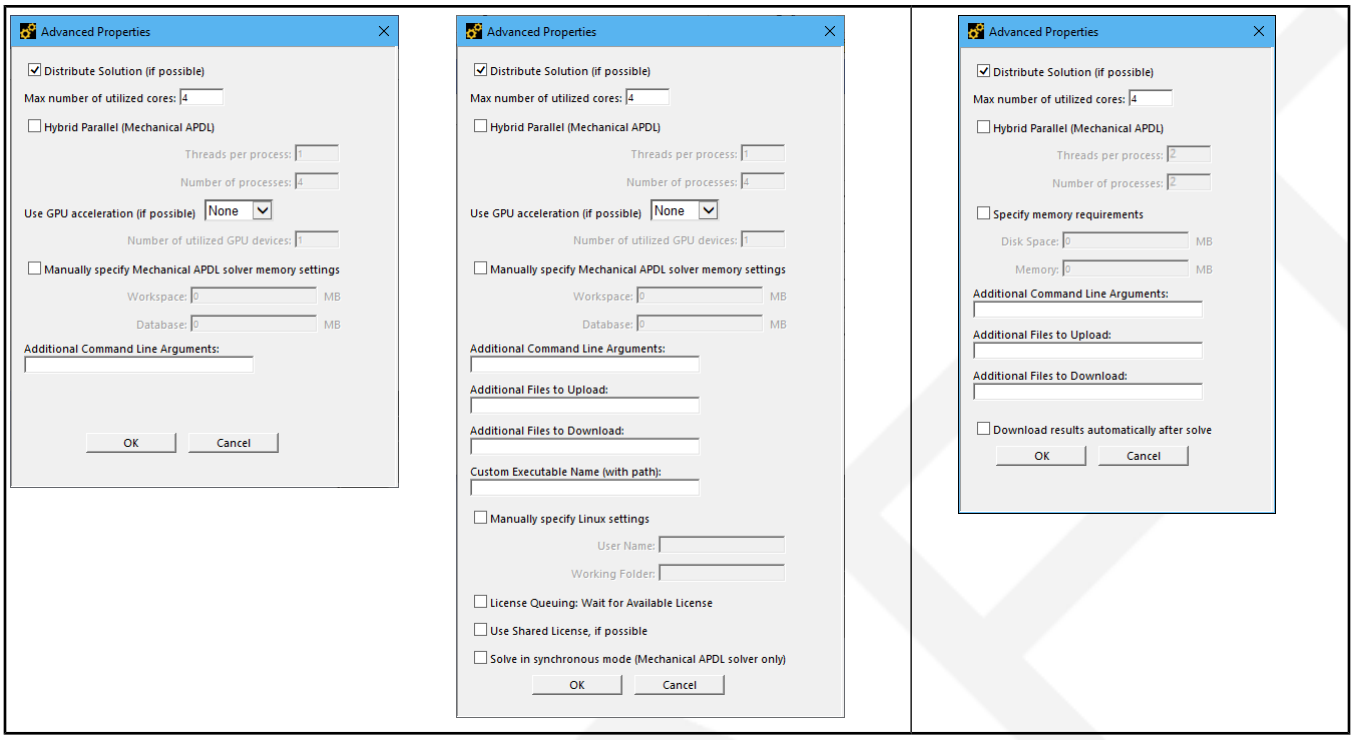
Advanced Properties

Selecting the **Advanced** button on the **Solve Process Settings** dialog displays one of the following **Advanced Properties** dialog boxes. The available options are based on whether you select **My Computer**, **My Computer, Background**, or **Remote**.

My Computer	My Computer, Background/Remote RSM	Remote HPS

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The **Advanced Properties** are described below.

Distribute Solution (if possible)	This option is selected by default. It enables and disables the ability for a distributed solution. **Note:** This option supports the Mechanical APDL Solver and Explicit Dynamics Solver only. For specific support specifications for the Mechanical APDL Solver, see the Supported Analysis Types and Features section of the *Mechanical APDL Parallel Processing Guide* for more information. **Important:** If you solve a Modal system using this setting, and you have linked the analysis to either a Harmonic Response, Response Spectrum, or a Random Vibration, or a follow on Mechanical APDL (turn on the **Distributed** property in Project Schematic), these downstream systems must also have this setting selected in order to properly solve. The application alerts you to this situation with a warning message.
Max number of utilized cores	Sets the number of cores to use during the solution. The default is value is **4**. Entering **0** does not send any request to the Mechanical APDL solver related to the number of cores to use. For shared-memory solutions, if you specify a number greater than the number of cores in the node, the highest available number of cores is used. However, for distributed-memory solutions, you cannot specify a number greater that the available number of physical processors.

	For more information on parallel processing in Mechanical APDL, see HPC Licensing in the *Parallel Processing Guide*. For analyses using the Explicit Dynamic solver, this setting is used to determine the number of cores unless this has been specified in the Additional Command Line Arguments. **Note:** • Available only for the Mechanical APDL and Explicit Dynamic solvers. This setting is not available when using the LS-DYNA solver. • In order to run an Explicit Dynamics or Mechanical APDL solve in parallel, you need an Ansys HPC license for additional cores after the first 4. • For Explicit Dynamics analyses, this setting is used to determine the number of cores unless this has been specified in the Additional Command Line Arguments.
Hybrid Parallel (Mechanical APDL) **Important:** Not supported for Structural Optimization analyses.	This option is available when the **Distributed Solution** check box is selected (default setting). This capability reduces memory usage, more effectively uses hardware, and improves the scalability of large models with high element load balance ratios. For more information, see Chapter 5: Using Hybrid Parallel Processing (SMP + DMP) of the *Mechanical APDL Parallel Processing Guide*. Even if this option is not checked and Distributed Solution is selected, the application may automatically choose to use the Hybrid Parallel ("auto-hybrid"). Note that when the application performs an auto-hybrid solution, the Solution Statistics page displays the solution process as "Automatic Hybrid" instead of "Hybrid Parallel." The **Hybrid Parallel** process includes two associated settings: **Threads per process**: Specify the number of threads for each process. The default value is **1**. This value cannot exceed the **Max number of utilized cores** value. If this value is greater than the number of **Max number of utilized cores**, the application automatically reduces the threads value to be equal to the **Max number of utilized cores** value. **Note:** When you active the **Hybrid Parallel (Mechanical APDL)** option and the **Threads per process** property is set to **1**, the application performs a Forced Distributed Solution. • **Number of processes**: This read-only value is calculated as:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	`Max number of utilized cores / Threads per process` **Note:** – If the resulting value is a decimal, the application automatically uses the lower integer. – When Mechanical uses the lower integer, the number of utilized cores is the product of the number of processes and threads per process, instead of the actual number of specified cores. **Note:** There are instances when the Hybrid Parallel feature may not use the entire number of requested cores. In this case, the remaining cores are idle for the duration of the calculation.
Use GPU Acceleration (if possible)	Provides access to the Graphics Processing Unit (GPU) acceleration capability offered by the Mechanical APDL solver. It supports the NVIDIA and AMD acceleration cards. To enable the feature, select **NVIDIA** or **AMD** from the drop-down menu. Ansys Distributed Compute Services (HPS) does not support this option. See the **Requirements for the GPU Accelerator in Mechanical APDL** section in the Windows and Linux installation guides for specific card requirements.
Number of utilized GPU devices	Specifies the number of GPU accelerator devices to be used when the **Use GPU acceleration** property is set to use a valid accelerator type. This value can be an integer in the range of **1** to **20**. The default value is **1**. For additional information, see the GPU Accelerator Capability section of the *Mechanical APDL Parallel Processing Guide*.
Manually specify Mechanical APDL solver memory settings	Helps you specify the amount of system memory, in MB, used for the Ansys application workspace and database. **Note:** This setting can also be used to specify memory use for the ABAQUS and Samcef solvers. For more information, see Memory Tuning the Samcef Solver (p. 1814) and Memory Tuning the ABAQUS Solver (p. 1814). This setting is not valid when using the LS-DYNA solver.
Specify memory	Specify the amount of memory and disk space needed to run the solution. See the computational Requirements topic in the **Defining Tasks** section of the *Distributed Compute Services* documentation for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

requirements (HPS Only)	
Additional Command Line Arguments	Specifies arguments that you would normally enter into a command line input, for example, `-machine` option for a distributed solution. This setting is ignored for non-remote and non-background LS-DYNA calculation.
Additional Files to Upload	This field enables you to specify desired files to upload into the Remote Machine from the Solution Directory of your local machine. Use a semi-colon to separate file names. Use the asterisk character (*) as a wild card to automatically select multiple files for the given criterion. For example, you can include certain files based on extension, such as *.PNG, and/or files with a specific string in the name.
Additional Files to Download	This field enables you to specify desired files to download from the Remote Machine to the Solution Directory of your local machine. Use a semi-colon to separate file names. Use the asterisk character (*) as a wild card to automatically select multiple files for the given criterion. For example, you can include certain files based on extension, such as *.PNG, and/or files with a specific string in the name.
Download results automatically after solve	Selecting this option instructs the application to automatically download the results once the HPS solution is complete. Ansys recommends that you keep the Mechanical application open until the download finishes. For linked systems that you can submit synchronously (see Using Distributed Compute Services to Solve Linked Analyses (p. 1801)), the download of the results happens independently of the submission. As a result, the downstream system's solution does not need to wait for a completed download of the upstream results, unlike the solution process for the synchronous mode that is available for RSM.
Custom Executable Name (with path)	Specifies a custom Ansys application solver executable name and path. This executable will be used for the Ansys application solve rather than using the default. **Note:** Depending on the requirements of your operating system, you may need to specify the path in quotations.
Manually specify Linux settings	Enter a valid **User Name** and **Working Folder** to override the RSM compute server proxy settings. **Note:** • You must have write access to this folder on all potential compute proxies in the queue.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	• To use the RSM settings, leave this field blank.
Use Shared License, if possible	Enables the use of a Shared License **Note:** • This option works only for Explicit Dynamics and Rigid Dynamics analysis. For more information, see Shared Licensing • License sharing is only possible within a single Workbench session with the solver running on the same machine. A remote solve on another machine via RSM will require a license for the Workbench session and a license for the remote solve.
License Queuing: Wait for Available License	Instruct the Mechanical APDL solver to wait for an available license when solving remotely via RSM.
Solve in synchronous mode (Mechanical APDL solver only)	Select to mimic the default **My Computer** behavior while leveraging the computation power of a remote machine. See this section from the Mechanical APDL help for more information: HPC Licensing in the *Parallel Processing Guide*. For Explicit Dynamics analyses, this setting is used to determine the number of processors unless this has been specified in the Additional Command Line Arguments. **Note:** • Applicable only for Mechanical APDL solver. • Requires an additional license. Clear the check box to deliberately download results from a remote machine (by issuing **Get Results** on a right-mouse click the **Solution** object). This precludes the solution of linked analyses or multiple convergence loops automatically on a single click of the **Solve** button. This is the default and enables you to close the Mechanical editor or solve an unrelated analysis. See the Understanding Solving (p. 1791) help section for additional information.
OK	Selecting the **OK** button commits all changes in the **Solve Process Settings** dialog box and closes the dialog box. You must choose **OK** for the Solve Process Setting configurations to be used when you initiate the solve.
Cancel	Selecting the **Cancel** button closes the dialog box and ignores all changes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Memory Tuning the Samcef Solver

Memory settings for the Samcef solver are specified using the **Manually specify Mechanical APDL solver memory settings** option. The default value for the **Workspace** setting is 600 MB, and the default value for **Database** is 300 MB. The amount of memory allocated to the Samcef solver is determined by the following rules:

- The value of the **Workspace** option is the total memory available to the solve process.
- The **Database** memory cannot exceed 35% of the total system memory of the node. If it exceeds this value, it is reset to 35% of the total system memory.
- If the processor architecture is 32-bit, the total memory available to the solve process cannot exceed 1 GB.
- If **Database** is 0, Workbench automatically sets it to 300 MB.
- If **Workspace** is 0, Workbench automatically sets it to twice the **Database** setting.

Before launching the Samcef executable, the **SAM_ZONE** environment variable is set to the value of the total memory (**Workspace**) in words, where 1 MB = 125000 words. The **SAM_DBRAM** environment variable is set to the ratio (in percent) between the **Database** and **Workspace** setting. The default value is 50%.

Memory Tuning the ABAQUS Solver

Memory settings for the ABAQUS solver are specified using the **Manually specify Mechanical APDL solver memory settings** option. The default value for the **Workspace** setting is 512 MB, and the default value for **Database** is 256 MB. The amount of memory allocated to the ABAQUS solver is determined by the following rules:

- The value of the **Workspace** option is the total memory available to the solve process.
- The **Database** memory cannot exceed 35% of the total system memory of the node. If it exceeds this value, it is reset to 35% of the total system memory.
- If the processor architecture is 32-bit, the total memory available to the solve process cannot exceed 1 GB.

Workbench tunes ABAQUS memory by changing keywords and values in the ABAQUS_v6.env file in the solve working directory:

- If the ABAQUS release number is less than 6.8, the keyword `pre_memory` is set to the **Database** value and the keyword `standard_memory` is set to the **Workspace** value.
- If the ABAQUS release number is greater than or equal to 6.8, the keyword `memory` is set to the **Workspace** value and the **Database** value is ignored.

Workbench does not change the memory values if they are out of bounds for the ABAQUS solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using Solution Restarts

Note:

Solution Restarts are supported in Coupled Field Analyses (p. 376), Static Structural (p. 652), and Transient Structural (p. 665) analyses only. However, there are exceptions for these analysis types. Restarts are **not** supported for a:

- Purely electrostatic Coupled Field analyses.
- Static Structural analysis when computing fracture parameters with **SMART Crack Growth**. See the Computation of Fracture Parameters (p. 483) discussion in the *Solving a Fracture Analysis* section for more information.

The solution process is composed of a sequence of calculations that predict a structure's response when applied to a specific analysis type and loading condition. Restarts provide the ability to continue an initial or existing solution which can save time during the solve phase. This feature facilitates a variety of workflows, which include:

1. Pausing or stopping a job to review results and then restarting the job.
2. Review and correction of a non-converging solution. Solution parameters in the analysis settings could be fine-tuned or adjusted allowing the solution to proceed while retaining prior solution progress. Similarly a load history can be modified to aid in the convergence.
3. Extending a solution that has already completed, for example, to allow system transients to progress further into time.
4. Submitting post processing instructions into Mechanical APDL after the model has been fully solved (see below).

The following topics are covered in this section:

- Restart Points (p. 1816)
- Generating Restart Points (p. 1816)
- Retaining Restart Points (p. 1816)
- Viewing Restart Points (p. 1816)
- Using Restart Points (p. 1817)
- Deleting Restart Points (p. 1818)
- Modifications Affecting Restart Points (p. 1818)
- Loads Supported for Restarts (p. 1822)
- Conditions Supported for Restarts (p. 1823)
- Solution Information Files During Restart (p. 1823)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

• Restarts and Distributed-Memory Solutions (p. 1823)

Restart Points

Solution restarts are based on the concept of a restart point. Each restart point can be considered as a snapshot of the system solution state at a discrete point along the sequence of calculations. The solver stores this state of the solution in a restart file on disk. Every restart file on disk will have a corresponding restart point in Mechanical. See Viewing Restart Points (p. 1816) below.

A solution can only be restarted from an available restart point. It is thus important to understand how to work with these restart points.

Generating Restart Points

Restart points are automatically created by Mechanical depending on the analysis type. The program controlled option will create one restart point at the last successful solve point for a nonlinear analysis. However, you may directly control their frequency to alter the balance between flexibility and disk usage with the Restart Controls (p. 1438) group of the Analysis Settings object. Restart points could be generated at all substeps or specific substep intervals in the analysis or at none at all.

Note:

- You can manually interrupt a solution and preserve any restart points that may have been produced from a converged iteration by clicking the **Interrupt Solution** button on the **Solution Status** window.
- A stand-alone linear analysis will not produce any restart points with the program controlled option. It has to be explicitly turned on using the manual setting. However, if the analysis is linked to a follow on modal analysis, it will generate restart points by default.

Retaining Restart Points

An incomplete solution (for example, a convergence failure) will always retain the restart points. However, for a complete solution, this is controlled by **Retain Files After Full Solve** property located in the Details view of **Analysis Settings** under **Restart Controls**. This property is set to **No** by default and hence will delete all restart points after the solution is completed. It can be set to **Yes** which will retain the restart files for the current project.

You can change the default setting of the **Retain Files After Full Solve** property via the **Options** dialog. Select the **Options** dialog option from the **File** tab to open the dialog. Select **Analysis Settings and Solution** and under the **Restart Controls** category you can change the default setting.

Viewing Restart Points

Once restart points are generated, they will be visible in several forms. For an overview, select the **Analysis Settings** object and refer to the **Graph** window where restart points are symbolized by triangular markers atop the timeline. The **Tabular Data** window lists the restart points within each load step.

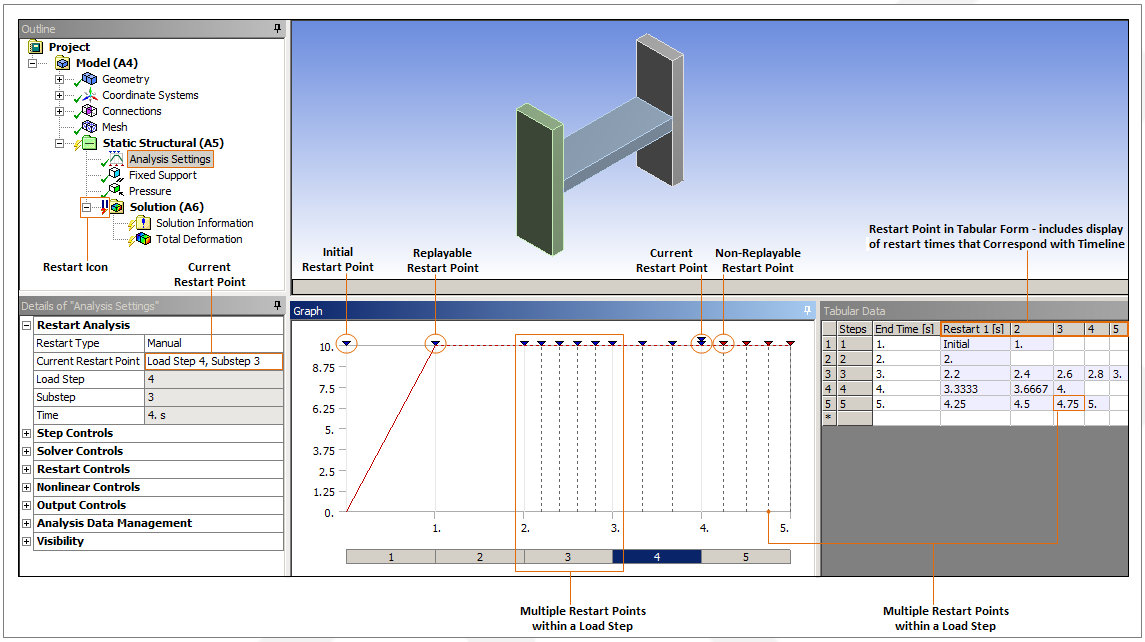
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

A restart point is color coded to distinguish between replayable and a non-replayable. A replayable solution is one which will produce the exact solution when run from start to finish or completed incrementally using intermediate restart points. A blue triangle indicates a replayable restart point. A red triangle indicates a potentially non-replayable restart point and can only be used in manual mode.

Note:

The Initial Restart Point does not represent a restart file on disk. It is only a place holder to facilitate selection to run the solution from the beginning even when other restart points are available.



Using Restart Points

You can manually choose the restart point to be used in a solution. Alternatively, you can configure Mechanical to suggest one for you. To allow Mechanical to automatically select a restart point, set **Restart Type** to **Program Controlled**. If you prefer a different point, you may specify it directly by setting **Restart Type** to **Manual** and by:

- Choosing **Current Restart Point** in the Details view of the **Analysis Settings** object.
- Selecting the desired marker on the **Graph** window and choosing **Set Current Restart Point** in the context menu.
- Selecting the desired cell in the **Tabular Data** window and choosing **Set Current Restart Point** in the context menu.

The **Current Restart Point** in the **Restart Analysis** group of the **Analysis Settings** object will indicate which restart point will be used the next time a solution is attempted. The current restart point in the graph/timeline window will be denoted with a double triangle in the timeline.

The program controlled setting takes a conservative approach to guarantee a replayable solution and will always select the last replayable restart point. In manual mode, the software will not automatically

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

change the current restart point and has to be selected explicitly. Picking a non-replayable restart point in manual mode is only recommended for experienced users who understand the implications of the results produced.

Mechanical automatically tracks how restart points are affected as you work and modify your model. So they may get flagged as non-replayable (red triangle) or be removed altogether depending on the operation. See Modifications Affecting Restart Points (p. 1818) for details.

Also see Restart Analysis (p. 1437) under Configuring Analysis Settings (p. 1417).

Note:

- An analysis should use the same units (set at the beginning of a solve) throughout the solve including all restarts. If the units are changed at any restart point, the solve is aborted and an error message is displayed.
- Named Selections (p. 1057) created/modified following the solution process are not recognized during a restart. For example, you may wish to list the nodes of a newly created Named Selection using the Command (p. 2303) feature. Because the Named Selection's geometric data was not defined during the initial solution process, no data is available for the command to process.

Deleting Restart Points

In order to delete existing restart points, you may use the **Delete All Restart Points** in the context menu at the **Environment** and **Solution** folders. For more granularity, one or more restart points may also be deleted by selecting them on either the **Graph** or Tabular Data windows and issuing **Delete Restart Points**.

Note:

The **Clear Generated Data** option in the context menu from either the **Solution**, **Environment**, **Model** or **Project** objects also **deletes all restart points**.

Modifications Affecting Restart Points

The following table summarizes the effects of making changes to the controls of the Analysis Settings object and the effects on restart points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If a change is made to one of the following Controls…		Then…				
		All Restart Points are Deleted	Current Restart Point is Set to Initial[a]	Current Restart Point set to the Beginning of the Modified Load Step	Non-replayable Restart Points may be Available[b]	Restart Points are Unaffected
Step Controls	Step End Time [c]			X		
	Auto Time Stepping			X	X	
	Define By			X	X	
	Carry Over Time Step			X	X	
	Time Integration			X	X	
Solver Controls		X				
Rotordynamics Controls		X				
Restart Controls						X
Restart Analysis						X
Non Linear Controls				X	X	
Output Controls[d]	Stress					X
	Strain					X
	Nodal Force					X
	Contact Miscellaneous					X
	General Miscellaneous					X
	Store Results At			X	X	
Damping Controls		X				
Analysis Data Management	Save MAPDL dB					X
	Delete Unneeded Files					X
	Solver Units	X				

[a] Restart Type specified as Program Controlled.

[b] It can only be selected when Restart Type is specified as Manual.

[c] When the Step End Time option in the Step Controls category is changed, the restart point is deleted as well as all the steps after this modified restart points are deleted and are not available, not even for manual restarts. Exception is the case when Fluid Solid Interface load exists and all the restart points are retained.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

[d] It is recommended that you not change Output Controls settings during a solution restart. Modifying Output Controls settings changes the availability of the respective result type in the results file. Consequently, result calculations cannot be guaranteed for the entire solution. In addition, result file values may not correspond to GUI settings in this scenario. Settings turned off during a restart generate results equal to zero and may affect post processing of results and are therefore unreliable.

Note:

If you 1) restart a solution from a time point that is within a load step and 2) the **Auto Time Stepping** property is set to **Program Controlled**, the application will include the Substep/Time Step details for that load step based on the prior solution that was performed without restarts.

The following table summarizes the effects of step modifications on restart points.

If a change is made to one of the following Controls…	**Then…**		
	All Restart Points are Deleted	**Current Restart Point is set to the Beginning of the Modified Load Step**	**Non-replayable Restart Points may be Available**[b]
Activate/Deactivate		**X**	**X**
Add Step/Insert Step		**X**	
Delete Step	**X**		

A solution can be restarted after modification to the load history. However, any other changes to the definition delete all of the Restart Points.

Note:

- **Displacements**, **Remote Displacements**, and **Nodal Displacements** only support **Tabular** data modifications. See the Loads Supported for Restarts (p. 1822) topic for a detailed list.
- Changing a **Displacement** boundary condition may cause the program to return to the initial restart point, depending upon the change you make. The restart point where the change occurred is maintained - not deleted. For example, changing the magnitude of either of these loads from a zero value to a non-zero value, or vice versa, prompts the application to return to the beginning of the solution process. Similarly, if you change the independent time value of either load, the solution process restarts from the beginning.

If a change is made to one of the following Controls…		**Then…**		
		Current Restart Point is Set to Initial[a]	**Current Restart Point is set to the Beginning of the Modified Load Step**	**Non-replayable Restart Points may be Available**[b]
Modify Load History	Constant	**X**		**X**
	Tabular		**X**	**X**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

	Function	**X**		**X**
Change Load Type (Constant, Tabular, Function)		**X**		**X**

[a] Restart Type specified as Program Controlled.

The following table summarizes the effects of adding/modifying/deleting a **Commands** (p. 2303) object. When Restart Points are available, adding a new **Commands** object defaults to the last step so as to preserve the Restart Points. Adding a **Commands** object without Restart Points defaults to first step.

If a change is made to one of the following Controls…		**Then…**			
		All Restart Points are Deleted	**Current Restart Point is set to the Beginning of the Modified Load Step**	**Non-replayable Restart Points may be Available**[b]	**Restart Points are Unaffected**
Add/Modify/Delete Command Snippets	Under Environment		**X**	**X**	
	Under Solution/Results				**X**
	Under Model/Trunk Objects	**X**			

Modifications such as adding or changing boundary conditions (for example, scoping changes), constraints, initial conditions, or editing model level objects (**Geometry**, **Contact Region**, **Joint**, **Mesh**) *invalidates* and *deletes* existing Restart Points. Exceptions include Direct FE (p. 1719) loads with a zero magnitude or a Pressure (p. 1521) with a zero magnitude. The restart points are retained.

If a change is made to one of the following Controls…		**Then…**	
		All Restart Points are Deleted	**Restart Points are Unaffected**
Add/Delete Boundary Condition		**X**	
Add/Delete Direct Boundary Condition	Force (zero)		**X**
	Force (non zero)	**X**	
	Pressure (zero)		**X**
	Displacement	**X**	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a change is made to one of the following Controls…		Then…	
		All Restart Points are Deleted	Restart Points are Unaffected
Model Level Changes		X	

Note:

Restart is not supported for an analysis with Adaptive Convergence (p. 1852). So the presence of an adaptive convergence will not retain any restart points.

Loads Supported for Restarts

The following table outlines which loads may be modified for a solution restart.

Load Type	Load Specified As...		
	Constant	Tabular	Function
Pressure	X	X	X
Line Pressure	X	X	X
Force	X	X	X
Remote Force* (p. 1823)	X	X	X
Moment* (p. 1823)	X	X	X
Displacement	X	X	N/A
Remote Displacement* (p. 1823)	X	X	N/A
Rotational Velocity	X	X	X
Rotational Acceleration	X	X	X
Bolt Pretension	X	X	N/A
Acceleration	X	X	X
Earth Gravity	N/A	N/A	N/A
Hydrostatic Pressure	X	N/A	N/A
Bearing Load	X	X	N/A
Joint Load	X	X	X
Pipe Temperature	X	X	X
Pipe Pressure	X	X	X
Thermal Condition	X	X	X
Imported Load	N/A	N//A	N/A
Nodal Force	X	X	X
Nodal Pressure	X	X	X

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Nodal Displacement	**X**	**X**	**N/A**

Note:

The Remote Force, Moment, and Remote Displacement boundary conditions support restarts **only** when scoped to a Remote Point.

Conditions Supported for Restarts

The **Nonlinear Adaptive Region** (p. 1707) condition supports restarts. Adding a **Nonlinear Adaptive Region** condition to a non-convergent and/or terminated solution may enable the solution to converge.

Solution Information Files During Restart

During a restart, solution information files (input file `ds.dat` and output file `solve.out`) from the previous solve are retained for reference by renaming it just before the restart solve is initiated. The naming convention is `filename_loadstep_substep.ext`. For example, if the previous solve occurred at loadstep = 2 and substep = 5, the file name would be `ds_2_5.dat` and `solve_2_5.out`. Files from the initial solve will be named `ds_0_0.dat` and `solve_0_0.dat`. Based on the restart point, Mechanical will ensure that obsolete and invalid solution files are cleaned up.

Restarts and Distributed-Memory Solutions

For a distributed solution, the application merges all individual result files into a single file at the end of the solution so that Mechanical can postprocess the results or continue a restarted solution. If you experience a solution failure, or if the solution is stopped, before the files have been combined, you can manually recombine the files by executing the **COMBINE**, **RST** command from the /AUX2 module in Mechanical APDL.

Note:

The current working directory must include all of the local (`file.rst`) files.

Distributed Ansys needs to be launched with the same number of processes that were used when generating the local results files.

Following the execution of the **COMBINE** command, the working directory includes a complete results file (`file.rst`) and you may use it for any additional Mechanical post-processing or subsequent restarts.

Restart Analysis Extension

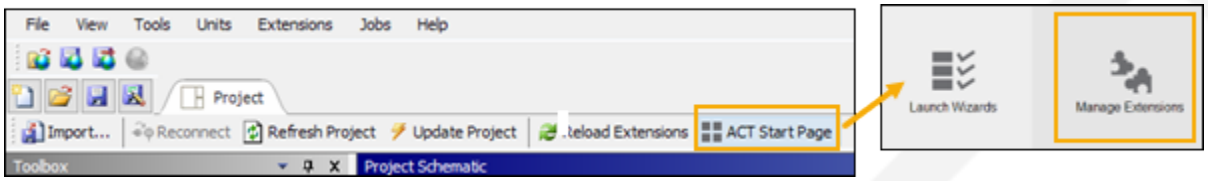
The Restart Analysis extension makes Static-Static or Transient-Transient Structural restart capabilities available within the Mechanical environment.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

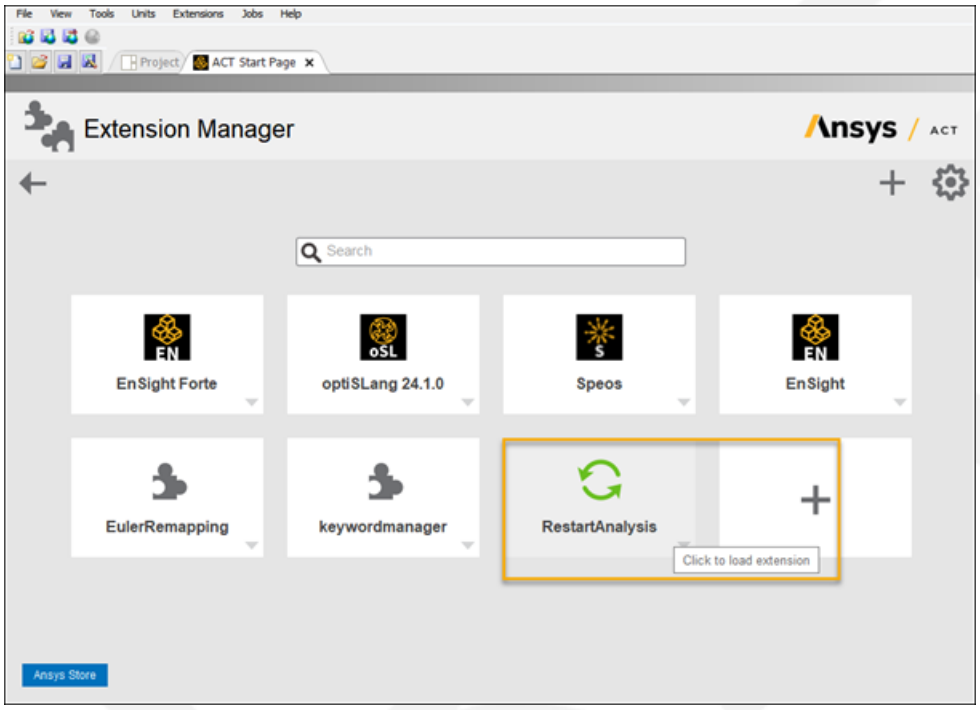
Loading the Restart Analysis Extension

This extension must be loaded first from the ACT Start Page. Follow these steps to load the extension correctly:

1. In the Workbench Project dialog, click **ACT Start Page** and select **Manage Extensions**.



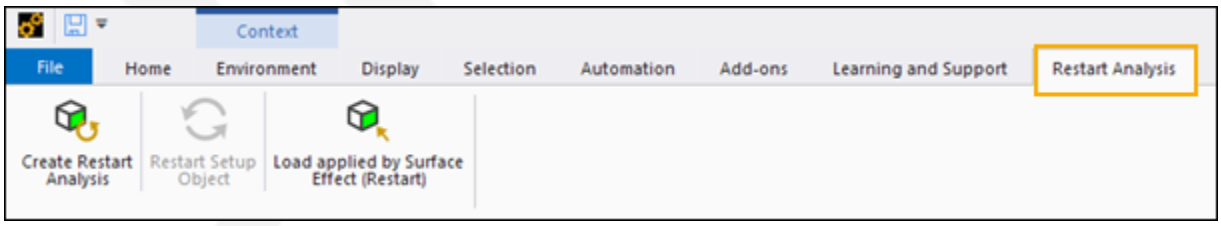
2. Click the **RestartAnalysis** tile to load the extension.



Note:

To unload it, simply click this tile again.

Then, a new **Restart Analysis** tab will appear in Mechanical.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Creating a Restart Analysis

Creating a Restart Analysis produces a new analysis system in Mechanical, which can be restarted from a selected Base Analysis system by following these steps:

1. Select any Static or Transient Structural analysis system in the tree view in Mechanical.
2. Click **Create Restart Analysis** in the Restart Analysis toolbar. If all criteria are satisfied (i.e., Restart Files must be retained, etc.), a new analysis system is created with a **Restart Setup** object already added to it. Otherwise, set the options in the Base Analysis. The new analysis system is created with all the appropriate settings. For more information, see Restart Analysis System Criteria (p. 1828).
3. Add your loads and boundary conditions. For more information on acceptable loads and boundary conditions, see Limitations for Loads and Boundary Conditions (p. 1829).
4. Configure the settings in the Details panel of the **Restart Setup** object.

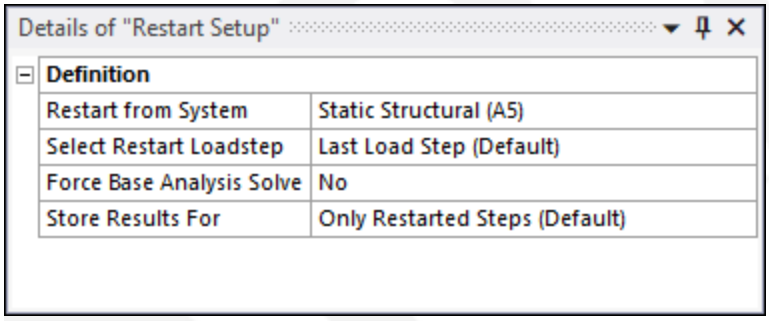
Restart Setup Object

This object is used for an existing analysis system that you want to use as the Restart system.

Restart the setup by following these steps:

1. Select your chosen analysis system.
2. Click **Restart Setup Object** in the Restart Analysis toolbar to insert this object under the selected analysis and proceed with restarting the analysis.

These properties are available for the **Restart Setup** object:



Restart from System

Choose any valid analysis system from the drop-down menu in this property. The analysis list is filtered based on the Restart Analysis type. If the Restart Analysis system is Static Structural, only the remaining Static Structural systems will appear in the list.

To perform a Static-Transient or Transient-Static restart, use Transient Structural systems for both the Base and Restart Analyses and set **Time Integration** to either **On** or **Off** as appropriate.

Select Restart Loadstep

This property has two options:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Last Load Step (Default)**: This is the default option. It uses the last load step of the selected Base Analysis to restart the analysis.
- **Manual**: If there are restart points available at multiple load steps in the Base Analysis, use this option to manually specify a load step number to restart from.

Force Base Analysis Solve

No is the default option for this property. When solving the Restart Analysis, selecting **Yes** forces the Base Analysis to solve before starting the Restart Analysis solution. This option is recommended only for batch analyses such as a Design Point study.

Store Results For

This property has two options:

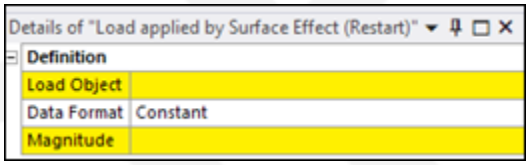
- **Only Restarted Steps (Default)**: When this option is selected, only the results for the restarted steps are saved and available for post-processing.
- **Base + Restarted Steps (Full Time History)**: When this option is selected, all the load step results are saved, that is, both the Base Analysis steps and restart load steps. This increases the size of the results file for the Restart Analysis.

Load Applied by Surface Effect (Restart)

Adding loads that create new elements is not permitted when using the Restart Analysis extension. An example is loads such as Pressure or Force when **Applied By** is set to **Surface Effect** instead of **Direct**.

If the Base Analysis has loads (Pressure or Force) applied by the **Surface Effect** option, these loads can be further modified in the Restart Analysis by following these steps:

1. Select the analysis system where the restart is being performed.
2. Click **Load Applied by Surface Effect (Restart)** in the Restart Analysis toolbar, or right-click the analysis system and select **Insert** > **Load Applied by Surface Effect (Restart)**.
3. Configure the settings in the Details panel:



Each property is described below:

Load Object

This contains a drop-down menu of all the Pressure and Force loads present in the selected Base Analysis (the analysis selected in the **Restart From System** property in the **Restart Setup** object). Only Pressure and Force loads with the **Applied By** property set to **Surface Effect** and **Magnitude** defined as **Constant** or **Tabular** are supported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Data Format

It has two options:

- **Constant**: This is the default option. It applies a constant load ramped over the Restart load step.
- **Tabular**: This option defines the Time vs Load data.

Magnitude or Component X,Y, and Z

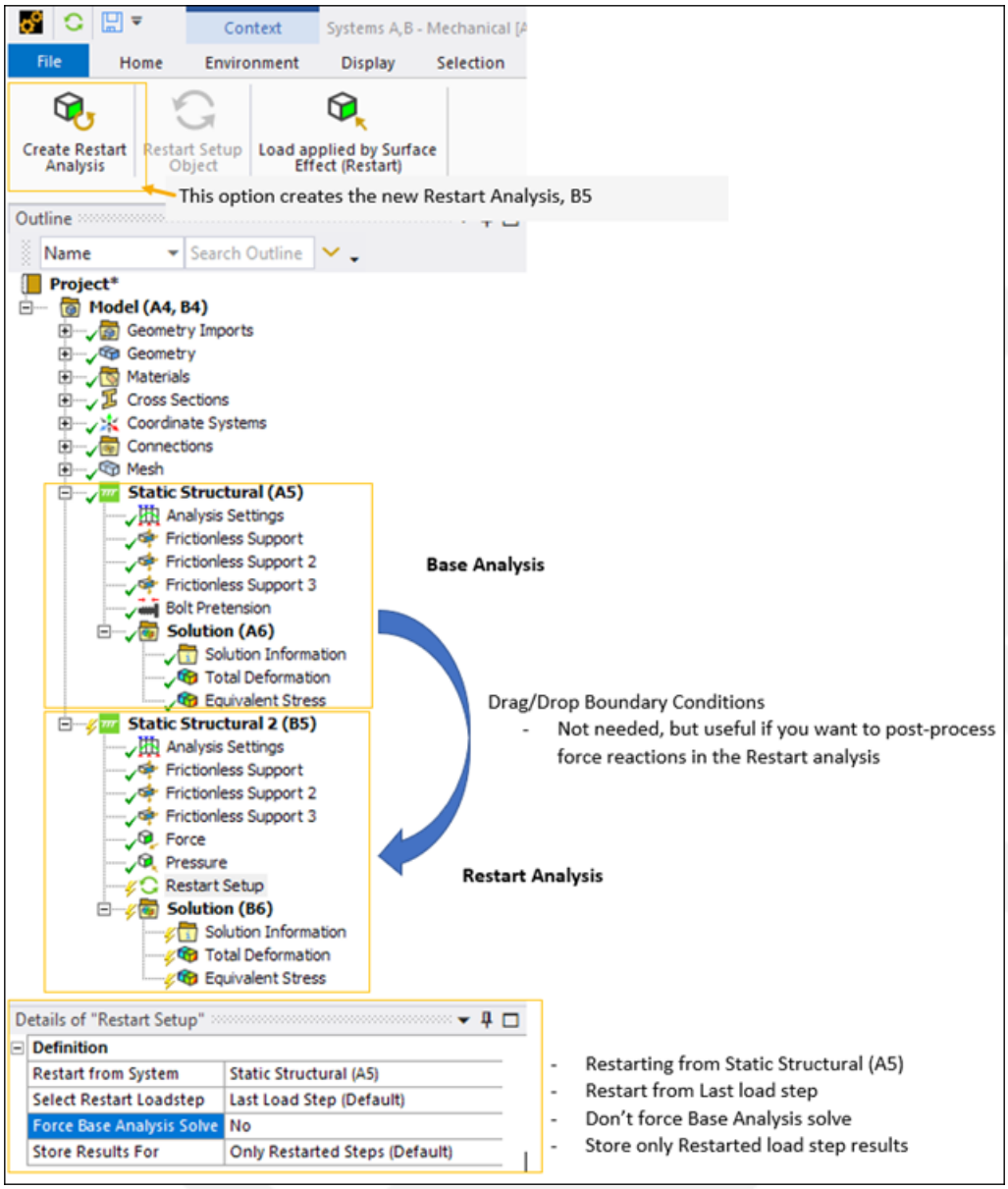
Depending on whether the **Define By** property in the Details panel of the load (Pressure or Force) is set to **Components**, **Vector**, or **Normal To**, this property can be set to **Magnitude** (for **Define By** set to **Vector** or **Normal To**) or **X Component**, **Y Component**, and **Z Component** (for **Define By** set to **Components**).

Restart Workflow Example

Below is one of the typical workflows that the Restart Analysis extension is used for. Here, a Bolt Pretension load (2 load steps—Bolt load + Bolt lock) is present in the Base Analysis. In the restarted system (i.e., B5), the Base Analysis is restarted, and Force and Pressure loads are applied in Load step 3.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Restart Analysis System Criteria

The criteria below must be met for the Restart Analysis. If you use **Create Restart Analysis** to create the Restart Analysis directly, all the criteria are automatically satisfied. If you use the **Restart Setup** object and all the criteria are not met, warning messages are issued, and the **Restart Setup** object will be invalidated.

- As the Restart Analysis continues from the Base Analysis, more steps (load steps) are required in the Restart Analysis compared to the Base Analysis.
- For Static-Static restarts, the Base and Restart Analyses must have identical Large Deflection, Newton Raphson, Solver Type, and Quasi-Static settings under Analysis Settings.
- For Transient-Transient restarts, in addition to the preceding criteria, Application Based Settings and Damping Controls must be identical between the Base and Restart Analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The step end times and substep settings (of the Base Analysis) are retained in the Restart Analysis.
- Weak springs are not permitted in the Restart Analysis.
- If the Base Analysis is not solved, the **Solver Unit System** setting (under Analysis Settings) is the same for the Base and Restart Analyses. If the Base Analysis is solved, the Solver Unit System setting for the Restart Analysis is the same as the unit system in which the Base Analysis is solved (irrespective of the active unit system in Mechanical).
- Contact Split is not permitted in the Restart Analysis.

General Limitations

- This extension is available for connected Static-Static or Transient-Transient Structural Analysis only. That is, they share the mesh and appear within the same Mechanical window.
- It is compatible with the Windows operating system only.
- Distributed Compute Services (HPS) and Multistage Cyclic Symmetry are not supported by the Restart Analysis extension.
- When the **Only Restarted Steps (Default)** option is used during post-processing and you attempt to retrieve the result at a time point corresponding to the Base Analysis steps, the result corresponding to the nearest available time point in the solved model is displayed.
- When a Modal analysis is restarted from Load Step 2 or higher, it does not activate certain elements in the presence of Element Birth and Death or Contact Step control in the pre-stress Static Structural analysis.
- If **Retain Files After Full Solve** under Analysis Settings of the Restart Analysis is set to **Yes**, set **Combine Restart Files** to **Yes**.

Limitations for Loads and Boundary Conditions

Any loads and boundary conditions that create new elements are not permitted during the restart, as new elements cannot be created in a Restart Analysis. However, all loads and boundary conditions applied in the Base Analysis are still active by default in the Restart Analysis. Consider these limitations:

- Force, Pressure, Imported Pressure, or Hydrostatic Pressure loads are permitted only with the **Applied By** option set to **Direct**, unless a Nonlinear Adaptive Region is defined in the Base Analysis. However, using the **Load Applied by Surface Effect (Restart)** option modifies the existing surface-based Force or Pressure loads in the Base Analysis.
- Force loads scoped to vertices are not permitted. As an alternative, use Nodal Force to apply Force loads on a nodal-named selection.
- Remote loads (Remote Force, Remote Displacement, and Moments) are permitted only with the **Remote Point** scoping method.
- Elastic Support, Compression Only Support, Bearing Load, Nodal Orientations, Imported Initial Stress or Strains, and Line Pressure objects are not permitted in the Restart Analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Modifying Bolt Pretension loads in the Restart Analysis is not supported. For evaluating Bolt results using the Bolt Tool, drag and drop the Bolt Pretension loads from the Base Analysis onto the Restart Analysis. Do not change the Scoping or Formulation of the Bolt Pretension objects. The **Define By** property will be set to **Open** for clarity.
- Nonlinear Adaptive Regions are permitted in the Base Analysis only. However by default (the **Time Range** property set to **Entire Load Step**), a Nonlinear Adaptive Region is active for all the load steps in the Base and Restart Analyses. To make a Nonlinear Adaptive Region active for specific load steps only (for instance, activating it for the Base Analysis only), use the **Time Range** property.

  If a Nonlinear Adaptive Region is present in the Base Analysis:

  – Imported loads are not permitted in the restarted analysis.

  – Nodal Force, Nodal Pressure, and Nodal Displacement objects are not permitted in the Restart Analysis, if the scoping belongs to the bodies scoped in the Nonlinear Adaptive Region objects in the Base Analysis.

  – Only the **Base + Restarted Steps (Full Time History)** option is permitted.

  – If the **Time Range** of the Nonlinear Adaptive Region objects in the Base Analysis is set such that remeshing is also active in the Restart Analysis, Element Birth and Death or Contact Step Control objects are not permitted in the Restart Analysis.
- Element Birth and Death and Contact Step Control are permitted in the Base and Restart Analyses, unless **Contact Split** is active in the Base Analysis.

Note:

Do not set the status (under **Step Controls**) for either **Element Birth and Death** or **Contact Step Control** objects to **Dead**, if the load step number is less than or equal to the Restart load step number (in the **Restart Setup** object). This is an interface limitation that has no effect on the solution. The **Status** setting is only considered for a load step number greater than the Restart load step number.

Restrictions on Displacements

If a Nonlinear Adaptive Region is present in the Base Analysis, displacements are permitted with the following restrictions:

- If Displacement boundary conditions (in the Base or Restart Analysis) that are scoped to geometric entities (vertices, edges, or faces) belonging to the bodies scoped to a Nonlinear Adaptive Region in the Base Analysis are present, only Named Selection Scoping is permitted.
- Multiple Displacement objects cannot be applied using the same Named Selection.
- Displacement objects cannot be applied in a Restart Analysis alone. If a Displacement is applied in the Restart Analysis, a corresponding Displacement (using the same Named selection scoping) must be applied in the Base Analysis. This also applies to component Displacements (i.e., a component cannot be Free in the Base Analysis and loaded in the Restart Analysis).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- For Displacements, only Tabular Loading is permitted. In the Details panel of Displacement objects, only setting **Independent Variable** as **Time** is permitted (i.e., step loading or function loading is not permitted).

Understanding Solve Scenarios

This section describes the various configuration steps involved for the following solving scenarios:

- Solve on the Local Machine within the Workbench process (synchronous) (p. 1831)
- Solve on My Computer in the Background (asynchronous) (p. 1831)
- Solve using Distributed Compute Services (HPS) (p. 1831)
- Solve Using Remote Solve Manager (RSM) (p. 1831):
  - Solve to a Remote Windows Cluster (p. 1831)
  - Solve Directly from My Computer to a Remote Linux Cluster (p. 1832)
  - Solve to LSF Cluster with Remote Solve Manager (p. 1832)
  - Solve to Microsoft HPC Cluster with Remote Solve Manager (p. 1833)

Solve on the Local Machine within the Workbench process (synchronous)

- Use the built-in **My Computer** solve process setting (p. 1804).

Solve on My Computer in the Background (asynchronous)

- Use the built-in **My Computer, Background** solve process setting (p. 1804). The option is only functional if Remote Solve Manager (RSM) was installed along with Workbench. RSM has a built-in "Local" queue and server for running jobs on the client computer.

Solve using Distributed Compute Services (HPS)

Use the options of the Solve Process Settings (p. 1804) dialog to select HPS as your target and specify the settings for the remote location.

Solve Using Remote Solve Manager (RSM)

The following scenarios describe how to submit to a cluster using RSM.

Solve to a Remote Windows Cluster

Important:

RSM must be installed on your local computer. The RSM launcher service must be installed and running on the remote cluster submit host. See Installing and Configuring the RSM Launcher Service for Windows in the *Distributed Compute Gateway User's Guide*. The

Mechanical application must be installed on your local computer as well as the cluster execution node that will be used for the solve.

This step requires the following configuration steps:

1. Define a cluster configuration for the Windows cluster in RSM. Or, if an RSM administrator has already defined a cluster configuration, load it into your local installation of RSM. See Sharing and Accessing Cluster Configurations in the *Distributed Compute Gateway User's Guide*. The cluster configuration will contain RSM queues that map to HPC queues.

   RSM will use your account credentials to submit the job on your behalf to the cluster submit host.

   If jobs will be submitted to a remote Microsoft HPC cluster, refer to Integrating RSM with a Microsoft HPC Cluster for additional considerations.

2. Create a solve process setting that specifies the cluster configuration and RSM queue defined in step 1 (see Using Solve Process Settings (p. 1804)).

3. Use the **Solve Process Setting** created in step 2 using the **Solve** drop-down option.

Solve Directly from My Computer to a Remote Linux Cluster

Important:

RSM must be installed on your local computer. The RSM launcher service must be installed and running on the remote cluster submit host. See Installing and Configuring the RSM Launcher Service for Windows in the *Distributed Compute Gateway User's Guide*. The Mechanical application must be installed on your local computer as well as the cluster execution node that will be used for the solve.

This step requires the following configuration steps:

1. If your IT policy requires it, you may need to configure RSM to use SSH for job submission to a remote Linux cluster.

2. Define a configuration for the Linux cluster in RSM. Or, if an RSM administrator has already defined a configuration, load it into your local installation of RSM. See Sharing and Accessing RSM Configurations in the *Distributed Compute Gateway User's Guide*. The configuration will contain RSM queues that map to cluster queues.

3. Create a solve process setting that specifies the RSM configuration and RSM queue defined above (see Using Solve Process Settings (p. 1804)).

4. Use the **Solve Process Setting** created above using the **Solve** drop-down option.

Solve to LSF Cluster with Remote Solve Manager

The configuration from a Mechanical application user perspective is the same as above. A Solve Process Setting is required that specifies an LSF cluster configuration and queue to which the solve will be submitted. For RSM and cluster configuration information, see RSM Configuration and RSM Integration with a Cluster or Cloud Portal in the *RSM User's Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Solve to Microsoft HPC Cluster with Remote Solve Manager

The configuration from a Mechanical application user perspective is the same as above. A Solve Process Setting is required that specifies an RSM queue to which the solve will be submitted. For RSM and cluster configuration information, see RSM Configuration and Integrating RSM with a Microsoft HPC or Windows-Based Cluster in the *RSM User's Guide*.

Specifying Solution Information

You can track, monitor, or diagnose problems that arise during any solution as well as view certain finite element aspects of the engineering model, using a **Solution Information** object, which is inserted automatically under a **Solution** object of a new environment or an environment included in a database from a previous release. You can also manually insert a **Solution Information** object under a **Connections** object for solver feedback.

This section examines the following uses of the feature:

- Specifying Solution Information Properties (p. 1833)
- Viewing Finite Element Connections (p. 1840)

Specifying Solution Information Properties

When you select a **Solution Information** object in the Outline, the following properties are available in the **Details** pane under the **Solution Information** category:

Solution Output

The **Solution Output** property defines how you want solution response results to display. All of the options, described below, display results in real time as the solution progresses.

Important:

The **Solution Output** property is **not** applicable to the **Connections** object.

- **Solver Output** (default): Displays the solution output file (in text format) from the appropriate solver (Mechanical APDL, Explicit Dynamics, etc.). This option is valuable to users who are accustomed to reviewing this type of output for diagnostics on the execution of their solver of choice. During the solution process, the default behavior of the solver output display is to scroll to the bottom of the log. However, if you scroll the log to another position, then the application maintains that view position. You resume automatic scrolling by pressing the **End** key.

Note:

The text contents may be incomplete for a distributed solution involving Harmonic analyses or analyses featuring Cyclic Symmetry. See the **DDOPTION** command for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Solution Statistics**: This option is supported for all Mechanical analysis systems using Mechanical APDL solver and for LS-DYNA analysis systems. It becomes available in the property drop-down list once you have completed your solution. Selecting this option displays the **Worksheet**. The **Worksheet** displays statistical information about your completed solution process as well as recommendations about how to improve the process. Using this information, you can make decisions to improve your solution time and performance. For example, you may decide to use a different solver or to change the number of cores being employed.

- **Solution History**: This option is supported for all Mechanical analysis systems using Mechanical APDL solver. When selected, the **Solution History** option displays the Worksheet that contains two tabs to review information the solutions performed and about the evaluated results: **Solution Tracking** and **Result Tracking**. The **Solution Tracking** tab tracks information such as time to solve as well as the number of nodes and elements for each solution. The **Result Tracking** tab tracks a result's minimum and maximum values for each solution. Most common result types are supported for this feature, such as Total Deformation, stress, strain, etc.

  Each tab presents a chart that graphs the data based on the data columns you select in the table. Each table heading provides a filter as well as a checkbox. Only two check boxes can be selected at one time. You can change chart display settings (lines or columns) as well as the labeling by right-clicking in the chart. In addition, you can right-click on a row and select the Delete option.

  Note:

  You can change the default settings for the number of the solutions to be tracked using the **Maximum Solutions to Store** property and turn result tracking off using the **Track Results** property in the **Analysis Settings and Solution** (p. 238) of the **Options** dialog.

- **Resource Prediction (p. 170)**: This option is available for Static Structural and stand-alone Modal analyses and displays resource prediction information in the Worksheet. Information includes time and memory predictions for the requested number of cores, charts of time and memory allocations for different core settings (maximum of 32), and suggestions, if applicable, such as whether an alternative solver type setting (structural only) improves resource allocations or whether there is insufficient memory.

Choosing any of the following options for the **Solution Output** property displays a graph of that option as a function of **Cumulative Iteration/Cycle** (availability depends on the solver).

- **Force Convergence** [1] (p. 1837)
- **Displacement Convergence** [1] (p. 1837)
- **Rotation Convergence** [1] (p. 1837)
- **Charge Convergence**[1] (p. 1837)
- **Moment Convergence** [1] (p. 1837)
- **Participation Factor Summary** (Modal analysis only): This option displays the **Worksheet** with the label **Participation Factor Summary**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Complex eigenvalues are available when the **Damping** property is set to **Yes** (except when the **Reduced Damped** Solver Type is selected and the **Store Complex Solution** property is set to **No**). When the **Damping** property is set to **No**, only the **Unsymmetric** Solver Type provides complex Eigenvalues.

When you have a complex eigenvalue solution, the **Participation Factor Summary** Worksheet displays Mode, Frequency, X Direction (Real), Y Direction (Real), Z Direction (Real), X Direction (Imaginary), Y Direction (Imaginary) and Z Direction (Imaginary). Otherwise the worksheet tables display Mode, Frequency, X Direction, Y Direction, Z Direction, Rotation X, Rotation Y and Rotation Z.

When you select **Participation Factor Summary** option, the property **Summary Type** also displays and its options include:

– **All** (default): All tables are displayed in the **Worksheet**, including **Participation Factor**, **Effective Mass**, **Cumulative Effective Mass Fraction**, and **Ratio of Effective Mass to Total Mass**.

– **Participation Factor**: Only the **Participation Factor** table is displayed in the worksheet.

– **Effective Mass**: Only the **Effective Mass** table is displayed in the worksheet.

– **Cumulative Effective Mass Fraction**: Only the **Cumulative Effective Mass Fraction** table is displayed in the worksheet.

– **Ratio of Effective Mass to Total Mass**: Only the **Ratio of Effective Mass to Total Mass** table is displayed in the worksheet.

Important:

The unit system for all of the data displayed in the **Participation Factor Summary Worksheet** is the unit system for the **Solver Unit System** property in the **Analysis Data Management** category of the **Analysis Settings** object.

Note:

– The **Participation Factor Summary** output option is not available when Cyclic Symmetry is active.

– If the Campbell Diagram (p. 1490) property is set to **On** in the Modal system's **Analysis Settings** (multi-step Modal analysis), the **Participation Factor Summary** reported in the Worksheet is for the last rotational velocity/load step.

- **Max DOF Increment**
- **Line Search**
- **Time**
- **Time Increment**
- **Maximum Residual Force**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Kinetic Energy** (Quasi-Static Solution (p. 1428) only)
- **Stiffness Energy** (Quasi-Static Solution (p. 1428) only)
- **CSG Convergence** [1] (p. 1837) (magnetic current segments)
- **Heat Convergence** [1] (p. 1837)
- **Energy Conservation**: Shows plots of total energy, reference energy, work done, and energy error.
- **Momentum Summary**: Shows plots of X, Y and Z momentum and X, Y and Z impulse for the model.
- **Energy summary**: Shows plots of internal energy, kinetic energy, hourglass energy and contact energy.
- **Post Output**: displays the output of a Post Command snippet (p. 2303).
- **Optimization Output**: This option displays the solution output (in text format) from the Topology Optimizer solver.

  Mechanical also provides the following convergence charts for the **Structural Optimization** analysis. These options provide plots (in the **Worksheet**) for convergence values determined during the solution. This is useful for determining if the simulation is nearing convergence. All of these options have a plot for the *combined* objective value versus the **Convergence Accuracy** as defined in the Definition Analysis Settings for the Topology Optimization environment. Note that individual objective values can occur separately in a multi-step analysis. These options also include a plot for the convergence of the Response Constraint that you wish to observe (Mass, Volume, etc.).

  - **Objective and Mass Response Convergence**: This option is the default option for a Structural Optimization analysis. This property plots the **Mass Response Convergence** against the criterion you specify in the **Percent to Retain** property of the **Mass Constraint** object (Response Constraint). The convergence chart will plot convergence against **Percent To Retain (Min)** and **Percent To Retain (Max)** criterion, if the constraint is defined by **Range**.
  - **Objective and Volume Response Convergence**: This property plots the **Volume Response Convergence** against the criterion you specify in the **Percent to Retain** property of the **Volume Constraint** object (Response Constraint). The convergence chart will plot convergence against **Percent To Retain (Min)** and **Percent To Retain (Max)** criterion, if the constraint is defined by **Range**.
  - **Objective and Global Stress Response Convergence** (Static Structural analyses only): Plots the stress response convergence against the criterion specified in the **Maximum** property of the Global Von-Mises Stress Constraint object. For multi-step analyses, the application provides fields to enable you to choose which Step Number corresponds to the stress constraint you wish to observe.
  - **Objective and Local Stress Response Convergence** (Static Structural analyses only): Plots the local stress response convergence against the criterion specified in the **Maximum** property of the **Local Von-Mises Stress Constraint** object. For multi-step analyses, the application provides fields to enable you to choose which **Step Number** corresponds to the stress constraint you wish to observe.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **Objective and Displacement Response Convergence** (Static Structural analyses only): Plots the displacement response convergence against the criterion specified in **X Component (Max)**/**Y Component (Max)**/**Z Component (Max)** of the **Displacement Constraint** object. For multi-step analyses, the application provides fields to enable you to choose which **Step Number** corresponds to the displacement constraint you wish to observe.

– **Objective and Reaction Force Response Convergence** (Static Structural analyses only): Plots the reaction force response convergence against the criterion specified in **X Component (Max)**/**Y Component (Max)**/**Z Component (Max)** of the **Reaction Force Constraint** object. For multi-step analyses, the application provides fields to enable you to choose which **Step Number** corresponds to the reaction force constraint you wish to observe.

– **Objective and Natural Frequency Convergence** (applicable for Modal analyses): Plots the Natural Frequency Response convergence against the criterion specified in the **Minimum Frequency** and **Maximum Frequency** properties of the **Natural Frequency Constraint** object (Response Constraint). The application provides a field to enable you to specify a **Mode Number** corresponding to a natural frequency range you wish to observe.

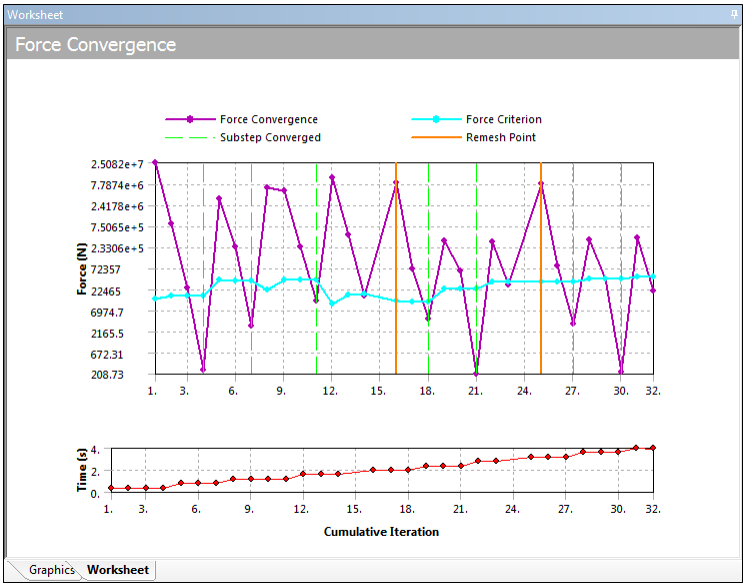
– **Objective and Manufacturing Convergence** (applicable when criterion is entered for the Maximum property of the Manufacturing Constraint object): Plots the manufacturing response convergence against the criterion specified in the **Maximum** property of the **Member Size** category of the **Manufacturing Constraint** object when the **Maximum** property is set to **Manual**.

Note:

- The frequency at which data is written can be specified as a time step frequency or a physical time frequency. By default information is displayed for every 100 time steps.
- For ease of viewing solutions with many substeps/iterations, the **Substep Converged** and **Load Step Converged** lines are not displayed when the number of lines exceeds 1000. Also, graphs are shown as lines only, rather than lines and points, when the number of *points* exceeds 1000.

1: All convergence plots include designations where any bisections, converged substeps, converged steps, or remesh points occur. These designations are the red, green, blue, or orange lines (solid or dotted) shown in the example below of a **Force Convergence** plot.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Newton-Raphson Residuals

This property is applicable only to **Structural** environments solved with the Mechanical APDL application. It specifies the maximum number of Newton-Raphson residual forces to return. The default is **0** (no residuals returned). You can request that the Newton-Raphson residual restoring forces be brought back for nonlinear solutions that either do not converge or that you aborted during the solution. The Newton-Raphson force is calculated at each Newton-Raphson iteration and can give you an idea where the model is not satisfying equilibrium. If you select 10 residual forces and the solution doesn't converge, those last 10 residual forces will be brought back. The following information is available in the Details view of a returned **Newton-Raphson Residual Force** object:

- **Results**: **Minimum** and **Maximum** residual forces across the model.
- **Convergence**: Global convergence **Criterion** and convergence **Value**.
- **Information**: Time and load step information.

These results cannot be scoped and will automatically be deleted if another solution is run that either succeeds or creates a new set of residual forces.

Charge Residuals

This property is available for all Coupled Field analyses when 1) the **Electric** property of the **Physics Region** is set to **Charge** and 2) the **Newton-Raphson Residual** property is set to a non-zero value. Option includes **Yes** and **No** (default). When set to **Yes**, the application creates Newton-Raphson residual objects for charge data, during nonlinear solutions, that either did not converge or because of an aborted solution. The number of residual charge objects is the same as the value entered in the **Newton-Raphson Residual** property.

Identify Element Violations

This property is a diagnostic tool that enables you to identify and view elements on your model that have failed to meet certain solver criteria during the solution process. The application generates an error message and creates Named Selection objects that contain the elements that violate the following criteria:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Distortion too large (HDST)
- Elements that contain nodes that have near zero pivots (PIVT) for nonlinear analyses
- Plastic/creep strain increment is too large (EPPL/EPCR)
- Elements for which mixed u-P constraints are not satisfied (MXUP) - mixed U-P option of 18x solid elements only
- Radial displacement (RDSP) is not converged

For the system generated Named Selections that are scoped to the failed elements, the application generated "name" includes a reference to the specific failed criterion, such as "HDST" for a distortion that is too large. These Named Selections are placed under the **Solution Information** object.

The default setting for this property is **0** (no violations are returned). This value can be set to an **n**, where **n** is an integer value greater than **0**. This value defines the last n solver iterations for which the failed elements are stored.

The system generated Named Selections behave as user-defined element-based named selections (p. 276) and as desired, you can scope results to these named selections. In addition, unlike other diagnostic features, these Named Selections are not automatically deleted or overwritten upon subsequent solutions. As needed, you need to delete then manually.

You can find additional details in the **Element Components That Violate Criteria** topic of the **NLDIAG** command section of the *Mechanical APDL Command Reference*. Also see the **Performing Nonlinear Diagnostics** topic in the **Nonlinear Structural Analysis** section of the *Mechanical APDL Structural Analysis Guide*.

Update Interval

This property appears only for synchronous (p. 1803) solutions. It specifies how often any of the result tracking items under a **Solution Information** object get updated while a solution is in progress. The default is **2.5** seconds.

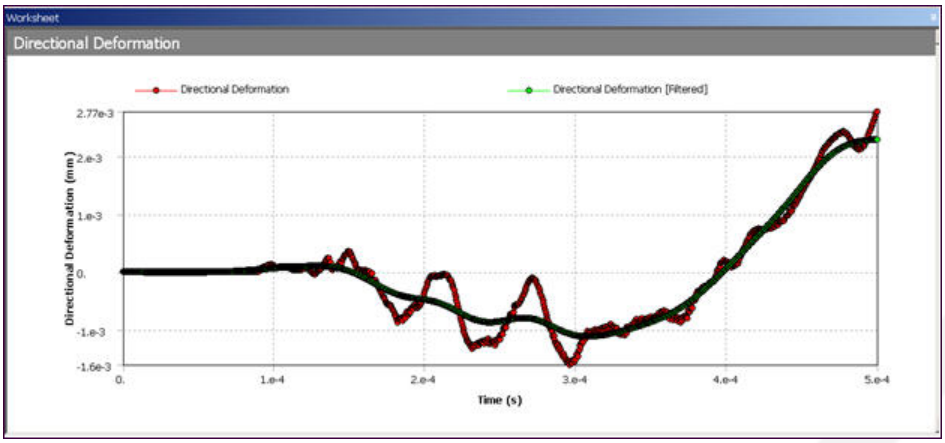
Display Points

This property is not applicable to **Connections** object. It specifies the number of points to plot for a graphical display determined by the **Solution Output** setting (described above).

Display Filter During Solve

This property is applicable only when using Result Tracker filtering in **Explicit Dynamics** analyses. When set to **Yes**, it displays filtered data from Result Trackers in the Worksheet at each refresh interval of the Result Tracker. As shown below, a legend is included in the Worksheet to help distinguish the filtered data from the non-filtered data. Typically there are two curves, non-filtered data is displayed in red, and filtered data is displayed in green.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

If an error occurs during a solve when using the Mechanical APDL solver, the **Solution Information** worksheet may point you to files (for example, `file.err`) in temporary scratch folders whose purpose is for solving only (this is the folder where Ansys actually ran). After the solution, these files are moved back to the project structure, so you may not find them in the scratch folders (or sub-folders).

Viewing Finite Element Connections

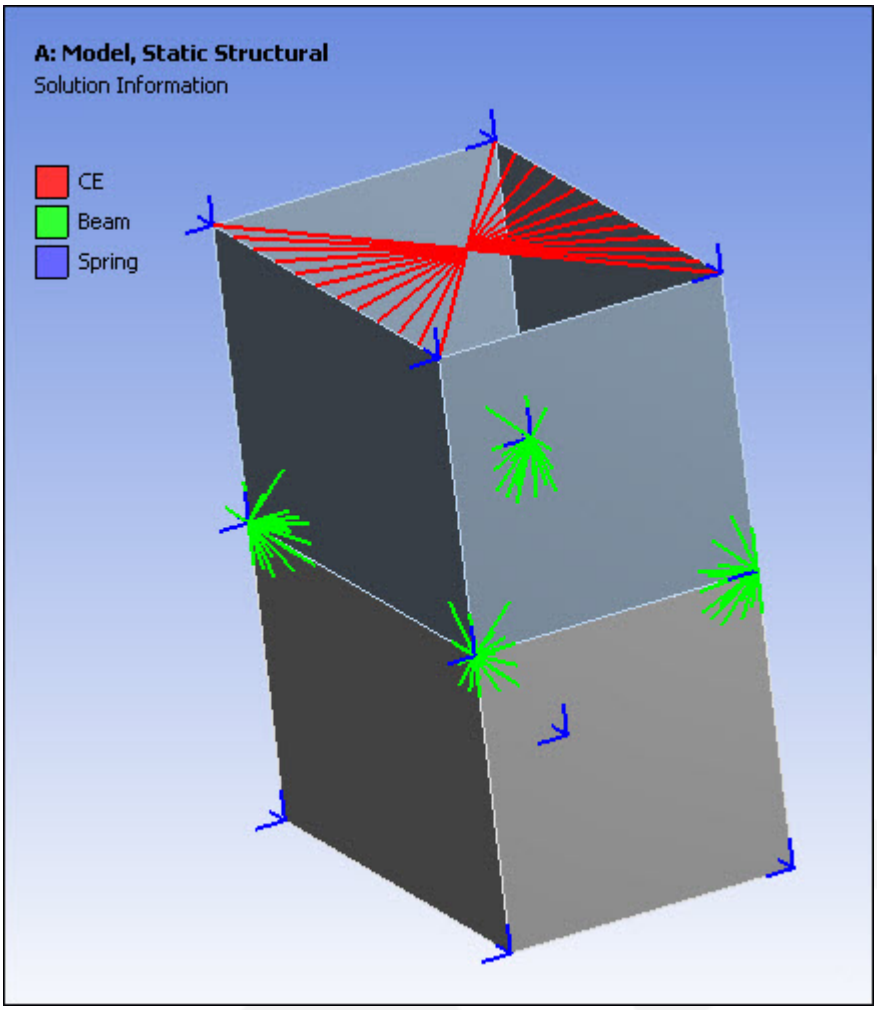
During the solution, the Mechanical application will sometimes create additional elements or Constrain Equations (CE) for certain objects, such as a remote boundary condition, spot weld, joint, MPC-based contact, or weak springs. So that you might better understand how the boundary conditions are applied, the Mechanical application enables you to "view" these connections after a solution is completed using the properties of the **FE Connection Visibility** category. Note that this display feature requires a complete solution. For example, connections are not displayed when reading a result file.

The following properties are available in the Details view under the **FE Connection Visibility** category:

- **Activate Visibility**: Enables control on whether or not the finite element connection data is stored during the solution. If visualization of the finite element connections will never be desired or to maximize performance on extreme models in which many constraint equations exist, this feature can be deactivated by setting the value to **No** before solving the model. Note that in the case of a multiple step analysis, if constraint equations are present, they will be reported from the first load step. The default value for this property can be changed under Analysis Settings and Solution (p. 238) category of the **Options** dialog.
- **Display**: Controls which finite element connections to display. The options include:
  - **All FE Connectors** (default)

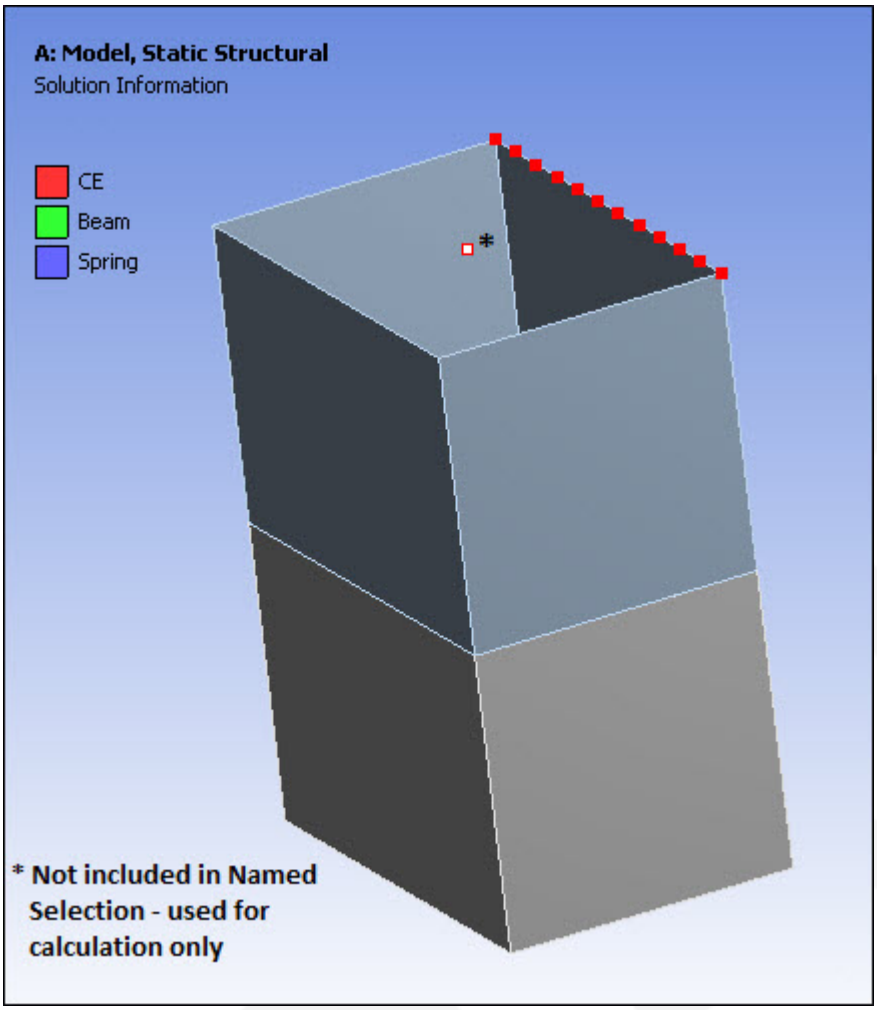
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



– **CE Based** (As illustrated below, outlined or hollow nodes indicate use for calculation purposes only.)

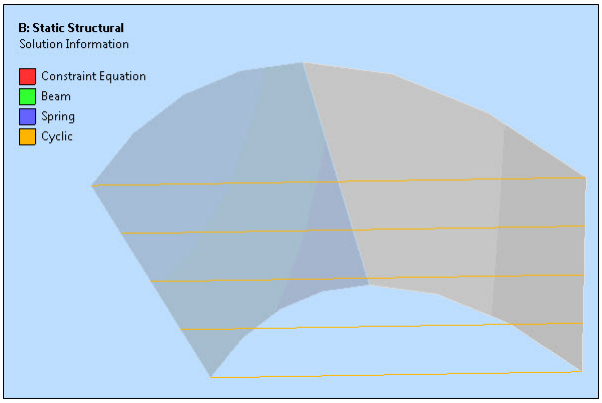
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- **Beam Based**
- **Weak Springs**
- **None**: This option is especially useful to separate the constraint equation connections from the beam connections. The option **None** is available to assist in avoiding potential performance issues from this feature.
- **Cyclic**: For a solution containing a Pre-Meshed Cyclic Region (p. 1138), this option displays matching (cyclic) node pairs if they are detected by the solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Important:

There is a mesh display limitation for this object when your analysis includes a Nonlinear Adaptive Region (p. 1707) condition and a remote boundary condition. Currently, this object only shows the base mesh for the geometry, even if it has changed, as it does when a Nonlinear Adaptive Region condition is defined. When you have a Nonlinear Adaptive Region condition combined with a remote boundary condition, the display of the **Constrain Equations (CE)** is based on the modified mesh, not the displayed mesh (**Show Mesh** option set to **On**). That is, the CE display is laid over the base mesh instead of a modified mesh.

- **Draw Connections Attached To**: Based on the availability and visibility of the bodies of your model, this property provides the following options that draw finite element connection annotations for those nodes that are involved in constraint equations.
  - **All Nodes** (default): Nodes attached to available bodies.
  - **Visible Nodes**: Nodes attached to visible bodies.
  - Any node-based **Named Selections** (p. 1057): Nodes in the selected node-based Named Selection.
- **Line Color**: Assigns colors to allow you to differentiate connections. The options include:
  - **Connection Type** (default): Displays a color legend that presents one color for constraint equation connections and another color for beam connections.
  - **Manual**: Displays a color that you choose.
  - **Color**: Appears if **Line Color** is set to **Manual**. By clicking in this field, you can choose a color from the color palette.
- **Visible on Results**: When set to **Yes** (default), the finite element connections are displayed with any result plot (with the exception of a base mesh). When set to **No**, the connections are displayed only when the **Solution Information** object is selected.
- **Line Thickness**: Displays the thickness of finite element connection lines in your choice of **Single** (default), **Double**, or **Triple**.
- **Display Type**: enables you to view FE connections as **Lines** (default) or as **Points**. If you wish to view the **Points** of a specified Named Selection, the nodes belonging to the Named Selection display as

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

solid colors. Any other associated nodes not belonging to the Named Selection, display with an outline only.

Note:

Finite element connection information is not available for Response Spectrum analyses when the **Spectrum Type** property is set to **Single Point**.

Postprocessing During Solve

This feature is only available when you solve an analysis on a remote computer or as a background process running as a non-distributed solution. When you run the solution as a background process, you can add new results under the **Solution** object or use postprocessing features such as viewing results contours, animation, min and max labels, and so on.

Postprocessing during a solve allows you to use postprocessing tools while an analysis is still in progress. This feature is useful for analyses that produce partial results (that is, analyses that produce intermediate results files that are readable but incomplete) such as all Static and Transient Structural, all Static and Transient Thermal, and Explicit Dynamics analyses.

To postprocess results during a solve:

1. Set up the Remote Solve Manager or Distributed Compute Services and run a solution. Make sure that the **Distribute Solution** option on the Solver Process Setting (p. 1804) dialog is unchecked.

   Request results for a specific time by entering the time in the **Display Time** field within the Details view of the **Solution** object.

2. Right-click the **Solution** object and choose **Evaluate All Results**.

   If you chose a specific time point that is not yet solved, the result of the *most recent* solved point will be displayed in the output fields within the Details view.

Note:

When using this feature, it is important that you allow adequate time after the solve for the results files to be created and present before any postprocessing can be successful. Requesting a postprocessing function too prematurely could generate an error message stating that the result file could not be opened.

Using Result Trackers

In addition to the real time solution response graphs you can view from the **Solution Information** object, you can also view graphs of specific displacement and contact results as a function of time using **Result Tracker** objects. These objects are inserted as branch objects under a **Solution Information** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Other than contact trackers that read data from the CND file (`file.cnd`), you cannot add new Result Trackers to completed solutions. In order to add and solve a new result, you must clear (p. 1981) the **Solution**, add a new **Result Tracker**, and then resolve the simulation.

Note:

- Result Trackers employ the instructions of the Mechanical APDL command, **NLHIST**.
- Contact Trackers reading from the CND file use the Mechanical APDL command, **NLDIAG**.

Application

To insert a **Result Tracker** object, select a **Solution Information** object and either choose an option under the **Result Tracker** drop-down menu in the **Solution Information** Context tab, or right-click on a **Result Tracker** object and then select Insert > and the desired result tracker type.

Note:

You can add **Contact Result Trackers**while the solution is in progress. The application automatically sets the **Enhanced Tracking** property to **Yes** and the property is read-only. Normal scoping requirements apply.

Result Tracker Types

The Result Tracker feature is available for the following:

- Structural Result Tracker (p. 1847)
- Thermal Result Tracker (p. 1852)
- Explicit Dynamics Result Tracker

Result Tracker Features

The following options are offered by the Result Tracker object:

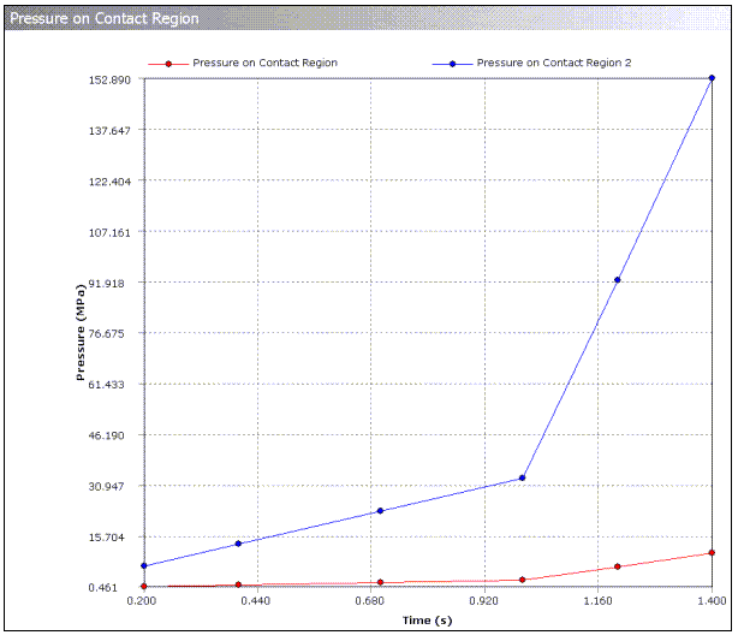
- Plotting (p. 1845)
- Renaming (p. 1846)
- Exporting (p. 1846)

Plotting a Result Tracker

Any of the graphs created by either the **Result Tracker** or nonlinear convergence items have the following features:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Multiple **Result Tracker** objects may be selected at the same time to create a combined chart assuming they share the same X and Y output types (such as pressure for Y and time for X). An example is shown here:



- The graph can be zoomed by using the **ALT** key + left mouse button. Moving down and to the right zooms in, and moving up and to the left zooms out.

Caution:

Because nodes may be rotated in solutions obtained with the Mechanical APDL application, deformation **Result Trackers** may not record the expected component of the deformation. Should this occur, a warning message alerting you to this will appear after the solve in the Details view of the **Solution** object, in the **Solver Messages** field. This situation can occur when **Result Trackers** are adjacent to supported faces, lines, or vertices. One possible approach to avoid this situation is to add 3 deformation **Result Trackers**, one for each of the x, y, and z directions. This will ensure that the tracker is showing all deformation of that vertex of the model.

Renaming a Result Tracker

The **Result Tracker** has an option for renaming the object based on the definition of the result and the scoping. You choose the option in the right-click (context) menu. This option is useful in having the program create meaningful names of the result trackers. An example would be **Result Tracker 5** being renamed to **Pressure on Contact Region 2**.

Exporting a Result Tracker

Result Tracker objects can be exported to a text file and an Excel file by selecting **Export Text File** from the context (right-click) menu. This option appears in the menu after the solution is obtained. This option enables you to save a text file of the result data. And, based on the setting of the **Automatically**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Open Excel preference in the **Export** (p. 222) category of the **Options** dialog, this option will also open an excel spreadsheet that includes the result data.

Note:

You must right-click the selected *object* to use this **Export** feature. On Windows platforms, if you have the Microsoft Office 2002 (or later) installed, you may see an **Export to Excel** option if you right-mouse click in the **Worksheet** window. This is not the Mechanical application **Export** feature but rather an option generated by Microsoft Internet Explorer.

Structural Result Trackers

Coupled Field Static, Static Structural, and Transient structural analyses support the following result trackers.

- Deformation (p. 1847)
- Contact (p. 1848)
- Energy (p. 1851)

Deformation

The **Deformation** result tracker tracks displacement for one vertex only using the geometry picker or a geometry-based Named Selection or a node-based Named Selection for a single node. **Details** view categories and properties include:

Category	Property/Description
Scope	**Scoping Method**: Options include **Geometry Selection** or **Named Selection**. **Geometry**: Visible when the **Scoping Method** property is set to **Geometry Selection**. This field allows you to select and define a single vertex or a single node as the geometry. **Named Selection**: Visible when the **Scoping Method** property is set to **Named Selection**.. This field provides a list of user-defined Named Selections that are either geometry-based or node-based.
Definition	**Type**: Read-only field that displays the type of Result Tracker - **Directional Deformation**. **Orientation**: Specifies **X-Axis**, **Y-Axis**, or **Z-Axis**. **Suppressed**: *Prior to solving*, you can include or exclude the result from the analysis. The default is value is **No**.
Results	**Minimum**: Read-only indication of the minimum value of the result tracker type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property/Description
	Maximum: Read-only indication of the maximum value of the result tracker type.

Note:

When using the Deformation result tracker to graph displacement for a Nonlinear Adaptive Region (p. 1707), there is a display limitation for the graph. The tracker reads and displays data contained in the `jobname.nlh` file. This file contains incremental displacement data collected after re-meshing occurs. That is, the re-meshed model is considered as a new reference.

Contact

The **Contact** result tracker is for contact outputs scoped to a given contact pair. **Details** view categories and properties include:

Note:

You can add **Contact Result Trackers**while the solution is in progress. The application automatically sets the **Enhanced Tracking** property to **Yes** and the property is read-only. Normal scoping requirements apply.

Category	Property/Description
Scope	**Contact Region**: Specifies the particular contact region in the pair. Default names are **Contact Region** and **Contact Region 2**. **Enhanced Tracking**: This property causes your Contact Result Tracker to obtain contact pair-based information as specified by the **Type** property (**Number Contacting**, **Penetration**, etc.). This takes place during the solution phase or following a completed solution from the CND file (`file.cnd`). The default setting is **Yes**. See **NLDIAG** command for additional information. The **No** setting reads data from the NLH file. See **NLHIST** command for additional information. **Note:** Enhanced Tracking results are only available when a complete solution is performed in Mechanical 16.0 or a later version. **Contact Side**: Specify the desired side of the associated **Contact Region** for which you wish to obtain results, either the **Contact** (default) side or the **Target** side.
Definition	**Type**: Specifies the particular contact output. For each of these options, the result tracking is performed on the Contact side of the pair. If you want to perform the result tracking on the Target side, you should flip the source and target sides. If this occurs you can change the contact region to **Asymmetric** (p. 1236) and flip the source and target faces in order to specify the side of interest that is to be the contact side. If **Auto Asymmetric** (p. 1237) contact is active (either by the **Behavior** (p. 1236) contact region setting equaling **Auto Asymmetric** or by the **Formulation** (p. 1241) setting equaling **Augmented Lagrange** (p. 1242) or **MPC** (p. 1243)) and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property/Description
	the contact side is chosen by the program to be disabled, the **Results Tracker** will not contain any results (as signified by a value of **-2** for **Number Contacting** output). Contact results will be valid depending on the type of contact (for example, edge-edge) and the contact formulation. • **Pressure**: Maximum pressure. • **Penetration**: Maximum penetration. • **Geometric Gap**: Minimum gap. The values will be reported as negative numbers to signify a gap. A value of zero is reported if the contact region is in contact (and thus has a penetration). Also, if the region is in far-field contact (contact faces are outside the pinball radius), then the gap will be equal to the resulting pinball size for the region. • **Frictional Stress**: Maximum frictional stress. • **Sliding Distance**: Maximum total sliding distance (algebraic sum) when the contact status is sticking or sliding. • **Number Sticking**: Number of elements that are sticking. • **Number Contacting** (default): Number of elements in contact. A value of **-1** means the contact pair is in Far-field contact (meaning the faces lie outside the contact pinball region). • **Chattering**: Maximum chattering level. • **Elastic Slip**: Maximum elastic slip. • **Max Normal Stiffness**: Maximum normal stiffness. • **Min Normal Stiffness**: Minimum normal stiffness. • **Max Tangential Stiffness**: Maximum tangential stiffness. • **Min Tangential Stiffness**: Minimum tangential stiffness. • **Contacting Area**: The total area of the elements that are in contact. • **Max Damping Pressure**: Maximum damping pressure. • **Min Geometric Sliding Distance**: Minimum total sliding distance, including sticking, sliding, and near-field. For more information, see the GSLID output parameter in the *Mechanical APDL Contact Technology Guide*. • **Max Geometric Sliding Distance**: Maximum total sliding distance, including sticking, sliding, and near-field. For more information, see the GSLID output parameter in the *Mechanical APDL Contact Technology Guide*. • **Resulting Pinball**: Maximum pinball radius.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property/Description
	• **Normal Fluid Pressure**: Maximum normal fluid penetration pressure. Requires the **Enhanced Tracking** (p. 1848) property to be set to **Yes** (default). • **Tangential Fluid Pressure**: Maximum tangential fluid penetration due to fluid penetration loading. Requires the **Enhanced Tracking** (p. 1848) property to be set to **Yes** (default). • **Fluid Pressure**: Maximum fluid penetration pressure. Requires the **Enhanced Tracking** (p. 1848) property to be set to **No**. • **Contact Depth**: Average contact depth. • **Closed Penetration**: Maximum closed (geometrical) penetration. • **Number with Large Penetration**: Number of contact points having too much penetration. • **Tangential Damping Stress**: Maximum tangential contact damping stress. • **Volume Loss Due to Wear**: Total volume lost due to wear for the contact pair (not available for general contact). • **Strain Energy**: Total strain energy due to contact constraint. • **Frictional Dissipation Energy**: Total frictional dissipation energy. • **Stabilization Energy**: Total contact stabilization energy. • **Number With Too Much Sliding**: Number of contact points having too much sliding for small sliding contact (p. 1244). • **Total Force From Contact Pressure X**: Total Force along the X-axis due to contact pressure. • **Total Force from Contact Pressure Y**: Total Force along the Y-axis due to contact pressure. • **Total Force from Contact Pressure Z**: In a 3D model, the reported item is the total force along the Z-axis. In a 2D axisymmetric model, the reported item is maximum torque that can potentially act on the Y-axis. • **Total Force from Tangential Stress X**: Total Force along the X-axis due to tangential stress. • **Total Force from Tangential Stress Y**: Total Force along the Y-axis due to tangential stress. • **Total Force from Tangential Stress Z**: Total Force along the Z-axis due to tangential stress. • **Sliding Indication**: Specifying this option requires that the **Type** property of the specified **Contact Region** be set to **Frictional**. Sliding Indication is calculated using the following formula.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property/Description
	$$\text{Sliding Indication} = \frac{\text{Magnitude(Total Force from Tangential Stress)}}{\text{Magnitude(Total Force from Contact Pressure)} \times \text{MU}}$$ For the equation, **MU** is the Friction Coefficient of the Frictional Contact. When Sliding Indication equals **-1**, this indicates that the normal contact force equals **0**. When Sliding Indication equals **1**, this indicates that the Friction Coefficient equals **0** (a great deal of sliding). Any other result for the Sliding Indication value means that there is some amount of sliding. The higher the value, the greater the amount of sliding. • **Force Convergence**: Pair-based force convergence norm and criterion used in the local convergence check. The local convergence check is met if the pair-based force convergence norm is less than the pair-based force convergence criterion. • **Possible Overconstraint**: Supported for contact regions using the **Program Controlled** or **MPC** settings for the Formulation (p. 1241) property. When extracted, negative value indicates a possible overconstraint for the scoped contact region and contact side. **Suppressed**: *Prior to solving*, you can include or exclude the result from the analysis. The default is value is **No**.
Definition	**Minimum**: Read-only indication of the minimum value of the result tracker type. **Maximum**: Read-only indication of the maximum value of the result tracker type.

Energy

The **Details** pane properties and options for energy-based result trackers are described below.

Category	Property/Description
Definition	**Type**: This property is a read-only field that displays the type of results tracker. Energy result trackers include: • **Kinetic Energy** • **Stiffness Energy** • **Damping Energy** • **Artificial Energy** • **Nonlinear Stabilization Energy** **Suppressed**: *Prior to solving*, you can include or exclude the result from the analysis. The default is value is **No**.
Results	**Minimum**: Read-only indication of the minimum value of the result tracker type. **Maximum**: Read-only indication of the maximum value of the result tracker type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Thermal Result Trackers

A Thermal Analysis supports the Temperature Result Tracker only. The Temperature can be applied to one vertex only using the geometry selection tools or using a geometry-based Named Selection or for a single node, node-based Named Selection.

Note:

Direct graphical node selection requires you to generate the mesh.

The Details view properties and options for the **Temperature** Result Tracker are described below.

- **Scope**:
  - **Scoping Method**: Specifies the option **Geometry Selection**, **Named Selection**, **Remote Point**, **Global Minimum**, or **Global Maximum** for a solution point.
  - **Geometry**: Visible when **Geometry Selection** is specified as the **Scoping Method**. This property enables you to select and define a single vertex as the geometry or a single node.
  - **Named Selection**: Visible when **Named Selection** is specified as the **Scoping Method**. This property provides a drop-down menu of user-defined Named Selections that are either geometry-based or node-based.
  - **Remote Point**
  - **Remote Point**: Visible when **Remote Point** is specified as the **Scoping Method**. This property provides a drop-down menu of user-defined Remote Points.
  - **Global Minimum**
  - **Global Maximum**
- **Definition**
  - **Type**: Read-only field that displays the type of Results Tracker.
  - **Suppressed**: *Prior to solving*, you can include or exclude the result from the analysis. The default is value is **No**.
- **Results**
  - **Minimum**: Read-only indication of the minimum value of the result tracker type.
  - **Maximum**: Read-only indication of the maximum value of the result tracker type.

Using Adaptive Convergence

You can control the relative accuracy of a solution in two ways. You can use the meshing tools to refine the mesh before solving, or you can use convergence tools as part of the solution process to refine solution results on a particular area of the model. This section discusses the latter.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Through its convergence capabilities, the application can fully automate the solution process, internally controlling the level of accuracy for selected results. You can seek approximate results or adapted/converged results. This section explains how to interpret accuracy controls.

Go to a section topic:

- Converged Results Control (p. 1853)
- Adaptivity (Refinement of Meshes Based on Solutions) (p. 1853)
- Adaptive Convergence in Multiple Result Sets (p. 1855)
- Ansys Workbench Product Adaptive Solutions (p. 1855)
- Requirements and Limitations (p. 1856)
- Notes About Accuracy (p. 1857)

Converged Results Control

You can control convergence to a predefined level of error for selected results. In the calculation of stresses, displacements, mode shapes, temperatures, and heat fluxes, the application employs an adaptive solver engine to identify and refine the model in areas that benefit from adaptive refinement. The criteria for convergence is a prescribed percent change in results. The default is 20%. You can change this default using the **Convergence** (p. 221) setting in the **Options** dialog box (p. 217).

Adaptivity (Refinement of Meshes Based on Solutions)

You can continue to refine the mesh based on a specific solution result. When you pick a result (**Equivalent Stress**, **Total Deformation**, **Total Flux Density**, etc.), indicate that you want to converge on this solution. You pick a value and the solution is refined such that the solution value does not change by more than that value.

To add convergence, click the result you added to your solution; for example, **Equivalent Stress**, **Total Deformation**, or **Total Flux Density**. If you want to converge on deformation, right-click **Total Deformation** and select **Insert** > **Convergence**. In the **Details**, you can specify convergence on either the **Minimum** or **Maximum** value. Additionally, you can specify the **Allowable Change** between convergence iterations.

For an adaptive solution, a solution is first performed on the base mesh, and then the elements are queried for their solution information (such as deflection, X-stress, Y-stress, etc.). If the element's results have a high Zienkiewicz-Zhu, or ZZ error (see the *Mechanical APDL Theory Reference* for more information on adaptivity theory), the element is placed in the queue to be refined. The application then continues to refine the mesh and perform additional solutions. Adaptivity will be more robust if your initial mesh is with tetrahedrons. Adaptive refinement starting from a hex-dominant mesh will automatically result in a re-meshing of the structure with tetrahedrons. The face mesh given to the tet mesher is the initial quad mesh split into triangles. That face mesh is then filled with tetrahedrons so it is recommended that you insert an all tetrahedron mesh method before you start an adaptive solution.

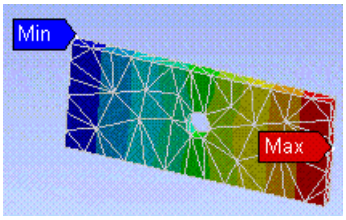
You can control the aggressiveness of the adaptive refinement by adjusting the **Refinement Depth** setting under **Adaptive Mesh Refinement** in the Details view of a **Solution** object. The default value is 2 for structural analyses, and 0 for magnetostatic analyses. The range is from 0 to 3. By default, when

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

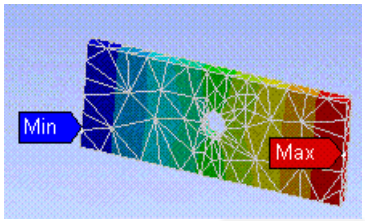
adaptive convergence occurs, the program will refine to a depth of 2 elements to help ensure smooth transitions and avoid excessive element distortion for repeated refinement. However, you can adjust this refinement depth to a value of 0 or 1 if for a particular problem, the deep refinement is not required and problem size is a major concern. In general, for mechanical analyses, the default value of 2 is highly recommended. However, you can lower the value if too much refinement is occurring and is overwhelming the solution in terms of size of solution time. If you use a value less than 2, be aware of the following:

- Verify that false convergence is not occurring because of too little refinement.
- More refinements may be required to achieve the desired tolerance, which may increase the total solution time.

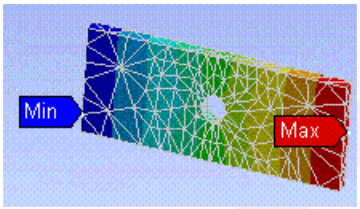
The following pictures show the effects of various settings of **Refinement Depth** on plots of Total Deformation.



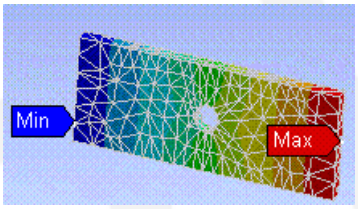
Base Mesh: No Refinement



Refinement Depth = 0



Refinement Depth = 1



Refinement Depth = 2

For magnetostatic analyses, there are additional settings that allow you to change the percentage of the element selected for adaptive refinement during solution. These settings use an **Energy Based** percentage and an **Error Based** percentage. The internal selection process first uses the **Energy Based** percentage to select the number of elements in the full model that have the highest values of magnetic energy. From this number, it uses the **Error Based** percentage to select the number of elements with the highest error in the particular body. **Magnetic Error** results are also available to display on the geometry for verification.

These adaptive refinement settings for magnetostatic analyses are in the **Refinement Controls** group, located in the Details view of the **Solution** object, provided you have a **Convergence** object inserted under any magnetostatic result (p. 2157). An **Element Selection** setting in this group has the following options:

- **Program Controlled** (default): The percentage of elements selected for adaptive refinement equals the default values of 10% for the **Energy Based** percentage and 20% for the **Error Based** percentage.
- **Manual**: The percentage of elements selected for adaptive refinement equals the values you enter in the **Energy Based** and **Error Based** fields that appear only when you choose **Manual**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

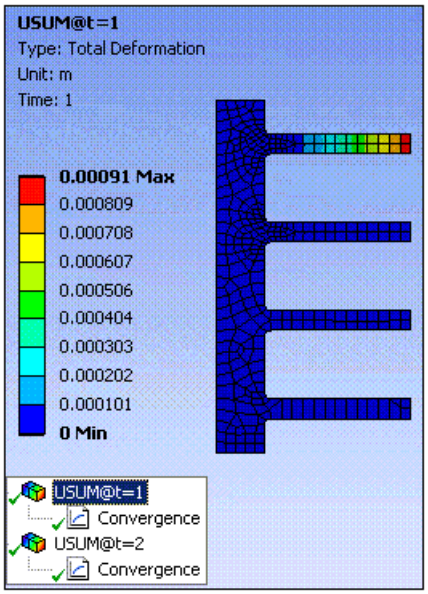
Published: 2023-10-23

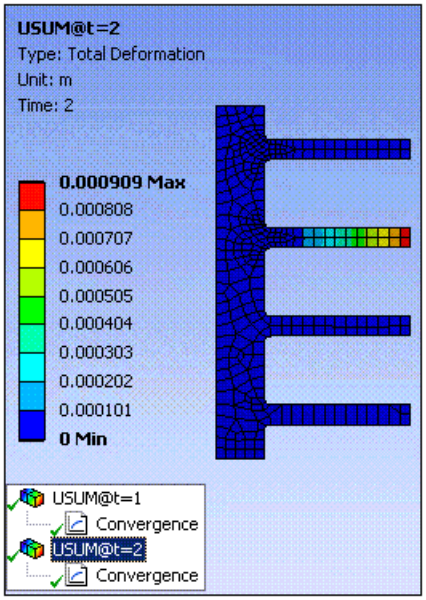
For magnetostatic analyses, **Directional Force** results allow seeking convergence based on **Force Summation** or **Torque** as opposed to other results converging on **Maximum** or **Minimum** values.

Adaptive Convergence in Multiple Result Sets

You can apply adaptive convergence on multiple result sets that may include different loadings or time points. To do so, create a result for each loading or time point and insert a **Convergence** object under each result.

The following example shows **Total Deformation** results at two time points where a **Convergence** object was inserted under each result.





Ansys Workbench Product Adaptive Solutions

Nearly every Ansys Workbench product result can be calculated to a user-specified accuracy. The specified accuracy is achieved by means of adaptive and iterative analysis, whereby h-adaptive methodology is

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

employed. The h-adaptive method begins with an initial finite element model that is refined over various iterations by replacing coarse elements with finer elements in selected regions of the model. This is effectively a selective remeshing procedure. The criterion for which elements are selected for adaptive refinement depends on geometry and on what Ansys Workbench product results quantities are requested. The result quantity ϕ, the expected accuracy E (expressed as a percentage), and the region R on the geometry that is being subjected to adaptive analysis may be selected. The user-specified accuracy is achieved when convergence is satisfied as follows:

$$100\left(\frac{\phi_{i+1}-\phi_i}{\phi_i}\right) < E, \quad i = 1,2,3,\ldots,n \text{ (in R)}$$

where **i** denotes the iteration number. It should be clear that results are compared from iteration **i** to iteration **i+1**. Iteration in this context includes a full analysis in which h-adaptive meshing and solving are performed.

The Ansys Workbench product uses two different criteria for its adaptive procedures. The first criterion merely identifies the largest elements (LE), which are deleted and replaced with a finer finite element representation. The second employs a Zienkiewicz-Zhu (ZZ) norm for stress in structural analysis and heat flux in thermal analysis.

Table 4: Ansys Workbench Product Adaptivity Methods

Result	Adaptive Criterion
Stresses and strains	ZZ norm
Structural margins and factors of safety	ZZ norm
Fatigue damage and life	ZZ norm
Heat flows	ZZ norm
Temperatures	ZZ norm
Deformations	ZZ norm
Mode frequencies	LE

As mentioned above, geometry plays a role in the Ansys Workbench product adaptive method. In general, accurate results and solutions can be devised for the entire assembly, a part or a collection of parts, or a surface or a collection of surfaces. The user makes the decision as to which region of the geometry applies. If accurate results on a certain surface are desired, the Ansys Workbench product ignores the aforementioned criterion and simply refines all elements on the surfaces that make up the defined region. The reasoning here is that the user restricts the region where accurate results are desired. In addition, there is nothing limiting the user from having multiple accuracy specification. In other words, specified accuracy in a selected region and results with specified accuracy over the entire model can be achieved.

Requirements and Limitations

To use **Convergence** feature:

- To use Convergence, you must set **Calculate Stress** to **Yes** under **Output Controls** in the **Analysis Settings** details panel. However, you can perform Modal and Buckling Analysis without specifying this option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Convergence objects inserted under an environment that is referenced by an Initial Condition object or a **Thermal Condition** (p. 1740) load object, will invalidate either of these objects, and not allow a solution to progress.
- The **Error** (p. 2030) result that is used for convergence is based on linear stresses and as such may be inaccurate in certain nonlinear analyses, such as when plasticity is active.
- When performing an out of process (p. 1803) solution asynchronously, wherein the solve may finalize during another Workbench session, the application performs only one maximum refinement loop. As necessary, you must manually perform additional loops. To solve with a single user action, solve synchronously.
- Results cannot be converged when you have a **Mesh Connection** object.
- You cannot use Convergence if your system is linked to an upstream or a downstream analysis.
- Convergence is not available when you activate either of the following conditions:
  - Nonlinear Adaptive Region (p. 1707)
  - Contact Step Control (p. 1711)
- Convergence is not supported for:
  - A model with Layered Sections.
  - Meshes generated using the MultiZone Quad/Tri, MultiZone Quad, or Patch Independent mesh methods.
  - Mixed Order Meshing
  - Orthotropic materials.
  - Hyperelastic materials.
  - Solid shell elements (the SOLSH190 series elements).
  - If a **Periodic Region** or **Cyclic Region** symmetry object is defined.
  - Solution Combinations (p. 2288).

Notes About Accuracy

Accuracy acceptance:

- Low levels of accuracy are acceptable for demonstrations, training, and test runs. Allow for a significant level of uncertainty in interpreting answers. Very low accuracy is never recommended for use in the final validation of any critical design.
- Moderate levels of accuracy are acceptable for many noncritical design applications. Moderate levels of accuracy should not be used in a final validation of any critical part.
- High levels of accuracy are appropriate for solutions contributing to critical design decisions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

At maximum accuracy, when convergence is *not* sought, studies of problems with known answers yield the following behaviors and approximated errors:

- Less than 20% error for peak stresses and strains, and minimum margins and factors of safety.
- Between 5% and 10% error for average (nominal) stresses and elastic strains, and average heat flows.
- Between 1% and 5% error for average stress-related displacements and average calculated temperatures.
- 5% or less error for mode frequencies for a wide range of parts.

When seeking highly accurate, converged Results, more computer time and resources will be required than manual control, except in some cases where the manual preference approaches highest accuracy.

Given the flexible nature of the solver engine, it is impossible to explicitly quantify the effect of a particular accuracy selection on the calculation of results for an arbitrary problem. Accuracy is related only to the representation of geometry. Increasing the accuracy preference will not make the material definition or environmental conditions more accurate. However, specified converged results are nearly as accurate as the percentage criteria.

Critical components should always be analyzed by an experienced engineer or analyst prior to final acceptance.

Saving and Managing Results

Go to s section topic:

- Saving Files (p. 1858)
- Managing Files (p. 1859)

Saving Files

There are three ways to save your results in the Mechanical application:

- As a Mechanical APDL application database file.

  To save the Mechanical application results in a Mechanical APDL application database file, click **Analysis Settings** on the Outline (p. 105) and in its Details, click **Yes** next to **Save Ansys db** under Analysis Data Management (p. 1484).

- As an input file for the Mechanical APDL application. See Writing and Reading the Mechanical APDL Application Files (p. 1861).
- As a Mechanical application database file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To save your solution as a Mechanical application database file, select **File> Export**. Select **File> Save As** in the Project Schematic to save the project. The **Save As** dialog box appears, allowing you to type the name of the file and specify its location.

Note:

The application creates reference files that contain analysis information that is read back into the application during solution processing. Certain textual characters can create issues during this reading process. Avoid the use of the following characters in your file naming conventions:

- Quote character (")
- Ampersand (&)
- Apostrophe (')
- Greater than and less than characters (< >)
- Japanese script

Managing Files

During the solution, several files are created. Some of these can be deleted after the solution but some need to be retained for postprocessing or for feeding other subsequent analyses. Since you can perform several different analyses on a single model or even have several models in the same Mechanical application project, you must manage the solution files in a consistent and predictable manner.

Note:

If you open a solved project and make a change to either the geometry or the mesh, Mechanical automatically deletes the content of the original project files (*.rst, *.out, etc.).

Consistent Directory Structure for Mechanical Application Analyses

Ansys Workbench's file management system keeps multiple databases under a single project. See Saving and Managing Project Files for a description of the file management system.

Note:

The **Analysis Settings** Details view has an **Analysis Data Management** (p. 1484) grouping that shows the solution directory location for each analysis.

Solution Files

Default behavior: By default an analysis in the Mechanical application saves only the minimal files required for postprocessing. Typically these include results files (`file.rst`, `file.rth`, `file.rmg`, `file.psd`, `file.mcom`), input file (`ds.dat`), output file (`solve.out`), and some other files that

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

have valuable information about the solution (`file.BCS`, `file.nlh`, `file.gst`). Of these only the results file is generally of significant size.

For Windows users, the solution files folder can be displayed using the **Open Solver Files Directory** (p. 1485) feature.

Future Analysis: If the results of this analysis are to be used as a load or an initial condition in a subsequent analysis then additional files may need to be saved. Declaring your intent to use this in the future (p. 1484) will automatically save the required files and reuse them in the subsequent analysis. Refer to Define Initial Conditions (p. 349) for details of these analyses.

Delete Unneeded Files: The solution process creates other files that are typically not needed for postprocessing or are not used in subsequent analyses. By default, the Mechanical application deletes these files at the end of solution. However, if for any reason, you want to keep all the files you could choose to do so (p. 1484).

You can use the **Output Controls** (p. 1473) on the analysis settings page to write only the desired result types to the `.rst` file (for example, if strains are not needed, you can turn them off which would create a smaller result file). In addition, for advanced Mechanical APDL application users, **Command** objects (p. 2303) can be used to further limit output via the OUTRES command.

An external result file is needed to post results. The following behavior will occur:

- If you save a simulation, any simulation files (result and other required files) will be saved to the new location.
- If you use the **Duplicate Without Results** option (Environment and Model objects only), all subordinate objects are reproduced with the exception of the *data* for all result objects. This is based on the intention that loading changes are performed and the solution process is repeated.
- If you attempt to resolve a previously solved and saved database, the corresponding saved result files are backed up automatically in case the current solve is not saved.
- The /post1 XML transfer of result files used in previous releases is no longer used so any existing solution **Command** objects which were modifying the Mechanical APDL application results to be brought back into the Mechanical application no longer function.
- For loads imported from External Model, the **Options** dialog preference, **Solver File Verbosity** (p. 240), enables you to control how much information is written to the input file (ds.dat) and output file (solve.out) during the solution process. The default does not write comments to the input and solver file. This option provides faster processing time from the Mechanical APDL solver. There are two additional options that enable you to increment the amount of information or to write all information to the file.

Writing and Reading Solution Data

The application supports several options for writing input data and reading result data, including:

Writing and Reading the Mechanical APDL Application Files

Writing and Reading the LS-DYNA Application Files

Writing Ansys Rigid Dynamics Files

Writing NASTRAN Files

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Writing and Reading the Mechanical APDL Application Files

The **Solution** Context (p. 62) tab includes options for writing the Mechanical APDL application input files and for reading the Mechanical APDL application results files.

To write the Mechanical APDL application input file:

1. Highlight the **Solution** object folder in the **Outline**.
2. From the **Tools** group on the **Solution** Context tab, select the **Write Input File** option.
3. In the **Save As** dialog box, specify a location and name for the input file.

To read the Mechanical APDL application result files:

1. Highlight the Solution object in the **Outline**, open the drop-down menu of the **Read Result Files** option, and select one of the following options:
   - **By Copy**: The application copies all files contained in this folder to the **Solver Files Directory**.
   - **By Reference**: Using this option, you specify the path to the directory that includes your result files. It does not copy any files.

   Note:

   When reading files on a server, you could see a performance delay.

   Note:

   The **Read Result File** option is not available if a Convergence object is specified in the analysis.

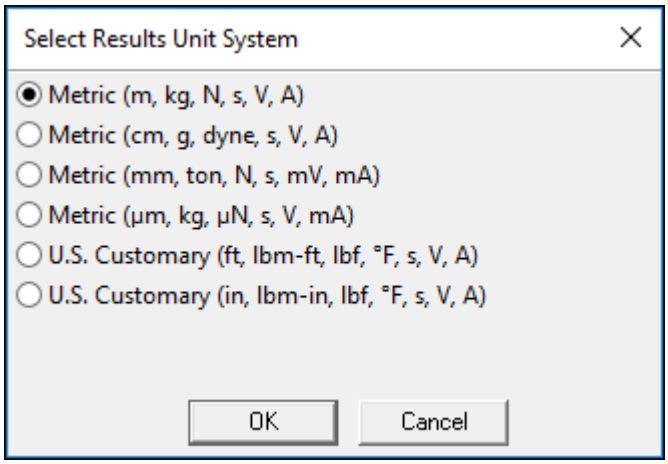
2. A dialog displays that enables you to open a file. Browse to the folder that contains the desired result file, select the file, and then select the **Open** button.

   Note:

   - When reading result files from a directory other than the solver directory, the current solution is cleared and the result files are deleted.
   - When reading result files from a directory other than solver directory when using the **By Copy** option, the application changes the job name to "file" when copying the files to the solver directory. For example, "jobname.rst" would be renamed "file.rst."
   - The target folder should only contain files pertinent to the solution of the selected result file.

3. The following dialog displays. Select a desired unit system and then click **OK**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

By default, the application uses the unit system stored in the result file (under the Mechanical APDL **/UNITS** command). The **Select Results in Unit System** dialog box displays to have you verify the unit system. If you select a unit system that is different than the one already specified result file, the application displays a warning message.

If the application does not have 1) a specified unit system (**/UNITS,0**), or 2) the solver input files include the command entries **/UNITS,SI** or **/UNITS,USER** commands or 3) the file does not include the **/UNITS** command, then the application alerts you that you have updated the unit system based on your select on the **Select Results in Unit System** dialog box. Possible system messages include:

- The source result file does not have the unit system information. If the result file was generated in MAPDL, consider using the /UNITS command to annotate the result file with the unit system information.
- Since the result file does not indicate the unit system, the following file unit system is assumed…

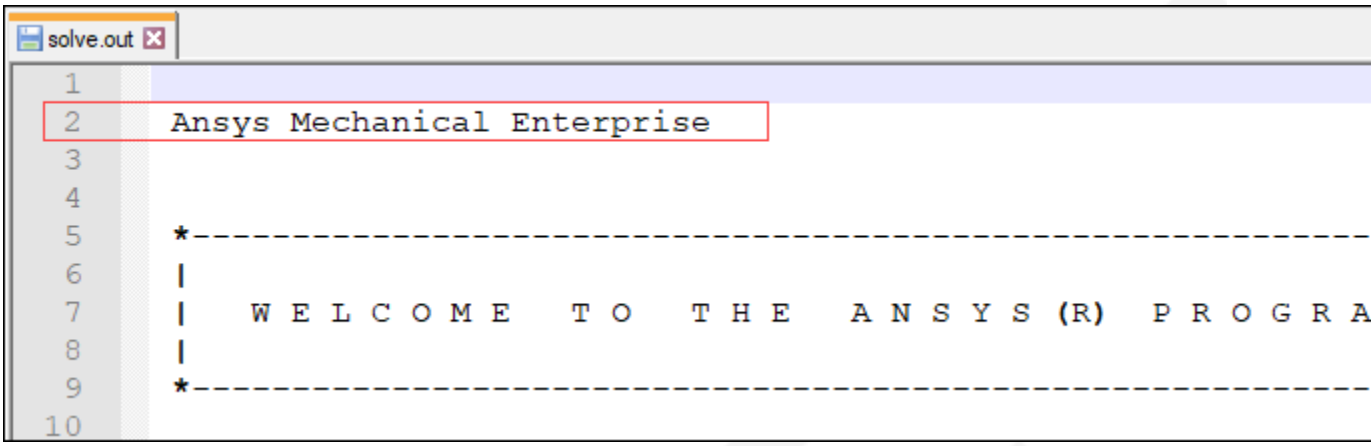
4. Once solution files are either copied to the working directory or properly referenced, the application automatically evaluates the solution.

Caution:

- Errors occur if the application result files are from a version of Mechanical that is older than the version currently running.
- During the process of reading the results files, the application reads the file, `solve.out`, for the product license. This license name needs to be included on Line 2 of this file. If not, the application could stop processing. Scenarios exist where the application could insert text into the first few lines of this file, cause the license to not be included in Line 2. If you experience processing anomalies, check this file and make sure it is formatted properly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Mechanical APDL Application Analysis from a Mechanical Application Mesh

The option for writing the Mechanical APDL application file can be used to perform analyses in the Mechanical APDL application while taking advantage of the meshing capabilities within the Mechanical application. The procedure is as follows:

1. Attach the model into the Mechanical application (p. 330).
2. Mesh the model.
3. Select the **Solution** folder in the tree.
4. Select **Write Input File...** from the tab and specify a location and name for the input file.
5. Use this input file to complete your analysis in the Mechanical APDL application. The meshed model will contain generic elements encoding only shape and connectivity information. Such elements can then be replaced by others that are appropriate to your desired analysis.

Note:

Any named selection group from the Mechanical application is transferred to the Mechanical APDL application as a component according to specific naming rules and conventions (p. 1087).

Using Writing and Reading Files Together

The writing and reading options are useful when used together. You can use the write option, then solve at your leisure on the machine of your choice. When the solution is done, you can use the read option to browse to the directory that contains the Mechanical APDL application output files (for example, result file, file.err, solve.out, file.gst, file.nlh). Workbench will then copy all files into your solution directory and proceed to use those files for postprocessing.

The reading option only requires that the target directory include the result (`.rst`) file. You can also use uncombined result files from a distributed-memory parallel processing solution instead of a single combined result file. This requires that your Jobname is named "file" and you must also select the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

file `file0.rst` (or `file0.rth`) from the directory. You may find it useful to read uncombined result files from a solution that was stopped before the combination took place or if you requested that the result file not be combined (**DMPOPTION,RST,NO**).

Note:

You must ensure that the mesh in the result file matches the mesh in Workbench. This includes the Workbench generated mesh from the geometry as well as any nodes or elements defined in the input file (such as for contact or remote boundary conditions). Failure to do so could result in incorrect results and unexpected behavior.

The *reading* Mechanical APDL application file option is available for all analysis types except rigid dynamic analyses and shape analyses.

The *writing* Mechanical APDL application file option is available for all analysis types except rigid dynamic analyses.

System units must be specified in the Mechanical APDL application result files being read for **Result Tracker** graphs to display properly. **Result Tracker** graphs will display in the Mechanical APDL application result file units if the units specified when reading the files are inconsistent with those in the files.

Writing and Reading the LS-DYNA Application Files

The **Solution** Context tab (p. 62) includes options for writing the LS-DYNA application input files and for reading the LS-DYNA application results files.

To write the LS-DYNA application input file:

1. Highlight the **Solution** object folder in the tree.
2. From the **Tools** group on the **Solution** Context tab, select the **Write Input File** option.
3. In the **Save As** dialog box, specify a location and name for the input file.

To read the LS-DYNA application result files:

1. Highlight the **Solution** object folder in the tree.
2. From the **Tools** group on the **Solution** Context tab, select the **Read Result Files** option.

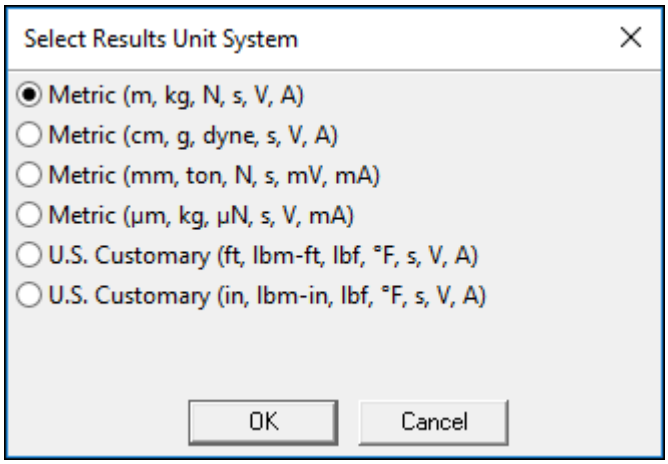
Note:

The **Read Result File** option is not available if a Convergence object is specified in the analysis.

3. Browse to the folder that contains the LS-DYNA application result files and click **Open**.
4. In the dialog box that follows, select the unit system, then click **OK**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The Unit System used during the solution is stored in the file *file*.actunits. The **Select Results in Unit System** dialog box displays to have you explicitly set the unit system in which read results are to be interpreted, which you can choose to be different from the Solver unit system defined in the Analysis Settings. Selecting a unit system that differs from the unit system defined in the analysis settings causes a warning message to display.

Note:

The mm, kg, kN, ms solver unit system is not currently supported with Read Result Files.

Important:

- The application overwrites existing result files in the Solver Files Directory if these result files names are present (perfect match or begin the same) in the folder of the LS-DYNA application result files.
- The application does not overwrite existing result files in the Solver Files Directory if these result files names are not already present (perfect match or begin the same) in the folder of the LS-DYNA application result files.
- If `file.actunits` already exists in the Solver Files Directory or in the folder that contains the LS-DYNA application result files, it will be overwritten by the application.
- If `file.actunits` does not exist in the Solver Files Directory or in the folder that contains the LS-DYNA application result files, it will be created by the application.

Caution:

- Errors will occur if the LS-DYNA application result files are from a version of the LS-DYNA application that is older than the version currently running.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The procedure above instructs you to browse to the folder that contains the LS-DYNA application result files. This folder should only contain files pertinent to that solution because LS-DYNA copies all the files contained in this folder to the **Solver Files Directory**.

LS-DYNA Application Analysis from a Mechanical Application Mesh

The option for writing the LS-DYNA application file can be used to perform analyses in the LS-DYNA application while taking advantage of the meshing capabilities within the Mechanical application. The procedure is as follows:

1. Attach the model into the Mechanical application (p. 330).
2. Mesh the model.
3. Select the **Solution** folder in the tree.
4. Select **Write Input File...** from the tab and specify a location and name for the input file.
5. Use this input file to complete your analysis in the LS-DYNA application. The meshed model will contain generic elements encoding only shape and connectivity information. Such elements can then be replaced by others that are appropriate to your desired analysis.

Note:

Any named selection group from the Mechanical application is transferred to the LS-DYNA application as a component according to specific naming rules and conventions (p. 1087).

Using Writing and Reading Files Together

The writing and reading options are useful when used together. You can use the write option, then solve at your leisure on the machine of your choice. When the solution is done, you can use the read option to browse to the directory that contains the LS-DYNA application output files (for example, result file, file.err, solve.out, file.gst, file.nlh). Workbench will then copy all files into your solution directory and proceed to use those files for postprocessing.

Note:

You must ensure that the mesh in the result file matches the mesh in Workbench. This includes the Workbench generated mesh from the geometry as well as any nodes or elements defined in the input file (such as for contact or remote boundary conditions). Failure to do so could result in incorrect results and unexpected behavior.

The *reading* LS-DYNA application file option is available for all analysis types except rigid dynamic analyses and shape analyses.

The *writing* LS-DYNA application file option is available for all analysis types except rigid dynamic analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

System units must be specified in the LS-DYNA application result files being read for **Result Tracker** graphs to display properly. **Result Tracker** graphs will display in the LS-DYNA application result file units if the units specified when reading the files are inconsistent with those in the files.

Writing Ansys Rigid Dynamics Files

When using a Rigid Dynamics analysis system, the **Tools** group on the **Solution** Context tab provides the option **Write Input File**. You can write a file to use for co-simulation with Twin Builder (p. 842) (`.fmu`), or the `.fmu` file can be used for co-simulation with an external tool (co-simulation leader). For more details about the FMI standard and tools that support co-simulation with FMU, see fmi-standard.org.

To write the Rigid Dynamics input file:

1. Highlight the **Rigid Dynamics** object folder (or any object under it) in the tree.
2. From the **Tools** group on the **Solution** Context tab, select the **Write Input Files** option.
3. In the **Save As** dialog box, choose the type of file to write (`.fmu`) and specify a location and name for the file.

Note:

From Release 2024 R1, the Rigid Dynamics solver exports `.fmu` files that comply with the FMI 2.0 standard by default.

Writing NASTRAN Files

This feature is supported for stand-alone Static Structural and Modal analysis types only. It enables you to export your simulation as a NASTRAN Bulk Data file. You can use the capabilities of Mechanical and run the resulting simulation with your choice of NASTRAN solver. Currently, the NASTRAN Export capability is designed to export the common features of static structural and modal simulations. However, not all features are supported. Review the following subsections for additional information about exported what is supported as well as the limitations associated with this feature:

- NASTRAN Export Supported Features (p. 1870)
- NASTRAN Export Limitations (p. 1872)

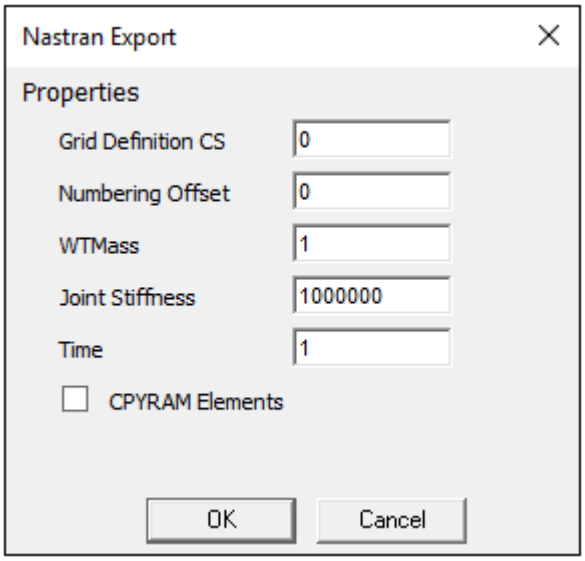
Application

1. Select the Environment object and select the **Export NASTRAN File** option from the **Tools** group of the Environment Context Tab (p. 61) (the option is also available via right-click on the **Environment** object). Based on your analysis type, one of the following dialogs displays.

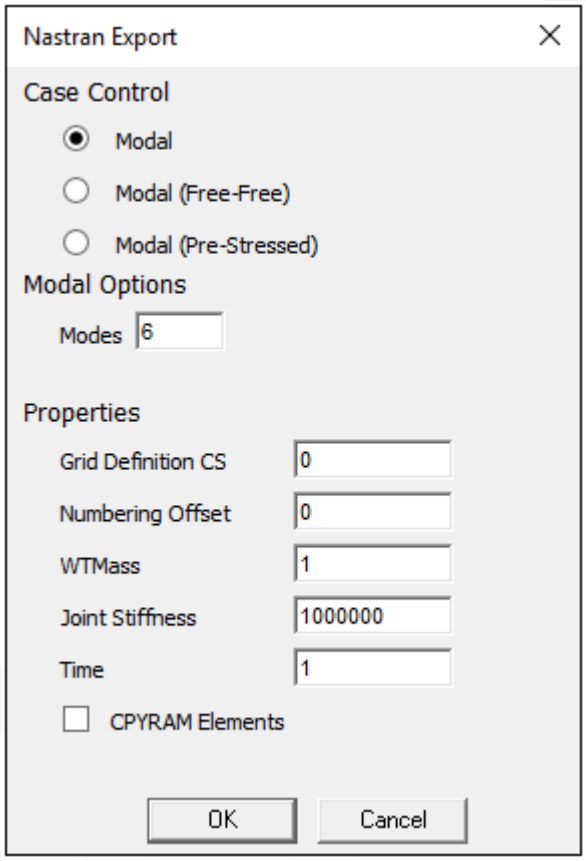
   You use these property options to further define how you wish to export your simulation. Review the property descriptions (p. 1868) listed below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Static Structural



Modal



2. Click **OK** to begin the export process. You are prompted to enter a file name and specify the file extension, `.dat`, `.bdf` or `.nas`.

Note:

The contents of each file type (`.bdf`, `.nas`, and `.dat`) are identical.

3. As needed, you can open the log file (filename.log) produced during the NASTRAN Export process to review certain translation details of the export. This file is placed in the same location as your NASTRAN file.

Property Definition

General Properties

Each dialog includes the following identical **Properties**:

- **Grid Definition CS**: This property determines which Mechanical Cartesian coordinate system is used as the NASTRAN basic coordinate system. When set to 0 (the usual case), the Mechanical global coordinate system will be used. You may instead choose any local Cartesian coordinate system in Mechanical to be used as the NASTRAN basic coordinate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

system, thereby translating and/or rotating the grid for export. To do so, set this property value to the desired Mechanical coordinate system reference number.

Unless you have a specific reason for doing so, do not choose a cylindrical or other non-Cartesian system for this property. Doing so will cause R, Θ, and, Z coordinates, for example, to be interpreted as X, Y, and Z values, causing a cylindrical mesh to be unrolled (developed) into a planar mesh.

- **Numbering Offset**: This property is used to add a specified integer offset to the numbering of all node, element, material, property, and coordinate system IDs. This property can be used to prevent numbering conflicts if the exported data is to be combined with data from additional exports or other sources. Enter the offset value for the property value.
- **WTMass**: This property writes the parameter WTMASS into the export file to specify a global change in Mass units. Mechanical does not use this property to scale any values that are written to the exported file.
- **Joint Stiffness**: This property is used to specify the stiffness of any rigidly restrained degrees of freedom for NASTRAN CBUSH joint elements (created from Mechanical MPC184 joint elements). This stiffness is used as-is and is unaffected by any unit conversions. A value greater than 1e6 may cause an ill-conditioned matrix in the NASTRAN solver.
- **Time**: In the case of a Mechanical model with time-varying applied loads, this property specifies which load values are to be used as the static loads for the export. Repeating the export with a different Time property value will calculate a new set of load values at the new time.
- **CPYRAM Elements**: This option suppresses (unchecked) or allows (checked) the creation of NASTRAN pyramid elements. Not all NASTRAN versions support pyramid elements. Using this option, you can customize the export to the capabilities of the target NASTRAN version.

Note:

If pyramid elements are present in the Mechanical model and this option is left unchecked (pyramid elements are suppressed), there will be voids in the exported mesh in place of the pyramid elements.

Modal Analysis Specific Properties

The modal analysis dialog includes the additional options described below.

Case Control: From this group of options, select the type of modal analysis you want the NASTRAN Export to do:

- **Modal**: Modal analysis of a constrained model, without pre-stress.
- **Modal (Free-Free)**: Modal analysis of an unconstrained model, without pre-stress. The first modes will be rigid body modes at nominally 0 Hz.
- **Modal (Pre-Stressed)**: Static analysis of a constrained model, using the exported applied loads, followed by a modal analysis of the pre-stressed model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Modal Options: This group contains the following properties/options:

- – **Modes**: Specify the number of modes (increasing in frequency) to be calculated.
  - – **Perform Ground Check**: This option displays when the **Modal (Free-Free)** Case Control is selected. Activating (checking) this option will direct that a ground check be performed to discover any unintentionally constrained rigid body modes.

NASTRAN Export Supported Features

The following table summarizes which Mechanical items are supported by NASTRAN Export, and the NASTRAN commands they are mapped to. The Mechanical items are categorized by Group and Subgroup. This table is only a guide and should not be used as the definitive list of supported items.

Group	Subgroup	Mechanical Item	Generated NASTRAN Command
Analysis	Modal	Modal	SOL103
		Modal (Free-Free) with Perform Ground Check	SOL103,GROUNDCHECK
		Modal (Pre-Stressed)	SOL103, STATSUB
	Static	Linear Static	SOL101
	Transient	Direct Transient	SOL109
Boundary Conditions	Acceleration	Acceleration by components	GRAV
	Displacement	Fixed Support, by vertex	SPC, spc id, grid id, 123456
		Fixed Support, by vertex	SPC, spc id, grid id, 123456
		Fixed Support, by vertex	SPC, spc id, grid id, 123456
		Nodal Displacement	SPC+SPCD pair
		Remote Displacement, rigid, by edge	SPC, spc id, grid id, dof
		remote Displacement, rigid, by face	SPC, spc id, grid id, dof
		Remote Displacement, rigid, by remote point	SPC, spc id, grid id, dof
		Remote Displacement, rigid, by vertex	SPC, spc id, grid id, dof
		Remote displacement, rigid, by vertex	SPC, spc id, grid id, dof
	Force	Force, by edge on beam	PLOAD1: P1 term
		Force, by face	PLOAD4: P1,N1,N2,N3 terms
		Line Pressure, by edge on beam	PLOAD1: P1 term
		Moment, by face	MOMENT
		Nodal Force	FORCE

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

		Pressure, by face	PLOAD4: P1 term
		Remote Force, by face	FORCE
	Nodal Orientation	Nodal Orientation object	GRID: CD term
	Thermal	Thermal Condition, by body (line/solid)	Thermal loads
		Thermal Condition, by body (surface top)	Thermal loads
Connection	Contact	MPC Formulation	MPC
	Joint	Bushing	CBUSH
		Fixed	CBUSH
		General	CBUSH
		Revolute, rotational stiffness	CBUSH
	Remote Point	Deformable, scoped to faces	RBE3
		Rigid, scoped to edges	RBE2
		Rigid, scoped to solid body face	RBE2
Coordinate System	Coordinate System	Defined by Geometry	CORD2*
		Defined by Named Selection	CORD2*
		Offset	CORD2*
		Rotated	CORD2*
	Material Orientation	Surface Cody Coordinate System	CQUAD*/CTRIA*: MCID term
Element	Beam	cross_section, ASEC	PBEAM
		cross_section, CHAN	PBEAML,CHAN
		cross_section, CSOLID	PBEAML,ROD
		cross_section, CTUBE	PBEAML,TUBE
		cross_section, HAT	PBEAML,HAT
		cross_section, HREC	PBEAML,BOX1
		cross_section, I	PBEAML,I
		cross_section, L	PBEAML,L
		cross_section, RECT	PBEAML,BAR
		cross_section, T	PBEAML,T2
		cross_section, Z	PBEAML,Z
		Line Body, Model Type:Beam, linear	CBEAM
	Link	Line Body, Model Type:link	CROD
		Line Body, Model Type:link	PROD
	Shell	Layered Section, by body	PCOMP
		Surface Body, linear	CQUAD4 / CTRIA3
		Surface Body, quadratic	CQUAD8 / CTRIA6
		Thickness, by body	PSHELL

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Solid	Solid Body, hex20	CHEXA
		Solid Body, hex8	CHEXA
		Solid Body, pyr13	CPYRAM
		Solid Body, pyr5	CPYRAM
		Solid Body, tet10	CTETRA
		Solid Body, tet4	CTETRA
	Spring	Longitudinal	CBEAM w/ end releases
		Longitudinal, force preload	CBEAM w/ end releases and TEMP
Mass	Distributed Mass	Scoped to Line Body	PBEAM*/PROD: NSM term
		Scoped to Surface Body	PSHELL/PCOMP: NSM term
	Point Mass	Direct vertex attachment	CONM2
		Remote attachment, vertex	CONM2+RBE2
Property	Material	Isotropic	MAT1
		Isotropic, Temperature Dependant	MAT1, MATT1, TABLEM1
		Orthotropic	MAT8, MAT12
	Shell	Preintegrated Shell Section	PSHELL + 4x MAT2

NASTRAN Export Limitations

The NASTRAN Export feature is currently subject to the limitations listed below.

General

The NASTRAN Export feature is only supported for stand-alone systems (Static Structural and Modal).

Supported Elements Types

Only the following element types are supported: 14, 21, 154, 156, 170, 173, 174, 175, 177, 180, 181, 184, 185, 186, 187, 188, 189, 201, 250, 281.

Not all features of the element types above are supported. Check the NASTRAN Export log file for messages about unsupported KEYOPT settings or element behaviors.

Pyramid Elements

Not all NASTRAN versions support pyramid-shaped 3D solid elements. For this reason, the NASTRAN Export ignores pyramid elements in the Mechanical model by default. The menu does have an option to include pyramid-shaped elements in the export. If pyramid elements are present in the Mechanical model and this option is set to ignore them (left unchecked), there will be voids in the exported mesh in place of the pyramid elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Properties

Element/Specification	Limitation
LINK180	This element Is exported as CROD elements with the torsional constant J equal to 0.1 x the cross-sectional area.
COMBIN14	For this element: • Body-to-ground preloaded springs are not supported. • The damping portion of the element contributes only damping coefficients to the structural damping matrix. • 3D torsional elements are not supported. • Preloaded springs are exported as beams with end releases and a temperature load applied. New properties are created for these equivalent beams. The beam sections have orientation vectors that reference the displacement coordinate system of the end nodes; this may result in an improper orientation vector if the beam nodes reference the basic coordinate system and the model is rotated.
COMBI250	For this element: • Assumes KEYOPT(1)=0, (lumped mass between nodes I and J). • Offsets are not supported.
Beam Sections	For beam section specifications: • Section offsets are assumed to be relative to the shear center of the beam. • 'Hat' cross-sections assume average section thicknesses and bottom flange widths. • 'Z' cross-sections assume average flange thicknesses and widths. • Tapered sections are not supported. • Warping constants, shear corrections, and shear relief are not supported. Also review the Beam and Pipe Cross Sections documentation in the *Mechanical APDL Structural Analysis Guide*.
Pre-integrated Shell Sections	Pre-integrated shell sections do not support density.
Shell Offsets	Only user-defined offsets are supported (**Top** and **Bottom** are not supported).

Connections

Remote Points: CTE is not supported for remote points or surface-based constraints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Distributed Mass

When scoped to a solid body, Distributed Mass applied to a/an:

- Face of a solid body is not supported.
- Edge of a solid body is not supported.

When scoped to a surface body, Distributed Mass applied to a/an:

- Surface body must be applied to all faces of the surface body.
- Edge of a surface body is not supported.
- Surface body and also scoped to a layered section or thickness must be scoped to all faces of the surface bodies within the layered section.

Scoped to a line body, Distributed Mass must be scoped to:

- All edges of the line body.
- Every body that has the same cross-section assignment.

Loading

For the **SFE**/**SFEBLOCK** commands:

- Imaginary components of SFE loads are not supported.
- Line pressures and forces defined on surface or solid body edges are not supported.
- An SFE load scoped to a line body assumes the element coordinate system is aligned with the global coordinate system and the load is uniformly applied over the length of element.

For the **BFE**/**BFEBLOCK** commands: Temperatures applied with **BFE**/**BFEBLOCK** commands are only supported for shell element types SHELL181 and SHELL281.

Tabular force loading is only supported for Transient analyses. Otherwise, loads applied using tabular data or functions are not supported.

Solution Options

Multiple load steps are not supported. Use the **Time** option in the NASTRAN Export menu to specify the time at which time-varying loads will be exported.

Understanding Solving Units

There are a number of possible unit systems provided for the solution process in the Mechanical application. The tables shown below present the solver unit systems for the various quantities. For a given solution, the application detects the specified unit system and then converts all analysis quantities into a consistent unit system prior to sending data to the solver. This makes sure that the application interprets

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

all unit systems for all quantities correctly for processing as the inputs and the outputs to the Mechanical APDL application. Note that the GUI may display units that are different than the ones shown below.

Note:

- Units of measure specified in **Commands** objects (p. 2312) are not converted when they are sent to the solver.
- All magnetostatic analyses solve in the mks system regardless of the user-defined system.
- Note the following Units (p. 45) options.
  - Metric (mm, kg, N, s, mV, mA)
  - Metric (mm, t, N, s, mV, mA)
  - Metric (mm, dat, N, s, mV, mA)

  These options differ only by Mass unit: kilogram (kg), metric ton (t), and decaton (dat). However, the consistent unit system used for the solver is the same for all quantities, i.e. Mass is measured in tons [t]. For the purposes of this discussion, the following descriptions use only the first option (mm, kg, N, s, mV, mA), to represent all three of the above interface options.
- All "ton" designations in the table indicate *metric ton*.
- For additional information on unit systems for use with LS-DYNA, see Solving Units in the LS-DYNA User's Guide.

Acceleration (p. 1876)	Angle (p. 1877)	Angular Acceleration (p. 1877)	Angular Velocity (p. 1878)	Area (p. 1878)
Capacitance (p. 1879)	Charge (p. 1880)	Charge Density (p. 1880)	Conductivity (p. 1881)	Current (p. 1881)
Current Density (p. 1882)	Decay Constant (p. 1882)	Density (p. 1883)	Displacement (p. 1883)	Electric Conductance Per Unit Area (p. 1884)
Electric Conductivity (p. 1885)	Electric Field (p. 1885)	Electric Flux Density (p. 1886)	Electric Resistivity (p. 1886)	Energy (p. 1887)
Energy Density by Mass (p. 1887)	Energy Per Volume (p. 1888)	Film Coefficient (p. 1889)	Force (p. 1889)	Force Intensity (p. 1890)
Force Per Angular Unit (p. 1891)	Fracture Energy (Energy Release Rate) (p. 1891)	Frequency (p. 1892)	Gasket Stiffness (p. 1892)	Heat Flux (p. 1893)
Heat Generation (p. 1893)	Heat Rate (p. 1894)	Impulse (p. 1894)	Impulse Per Angular Unit (p. 1895)	Inductance (p. 1895)
Inverse Angle (p. 1896)	Inverse Length (p. 1896)	Inverse Stress (p. 1897)	Length (p. 1897)	Magnetic Field Intensity (p. 1898)
Magnetic Flux (p. 1898)	Magnetic Flux Density (p. 1899)	Mass (p. 1899)	Material Impedance (p. 1900)	Moment (p. 1900)
Moment of Inertia of Area (p. 1901)	Moment of Inertia of Mass (p. 1902)	Normalized Value (p. 1902)	Permeability (p. 1903)	Permittivity (p. 1903)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Poisson's Ratio (p. 1904)	Power (p. 1904)	Pressure (p. 1905)	PSD Acceleration (p. 1906)	PSD Acceleration (G) (p. 1906)
PSD Displacement (p. 1907)	PSD Force (p. 1907)	PSD Moment (p. 1908)	PSD Pressure (p. 1908)	PSD Strain (p. 1909)
PSD Stress (p. 1909)	PSD Velocity (p. 1910)	Relative Permeability (p. 1910)	Relative Permittivity (p. 1911)	Rotational Damping (p. 1911)
Rotational Stiffness (p. 1912)	RS Acceleration (p. 1876)	RS Displacement (p. 1883)	RS Strain (p. 1917)	RS Stress (p. 1918)
RS Velocity (p. 1924)	Seebeck Coefficient (p. 1912)	Section Modulus (p. 1913)	Shear Elastic Strain (p. 1913)	Shock Velocity (p. 1914)
Specific Heat (p. 1914)	Specific Weight (p. 1915)	Square Root of Length (p. 1916)	Stiffness (p. 1916)	Strain (p. 1917)
Stress (p. 1918)	Stress Intensity Factor (p. 1919)	Strength (p. 1918)	Thermal Capacitance (p. 1920)	Thermal Conductance - 3D Face and 2D Edge (p. 1920)
Thermal Conductance - 3D Edge and Vertex (p. 1920)	Thermal Expansion (p. 1921)	Temperature (p. 1921)	Temperature Difference (p. 1922)	Temperature Gradient (p. 1923)
Time (p. 1923)	Translational Damping (p. 1924)	Velocity (p. 1924)	Voltage (p. 1925)	Volume (p. 1925)

Table 5: Acceleration and RS Acceleration

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters/second^2 [m/s^2]
cm, g, dyne, °C, s, V, A (cgs)	centimeters/second^2 [cm/s^2]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters/second^2 [mm/s^2]
µm, kg, µN, °C, s, V, mA (µmks)	micrometers/second^2 [µm/s^2]
ft, lbm, lbf, °F, s, V, A (Bft)	feet/second^2 [ft/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	inches/second^2 [in/s^2]
mm, mg, ms [Explicit Dynamics solvers]	millimeters/millisecond^2 [mm/ms^2]
m, kg, s [LS-DYNA solver]	meters/second^2 [m/s^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
mm, t, s [LS-DYNA solver]	millimeters/second^2 [mm/s^2]
in,lbf, s [LS-DYNA solver]	inches/second^2 [in/s^2]

Table 6: Angle

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	radians [rad]
cm, g, dyne, °C, s, V, A (cgs)	radians [rad]
mm, kg, N, °C, s, mV, mA (nmm)	radians [rad]
µm, kg, µN, °C, s, V, mA (µmks)	radians [rad]
ft, lbm, lbf, °F, s, V, A (Bft)	radians [rad]
in, lbm, lbf, °F, s, V, A (Bin)	radians [rad]

Table 7: Angular Acceleration

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	radians/second^2 [rad/s^2]
cm, g, dyne, °C, s, V, A (cgs)	radians/second^2 [rad/s^2]
mm, kg, N, °C, s, mV, mA (nmm)	radians/second^2 [rad/s^2]
µm, kg, µN, °C, s, V, mA (µmks)	radians/second^2 [rad/s^2]
ft, lbm, lbf, °F, s, V, A (Bft)	radians/second^2 [rad/s^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	radians/second^2 [rad/s^2]

Table 8: Angular Velocity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	radians/second [rad/s]
cm, g, dyne, °C, s, V, A (cgs)	radians/second [rad/s]
mm, kg, N, °C, s, mV, mA (nmm)	radians/second [rad/s]
µm, kg, µN, °C, s, V, mA (µmks)	radians/second [rad/s]
ft, lbm, lbf, °F, s, V, A (Bft)	radians/second [rad/s]
in, lbm, lbf, °F, s, V, A (Bin)	radians/second [rad/s]
mm, mg, ms [Explicit Dynamics solvers]	radians/millisecond [rad/ms]
m, kg, s [LS-DYNA solver]	radians/second [rad/s]
mm, t, s [LS-DYNA solver]	radians/second [rad/s]
in,lbf, s [LS-DYNA solver]	radians/second [rad/s]

Table 9: Area

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters^2 [m^2]
cm, g, dyne, °C, s, V, A (cgs)	centimeters^2 [cm^2]
mm, kg, N, °C, s, mV, mA	millimeters^2 [mm^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
µm, kg, µN, °C, s, V, mA (µmks)	micrometers^2 [µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	feet^2 [ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	inches^2 [in^2]
mm, mg, ms [Explicit Dynamics solvers]	millimeters^2 [mm^2]
m, kg, s [LS-DYNA solver]	meters^2 [m^2]
mm, t, s [LS-DYNA solver]	millimeters^2 [mm^2]
in,lbf, s [LS-DYNA solver]	inches^2 [in^2]

Table 10: Capacitance

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Farads [F]
cm, g, dyne, °C, s, V, A (cgs)	Farads [F]
mm, kg, N, °C, s, mV, mA (nmm)	microFarads [µF]
µm, kg, µN, °C, s, V, mA (µmks)	picoFarads [pF]
ft, lbm, lbf, °F, s, V, A (Bft)	Farads [F]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Farads [F]

Table 11: Charge

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Coulombs [C]
cm, g, dyne, °C, s, V, A (cgs)	Coulombs [C]
mm, kg, N, °C, s, mV, mA (nmm)	milliCoulombs [mC]
µm, kg, µN, °C, s, V, mA (µmks)	picoCoulombs [pC]
ft, lbm, lbf, °F, s, V, A (Bft)	Coulombs [C]
in, lbm, lbf, °F, s, V, A (Bin)	Coulombs [C]

Table 12: Charge Density

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Coulombs/meter^2 [C/m^2]
cm, g, dyne, °C, s, V, A (cgs)	Coulombs/centimeter^2 [C/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	milliCoulombs/millimeter^2 [mC/mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	picoCoulombs/micrometer^2 [pC/µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	Coulombs/foot^2 [C/ft^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Coulombs/inch^2 [C/in^2]

Table 13: Conductivity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts/meter * degree Celsius [W/m * °C]
cm, g, dyne, °C, s, V, A (cgs)	dynes/second * degree Celsius [dyne/s * °C]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters/second^3 * degree Celsius [t * mm/s^3 * °C]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts/micrometers * degree Celsius [pW/µm * °C]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet/second^3 * degree Fahrenheit [(lbm/32.2)ft/s^3 * °F]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches/second^3 * degree Fahrenheit [(lbm/386.4)in/s^3 * °F]

Table 14: Current

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Amperes [A]
cm, g, dyne, °C, s, V, A (cgs)	Amperes [A]
mm, kg, N, °C, s, mV, mA (nmm)	milliAmperes [mA]
µm, kg, µN, °C, s, V, mA (µmks)	picoAmperes [pA]
ft, lbm, lbf, °F, s, V, A (Bft)	Amperes [A]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Amperes [A]

Table 15: Current Density

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Amperes/meter^2 [A/m^2]
cm, g, dyne, °C, s, V, A (cgs)	Amperes/centimeter^2 [A/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	milliAmperes/millimeter^2 [mA/mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	milliAmperes/micrometer^2 [mA/µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	Amperes/foot^2 [A/ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	Amperes/inch^2 [A/in^2]

Table 16: Decay Constant

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	1/seconds [1/s]
cm, g, dyne, °C, s, V, A (cgs)	1/seconds [1/s]
mm, kg, N, °C, s, mV, mA (nmm)	1/seconds [1/s]
µm, kg, µN, °C, s, V, mA (µmks)	1/seconds [1/s]
ft, lbm, lbf, °F, s, V, A (Bft)	1/seconds [1/s]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	1/seconds [1/s]

Table 17: Density

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	kilograms/meter^3 [kg/m^3]
cm, g, dyne, °C, s, V, A (cgs)	grams/cm^3 [g/cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	tons/millimeter^3 [t/mm^3]
µm, kg, µN, °C, s, V, mA (µmks)	kilograms/micrometer^3 [kg/µm^3]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/foot^3 [(lbm/32.2)1/ft^3]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/inch^3 [(lbm/386.4)1/in^3]
mm, mg, ms [Explicit Dynamics solvers]	grams/cm^3 [g/cm^3]
m, kg, s [LS-DYNA solver]	kilograms/meter^3 [kg/m^3]
mm, t, s [LS-DYNA solver]	tons/millimeter^3 [t/mm^3]
in,lbf, s [LS-DYNA solver]	(slinch/1)/inch^3 [(lbm/386.4)1/in^3]

Table 18: Displacement and RS Displacement

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters [m]
cm, g, dyne, °C, s, V, A (cgs)	centimeters [cm]
mm, kg, N, °C, s, mV, mA	millimeters [mm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
μm, kg, μN, °C, s, V, mA (μmks)	micrometers [μm]
ft, lbm, lbf, °F, s, V, A (Bft)	feet [ft]
in, lbm, lbf, °F, s, V, A (Bin)	inches [in]
mm, mg, ms [Explicit Dynamics solvers]	millimeters [mm]
m, kg, s [LS-DYNA solver]	meters [m]
mm, t, s [LS-DYNA solver]	millimeters [mm]
in,lbf, s [LS-DYNA solver]	inches [in]

Table 19: Electric Conductance Per Unit Area

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Siemens/meter^2 [S/m^2]
cm, g, dyne, °C, s, V, A (cgs)	Siemens/centimeter^2 [S/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	Siemens/millimeter^2 [S/mm^2]
μm, kg, μN, °C, s, V, mA (μmks)	picoSiemens/micrometer^2 [pS/μm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	Siemens/foot^2 [S/ft^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Siemens/inch^2 [S/in^2]

Table 20: Electric Conductivity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Siemens/meter [S/m]
cm, g, dyne, °C, s, V, A (cgs)	Siemens/centimeter [S/cm]
mm, kg, N, °C, s, mV, mA (nmm)	Siemens/millimeter [S/mm]
µm, kg, µN, °C, s, V, mA (µmks)	picoSiemens/micrometer [pS/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	Siemens/foot [S/ft]
in, lbm, lbf, °F, s, V, A (Bin)	Siemens/inch [S/in]

Table 21: Electric Field

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Volts/meter [V/m]
cm, g, dyne, °C, s, V, A (cgs)	Volts/centimeter [V/cm]
mm, kg, N, °C, s, mV, mA (nmm)	milliVolts/millimeter [mV/mm]
µm, kg, µN, °C, s, V, mA (µmks)	Volts/micrometer [V/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	Volts/foot [V/ft]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Volts/inch [V/in]

Table 22: Electric Flux Density

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Coulombs/meter^2 [C/m^2]
cm, g, dyne, °C, s, V, A (cgs)	Coulombs/centimeter^2 [C/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	milliCoulombs/millimeter^2 [mC/mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	picoCoulombs/micrometer^2 [pC/µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	Coulombs/foot^2 [C/ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	Coulombs/inch^2 [C/in^2]

Table 23: Electric Resistivity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Ohm * meters [Ohm * m]
cm, g, dyne, °C, s, V, A (cgs)	Ohm * centimeters [Ohm * cm]
mm, kg, N, °C, s, mV, mA (nmm)	Ohm * millimeters [Ohm * mm]
µm, kg, µN, °C, s, V, mA (µmks)	teraOhm * micrometers [Tohm * µm]
ft, lbm, lbf, °F, s, V, A (Bft)	Ohm * Cir-mils/foot [Ohm * Cir-mil/ft]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Ohm * Cir-mils/inch [Ohm * Cir-mil/in]

Table 24: Energy

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules [J]
cm, g, dyne, °C, s, V, A (cgs)	ergs [erg]
mm, kg, N, °C, s, mV, mA (nmm)	milliJoules [mJ]
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules [pJ]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/second^2 [(lbm/32.2)ft^2/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches^2/second^2 [(lbm/386.4)in^2/s^2]
mm, mg, ms [Explicit Dynamics solvers]	microJoules [µJ]
m, kg, s [LS-DYNA solver]	Joules [J]
mm, t, s [LS-DYNA solver]	milliJoules [mJ]
in,lbf, s [LS-DYNA solver]	slinch * inches^2/second^2 [(lbm/386.4)in^2/s^2]

Table 25: Energy Density by Mass

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules/kilograms [J/kg]
cm, g, dyne, °C, s, V, A (cgs)	dynes * centimeters/grams [dyne * cm /g]
mm, kg, N, °C, s, mV, mA	milliJoules/tons [mJ/t]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules/kilograms [pJ/kg]
ft, lbm, lbf, °F, s, V, A (Bft)	$feet^2$ /$seconds^2$ [ft^2/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	$inches^2/seconds^2$ [in^2/sec^2]
mm, mg, ms [Explicit Dynamics solvers]	Joules/kilograms [J/kg]
m, kg, s [LS-DYNA solver]	Joules/kilograms [J/kg]
mm, t, s [LS-DYNA solver]	milliJoules/tons [mJ/t]
in,lbf, s [LS-DYNA solver]	$inches^2/seconds^2$ [in^2/sec^2]

Table 26: Energy Per Volume

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules/$meter^3$ [J/m^3]
cm, g, dyne, °C, s, V, A (cgs)	ergs/$centimeter^3$ [erg/cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	milliJoules/$millimeter^3$ [mJ/mm^3]
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules/$micrometer^3$ [pJ * um^3]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * $foot^2/second^2$ * $feet^3$[(lbm/32.2) * ft^2/s^2 * ft^3]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inch^2/second^2 * inch^3 [(lbm/386.4) * in^2/s^2 * in^3)]

Table 27: Film Coefficient

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts/meter^2 * degree Celsius [W/m^2 * °C]
cm, g, dyne, °C, s, V, A (cgs)	dyne/second * centimeter * degree Celsius [dyne/s * cm * °C]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^3 * degree Celsius [t/s^3 * °C]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts/micrometer^2 * degree Celsius [pW/µm^2 * °C]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^3 * degree Fahrenheit [(lbm/32.2)1/s^3 * °F]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^3 * degree Fahrenheit [(lbm/386.4)1/s^3 * °F]

Table 28: Force

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons [N]
cm, g, dyne, °C, s, V, A (cgs)	dynes [dyne]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters/second^2 [t * mm/s^2]
µm, kg, µN, °C, s, V, mA (µmks)	microNewtons [µN]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet/second^2 [(lbm/32.2)ft/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches/second^2 [(lbm/386.4)in/s^2]
mm, mg, ms	milliNewtons [mN]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
[Explicit Dynamics solvers]	
m, kg, s [LS-DYNA solver]	Newtons [N]
mm, t, s [LS-DYNA solver]	Newtons [N]
in,lbf, s [LS-DYNA solver]	pound force (lbf)

Table 29: Force Intensity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons/meter [N/m]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter [dyne/cm]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^2 [t/s^2]
µm, kg, µN, °C, s, V, mA (µmks)	microNewtons/micrometer [µN/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^2 [(lbm/32.2)1/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^2 [(lbm/386.4)1/s^2]
mm, mg, ms [Explicit Dynamics solvers]	Newtons/meter [N/m] or milliNewtons/millimeter [mN/mm]
m, kg, s [LS-DYNA solver]	Newtons/meter [N/m]
mm, t, s [LS-DYNA solver]	Newtons/millimeter [N/mm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in,lbf, s [LS-DYNA solver]	pound force/inch [lbf/in]

Table 30: Force Per Angular Unit

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons/radian [N/rad]
cm, g, dyne, °C, s, V, A (cgs)	dynes/radian [dyne/rad]
mm, kg, N, °C, s, mV, mA (nmm)	Newtons/radian [N/rad]
µm, kg, µN, °C, s, V, mA (µmks)	microNewtons/radian [µN/rad]
ft, lbm, lbf, °F, s, V, A (Bft)	pounds mass/radian [lbf/rad]
in, lbm, lbf, °F, s, V, A (Bin)	pounds mass/radian [lbf/rad]

Table 31: Fracture Energy (Energy Release Rate)

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules/meter^2 [J/m^2]
cm, g, dyne, °C, s, V, A (cgs)	erg/centimeter^2 [erg/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	milliJoules/millimeter^2 [mJ/mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules/micrometer^2 [pJ/µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/seconds^2 * feet^2 [(lbm-ft^2)/(s^2) * ft^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inch^2/seconds^2 * inch^2 [(lbm-in^2)/(s^2) * in^2

Table 32: Frequency

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Hertz[Hz]
cm, g, dyne, °C, s, V, A (cgs)	Hertz[Hz]
mm, kg, N, °C, s, mV, mA (nmm)	Hertz[Hz]
µm, kg, µN, °C, s, V, mA (µmks)	Hertz[Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	Hertz[Hz]
in, lbm, lbf, °F, s, V, A (Bin)	Hertz[Hz]

Table 33: Gasket Stiffness

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals/meter [Pa/m]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter^3 [dyne/cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^2 * millimeter^2 [t/s^2 * mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	megaPascals/micrometer [MPa/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	slug/second^2 * foot^2 [(lbm/32.2)/s^2 * ft^2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	slinch/second^2 * inch^2 [(lbm/386.4)/s^2 * in^2]

Table 34: Heat Flux

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts/meter^2 [W/m^2]
cm, g, dyne, °C, s, V, A (cgs)	dynes/second * centimeter [dyne/s * cm]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^3 [t/s^3]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts/micrometer^2 [pW/µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^3 [(lbm/32.2)1/s^3]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^3 [(lbm/386.4)1/s^3]

Table 35: Heat Generation

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts/meter^3 [W/m^3]
cm, g, dyne, °C, s, V, A (cgs)	dynes/second * centimeter^2 [dyne/s * cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^3 * millimeter [t/s^3 * mm]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts/micrometer^3 [pW/µm^3]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^3 * foot [(lbm/32.2)1/s^3 * ft]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^3 * inch [(lbm/386.4)1/s^3 * in]

Table 36: Heat Rate

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts [W]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeters/second [dyne * cm/s]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters^2/second^3 [t * mm^2/s^3]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts [pW]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/second^3 [(lbm/32.2) * ft^2/s^3]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches^2/second^3 [(lbm/386.4) * in^2/s^3]

Table 37: Impulse

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * second [N * s]
cm, g, dyne, °C, s, V, A (cgs)	dyne * second [dyne * s]
mm, kg, N, °C, s, mV, mA (nmm)	Newton * second [N * s]
µm, kg, µN, °C, s, V, mA (µmks)	microNewton * second [µN * s]
ft, lbm, lbf, °F, s, V, A (Bft)	pounds mass * second [lbf * s]
in, lbm, lbf, °F, s, V, A (Bin)	pounds mass * second [lbf * s]
mm, mg, ms	microNewton * second [µN * s]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
[Explicit Dynamics solvers]	
m, kg, s [LS-DYNA solver]	Newton * second [N * s]
mm, t, s [LS-DYNA solver]	Newton * second [N * s]
in,lbf, s [LS-DYNA solver]	pound force * second (lbf * second)

Table 38: Impulse Per Angular Unit

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * second/rad [N * s/rad]
cm, g, dyne, °C, s, V, A (cgs)	dyne * second/radian [dyne * s/rad]
mm, kg, N, °C, s, mV, mA (nmm)	Newton * second/rad [N * s/rad]
µm, kg, µN, °C, s, V, mA (µmks)	microNewton * second/radian [µN * s/rad]
ft, lbm, lbf, °F, s, V, A (Bft)	pounds mass * second/radian [lbf * s/rad]
in, lbm, lbf, °F, s, V, A (Bin)	pounds mass * second/radian [lbf * s/rad]

Table 39: Inductance

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Henries [H]
cm, g, dyne, °C, s, V, A (cgs)	Henries [H]
mm, kg, N, °C, s, mV, mA (nmm)	milliHenries [mH]
µm, kg, µN, °C, s, V, mA (µmks)	teraHenries [TH]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
ft, lbm, lbf, °F, s, V, A (Bft)	Henries [H]
in, lbm, lbf, °F, s, V, A (Bin)	Henries [H]

Table 40: Inverse Angle

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	1/radians [1/rad]
cm, g, dyne, °C, s, V, A (cgs)	1/radians [1/rad]
mm, kg, N, °C, s, mV, mA (nmm)	1/radians [1/rad]
µm, kg, µN, °C, s, V, mA (µmks)	1/radians [1/rad]
ft, lbm, lbf, °F, s, V, A (Bft)	1/radians [1/rad]
in, lbm, lbf, °F, s, V, A (Bin)	1/radians [1/rad]

Note:

The units presented above are applicable when the **Units** are set to **Radians**. The applicable units are 1/degree [1/°] when the **Units** are set to **Degrees**.

Table 41: Inverse Length

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	1/meter [1/m]
cm, g, dyne, °C, s, V, A (cgs)	1/centimeter [1/cm]
mm, kg, N, °C, s, mV, mA (nmm)	1/millimeter [1/mm]
µm, kg, µN, °C, s, V, mA	1/micrometer [1/µm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(µmks)	
ft, lbm, lbf, °F, s, V, A (Bft)	1/foot [1/ft]
in, lbm, lbf, °F, s, V, A (Bin)	1/inch [1/in]

Table 42: Inverse Stress

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	1/Pascal [1/Pa]
cm, g, dyne, °C, s, V, A (cgs)	centimeters^2/dyne [cm^2/dyne]
mm, kg, N, °C, s, mV, mA (nmm)	second^2 * millimeters/ton [s^2 * mm/t]
µm, kg, µN, °C, s, V, mA (µmks)	1/megaPascal [1/MPa]
ft, lbm, lbf, °F, s, V, A (Bft)	second^2 * feet/slug [s^2 * ft/(lbm/32.2)]
in, lbm, lbf, °F, s, V, A (Bin)	second^2 * inch/slinch [s^2 * in/(lbm/386.4)]

Table 43: Length

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters [m]
cm, g, dyne, °C, s, V, A (cgs)	centimeters [cm]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters [mm]
µm, kg, µN, °C, s, V, mA (µmks)	micrometers [µm]
ft, lbm, lbf, °F, s, V, A (Bft)	feet [ft]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	inches [in]
mm, mg, ms [Explicit Dynamics solvers]	millimeters [mm]
m, kg, s [LS-DYNA solver]	meters [m]
mm, t, s [LS-DYNA solver]	millimeters [mm]
in,lbf, s [LS-DYNA solver]	inches [in]

Table 44: Magnetic Field Intensity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Amperes/meter [A/m]
cm, g, dyne, °C, s, V, A (cgs)	Oersteds [Oe]
mm, kg, N, °C, s, mV, mA (nmm)	milliAmperes/millimeter [mA/mm]
µm, kg, µN, °C, s, V, mA (µmks)	picoAmperes/micrometer [pA/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	Amperes/foot [A/ft]
in, lbm, lbf, °F, s, V, A (Bin)	Amperes/inch [A/in]

Table 45: Magnetic Flux

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Webers [Wb]
cm, g, dyne, °C, s, V, A (cgs)	Maxwells [Mx]
mm, kg, N, °C, s, mV, mA	milliWebers [mWb]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
μm, kg, μN, °C, s, V, mA (μmks)	Webers [Wb]
ft, lbm, lbf, °F, s, V, A (Bft)	Lines
in, lbm, lbf, °F, s, V, A (Bin)	Lines

Table 46: Magnetic Flux Density

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Teslas [T]
cm, g, dyne, °C, s, V, A (cgs)	Gauss [G]
mm, kg, N, °C, s, mV, mA (nmm)	milliTeslas [mT]
μm, kg, μN, °C, s, V, mA (μmks)	teraTeslas [TT]
ft, lbm, lbf, °F, s, V, A (Bft)	Lines/foot^2 [lines/ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	Lines/inch^2 [lines/in^2]

Table 47: Mass

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	kilograms [kg]
cm, g, dyne, °C, s, V, A (cgs)	grams [g]
mm, kg, N, °C, s, mV, mA (nmm)	kilograms [kg]
μm, kg, μN, °C, s, V, mA (μmks)	kilograms [kg]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
ft, lbm, lbf, °F, s, V, A (Bft)	slug [lbm/32.2]
in, lbm, lbf, °F, s, V, A (Bin)	slinch [lbm/386.4]
mm, mg, ms [Explicit Dynamics solvers]	milligrams [mg]
m, kg, s [LS-DYNA solver]	kilograms [kg]
mm, t, s [LS-DYNA solver]	tons [t]
in,lbf, s [LS-DYNA solver]	slinch [lbm/386.4]

Table 48: Material Impedance

Unit System	Measured in . . .
mm, mg, ms [Explicit Dynamics solvers]	milligrams/millimeter^2/second [mg/mm^2/s]
m, kg, s [LS-DYNA solver]	kilograms/meter^2/second [kg/m^2/s]
mm, t, s [LS-DYNA solver]	tons/millimeter^2/second [t/mm^2/s]
in,lbf, s [LS-DYNA solver]	slinch/inch^2/second [slinch/in^2/s]

Table 49: Moment

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * meters [N * m]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeters [dyne * cm]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters^2/second^2 [t * mm^2/s^2]
µm, kg, µN, °C, s, V, mA	microNewton * micrometers [µN * µm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(µmks)	
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/second^2 [(lbm/32.2) * ft^2/s^2]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches^2/second^2 [(lbm/386.4) * in^2/s^2]
mm, mg, ms [Explicit Dynamics solvers]	microNewton * meters [µN * m]
m, kg, s [LS-DYNA solver]	Newton * meters [N * m]
mm, t, s [LS-DYNA solver]	Newton * millimeters [N * mm]
in,lbf, s [LS-DYNA solver]	pound force * inch [lbf * in]

Table 50: Moment of Inertia of Area

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters^4 [m^4]
cm, g, dyne, °C, s, V, A (cgs)	centimeters^4 [cm^4]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters^4 [mm^4]
µm, kg, µN, °C, s, V, mA (µmks)	micrometers^4 [µm^4]
ft, lbm, lbf, °F, s, V, A (Bft)	feet^4 [ft^4]
in, lbm, lbf, °F, s, V, A (Bin)	inches^4 [in^4]
mm, mg, ms [Explicit Dynamics solvers]	millimeters^4 [mm^4]
m, kg, s [LS-DYNA solver]	meters^4 [m^4]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
mm, t, s [LS-DYNA solver]	millimeters^4 [mm^4]
in,lbf, s [LS-DYNA solver]	inches^4 [in^4]

Table 51: Moment of Inertia of Mass

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	kilogram * meter^2 [kg * m^2]
cm, g, dyne, °C, s, V, A (cgs)	gram * centimeter^2 [g * cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeter^2 [t * mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	kilogram * micrometer^2 [kg * µm^2]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2 [(lbm/32.2) * ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inch^2 [(lbm/386.4) * in^2]
mm, mg, ms [Explicit Dynamics solvers]	milligram * millimeter^2 [mg * mm^2]
m, kg, s [LS-DYNA solver]	kilogram * meter^2 [kg * m^2]
mm, t, s [LS-DYNA solver]	ton * millimeter^2 [t * mm^2]
in,lbf, s [LS-DYNA solver]	slinch * inch^2 [slinch * in^2]

Table 52: Normalized Value

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	*unitless*
cm, g, dyne, °C, s, V, A	*unitless*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(cgs)	
mm, kg, N, °C, s, mV, mA (nmm)	*unitless*
μm, kg, μN, °C, s, V, mA (μmks)	*unitless*
ft, lbm, lbf, °F, s, V, A (Bft)	*unitless*
in, lbm, lbf, °F, s, V, A (Bin)	*unitless*

Table 53: Permeability

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Henries/meter [H/m]
cm, g, dyne, °C, s, V, A (cgs)	Henries/centimeter [H/cm]
mm, kg, N, °C, s, mV, mA (nmm)	milliHenries/millimeter [mH/mm]
μm, kg, μN, °C, s, V, mA (μmks)	teraHenries/micrometer [TH/μm]
ft, lbm, lbf, °F, s, V, A (Bft)	Henries/foot [H/ft]
in, lbm, lbf, °F, s, V, A (Bin)	Henries/inch [H/in]

Table 54: Permittivity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Farads/meter [F/m]
cm, g, dyne, °C, s, V, A (cgs)	Farads/centimeter [F/cm]
mm, kg, N, °C, s, mV, mA (nmm)	microFarads/millimeter [μF/mm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
µm, kg, µN, °C, s, V, mA (µmks)	picoFarads/micrometer [pF/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	Farads/foot [F/ft]
in, lbm, lbf, °F, s, V, A (Bin)	Farads/inch [F/in]

Table 55: Poisson's Ratio

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	*unitless*
cm, g, dyne, °C, s, V, A (cgs)	*unitless*
mm, kg, N, °C, s, mV, mA (nmm)	*unitless*
µm, kg, µN, °C, s, V, mA (µmks)	*unitless*
ft, lbm, lbf, °F, s, V, A (Bft)	*unitless*
in, lbm, lbf, °F, s, V, A (Bin)	*unitless*
mm, mg, ms [Explicit Dynamics solvers]	*unitless*
m, kg, s [LS-DYNA solver]	*unitless*
mm, t, s [LS-DYNA solver]	*unitless*
in,lbf, s [LS-DYNA solver]	*unitless*

Table 56: Power

Unit System	Measured in . . .
m, kg, N, °C, s, V, A	Watts [W]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(mks)	
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeters/second [dyne * c/s]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters^2/second^3 [t * mm^2/s^3]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts [pW]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/second^3 [(lbm/32.2) * ft^2/s^3]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches^2/second^3 [(lbm/386.4) * in^2/s^3]
mm, mg, ms [Explicit Dynamics solvers]	milliWatts [mW]
m, kg, s [LS-DYNA solver]	Watts [W]
mm, t, s [LS-DYNA solver]	Newton * millimeters/second [N * mm/s]
in,lbf, s [LS-DYNA solver]	pound force * inch/second [lbf * in/s]

Table 57: Pressure

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals [Pa]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter^2 [dyne/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	ton/second^2/millimeters [t/s^2/mm]
µm, kg, µN, °C, s, V, mA (µmks)	megaPascals [MPa]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^2/foot [(lbm/32.2)1/s^2/ft]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^2/inch [(lbm/386.4)1/s^2/in]
mm, mg, ms [Explicit Dynamics solvers]	kiloPascals [kPa]
m, kg, s [LS-DYNA solver]	Pascals [Pa]
mm, t, s [LS-DYNA solver]	megaPascals [MPa]
in,lbf, s [LS-DYNA solver]	pounds/inch^2 [lb/in^2]

Table 58: PSD Acceleration

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	(meters/second^2)^2/Hertz [(m/s^2)^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(centimeters/second^2)^2/Hertz [(cm/s^2)^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	(millimeters/second^2)^2/Hertz [(mm/s^2)^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	(micrometers/second^2)^2/megahertz [(µm/s^2)^2/MHz]
ft, lbm, lbf, °F, s, V, A (Bft)	(feet/second^2)^2/Hertz [(ft/s^2)^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	(inch/second^2)^2/Hertz [(in/s^2)^2/Hz]

Table 59: PSD Acceleration (G)

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	G^2/Hertz [G^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	G^2/Hertz [G^2/Hz]
mm, kg, N, °C, s, mV, mA	G^2/Hertz [G^2/Hz]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
μm, kg, μN, °C, s, V, mA (μmks)	G^2/Hertz [G^2/Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	G^2/Hertz [G^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	G^2/Hertz [G^2/Hz]

Table 60: PSD Displacement

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters^2/Hertz [m^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	centimeters^2/Hertz [cm^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters^2/Hertz [mm^2/Hz]
μm, kg, μN, °C, s, V, mA (μmks)	micrometers^2/megaHertz [μm^2/MHz]
ft, lbm, lbf, °F, s, V, A (Bft)	feet^2/Hertz [ft^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	inches^2/Hertz [in^2/Hz]

Table 61: PSD Force

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons^2/Hertz [N^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	dynes^2/Hertz [dyne^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	((kilograms·millimeters)/second^2)^2/Hertz [((kg * mm)/s^2)^2/Hz]
μm, kg, μN, °C, s, V, mA (μmks)	microNewtons^2/Hertz [μN^2/Hz]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
ft, lbm, lbf, °F, s, V, A (Bft)	((pounds * mass/32.2) * feet)/second^2))^2/Hertz [((lb * m/32.2) * ft/s^2))^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	((pounds * mass/32.2) * inches)/second^2))^2/Hertz [((lb * m/32.2) * in/s^2))^2/Hz]

Table 62: PSD Moment

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	(Newtons * meters)^2/Hertz [(N * m)^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(dynes * centimeters)2/Hertz [(dyne * cm)2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	((kilograms * millimeters^2)/Second^2)^2/Hertz [((kg * mm^2)/s^2)^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	(microNewtons * micrometers)^2/Hertz [(µN * µm)^2/Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	((pounds * mass/32.2) * feet^2)/second^2)^2/Hertz [((lb * m/32.2) * ft^2)/s^2)^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	((pounds * mass/386.4) * inches^2)/second^2)^2/Hertz [((lb * m/386.4) * in^2)/s^2)^2/Hz]

Table 63: PSD Pressure

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals^2/Hertz [Pa^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(dynes/centimeter^2)^2/Hertz [(dyne/cm^2)^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	(kilograms/(millimeter * second^2))^2/Hertz [(kg/(mm * s^2))^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	megaNewtons^2/Hertz [MPa^2/Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/(foot * second^2))^2/Hertz [((lbm/32.2)/(ft * s^2))^2/Hz]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/(inch * second^2))^2/Hertz [((lbm/386.4)/(in * s^2))^2/Hz]

Table 64: PSD Strain

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	(meters/meter)^2/Hertz [(m/m)^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(centimeters/centimeter)^2/Hertz [(cm/cm)^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	(millimeters/millimeter)^2/Hertz [(mm/mm)^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	(micrometers/micrometer)^2/Hertz [(µm/µm)^2/Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	(feet/foot)^2/Hertz [(ft/ft)^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	(inches/inch)^2/Hertz [(in/in)^2/Hz]

Table 65: PSD Stress

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals^2/Hertz [Pa^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(dynes/centimeter^2)^2/Hertz [(dyne/cm^2)^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	(kilograms/(millimeter * second^2))^2/Hertz [(kg/(mm * s^2))^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	megaNewtons^2/Hertz [MPa^2/Hz]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/(foot * second^2))^2/Hertz [((lbm/32.2)/(ft * s^2))^2/Hz]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/(inch * second^2))^2/Hertz [((lbm/386.4)/(in * s^2))^2/Hz]

Table 66: PSD Velocity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	(meters/second)^2/Hertz [(m/s)^2/Hz]
cm, g, dyne, °C, s, V, A (cgs)	(centimeters/second)^2/Hertz [(cm/s)^2/Hz]
mm, kg, N, °C, s, mV, mA (nmm)	(millimeters/second)^2/Hertz [(mm/s)^2/Hz]
µm, kg, µN, °C, s, V, mA (µmks)	(micrometers/second)^2/megahertz [(µm/s)^2/MHz]
ft, lbm, lbf, °F, s, V, A (Bft)	(feet/second)^2/Hertz [(ft/s)^2/Hz]
in, lbm, lbf, °F, s, V, A (Bin)	(inches/second)^2/Hertz [(in/s)^2/Hz]

Table 67: Relative Permeability

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	*unitless*
cm, g, dyne, °C, s, V, A (cgs)	*unitless*
mm, kg, N, °C, s, mV, mA (nmm)	*unitless*
µm, kg, µN, °C, s, V, mA (µmks)	*unitless*
ft, lbm, lbf, °F, s, V, A (Bft)	*unitless*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	*unitless*

Table 68: Relative Permittivity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	*unitless*
cm, g, dyne, °C, s, V, A (cgs)	*unitless*
mm, kg, N, °C, s, mV, mA (nmm)	*unitless*
µm, kg, µN, °C, s, V, mA (µmks)	*unitless*
ft, lbm, lbf, °F, s, V, A (Bft)	*unitless*
in, lbm, lbf, °F, s, V, A (Bin)	*unitless*

Table 69: Rotational Damping

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * meter * seconds/radian [N * m * s/rad]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeter * seconds/radian [dyne * cm * s/rad]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeter^2 * seconds/second^2 * radian [t * mm^2 * s/s^2 * rad]
µm, kg, µN, °C, s, V, mA (µmks)	microNewton * micrometer * seconds/radian [µN * µm * s/rad]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * foot^2 * seconds/second^2 * radian [(lbm/32.2) * ft^2 * s/s^2 * rad]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inch^2 * seconds/second^2 * radian [(lbm/386.4) * in^2 * s/s^2 * rad]

Table 70: Rotational Stiffness

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * meters/radian [N * m/rad]
cm, g, dyne, °C, s, V, A (cgs)	dynes * centimeters/radian [dyne * cm/rad]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeters^2/second^2 * radian [t * mm^2/s^2 * rad]
µm, kg, µN, °C, s, V, mA (µmks)	microNewton * micrometers/radian [µN*µm/rad]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * feet^2/second^2 * radian [(lbm/32.2) * ft^2/s^2 * rad]
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inches^2/second^2 * radian [(lbm/386.4) * in^2/s^2 * rad]

Table 71: Seebeck Coefficient

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Volts/degree Celsius [V/°C]
cm, g, dyne, °C, s, V, A (cgs)	Volts/degree Celsius [V/°C]
mm, kg, N, °C, s, mV, mA (nmm)	milliVolts/degree Celsius [mV/°C]
µm, kg, µN, °C, s, V, mA (µmks)	Volts/degree Celsius [V/°C]
ft, lbm, lbf, °F, s, V, A (Bft)	Volts/degree Fahrenheit [V/°F]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	Volts/degree Fahrenheit [V/°F]

Table 72: Section Modulus

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters^3 [m^3]
cm, g, dyne, °C, s, V, A (cgs)	centimeters^3 [cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters^3 [mm^3]
µm, kg, µN, °C, s, V, mA (µmks)	micrometers^3 [µm^3]
ft, lbm, lbf, °F, s, V, A (Bft)	feet^3 [ft^3]
in, lbm, lbf, °F, s, V, A (Bin)	inches^3 [in^3]
mm, mg, ms [Explicit Dynamics solvers]	millimeters^3 [mm^3]
m, kg, s [LS-DYNA solver]	meters^3 [m^3]
mm, t, s [LS-DYNA solver]	millimeters^3 [mm^3]
in,lbf, s [LS-DYNA solver]	inch^3 [in^3]

Table 73: Shear Elastic Strain

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	radians [rad]
cm, g, dyne, °C, s, V, A (cgs)	radians [rad]
mm, kg, N, °C, s, mV, mA	radians [rad]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
µm, kg, µN, °C, s, V, mA (µmks)	radians [rad]
ft, lbm, lbf, °F, s, V, A (Bft)	radians [rad]
in, lbm, lbf, °F, s, V, A (Bin)	radians [rad]

Table 74: Shock Velocity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	seconds/meters [s/m]
cm, g, dyne, °C, s, V, A (cgs)	seconds/centimeters [s/cm]
mm, kg, N, °C, s, mV, mA (nmm)	seconds/millimeters [s/mm]
µm, kg, µN, °C, s, V, mA (µmks)	seconds/micrometers [s/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	seconds/feet [s/ft]
in, lbm, lbf, °F, s, V, A (Bin)	seconds/inches [s/in]

Table 75: Specific Heat

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules/kilogram * degree Celsius [J/kg * °C]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeters/gram * degree Celsius [dyne*cm/g * °C]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters^2/second^2 * degree Celsius [mm^2/s^2 * °C]
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules/kilogram * degree Celsius [pJ/kg * °C]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
ft, lbm, lbf, °F, s, V, A (Bft)	feet^2/second^2 * degree Fahrenheit [ft^2/s^2 * °F]
in, lbm, lbf, °F, s, V, A (Bin)	inches^2/second^2 * degree Fahrenheit [in^2/s^2 * °F]
mm, mg, ms [Explicit Dynamics solvers]	Joules/kilogram * degree Kelvin [J/kg * °K]
m, kg, s [LS-DYNA solver]	Joules/kilogram/degree Kelvin [J/kg/°K]
mm, t, s [LS-DYNA solver]	milliJoules/ton/degree Kelvin [mJ/t/°K]
in,lbf, s [LS-DYNA solver]	inch^2/second^2/°F [in^2/s^2/°F]

Table 76: Specific Weight

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons/meter^3 [N/m^3]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter^3 [dyne/cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	tons/second^2 * millimeters^2 [t/s^2 * mm^2]
µm, kg, µN, °C, s, V, mA (µmks)	microNewtons/micrometer^3 [µN/µm^3]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^2 * feet^2 [(lbm/32.2)1/s^2 * ft^2]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^2 * inch^2 [(lbm/386.4)1/s^2 * in^2]
mm, mg, ms [Explicit Dynamics solvers]	megaNewtons/meter^3 [MN/m^3]
m, kg, s [LS-DYNA solver]	Newtons/meter^3 [N/m^3]
mm, t, s	Newtons/millimeter^3 [N/mm^3]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
[LS-DYNA solver]	
in,lbf, s [LS-DYNA solver]	pound force/inch^3 [lbf/in^3]

Table 77: Square Root of Length

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meter$^{0.5}$ [m$^{0.5}$]
cm, g, dyne, °C, s, V, A (cgs)	centimeter$^{0.5}$ [cm$^{0.5}$]
mm, kg, N, °C, s, mV, mA (nmm)	millimeter$^{0.5}$ [mm$^{0.5}$]
µm, kg, µN, °C, s, V, mA (µmks)	micrometer$^{0.5}$ [µm$^{0.5}$]
ft, lbm, lbf, °F, s, V, A (Bft)	feet$^{0.5}$ [ft$^{0.5}$]
in, lbm, lbf, °F, s, V, A (Bin)	inch$^{0.5}$ [in$^{0.5}$]

Table 78: Stiffness

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newtons/meter [N/m]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter [dyne/cm]
mm, kg, N, °C, s, mV, mA (nmm)	Newtons/millimeter [N/mm]
µm, kg, µN, °C, s, V, mA (µmks)	microNewtons/micrometer [µN/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	pound force/foot [lbf/ft]
in, lbm, lbf, °F, s, V, A (Bin)	pound force/inch [lbf/in]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
mm, mg, ms [Explicit Dynamics solvers]	Newtons/meter [N/m] or milliNewtons/millimeter [mN/mm]
m, kg, s [LS-DYNA solver]	Newtons/meter [N/m]
mm, t, s [LS-DYNA solver]	Newtons/millimeter [N/m]
in,lbf, s [LS-DYNA solver]	pound force/inch [lbf/in]

Table 79: Strain and RS Strain

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meter/meter [m/m]
cm, g, dyne, °C, s, V, A (cgs)	centimeter/centimeter [cm/cm]
mm, kg, N, °C, s, mV, mA (nmm)	millimeter/millimeter [mm/mm]
µm, kg, µN, °C, s, V, mA (µmks)	micrometer/micrometer [µm/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	feet/foot [ft/ft]
in, lbm, lbf, °F, s, V, A (Bin)	inch/inch [in/in]
mm, mg, ms [Explicit Dynamics solvers]	millimeter/millimeter [mm/mm]
m, kg, s [LS-DYNA solver]	meter/meter [m/m]
mm, t, s [LS-DYNA solver]	millimeter/millimeter [mm/mm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in,lbf, s [LS-DYNA solver]	inch/inch [in/in]

Table 80: Strength

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals [Pa]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter^2 [dyne/cm^2]
mm, kg, N, °C, s, mV, mA (nmm)	ton/second^2 * millimeters [t/s^2 * mm]
µm, kg, µN, °C, s, V, mA (µmks)	megaPascals [MPa]
ft, lbm, lbf, °F, s, V, A (Bft)	(slug/1)/second^2 * foot [(lbm/32.2)1/s^2 * ft]
in, lbm, lbf, °F, s, V, A (Bin)	(slinch/1)/second^2 * inch [(lbm/386.4)1/s^2 * in]
mm, mg, ms [Explicit Dynamics solvers]	kiloPascals [kPa]
m, kg, s [LS-DYNA solver]	Pascals [Pa]
mm, t, s [LS-DYNA solver]	megaPascals [MPa]
in,lbf, s [LS-DYNA solver]	pounds/inch^2 [lb/in^2]

Table 81: Stress and RS Stress

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascals [Pa]
cm, g, dyne, °C, s, V, A (cgs)	dynes/centimeter^2 [dyne/cm^2]
mm, kg, N, °C, s, mV, mA	ton/second^2 * millimeters [t/s^2 * mm]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
µm, kg, µN, °C, s, V, mA (µmks)	megaPascals [MPa]
ft, lbm, lbf, °F, s, V, A (Bft)	slug/second^2 * foot [(lbm/32.2)/s^2 * ft]
in, lbm, lbf, °F, s, V, A (Bin)	slinch/second^2 * inch [(lbm/386.4)/s^2 * in]
mm, mg, ms [Explicit Dynamics solvers]	kiloPascals [kPa]
m, kg, s [LS-DYNA solver]	Pascals [Pa]
mm, t, s [LS-DYNA solver]	megaPascals [MPa]
in,lbf, s [LS-DYNA solver]	pounds/inch^2 [lb/in^2]

Table 82: Stress Intensity Factor

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Pascal * meter$^{0.5}$ [Pa * m$^{0.5}$]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeter^{-2} * centimeter$^{0.5}$ [dyne * cm^{-2}* cm$^{0.5}$]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeter^{-1} * second^{-2} * millimeter$^{0.5}$ [ton * mm^{-1} * s^{-2} * mm$^{0.5}$]
µm, kg, µN, °C, s, V, mA (µmks)	megaPascal * micrometer$^{0.5}$ [MPa * µm$^{0.5}$]
ft, lbm, lbf, °F, s, V, A (Bft)	lbm_ft * feet^{-1} * second^{-2} * feet$^{0.5}$ [lbm_ft * ft^{-1} * s^{-2} * ft$^{0.5}$]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	lbm_inch * $inch^{-1}$ * $second^{-2}$ * $inch^{0.5}$ [lbm_in * in^{-1} * s^{-2} * $in^{0.5}$]

Table 83: Thermal Capacitance

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Joules/degree Celsius [J/°C]
cm, g, dyne, °C, s, V, A (cgs)	ergs/degree Celsius [erg/°C]
mm, kg, N, °C, s, mV, mA (nmm)	milliJoules/degree Celsius [mJ/°C]
µm, kg, µN, °C, s, V, mA (µmks)	picoJoules/degree Celsius [pJ/°C]
ft, lbm, lbf, °F, s, V, A (Bft)	BTU/degree Fahrenheit [BTU/°F]
in, lbm, lbf, °F, s, V, A (Bin)	BTU/degree Fahrenheit [BTU/°F]

Table 84: Thermal Conductance - 3D Face and 2D Edge

For **Thermal Conductance** on 3D Faces and 2D Edges, the units are the same as those for Film Coefficient (p. 1889).

Table 85: Thermal Conductance - 3D Edges and Vertices

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Watts/degree Celsius [W/°C]
cm, g, dyne, °C, s, V, A (cgs)	dyne * centimeter/second * degree Celsius [dyne * cm/s * °C]
mm, kg, N, °C, s, mV, mA (nmm)	ton * $millimeter^2/second^3$ * degree Celsius [t * mm^2/s^3 * °C]
µm, kg, µN, °C, s, V, mA (µmks)	picoWatts/degree Celsius [pW/°C]
ft, lbm, lbf, °F, s, V, A (Bft)	slug * $feet^2/second^3$ * degree Fahrenheit [slug * ft^2/s^3 * °F]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
in, lbm, lbf, °F, s, V, A (Bin)	slinch * inch^2/second^3 * degree Fahrenheit [slinch * in^2/s^3 * °F]

Table 86: Thermal Expansion

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	1/degree Celsius [1/°C]
cm, g, dyne, °C, s, V, A (cgs)	1/degree Celsius [1/°C]
mm, kg, N, °C, s, mV, mA (nmm)	1/degree Celsius [1/°C]
µm, kg, µN, °C, s, V, mA (µmks)	1/degree Celsius [1/°C]
ft, lbm, lbf, °F, s, V, A (Bft)	1/degree Fahrenheit [1/°F]
in, lbm, lbf, °F, s, V, A (Bin)	1/degree Fahrenheit [1/°F]
mm, mg, ms [Explicit Dynamics solvers]	microJoules/degree Kelvin [µJ/°K]
m, kg, s [LS-DYNA solver]	1/degree Kelvin [1/°K]
mm, t, s [LS-DYNA solver]	1/degree Kelvin [1/°K]
in,lbf, s [LS-DYNA solver]	1/degree Fahrenheit [1/°F]

Table 87: Temperature

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	degrees Celsius [°C]
cm, g, dyne, °C, s, V, A (cgs)	degrees Celsius [°C]
mm, kg, N, °C, s, mV, mA	degrees Celsius [°C]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(nmm)	
µm, kg, µN, °C, s, V, mA (µmks)	degrees Celsius [°C]
ft, lbm, lbf, °F, s, V, A (Bft)	degrees Fahrenheit [°F]
in, lbm, lbf, °F, s, V, A (Bin)	degrees Fahrenheit [°F]
mm, mg, ms [Explicit Dynamics solvers]	degrees Kelvin [°K]
m, kg, s [LS-DYNA solver]	degrees Kelvin [°K]
mm, t, s [LS-DYNA solver]	degrees Kelvin [°K]
in,lbf, s [LS-DYNA solver]	degrees Fahrenheit [°F]

Table 88: Temperature Difference

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	degrees Celsius [°C]
cm, g, dyne, °C, s, V, A (cgs)	degrees Celsius [°C]
mm, kg, N, °C, s, mV, mA (nmm)	degrees Celsius [°C]
µm, kg, µN, °C, s, V, mA (µmks)	degrees Celsius [°C]
ft, lbm, lbf, °F, s, V, A (Bft)	degrees Fahrenheit [°F]
in, lbm, lbf, °F, s, V, A (Bin)	degrees Fahrenheit [°F]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
mm, mg, ms [Explicit Dynamics solvers]	degrees Kelvin [°K]

Table 89: Temperature Gradient

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	degrees Celsius/meter [°C/m]
cm, g, dyne, °C, s, V, A (cgs)	degrees Celsius/centimeter [°C/cm]
mm, kg, N, °C, s, mV, mA (nmm)	degrees Celsius/millimeter [°C/mm]
µm, kg, µN, °C, s, V, mA (µmks)	degrees Celsius/micrometer [°C/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	degrees Fahrenheit/foot [°F/ft]
in, lbm, lbf, °F, s, V, A (Bin)	degrees Fahrenheit/inch [°F/in]
mm, mg, ms [Explicit Dynamics solvers]	degrees Kelvin/millimeter [°K/mm]

Table 90: Time

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	seconds [s]
cm, g, dyne, °C, s, V, A (cgs)	seconds [s]
mm, kg, N, °C, s, mV, mA (nmm)	seconds [s]
µm, kg, µN, °C, s, V, mA (µmks)	seconds [s]
ft, lbm, lbf, °F, s, V, A (Bft)	seconds [s]
in, lbm, lbf, °F, s, V, A	seconds [s]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(Bin)	
mm, mg, ms [Explicit Dynamics solvers]	milliseconds [ms]
m, kg, s [LS-DYNA solver]	seconds [s]
mm, t, s [LS-DYNA solver]	seconds [s]
in,lbf, s [LS-DYNA solver]	seconds [s]

Table 91: Translational Damping

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Newton * seconds/meter [N * s/m]
cm, g, dyne, °C, s, V, A (cgs)	dyne * seconds/centimeter [dyne * s/cm]
mm, kg, N, °C, s, mV, mA (nmm)	ton * millimeter * seconds/second^2 * millimeter [t * mm * s/s^2 * mm]
µm, kg, µN, °C, s, V, mA (µmks)	microNewton * seconds/micrometer [µN * s/µm]
ft, lbm, lbf, °F, s, V, A (Bft)	slugfoot * seconds/second^2 * foot [(lbm/32.2)ft * s/s^2 * ft]
in, lbm, lbf, °F, s, V, A (Bin)	slinchinch * seconds/second^2 * inch [(lbm/386.4)in * s/s^2 * in]

Table 92: Velocity and RS Velocity

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	meters/second [m/s]
cm, g, dyne, °C, s, V, A (cgs)	centimeters/second [cm/s]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters/second [mm/s]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Unit System	Measured in . . .
µm, kg, µN, °C, s, V, mA (µmks)	micrometers/second [µm/s]
ft, lbm, lbf, °F, s, V, A (Bft)	feet/second [ft/s]
in, lbm, lbf, °F, s, V, A (Bin)	inches/second [in/s]
mm, mg, ms [Explicit Dynamics solvers]	meters/second [m/s] or millimeters/millisecond [mm/ms]
m, kg, s [LS-DYNA solver]	meters/second [m/s]
mm, t, s [LS-DYNA solver]	millimeters/second [mm/s]
in,lbf, s [LS-DYNA solver]	inches/second [in/s]

Table 93: Voltage

Unit System	Measured in . . .
m, kg, N, °C, s, V, A (mks)	Volts [V]
cm, g, dyne, °C, s, V, A (cgs)	Volts [V]
mm, kg, N, °C, s, mV, mA (nmm)	milliVolts [mV]
µm, kg, µN, °C, s, V, mA (µmks)	Volts [V]
ft, lbm, lbf, °F, s, V, A (Bft)	Volts [V]
in, lbm, lbf, °F, s, V, A (Bin)	Volts [V]

Table 94: Volume

Unit System	Measured in . . .
m, kg, N, °C, s, V, A	$meters^3$ [m^3]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Unit System	Measured in . . .
(mks)	
cm, g, dyne, °C, s, V, A (cgs)	centimeters^3 [cm^3]
mm, kg, N, °C, s, mV, mA (nmm)	millimeters^3 [mm^3]
µm, kg, µN, °C, s, V, mA (µmks)	micrometers^3 [µm^3]
ft, lbm, lbf, °F, s, V, A (Bft)	feet^3 [ft^3]
in, lbm, lbf, °F, s, V, A (Bin)	inches^3 [in^3]
mm, mg, ms [Explicit Dynamics solvers]	millimeters^3 [mm^3]
m, kg, s [LS-DYNA solver]	meters^3 [m^3]
mm, t, s [LS-DYNA solver]	millimeters^3 [mm^3]
in,lbf, s [LS-DYNA solver]	inches^3 [in^3]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using Results

The Help for Results is organized in the following sections based on analysis type as well as the treatment and usage for the various result options.

Introduction to the Use of Results
Result Outputs
Result Definitions
Result Scoping
Structural Results
Line Pressure Result
Volume Result
Thermal Results
Magnetostatic Results
Electric Results
Fatigue Results
Noise, Vibration, and Harshness
Forced Response
User Defined Results
Python Result
User Defined Criteria
Result Utilities

Introduction to the Use of Results

Generating results to understand the behavior of an analyzed model is fundamental to any analysis in Mechanical. The application supports a variety of result types and tools to facilitate this process.

Capabilities include:

- Display result contours over the entire, or a portion, of the model for various solution quantities, such as displacement, stress, temperature, and electric field density.
- Customized result access using user-defined results (p. 2192).
- Chart (p. 1929) minimum and maximum values over time for multiple result sets.
- Options (p. 67) to quantify and visualization result contours that represent vectors, iso-surfaces, slice planes, path operations, surface cuts, and capped iso-surfaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Probes (p. 1946) to calculate abstract engineering quantities such as reaction forces, reaction moments, and virtual strain gauges.
- Export result data in a variety of formats, such as ASCII files for raw data, static images such as .png, .avi animations, as well as HTML reports.

Result Application

To apply results:

- Highlight the **Solution** object and select a result, result probe, or result tool from the options on the Solution Context Tab (p. 62).

  Or...

- Right-click the **Solution** object, select **Insert**, and then choose from the result options.

Note:

See the User Defined Result (p. 2192) section of the Help for more information about the specification and definition of this result type.

Once inserted into the tree, you need to scope (p. 1997) your result objects to geometric or meshing entities of the model.

Note:

- If you suppress any result object, the application clears all generated data.
- Because of software limitations, Mechanical currently cannot display the results of some types of nodes and elements. For example, a total deformation display in Mechanical excludes the deformations of nodes that are not attached to elements (like nodes associated with remote points).
- Mechanical requires the mesh information to be present on the Mechanical APDL result file in order to postprocess. Therefore, result files created from Mechanical APDL cannot have the **/CONFIG**,NORSTGM option set to 1 (the default is zero).

Result Outputs

The following topics related to result outputs are covered in this section.

Chart and Table
Line Chart Results (LS-DYNA Only)
Contour Results
Coordinate Systems Results
Interpolation of Result Values
Line Body Results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Path Results
Probes
Result Set Listing
Surface Results
Solution Summary Worksheet
Vector Plots

Chart and Table

The Home tab (p. 40) option **Chart** is a feature that enables you to chart loading and result data versus time, versus additional loading conditions, as well as versus other result data. This includes the loading conditions and/or results of different analyses. For example, you may wish to compare the displacement responses from two different transient analyses with multiple damping characteristics.

Go to a section topic:

- Overview (p. 1929)
- Create a Chart (p. 1929)
- Determining Data Points (p. 1930)
- Details Pane Content (p. 1930)
- Graph Display (p. 1932)
- Tabular Data Display (p. 1932)

Overview

You use the **Chart** feature to:

- Chart load(s) and result(s) vs time.
- Chart multiple harmonic response plots vs. Frequency.
- Change x-axis to plot a result against a load or another result.
- Compare results across analyses.
- Visualize and compress data into an easy-to-understand report.

Create a Chart

1. Select the objects you wish to chart. You can select multiple objects of interest. In doing so, note that:
   - You can choose objects in the tree that belong to different analyses of a model. However all objects must belong to the same Model.
   - Only loads, probes and results that can be contoured are added to the chart.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For result items the variation of minimum and maximum values is plotted as a function of time

2. Select the **Chart** button on the **Home** tab (p. 40). This adds a new Chart object to the **Outline**. You can add multiple charts.

Determining Data Points

You can choose a mixture of loads and results that may even span different analyses. In these cases there can be a mismatch between the time points at which the loads are defined and the time points at which results are available. For example in case of a nonlinear transient stress analysis under constant load, the load has a single value but there can be many time points where results are available. The below interpolation scheme is used to create charts when such mismatch occurs.

- Loads are interpolated or extrapolated to the time points at which result values or other load values.
- Results are not interpolated or extrapolated.

Details Pane Content

The Details categories include:

Definition

Outline Selection: This property lists how many objects are used in the chart. Clicking on the number of objects highlights the objects in the tree allowing you to modify the selection if needed.

Chart Controls

This category includes the following properties:

- **X-Axis**: By default the data of the selected objects are plotted against time. You may choose a different load or result quantity for the x-axis. For example you can plot a Force – Deflection curve by choosing the deflection to be the X-axis.
- **Plot Style**: display as **Lines**, **Points**, or **Both** (default).
- **Scale**:
  - **Linear** (default) - plot as linear graph.
  - **Semi-Log (X)** - X-Axis is plotted logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.
  - **Semi-Log (Y)** - Y-Axis is plotted logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.
  - **Log-Log** - X-Axis and Y-Axis are plotted logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.
- **Gridlines**: Show gridlines for plotting 2D X-Y curves.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **Both** - The gridlines for both the X-axis and Y-axis are shown.

– **X Axis** - The gridline for the X-axis is shown.

– **Y Axis** - The gridline for the Y-axis is shown.

– **None** - No gridlines are shown.

Axis Labels

This category includes the **X-Axis** and **Y-Axis** properties. You can enter appropriate labels for the X and Y axes. In doing so, note that:

- The X and Y axes always show the units of the item(s) being charted. These units are appended to any label that you enter.
- When multiple items are plotted on the Y-axis the units are determined as follows: If all the items plotted on the Y-axis have the same units then the unit is displayed. For example, if all items are of type deformation and the active unit system is British Inch unit system then the unit is displayed as Inch. If the items plotted on the Y-axis are of different types for example, stress and strain then Normalized is displayed for unit.
- When determining pairs of points to plot on the chart when X-axis is not time be aware that time is still used to determine the pairs of points to plot when an item other than time is used for the x-axis. Both the X-axis quantity and the Y-axis quantity must share a common time point to be considered a valid pair.

Report

This category includes the following properties:

- **Content**: By default both the chart as well as the data listing of the objects gets added to reports. Instead you may choose to only add the chart or only add the data listing or exclude the chart from report. Note that only tabular data or chart data with two or more points is displayed in the report.
- **Caption**: You may enter a caption for the chart. The caption will be included in the report.

Input Quantities

Any valid load object added to the chart gets displayed under Input Quantities. If a load has multiple components then each component will get a line in this details group.

Output Quantities

Any valid result object added to the chart gets displayed under Output Quantities. If a result has multiple components then each component will get a line in this details group.

- **Input/Output Quantities**:

Note:

When using Input and Output Quantities, note that:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Naming and legend**: Each object added to a chart is assigned a name and a legend label. The name is simply the object name in the tree if there are no components associated with the object. An example would be a Y displacement probe. For objects that have multiple components the component direction or name will get added to the object name. For example adding 'Equivalent Stress' result item to a chart will result in two items getting added – 'Equivalent Stress (min)' and 'Equivalent Stress (max)'.
- Each name is preceded by a one letter label such as [A] or [B]. This label is also displayed on the corresponding curve in the chart and is used to associate the object name with the curve.
- The default setting is to display the item in the chart and data grid. You can exclude an item by setting this field to Omit. Omitting an item removes the corresponding data from both data grid and chart. Be aware that an item chosen for X-axis cannot be omitted and this field will be reset to Display for that item.

Graph Display

- **Legend**: You can use the right-click context menu options **Show Legend**/**Hide Legend** to display or hide annotations in the **Graph** window for the selected Chart.
- **Normalization**: Scaling of Y-axis is determined as follows.
  - **Single item on Y-axis**: Scaling is based on the minimum and maximum values of the item plotted
  - **Multiple items on Y-axis that have same unit type**: Scaling is based on the minimum and maximum values of the items plotted. For example, plot applied pressure load and a stress result against time.
  - **Multiple items on Y-axis that have different unit types**: In this case each curve is normalized to lie between 0 and 1, that is the minimum value is treated as zero and the maximum value as one. The label of the Y-axis reflects this by appending Normalized to any user specified label. Note that the data grid displays the actual values always.

Tabular Data Display

The data contained in the **Tabular Data** window is read-only and reflects the data for the specified object/objects.

Line Chart Results (LS-DYNA Only)

For LS-DYNA analyses, use the Line Chart object to plot a two-dimensional graph, in the X and Y directions, for supported input data. Currently, input data for **Line Chart** objects can come from:

- User Defined Results (p. 2192).
- Result Trackers (p. 1844).
- Result charts, such as waterfall diagrams.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Line Chart** objects.
- Data from the Solution Information (p. 1833) object.
- Imported text files (of supported formats).
- User-defined datasets created using scripting APIs.

Application

Use the following steps to specify a **Line Chart** object. The feature is very flexible and enables you to change the data and plots with ease.

1. Insert the object by highlighting the **Solution** object and either selecting the **Line Chart** option from the Solution (p. 62) Context Tab or right-clicking and selecting **Insert** > **Line Chart**.

   You can also right-click within the **Geometry** window and select **Insert** > **Line Chart**.

2. Specify the **Source Type** property. Options include **Existing Objects** and **Imported File**.

3. Use the **Source Object** to specify objects in the Outline or to select an external file. When the **Source Type** property is set to:

   - **Existing Objects**, use the **Source Object** property to manually select the objects in the Outline you wish to chart. You can select multiple objects.
   - **Imported File**, use the option of the **Source Object** property to display a dialog to browse to a directory location and select a file.

   Note:

   When importing a file, the application supports tab separated X-Y Data file format as well as LS-PrePost Data Export format. See the Example File Formats (p. 1934) topic below for examples of these formats.

4. Once the above property is defined, the object plots the data. At this point you can apply an operation or filer to your dataset.

   Important:

   The **Operation Type** and **Filter Type** properties are only supported for objects or files that include one dataset.

   Operation Type

   Operation Type options include:

   - **None** (default): Reset any previous operations and redisplay the plot of the original dataset.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Integration**: Generate and plot a new dataset based on the integral operation.
- **Differentiation**: Generate and plot a new dataset based on the differentiation operation.

Filter Type

Filter Type property options include:

- **None** (default): Reset any previous filters and redisplay the plot of the original dataset.
- **Butterworth**: Apply a Butterworth filter to create and plot a new dataset.
- **SAE**: Apply a SAE filter to create and plot a new dataset.

When you specify a filter, the following additional properties display:

- **Cutoff Frequency**: Define a cutoff frequency value for the applied filter.
- **Cutoff Frequency Limit**: A read-only property that displays the cutoff frequency limit for the current dataset. This limit indicates the maximum value for the cutoff frequency that can still supports filtering. This value helps you fine tune your Cutoff Frequency property entry.

Note:

You cannot use the options of the **Operation Type** property and the **Filter Type** property in combination. Specifying an **Operation Type** automatically causes the **Filter Type** property to be set to **None**, and vice versa.

5. You can use the **Display Original** property to display the original curve as well as the plot you have creating using an operation or filter. Options include **No** (default) and **Yes**.
6. Create additional charts as desired.

Examples File Formats

Tab Separated X-Y Data Format

```
Time Force Reaction
s N
0 0 0.01 3.17E+05
0.02 7.04E+06
... ...
```

LS-PrePost Data Export Format

```
Curveplot
Time
butterworth
Glstat Data
int-0 #pts=1001 5.00E-02 1.00E+08*
Minval=0.0000000000e+00 at time= 0.0000000000*
Maxval=1.00e+08 at time= 5.00
0          0
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
0.05        3.17E+05
0           .17.04E+06
…           …
```

Contour Results

Most result types can be displayed using contours or vectors. The **Result** Context Tab (p. 62) applies to **Solution** level objects that display contour or vector results.

Coordinate Systems Results

The following topics are addressed in this section:

Nodal Coordinate Systems Results

Elemental Coordinate Systems Results

Rotational Order of Coordinate System Results

Important:

If a given result experiences any change in position due to the loading conditions, such as a displacement or deformation, the **Geometry** window always displays this position change in the **Global Coordinate System** (per global XYZ coordinate triad). If you have specified a local coordinate system for your result, the application exposes all data contained in the **Details** view as well as the **Tabular Data** window based the local coordinate system entry. Furthermore, if applicable, the application always displays result contour colors in the **Geometry** window based on a specified local coordinate system.

Nodal Coordinate Systems Results

Every node in a model is associated with a coordinate system that, by default, is aligned with the global Cartesian coordinate system. If any of the X, Y, or Z axes of an individual node is rotated, the resulting coordinate system will typically *not* be aligned with the global Cartesian coordinate system. Using this feature, you can display nodal result rotations either as Euler rotated triads at each node location, or as contours that represent an Euler rotation angle about an individual nodal axis. Boundary conditions are highly dependent upon Euler angles.

To display nodal coordinate systems results:

Highlight the **Solution** object, and choose one of the following options from the **Coordinate Systems** drop-down menu. A corresponding object will be inserted in the tree.

- **Nodal Triads**: Displays an XYZ triad at each node representing the resulting rotation of the node's coordinate system compared to the global Cartesian coordinate system. See Rotational Order of Coordinate System Results (p. 1937) for details.
- **Nodal Euler XY Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each node about the Z axis.
- **Nodal Euler YZ Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each node about the X axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Nodal Euler XZ Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each node about the Y axis.

> **Note:**
>
> For the Mechanical APDL solver, nodal coordinate systems will not vary from time step to time step.

Elemental Coordinate Systems Results

Every element in a model is associated with a coordinate system that, by default, is aligned with the global Cartesian coordinate system. If any of the X, Y, or Z axes of an individual element is rotated, the resulting coordinate system will typically *not* be aligned with the global Cartesian coordinate system. Using this feature, you can display elemental result rotations either as Euler rotated triads at each element's centroid, or as contours that represent an Euler rotation angle about an individual elemental axis. Shell stresses are highly dependent upon Euler angles.

> **Note:**
>
> Euler rotated triads and contours of Euler rotation angles are determined by the element type and not all elements types are supported. If no result data is available for the corresponding element type, then the triads display as global triads.

To display elemental coordinate systems results:

Highlight the **Solution** object, and choose one of the following options from the **Coordinate Systems** drop-down menu. A corresponding object will be inserted in the tree.

- **Elemental Triads**: Displays an XYZ triad at each element centroid representing the resulting rotation of the element's coordinate system compared to the global Cartesian coordinate system. See Rotational Order of Coordinate System Results (p. 1937) for details.

> **Note:**
>
> You may need to use the Wireframe viewing mode (p. 80) to see a particular triad in an element.

- **Elemental Euler XY Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each element centroid about the Z axis.
- **Elemental Euler YZ Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each element centroid about the X axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Elemental Euler XZ Angle**: Displays a contour plot representing the magnitude of the resulting Euler angle rotation at each element centroid about the Y axis.

Note:

For the Mechanical APDL solver, it is possible for elemental coordinate systems to vary from:

- Time step to time step.
- Node to node. In this case, Mechanical displays the angles for the first node in the node number sequence. For line elements, like beams, the result file does not contain the elemental Euler angles and as a result Mechanical calculates the coordinate system from the (x, y, z) positions of the nodes (including the orientation node). Currently, Mechanical does not have access to the internal nodes created by the solver. Because of these calculations, the triad for a line element can differ from the triad displayed by the Mechanical APDL application if you have issued the **//PSYMB,ESYS,ON** command and the Mechanical APDL application has access to the Mechanical APDL db file.

Rotational Order of Coordinate System Results

The following rotational convention is used for both Nodal Coordinate Systems Results (p. 1935) and Elemental Coordinate Systems Results (p. 1936):

1. The first rotation is called **... Euler XY** and is in the X-Y plane (X towards Y, about Z).
2. The second rotation is called **... Euler YZ** and is in Y1-Z1 plane (Y1 towards Z1, about X1).
3. The third rotation is called **... Euler XZ** and is in X2-Z2 plane (Z2 towards X2, about Y2).

X1, Y1, and Z1 refer to the coordinate system axes after the initial rotation about the global Z axis.

X2, Y2, and Z2 refer to the coordinate system axes after the initial rotation about the global Z axis and subsequent rotation about X1.

See Figure : Euler Rotation Angles from the *Modeling and Meshing Guide* for a pictorial representation of this convention.

Interpolation of Result Values

Interpolation is a result calculation performed when you wish to study results at a location where no nodes exist. For example, **Path** results (p. 1940), **Surface** results (p. 1959), and some probes result objects (p. 1947) can consist of arbitrary (that is, non-node-based) locations. To calculate a result at these types of locations, Mechanical employs an interpolation algorithm. For a given x, y, z in the object, the algorithm identifies the element that contains x, y, z. If x, y, z is not contained in any element, the algorithm produces no results.

A Newton's Method loop, with the aid of element shape functions and their derivatives, attempts to converge to the natural (element) coordinates corresponding to the global x, y, z. Using the natural coordinates, the element shape functions, and the results at the element's nodes, Mechanical computes the interpolated result value for the global x ,y, z location.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Limitations and Conditions

Interpolation primarily applies to degree of freedom results such as temperatures, displacements, and node-based element results like thermal flux, stress, and strain.

Therefore, it is recommended that you not use interpolation for results such as node-based element reactions (EHEAT/ENFO). The application allows for these calculations, but they are not good candidates for interpolation. In addition, you should not sum interpolated results of these types; such as to calculate total heat or total force.

Convergence

There is no assurance that convergence actually occurs. If Newton fails to converge, natural coordinates of (0, 0, 0) are chosen. Currently, no warnings are issued for convergence problems.

High Order Elements

Contours for high-order elements are displayed by decomposing element faces into low-order primitives. This decomposition introduces extra display-only nodes at the centroids of element faces. The values at these nodes are generated by interpolating the computed results and can differ from manually calculated values, especially when dealing with expressions which introduce discontinuities in the result field, such as SX/abs(SX).

Expression-Based User Defined Results

Interpolation should be limited to simple linear combinations of results.

If the application performs an interpolation on an expression-based User Defined Result (p. 2192), it generates the final result based on the order of the calculations taking place. For example, given the expression SX^3, for the cube of normal stress in the X direction, at each node the application calculates SX^3 values. The interpolation occurs next for each (x,y,z) that requires a result and the application interpolates nodal values of SX^3. The critical point is that this procedure would produce different results if it first interpolated the nodal values of SX to the x,y,z values and then calculated SX^3.

Given a complex expression, such as (SEQV / SX)^3, the final interpolated results on a line or a surface could may be counter intuitive. This is why it is recommended that interpolation be limited to simple linear combinations of results.

Line Body Results

This section examines specific characteristics for results applied to line bodies. Go to a section topic:

- Beam Section Results (p. 1938)
- Stress and Strain Results for Line Coating (Axisymmetric Shell) (p. 1939)
- Limitations (p. 1940)

Beam Section Results

Using the **Beam Section Results** property of the Solution object, you can choose whether to display beam results using expanded data when you specify your line body as a beam/pipe that includes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

cross sections (**SECDATA**). The choice of whether or not to use expanded data significantly impacts the evaluation, display, and storage requirements for a given line body result. Property options include:

- **No** (default): This setting enables you to reduce the amount of result data created by the result. However, the application displays the beam contour results using only the nodal results (one value per node).
- **Yes**: For this setting, the application displays beam results with realistic contours that can vary along all coordinates, show stresses and strains as well as deformations, such as torsional rotation.

Note:

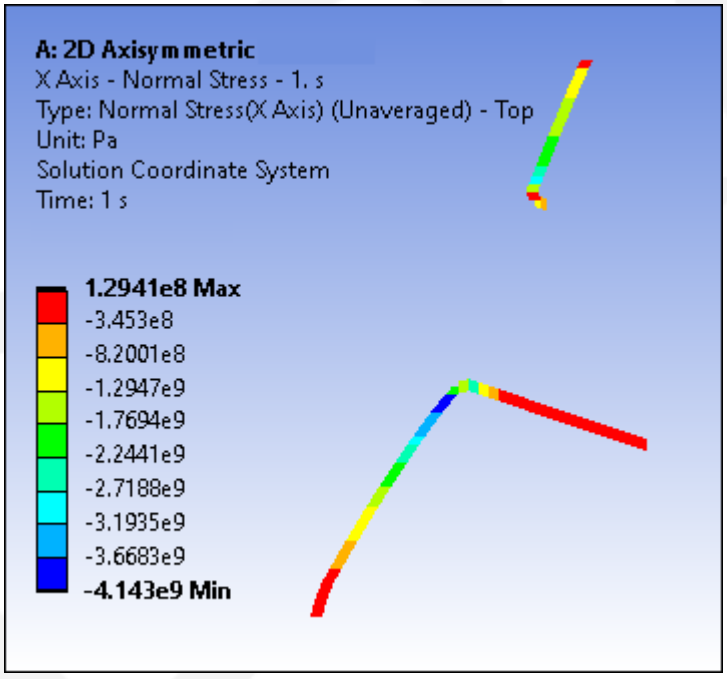
Ansys recommends that you set the **Beam Section Results** property to **Yes** working with beam and pipe elements. This makes sure that the actual contour values, the deformed shape, and result animations are all using the "true" expanded beam results.

Note:

- The application always uses expanded results when displaying animations.
- Probe annotation (p. 71) labels always display expanded results.

Stress and Strain Results for Line Coating (Axisymmetric Shell)

If you scope a **Line Coating** to stress and strain results (p. 2026), the result displays as a colored line only. Unlike beams, there is no visible element expansion (no visible cross section).



In addition, the **Position** property displays for stress and strain results scoped to a **Line Coating**. For the option **Top/Bottom**, the application only displays **Bottom** results and generates an associated Message.

Limitations

Note that line body result capabilities have certain limitations, including:

- Beam strains/stresses always use the solution coordinate system, even if you specify the global coordinate system.

  Because your model may contain solid/shell elements (which support the Global Coordinate System) as well as beams, the stress display of the entire model can contain results in different coordinate systems. Therefore, you will experience under-defined beam stress/strain results, that is, results that will **not** solve or evaluate, if you perform the following:

  1. Scope a component stress/strain to *any* line body and;
  2. Choose *any* coordinate system other than the global coordinate system or the solution coordinate system.

  Note:

  If you have shell or solid bodies in your model, you can choose a local coordinate system as long as you do not scope it to any line body.

- Pipes idealized using elbow elements are not supported.
- User Defined and User Integrated cross-sections are not supported.
- Do not currently perform Path interpolation on expanded beams or pipes for element-node-based data, like stress and strain.
- Linearized stress and stress error (SERR) and equivalent strain are not supported.

Path Results

A Path (p. 883) is categorized as a form of construction geometry that is represented as a spatial curve. Path results are evaluated at discrete points along this curve.

Go to a section topic:

- Scoping a Path to a Result (p. 1941)
- Scoping Two Point Path Results (p. 1942)
- Scoping Edge Path Results (p. 1945)
- Understanding Interpolation and Paths (p. 1946)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Scoping a Path to a Result

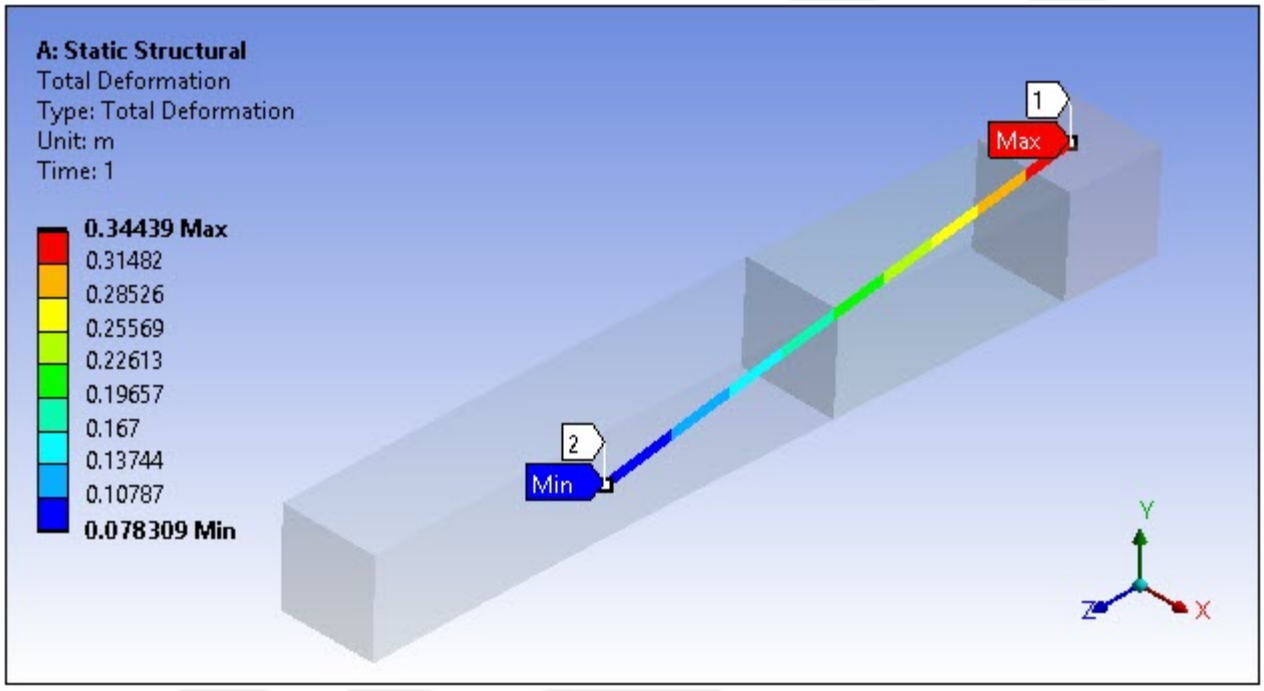
If you have already defined a path (p. 883), you can view the path results by highlighting the result object, and in the Details, setting **Scoping Method** to **Path**, then choosing the name of the particular path that you defined.

Note:

Path results are not supported for models using periodic symmetry.

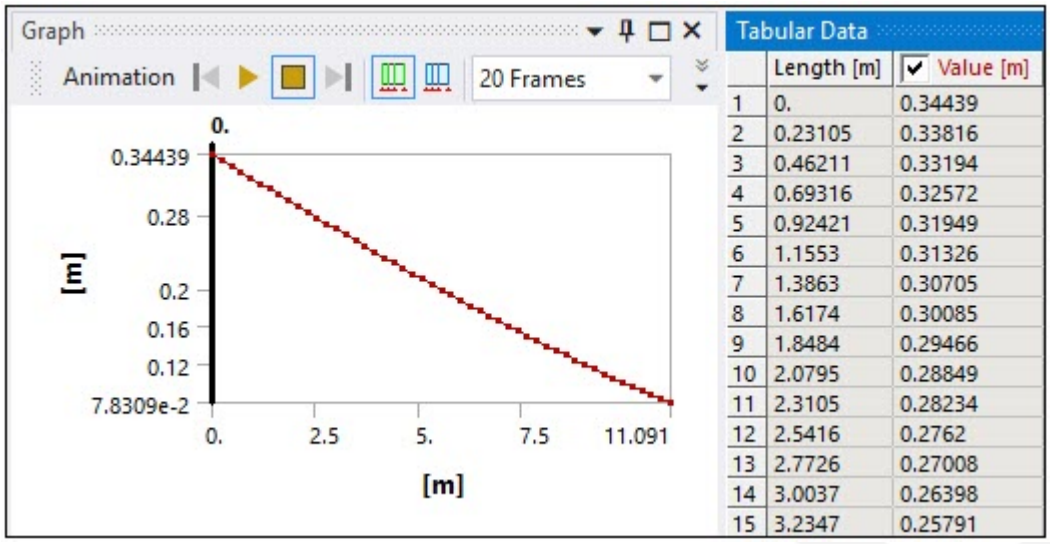
In Explicit Dynamics models, the path results are not supported for geometries that use an Eulerian reference frame.

An example path result plot is shown below.



In this example, the **Number of Sampling Points** for the **Path** object was set to 47. Results were calculated for each of these 47 points as shown in the **Graph** below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Tabular Data

	Length [m]	Value [m]
1	0.	0.34439
2	0.23105	0.33816
3	0.46211	0.33194
4	0.69316	0.32572
5	0.92421	0.31949
6	1.1553	0.31326
7	1.3863	0.30705
8	1.6174	0.30085
9	1.8484	0.29466
10	2.0795	0.28849
11	2.3105	0.28234
12	2.5416	0.2762
13	2.7726	0.27008
14	3.0037	0.26398
15	3.2347	0.25791

Scoping Two Point Path Results

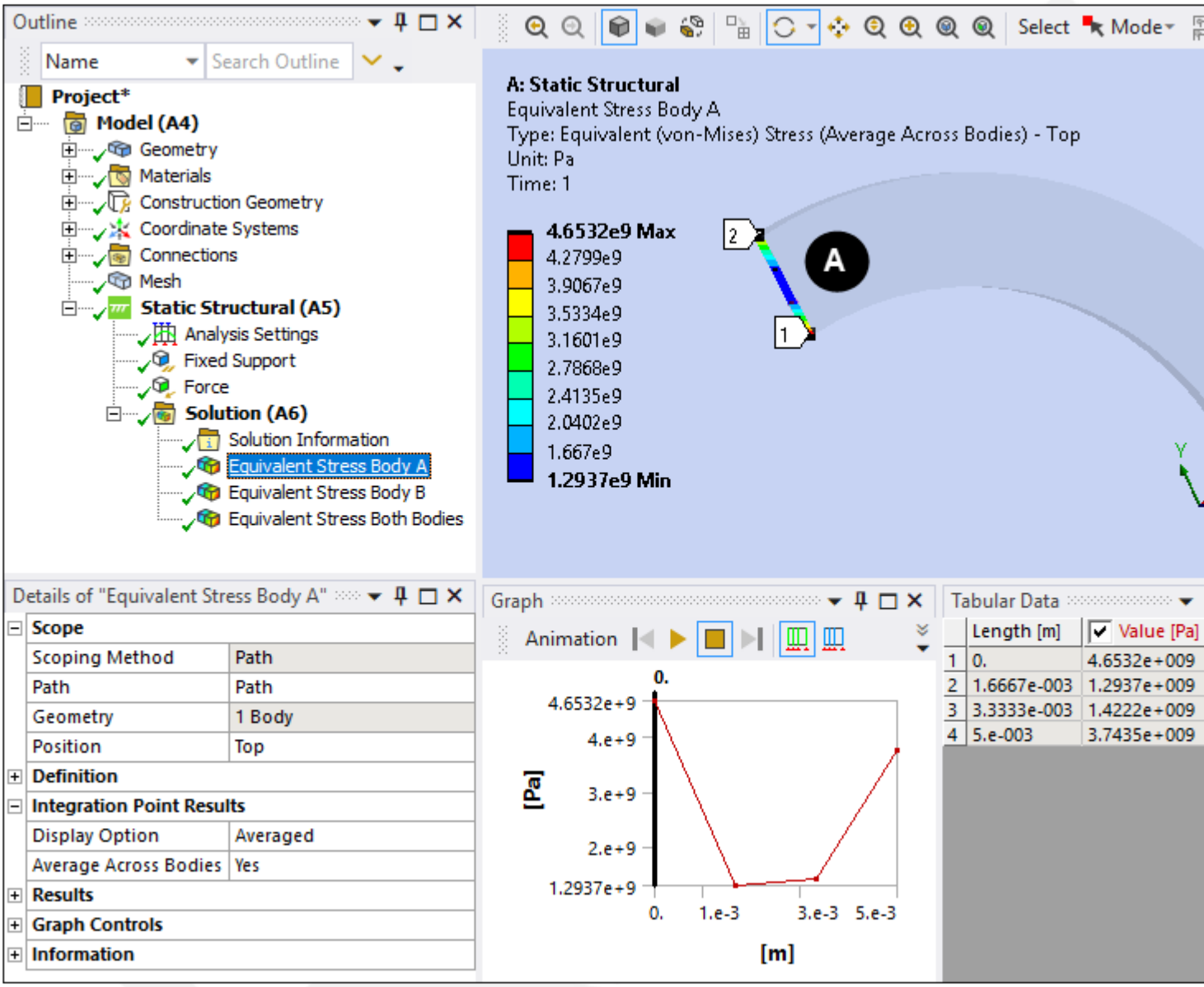
When you define a **Path** using two points (**Two Points** or **X Axis Intersection** options), for each point in the Path, Mechanical selects only **one body** from which to calculate the results. If you scope a Path to multiple bodies, Mechanical calculates the results from the body with the highest identifier as defined in the CAD application. Generally, the part/body with the highest identifier is the one most recently added. No averaging is done for a Path result across bodies, regardless if the setting of the **Average Across Bodies** property.

If a Path traverses multiple surface or solid bodies and if a Path point lies on the interface between the distinct bodies, the application only displays the body used to create the result. For example, as illustrated here, a Path is defined by the edge between two surface bodies. Note that both bodies are scoped. However, the result contours on the Path are only based on body **A**.

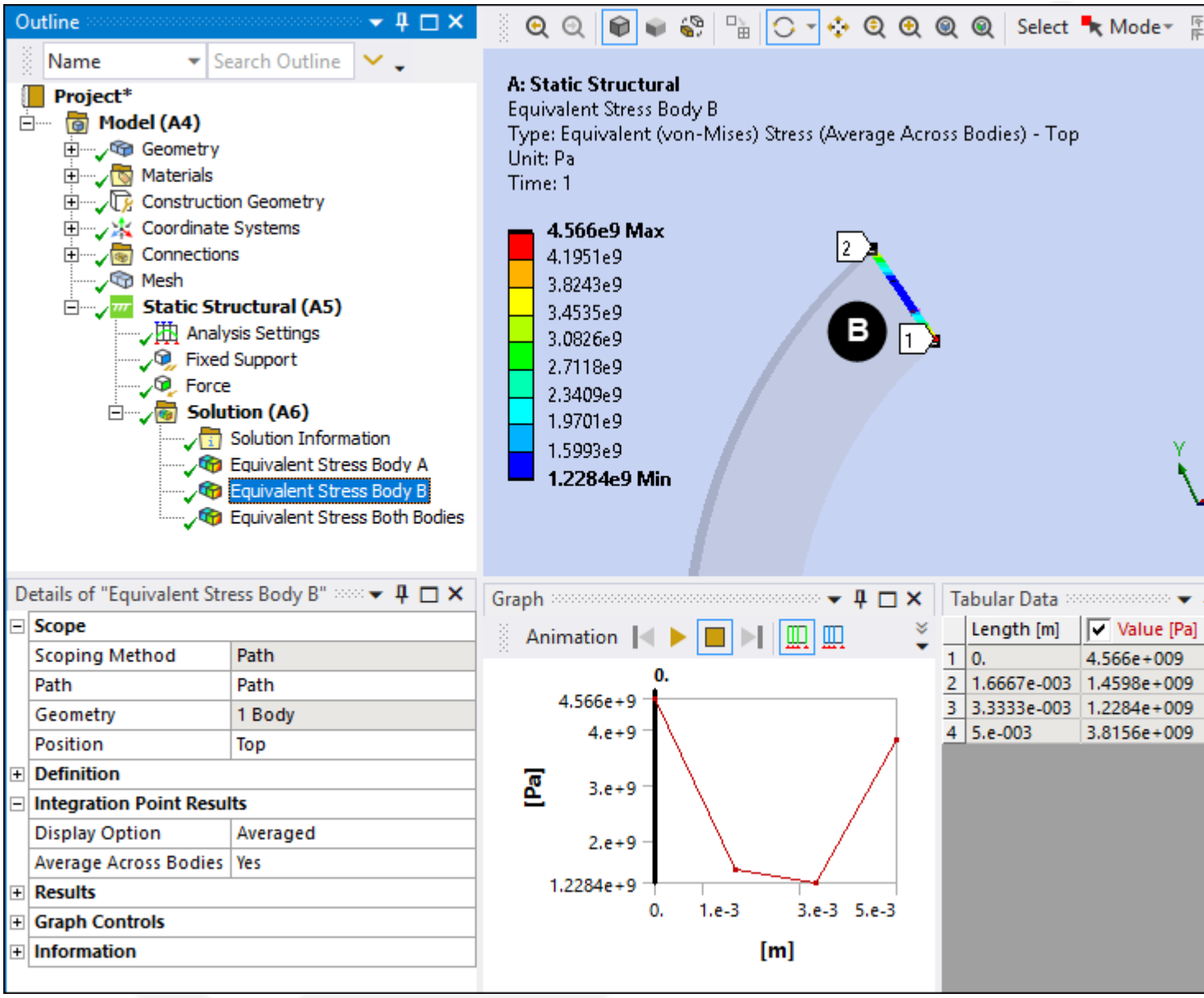
In the first two images, a body (**A** and **B**) is specified in the **Geometry** property for the result. Only one body is displayed in the Geometry window. And, note that the stresses for Body **A** and Body **B** differ. In the third image, the result is scoped to both bodies. The stresses displayed for this third image match those of Body A because it has the highest identifier and therefore selected by the application by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

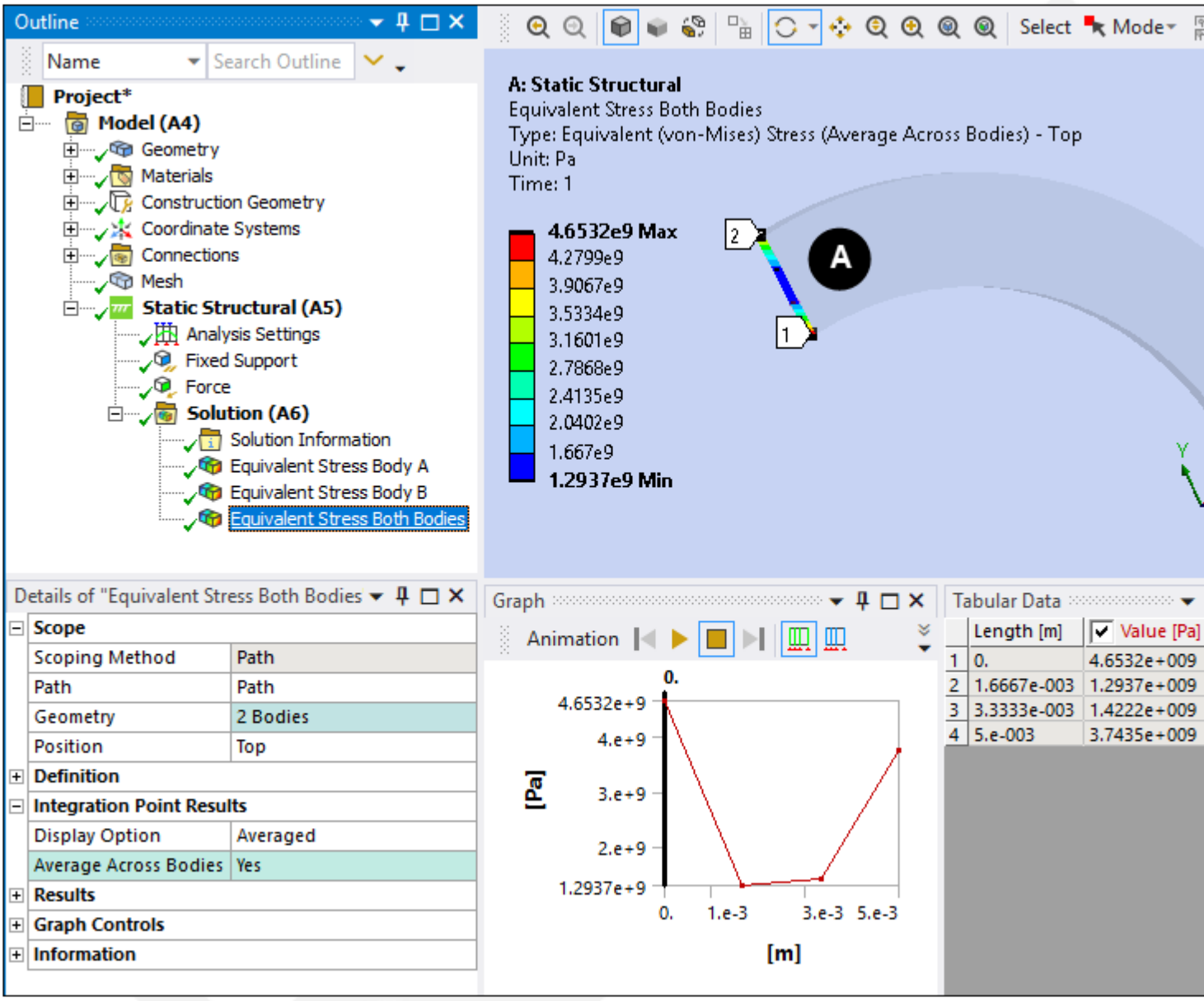


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Scoping Edge Path Results

When you define a **Path** using the **Edge** option and the **Path** traverses at least one line body, then for element-nodal results, such as stress, scoped to the **Path**, each node in the Path collects data from every scoped body and element that touch the node. Therefore, if a result is specified as **Averaged**, all the elements touching a node contribute to the averaged value at that node. If the result is specified as **Unaveraged**, all the elements touching a node contribute to the contour of the result at that node.

For a **Path** defined by the Edge option that **does not** traverse a line body, no behavior change occurs. The results solve as they would for non-path **Edge** scoping.

Note:

In previous releases, edge-based Paths that traversed at least one line body allowed at most two elements to contribute to a result at any path node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Understanding Interpolation and Paths

For a given path, Mechanical examines each element in the set of scoped bodies to determine the set of elements which contain a point on the path. A path point may reside on a face of an element, in the interior of an element, or in no element. The set of path points is, in essence, a set of interpolation points.

Assume, for example, that you request a normal x-axis stress result on the path (that is, *SX*). For a given interpolation point (*x*, *y*, *z*) lying on the face or residing in the interior of an element, Mechanical finds the natural (or normalized) coordinates of the point within the element. Mechanical then interpolates the corner values of *SX*, using the natural coordinates and shape functions, to find a value for *SX* at (x,y,z).

See the Interpolation (p. 1937) section for additional information.

Important:

1. If an interpolation point (x, y, z) lies on a face, edge, or a vertex that is shared by multiple elements, the application selects one element only to be used for result calculations. Because the application selects the element, there is no simple method to accurately identify the element selected.
2. If the result for a path is an element-nodal unaveraged result, the results at the interpolation point are interpolated from the unaveraged nodal stresses of the host element (x, y, z) only. Subsequently, the graph of the unaveraged result may not be smooth or differentiable.

Probes

Probes enable you to find results at a point on the model, or minimum or maximum results on a body, face, edge, or vertex; to find results on objects in the tree, such as elastic support or weak springs; or to obtain reaction forces and moments at supports.

Important:

Probe Scoping Limitation for Symmetry

Certain probes, such as a **Reaction** probe, support mesh scoping (node, element, or element face) through the Boundary Condition it is scoped to. However, if you specify **Symmetry** in your analysis, the application does not support mesh scoping on **any** probe.

Probe Scoping Limitation for Models with Changing Mesh

If you are using a feature that automatically changes the mesh during the solution process, such as the Nonlinear Adaptive Region (p. 1707) object or SMART Crack Growth (p. 468), any result probe scoped to the corresponding body will not evaluate.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Shell Geometry Considerations

1. Probe results at the mid-surface location of a shell geometry are obtained by interpolating the Top and Bottom layer result values. (This differs from Contour results which directly extracts the mid-surface location from the solver.)

2. When probes are scoped to a Coordinate System, the application first identifies the element corresponding to the scoped location, using a hit-detection algorithm. Once the element is identified, the application gathers the averaged nodal results for that element and then uses shape functions to find the interpolated result value at the scoped location. This interpolated value will account for both the Top and Bottom layer values for shell geometries.

Interpolation Limitation

Values for positions inside of an element may differ from hand-calculated values, especially for certain user defined result expressions. See the limitations topic in the Interpolation of Result Values (p. 1937) section.

This section examines the general function of the probe tool in Mechanical as well as the specific probe types that are available in the Mechanical application. It also describes the Details options associated with the probe object.

Probe Types and Application Overview

Probe Details View

Probe Types and Application Overview

The application supports the following probe types:

- Structural Probes (p. 2092)
- Thermal Probes (p. 2155)
- Magnetostatic Probes (p. 2161)
- Electric Probes (p. 2163)
- Volume Probe (p. 2152)

Review the following section topics, especially the requirements and limitations:

- Application (p. 1948)
- Scoping (p. 1948) (Requirements (p. 1949) and Limitations (p. 1949))
- Results Output Coordinate System (p. 1950)
- Limitations of Geometry-Based Probes (p. 1951)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Application

You insert probes from the **Solution** object. Probes are available from the **Probe** drop-down menu on the ribbon or through the right-click context menu. You specify the probe using the properties of the **Details** pane. Following the solution, the display of the probe reveals the displaced mesh for the specified time. The probe result shows values over time and for a specified time. Based on the probe type, the **Details** pane displays result data specific to that type, such as maximum or minimum values over time, emitted radiation, etc.

Note:

You cannot turn off the time history for result probes.

Scoping

Probes are customized for the particular result type, therefore, different probes enable different scoping mechanisms. For example, you can scope a reaction probe to a boundary condition while a stress probe allows you to scope to an x, y, z location (Hit Point Coordinate (p. 97) option) on the geometry. Refer to the "Characteristics" column of the tables in the linked probe type sections above for scoping details. The **Location Method** property provides the available scoping options (Geometry Selection, Named Selections, Remote Points, etc.).

Understanding When Scoping is Interpolated

When you create a probe by selecting a specific x, y, z location on the model (using the Hit Point Coordinate (p. 97) option) or using a coordinate system, Mechanical chooses a singular element of a singular body that contains the x, y, z location. In the event the probe lies between two bodies, the application still only picks one of the two bodies (and one element) to use for the result calculation. Result values are interpolated from the element's nodes to the x, y, z location.

When picking a specific x, y, z location, you can obtain the probe result directly at the closest corner node, without extra interpolation, by right-clicking on the probe object in the tree and choosing **Snap to mesh nodes** from the context menu. The identification number of the closest corner node is displayed as the **Node ID** in the Details pane of the probe in the **Results** category. See the Interpolation (p. 1937) section for additional information.

Note:

Line Body

If you attempt to intersect probes with a line body, Mechanical issues a warning message. No results (such as stresses or displacements) will appear in the Details of the probe.

Surface Body

For surface bodies with expanded thickness, because the snapping location is located on the expanded mesh, while other items such as the original x, y, z location and the node ID are on the non-expanded mesh, you are advised to turn the visual expansion (p. 84) off in order to best view these items.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Face

Only reaction probes support element face scoping.

Caution:

The application does not support probes applied to objects that you have scoped to multiple Remote Points, either directly or indirectly, such as a spring scoped to a Remote Point that is itself defined by multiple Remote Points.

Specific Scoping Requirements

Note the following specifications when scoping a probe:

- When you create a probe by scoping a vertex, edge, face, or volume, the results reported for the probe are for the undisplaced nodes and elements. The displaced location of the probe (if any) is *not* used in any way to calculate results.
- If a probe is scoped to any suppressed *or* hidden parts, then the probe will not solve or evaluate results. This strategy exists to prevent numeric contributions from elements and nodes that are not scoped.

Scoping Limitations

Review the following probe limitations:

Reaction, Summation, Torque, Et. Al.

As a result of an element selection limitation, the application can select unscoped adjacent elements that will then contribute to a probe's result.

When you set the **Location Method** property to **Geometry Selection**, the scoping algorithm initially selects a group of nodes on the highlighted geometry (face, edge, etc.). The algorithm then selects any elements attached to those nodes and can include unscoped adjacent elements, that is, elements not contained in the scoping (green highlight).

As a result, probe types that have element-nodal results, such as Structural Force Reaction/Moment Reaction (p. 2107) and Magnetostatic Force Summation/Torques (p. 2161), the scoping algorithm selects these unscoped adjacent elements and these additional elements can contribute to the probe results.

Although a limitation, this behavior enables you to compare element-nodal probes to nodal probes scoped to boundary conditions. For example, an element-nodal Force Reaction (p. 2107) probe scoped to a face can be compared to a nodal probe scoped to a boundary condition that occupies the face.

Volume, Energy, Joule Heat, Et. Al.

When the result that is associated with a probe is purely elemental (one value per element regardless of the number of nodes), then the probe does not include elements from un-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

scoped bodies. For example, a Volume (p. 2152) probe scoped to a body provides the exact volume of the body.

Note:

Bodies that are separated by contact do **not** share result nodes. In this case, results from elements on unscoped bodies are not included.

Shells

Shell element node-based results (like stress and strain) exist at the top, bottom, and middle of the shell element (or the layer). Therefore, a shell node can have three values for a given layer.

For result probes on shell models that are scoped by Geometry Selection, the probe normally considers the top value and bottom values at the scoped nodes and picks either the maximum or minimum value. Based on the probe type, the Spatial Resolution property enables you to select whether the application uses the maximum or minimum value.

If you scope your probe to a Coordinate System, the application performs an interpolation using the values at the top and bottom of the shell.

With these situations in mind, your scoping may present results with unexpected or non-intuitive values.

For example, consider a probe scoped to a coordinate system that is situated near a vertex at the mid-plane. For this situation, the interpolation is a simple arithmetic average. However, what if the Top value at the node is **`-1000`** and the Bottom value at the same node is **`1000`**, a very real scenario for shell models. The coordinate system probe would report (-1000 + 1000)/2 = 0.

Now consider the probe scoped by geometry to the same Vertex. It would report max(-1000, 1000) = 1000 if the Spatial Resolution property was set to Use Maximum.

Results Output Coordinate System

Some probes such as the Directional Deformation probe allow the results to be calculated and displayed in a coordinate system of your choice. Some other probes such as a Spring probe allow results to be output only in a specific coordinate system. Refer to **Orientation Coordinate System:** entry under the "Characteristics" column in the probe tables (see links above) regarding what coordinate systems are allowed and what the default coordinate system is. You can use **Orientation** in the Details of the probe to change the output coordinate system.

Note:

When the Orientation Coordinate System is Global Cartesian, the triad (p. 280) symbol is not displayed. The exception is for Torque probes in magnetostatic analyses, where the global triad is displayed and the direction vector is placed at the global origin.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Limitations of Geometry-Based Probes

The following table shows the limitations of geometry-based probes. If you make incorrect selections in the Details for any of the probes, all the probes under solution remain unsolved.

Probe	**Scope**	Must be **Scoped** to a rigid part	**Components** and **Principals** Result Selection invalid	**All** Result Selection invalid
Deformation	Vertices, Edges, Faces, or Volume			X
Stress			X	X
Strain			X	X
Thermal Flux[a]				X
Flux Density[a]				X
Flux Intensity[a]				X
Velocity				X
Acceleration				X
Position		X		
Angular Velocity[a]		X		
Angular Acceleration[a]		X		

[a] Not supported in explicit dynamics analyses.

Probe Details View

The following table describes the Probe Details view categories and properties. All Probes provide the same Details view categories, however, based on the probe type and/or how you specify the probe properties; the availability of the properties can differ.

Category	**Property Name and Description**
Definition	**Type** This read-only property displays the selected type of probe. **Location Method** Sets the probe location. Based upon the probe type, **Location Method** options include: **Geometry Selection** Default setting, indicating that the probe is applied to a geometry or geometries (X, Y, Z points, edge/edges, vertex/vertices, face/faces, or body/bodies), which are chosen using a graphical selection tools. If you select a point using the **Hit Point Coordinate** selection tool (see Graphics Toolbar (p. 97)), the read-only **X,Y, Z Coordinate** properties display and show the coordinate locations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	Geometry: Visible when the **Location Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools. **Coordinate System** Use this property to set the location according to a user-defined coordinate system. This choice displays a **Location** drop-down list where you pick the particular coordinate system. The **X,Y,Z Coordinates** of the location are also displayed. **Coordinate System**: Visible when the **Location Method** is set to **Coordinate System**. Provides a drop-down list of available coordinate systems. **Remote Points** Use this property to scope the probe to a remote point. **Remote Points**: Visible when the **Location Method** is set to **Remote Points**. Provides a drop-down list of available remote points. **Caution:** The application does not support probes applied to objects that you have scoped to multiple Remote Points, either directly or indirectly, such as a spring scoped to a Remote Point that is itself defined by multiple Remote Points. **Boundary Condition** This **Location Method** option is available for **Force Reaction** and **Moment Reaction** probes in structural analyses. It is also available for the **Reaction Probe** in thermal, electric, and thermal-electric (as Heat Reaction) analyses. And it is available for the **Radiation Probe** in thermal as well as thermal-electric analyses. Use this property to scope the probe to an existing boundary condition. **Boundary Condition**: Visible when the **Location Method** is set to **Boundary Condition**. This property provides a drop-down list of available boundary conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	Spring Use this property to scope a **Force Reaction** probe to an existing body-ground spring that you select from a drop-down list of available body-ground springs. **Spring**: This associated property is visible when the **Location Method** is set to **Spring**. This property provides a drop-down list of available body-ground springs. **Contact Region** Use this property to scope **Force Reaction**, **Moment Reaction**, and heat **Reaction** (thermal analyses only) probes to an existing contact region that you pick from a **Contact Region** drop-down list. **Contact Region**: Visible when the **Location Method** is set to **Contact Region**. This property provides a drop-down list of available contact regions. **Beam** Use to scope the probe to an existing boundary condition that you pick from a **Beam** drop-down list. **Beam**: Visible when the **Location Method** is set to **Beam**. Provides a drop-down list of available beams objects. **Mesh Connection** Use this property to scope the probe to an existing mesh connection in the tree. **Mesh Connection**: Visible when the **Location Method** is set to **Mesh Connection**. This property provides a drop-down list of available mesh connection objects. **Surface** Use the scope to probe to a surface and study reactions on cutting planes. **Surface**: Visible when the **Location Method** is set to **Surface**. Provides a drop-down list of available surface objects. **Geometry**: This property corresponds to the **Surface** property when the **Location Method** is set to **Surface**. Select a geometry Body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	Boundary Condition This property is available for a number of probe types. It provides a drop-down list of available boundary conditions that you use to scope the probe to. **Orientation** Sets the direction of the coordinate system specified by the **Coordinate System** described above. **Spring** This property displays when the **Location Method** is set to **Spring**. It provides a drop-down list of available body-ground springs that you use to scope the probe to. **X coordinate** A read-only property that displays X Axis value for the **Coordinate System** property. **Y coordinate** A read-only property that displays Y Axis value for the **Coordinate System** property. **Z coordinate** A read-only property that displays Z Axis value for the **Coordinate System** property. **Summation** Displayed only for **Moment Reaction** probes when **Orientation** is also displayed. Allows you to specify the summation point where the moment reaction is reported. • **Centroid**: The simple calculated average; unweighted by length, area, or volume. • **Orientation System**; The coordinate system you specified with the **Orientation** setting. **By** *Harmonic Response Analysis Only*. This property displays for the **Force Reaction** and **Moment Reaction** probes. Property options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	Frequency When this option is specified, a **Frequency** entry property and the **Sweeping Angle** property also display. **Set** When this option is specified, a **Frequency** entry property and the **Sweeping Angle** property also display. **Maximum Over Frequency** When this option is specified, the **Sweeping Angle** property also displays. **Frequency of Maximum** When this option is specified, a **Frequency** entry property and the **Sweeping Angle** property also display. **Maximum Over Phase** When this option is specified, the **Frequency** property and **Phase Increment** property also display. **Phase of Maximum** When this option is specified, a **Frequency** entry property and **Phase Increment** property also display. **Extraction** Displayed only for **Force Reaction** and **Moment Reaction** probes when the **Location Method** is set to **Contact Region** or **Mesh Connection**. **Options for Contact Region Setting** When the **Location Method** is set to **Contact Region**, options include: • **Contact (Underlying Element)** • **Target (Underlying Element)** • **Contact (Contact Element)** (p. 2117) (**Force Reaction** Probe only) **Options for Mesh Connection Setting** When the **Location Method** is set to **Mesh Connection**, options include: **Primary** or **Secondary**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Category	Property Name and Description
	Orientation Method Only displayed for a **Joint Probe**. Options include **Joint Reference System** and **User Specified**.
Options	**Result Selection** The options for this property vary based on the selected type of probe. See the Overview and Probe Types (p. 1947) section for additional information based on your desired probe type. **Display Time** **End Time** or **Time Step**. **Spatial Resolution** When edges, vertices, faces, or bodies are selected as the Geometry, this property displays. It allows you to calculate the maximum (**Use Maximum**) or minimum (**Use Minimum**) result values across the given geometry selection. **Result Type** This property provides a list of available results for a **Joint Probe**.
Results	This category provides read-only properties of result you select in the **Result Selection** or **Result Type** drop-down list. The **Node ID** is displayed if you used the **Snap to mesh nodes** feature.
Maximum Value Over Time	This category provides read-only properties that vary based on the probe type. They display maximum values of the results you select over time in stepped analysis.
Minimum Value Over Time	This category provides read-only properties that vary based on the probe type. They display minimum values of the results you select over time in stepped analysis.
Information	Based on the probe type, the following read-only result-based properties may be provided by this category. • **Time** • **Load Step** • **Substep** • **Iteration Number**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Category	Property Name and Description
	• **Surface Area**

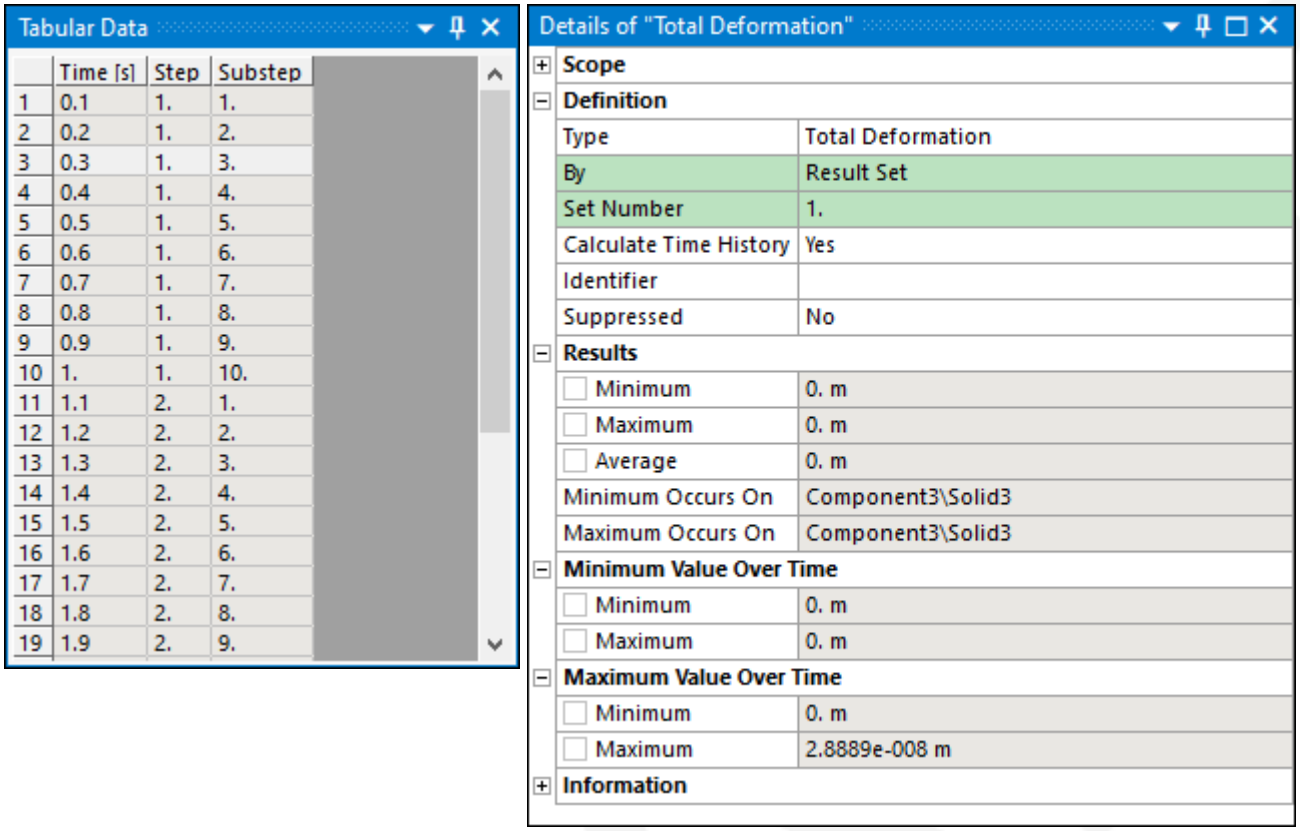
Note:

- When you set **Location Method** to **Coordinate System**, the probe traverses the primary axes to determine where the hits occur on the model. The hit closest to the origin of the coordinate system is used. This behavior is similar to placing a laser at the origin of the system and then shooting the laser sequentially along positive and negative direction of x, y, z axis.
- Probe objects scoped to x, y, z picked locations (using the **Hit Point Coordinate** selection tool) are achieved in such a way that a projection of the picked location in screen coordinates occurs onto the model based on the current view orientation, in other words, normal to the display screen onto the model at the picked location on the screen. If the geometry is updated, the update of the projection will follow the original vector that was established "behind the scenes" when the x, y, z pick was first made. Therefore the update of Probe objects scoped to x, y, z picking locations may not appear to be logical since it follows a vector that was established dependent on a view orientation when the original pick was made.
- The details view of the probe shows either the maximum or the minimum result values but not both.

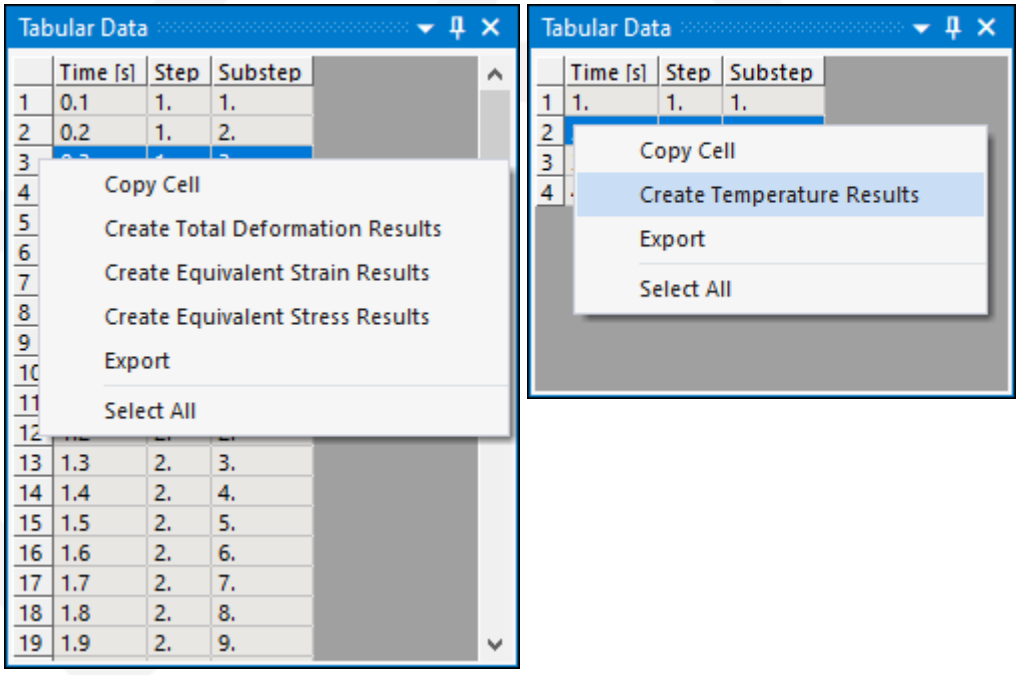
Result Set Listing

Once you solve a Static Structural, Transient Structural, or a thermal analysis, select the Solution object to display the **Tabular Data** window. The window displays time and load step values. This is a default feature. This data is a list of result sets. From the table, you can select a row and create certain results for a particular Set Number. Each row in the table is associated with the **Result Set** option of the **By** property and the value of the **Set Number** property for the supported result types (Total Deformation, Equivalent Total Strain, and Equivalent Total Stress).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The following example of the Tabular Data window illustrates the context menu for a structural analysis. Common selection/copy/export options are available as well as the ability to create new Total Deformation, Equivalent Strain, and Equivalent Stress results based on an available result set. However, these options are only available if displacement, strain, and stress data is present in the results file. If you are performing a thermal-based analysis, a menu option is available to create Temperature Results.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The **Changed Mesh** portion of this display is only applicable if your analysis has specified a **Nonlinear Adaptive Region** (p. 1707) condition or a **SMART Crack Growth** (p. 468) object. This column displays whether the application regenerated the mesh during the solution and it displays only when mesh regeneration takes place.

Tabular Data

	Time [s]	Changed Mesh	Step	Substep
1	1.e-001		1.	1.
2	0.2	Yes	1.	2.
3	0.3	Yes	1.	3.
4	0.4	Yes	1.	4.
5	0.5	Yes	1.	5.
6	0.6	Yes	1.	6.
7	0.7	Yes	1.	7.
8	0.8	Yes	1.	8.
9	0.9	Yes	1.	9.
10	1.	Yes	1.	10.
11	1.1	Yes	1.	11.
12	1.2		1.	12.

Surface Results

If you have already defined a surface (p. 890), you can view the surface results by first adding a standard result or user defined result (p. 2192), and in the Details view of the result object, setting **Scoping Method** to **Surface**, then choosing the name of the particular surface that you defined.

Note:

Surface results are not supported for models using periodic or cyclic symmetry.

The Details view for a surface result contains an additional item called **Average**, which can be parametrized.

For example, average stress over the surface is given by:

{ $\int$ Stress(X, Y, Z) dAREA} / {TOTAL_AREA}

For some results, the Details view will also contain a **Total** quantity, such as **Total Force**, which also can be parametrized.

The **Total** quantities are presented in the following table. Currently, if you desire a **Total** quantity for **Heat Flux**, **Magnetic Flux Density**, **Current Density**, or **Electric Flux Density**, you must choose a vector user defined result. **Total Force** (as integrated from principal stress vectors) is available to both standard and user defined results.

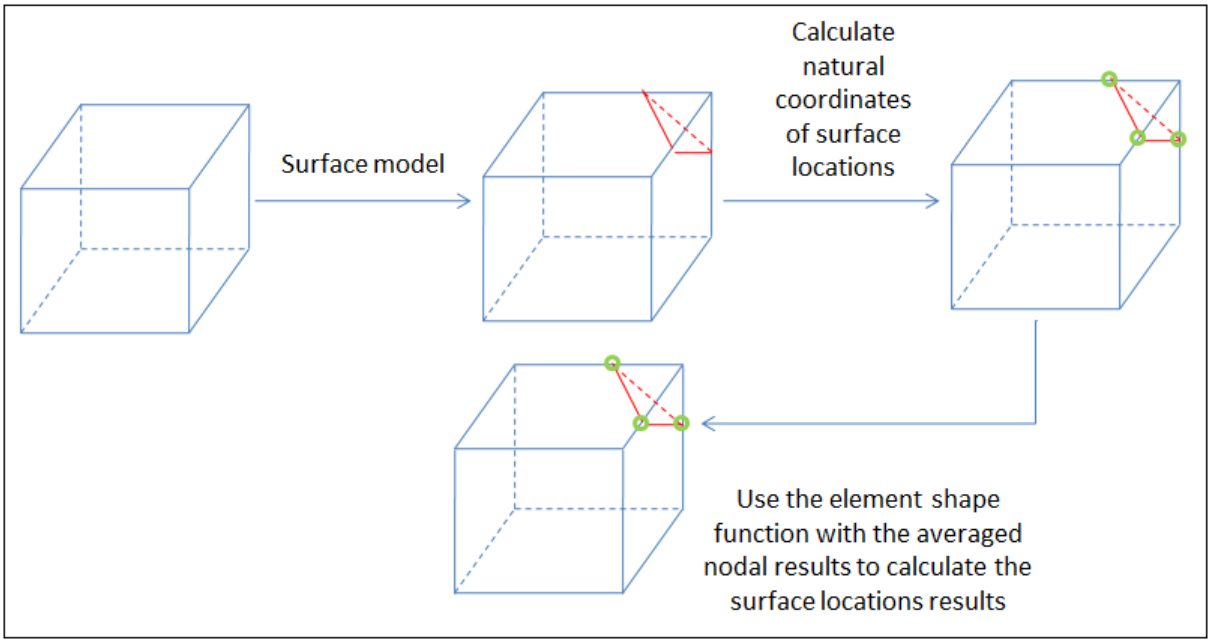
Identifier	Result	Surface Integral
TFVECTORS	Heat Flux	Heat Rate
BVECTORS	Magnetic Flux Density	Magnetic Flux
DVECTORS	Electric Flux Density	Charge
JTVECTORS, JCVECTORS	Current Density	Current

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

SVECTORS (also see Vector Principals (p. 2030))	Stress Tensor	Force

Interpolation of Data on a Surface

For a given surface (such as the intersection of a cutting plane and a finite element mesh), Mechanical examines each element in the set of scoped bodies to determine if any element edge was intersected by the surface. A surface may intersect multiple edges of a finite element, so Mechanical maintains a list of all (x, y, z) points and all element IDs from the edge intersections. This set of intersection points is, in essence, a set of interpolation points.



For example, you request a normal x-axis stress result on the surface (that is, *SX*). For a given interpolation point (x, y, z) lying on an edge of an element, Mechanical finds the natural (or normalized) coordinates of the point within the element. Mechanical then interpolates the corner values of *SX*, using the natural coordinates and shape functions, to find a value for *SX* at (x, y, z).

See the Interpolation (p. 1937) section for additional information.

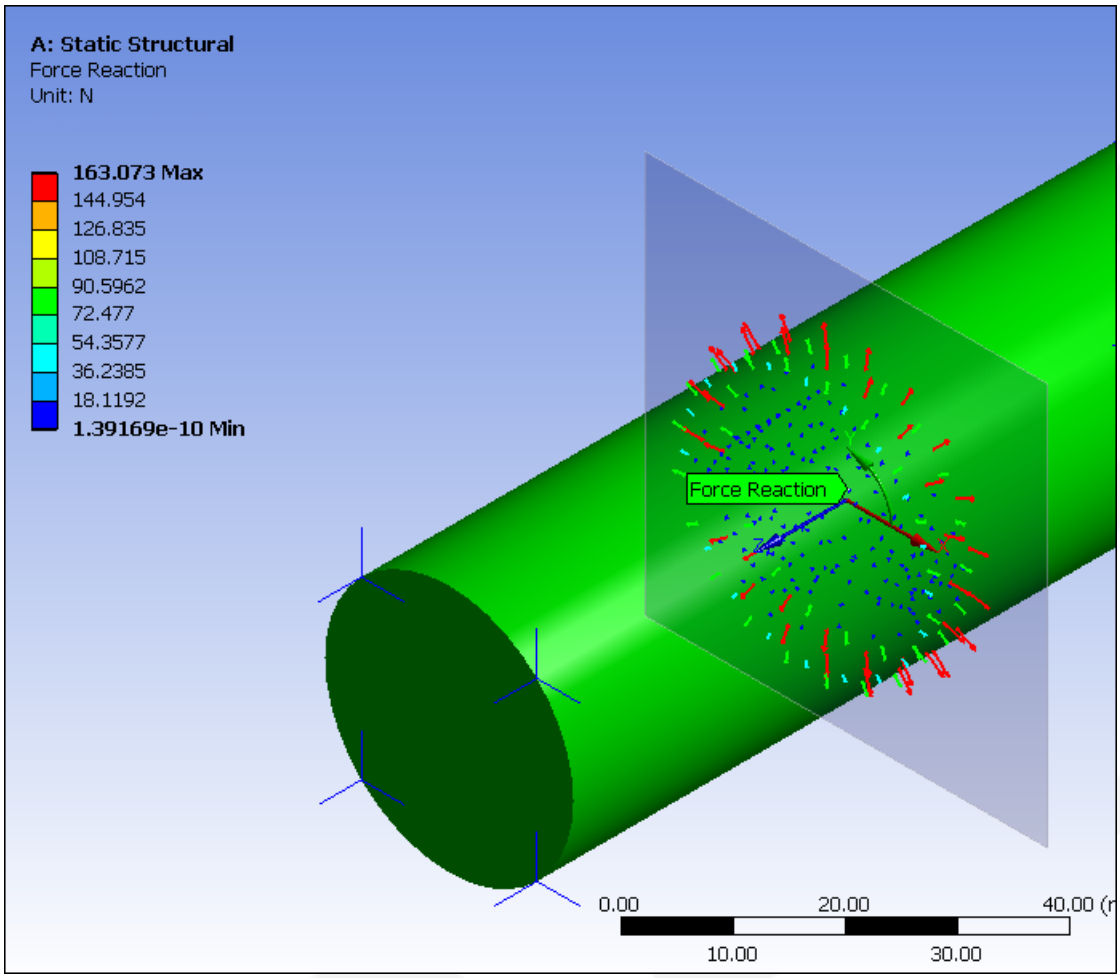
Force and Moment Reactions

For a pre-defined surface, a surface probe enables you to study reactions on cutting planes. You can extract generated member forces and reactions through a model by using a reaction probe scoped to a surface.

For this probe type, you must explicitly select the bodies to be sliced. You cannot apply this to "all bodies." You then specify for the Extraction detail whether you want to study nodes in front or behind the plane. The probe will only operate on elements cut by the plane (and only nodes on those elements which are on the selected side of the plane). Note that the surface probe will display nodal forces for all nodes that are involved in the reaction calculation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Surface Displays and Fracture

Mechanical analyzes for duplicates the sets of (x,y,z) for the facets in a surface construction object and compresses it by discarding all duplicate (x,y,z) sets. Mechanical employs compression to reduce the size of the data cache and to improve performance.

For each remaining (x,y,z) in the surface, Mechanical derives via interpolation the results (like displacements and stresses) from precisely one element. That is, even if an (x,y,z) resides in many elements, Mechanical only fetches the displacements from one element.

Hence, interpolated displacements at the (x,y,z) may currently fail to demonstrate the proper deformation of a crack.

Solution Summary Worksheet

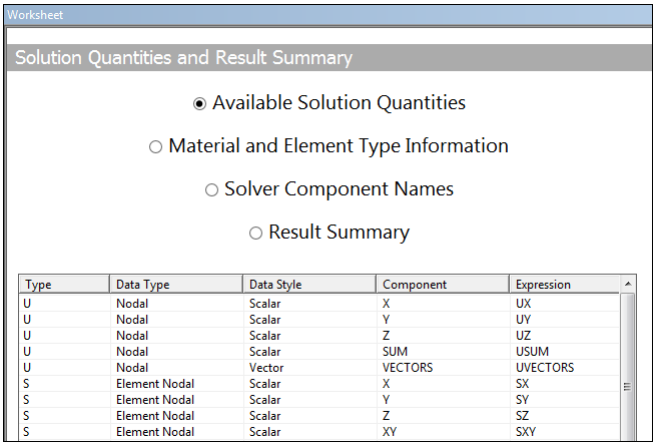
Once your solution is complete, you can display the **Solution Quantities and Result Summary** page of the Worksheet (p. 192) by either selecting the **Worksheet** button on the **Home (p. 40)** tab or by right-clicking on the Solution object and selecting the option **Worksheet: Result Summary**.

As illustrated in the images below, the Worksheet **Solution Quantities and Result Summary** provides the following options at the top of the page:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Available Solution Quantities

This option provides a list of the **User Defined Results** available in the result file. See the Application (p. 2194) section of the User Defined Results (p. 2192) section of the Help for the steps to use the Worksheet content available for this option.



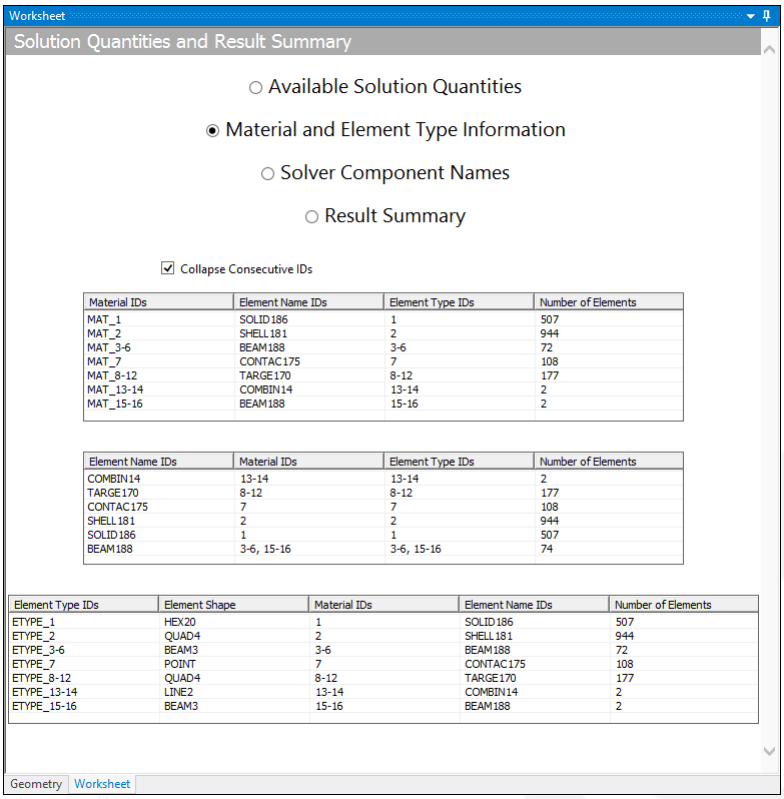
Type	Data Type	Data Style	Component	Expression
U	Nodal	Scalar	X	UX
U	Nodal	Scalar	Y	UY
U	Nodal	Scalar	Z	UZ
U	Nodal	Scalar	SUM	USUM
U	Nodal	Vector	VECTORS	UVECTORS
S	Element Nodal	Scalar	X	SX
S	Element Nodal	Scalar	Y	SY
S	Element Nodal	Scalar	Z	SZ
S	Element Nodal	Scalar	XY	SXY

Material and Element Type Information

This option displays a summary of all the Material and Element Type data present on the results file. You can use this data as a scoping option (p. 2002) for new result calculations. See the Result File Item Worksheet Features (p. 1965) topic below for the features available for this **Worksheet** selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



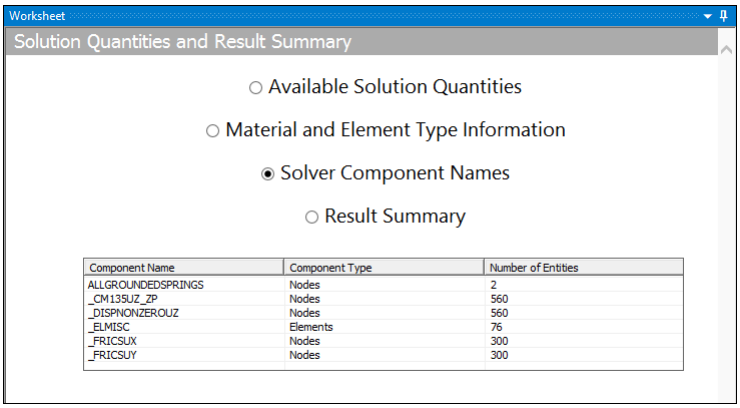
Important:

If your analysis includes a Condensed Part and a result is using the **Result File Item** option for the **Scoping Method**, **Material IDs** and **Element Name IDs** are not available for the **Material and Element Type Information** selection on the **Solution Quantities and Result Summary** page.

Solver Component Names

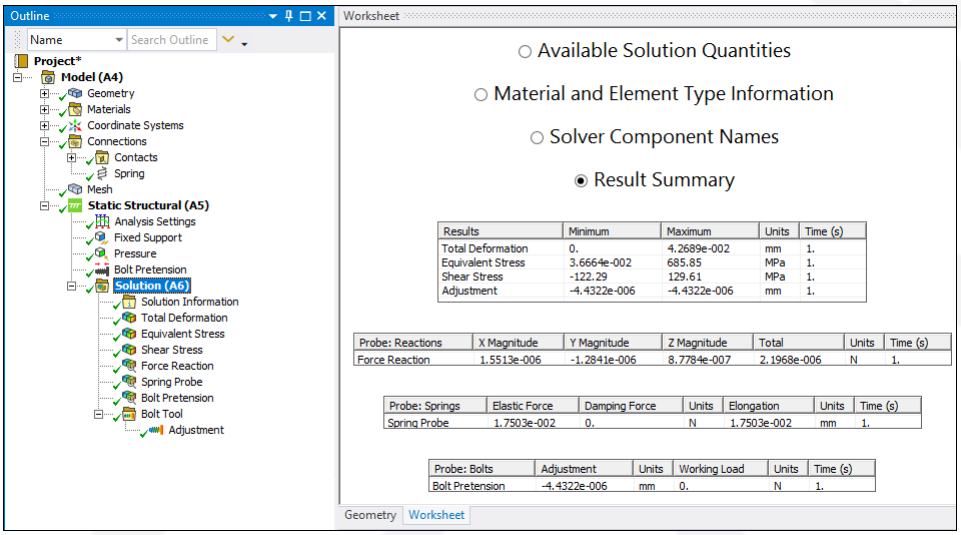
This option lists the Mechanical APDL Solver components present on the results file. These components are 1) node- or element-based groupings, 2) node- and element-based Name Selections that were written to the result file from Mechanical, or 3) components that the solver needed to generate in order to properly solve the analysis. Solver generated components typically contain an underscore (i.e. "_") at the beginning of the Component Name. You can use these components a scoping option (p. 2002) for new result calculations. See the Result File Item Worksheet Features (p. 1965) topic below for the features available for this **Worksheet** selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



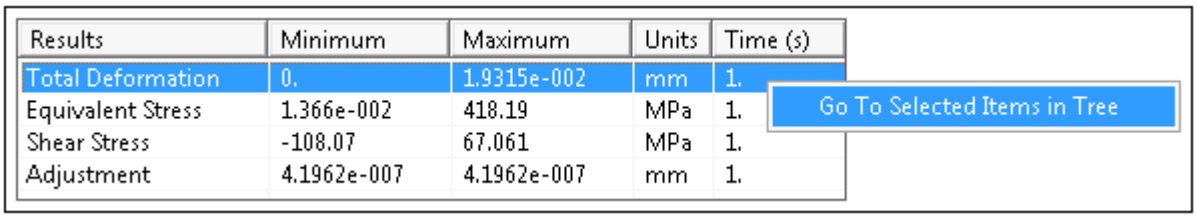
Result Summary

This option provides a tabular display of the minimum and maximum values of the available results, the associated unit of measure, and time step values.



Result Summary Features

For the results displayed by the **Result Summary** option, each table entry provides the right-click option, **Go To Selected Items In Tree**, to select and then graphically display the corresponding result object.



If a result is included in the tree but not yet evaluated, indicated by a yellow thunder bolt icon, the Worksheet displays a value of zero (note table entries above).

Result types supported by this feature include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

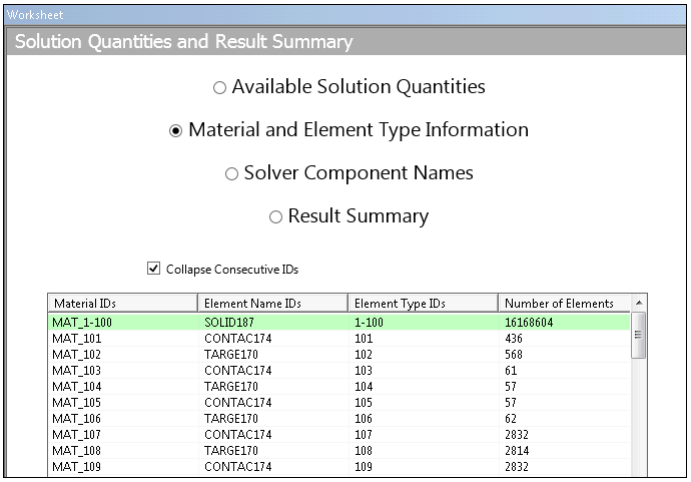
- Normal Contour/Vector type results such as Stress, Temperature, and Deformation.
- User Defined Results.
- Force and Moment Reaction Probes.
- Joint Probes reporting Force or Moment.
- Spring Probes.
- Bolt Pretension Probes.

Result File Item Worksheet Features

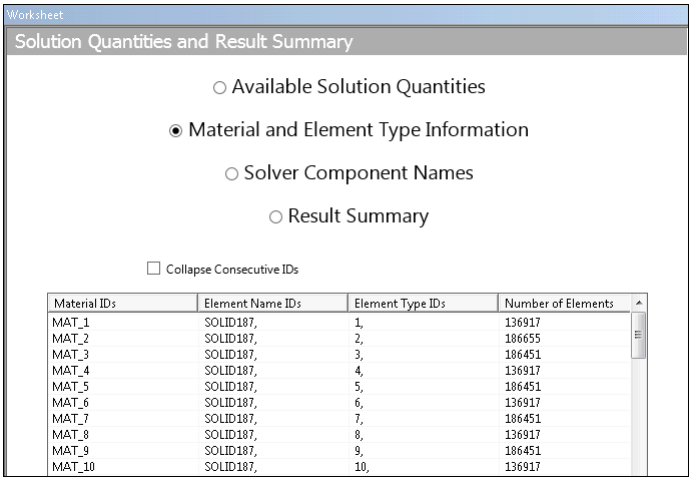
As described below, the **Worksheet** for the **Material and Element Type Information** and **Solver Component Names** options offers a number of use features.

Collapsing Consecutive IDs

The **Collapse Consecutive IDs** option is available for the **Material and Element Type Information** Worksheet display. As highlighted, this option organizes like materials and/or elements into singular rows in the table as opposed to creating a lengthy table list that repeats the same ID information. This option is active by default. Note that **Element Type IDs** do have a consecutive range of values and that this option displays this range when collapsed.

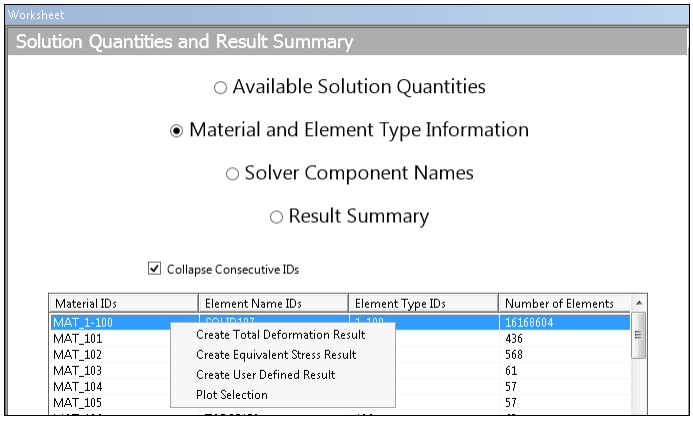


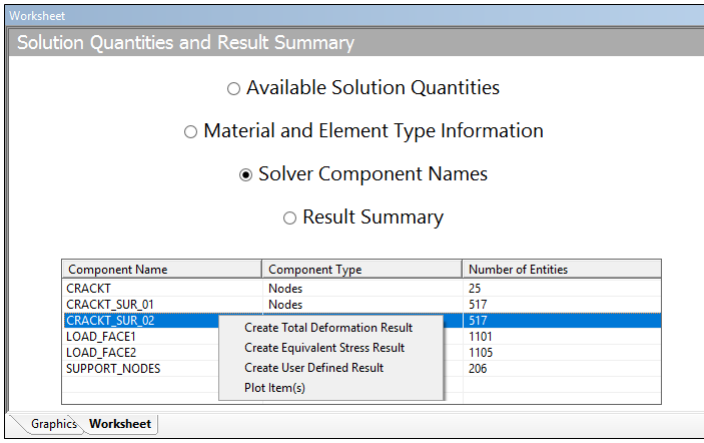
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Contextual Menu Options

You can right-click on a table listing to display options that enable you to automatically create a result. Based on your analysis type, the options include **Create Total Deformation Result**, **Create Equivalent Stress Result**, and **Create User Defined Result** (p. 2194) for structural analyses and **Create Temperature Result** and **Create User Defined Result** (p. 2194) for thermal analyses.



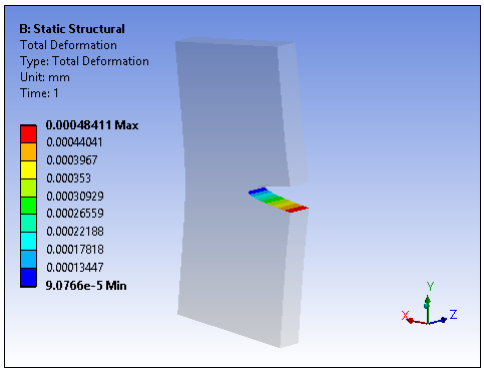


Important:

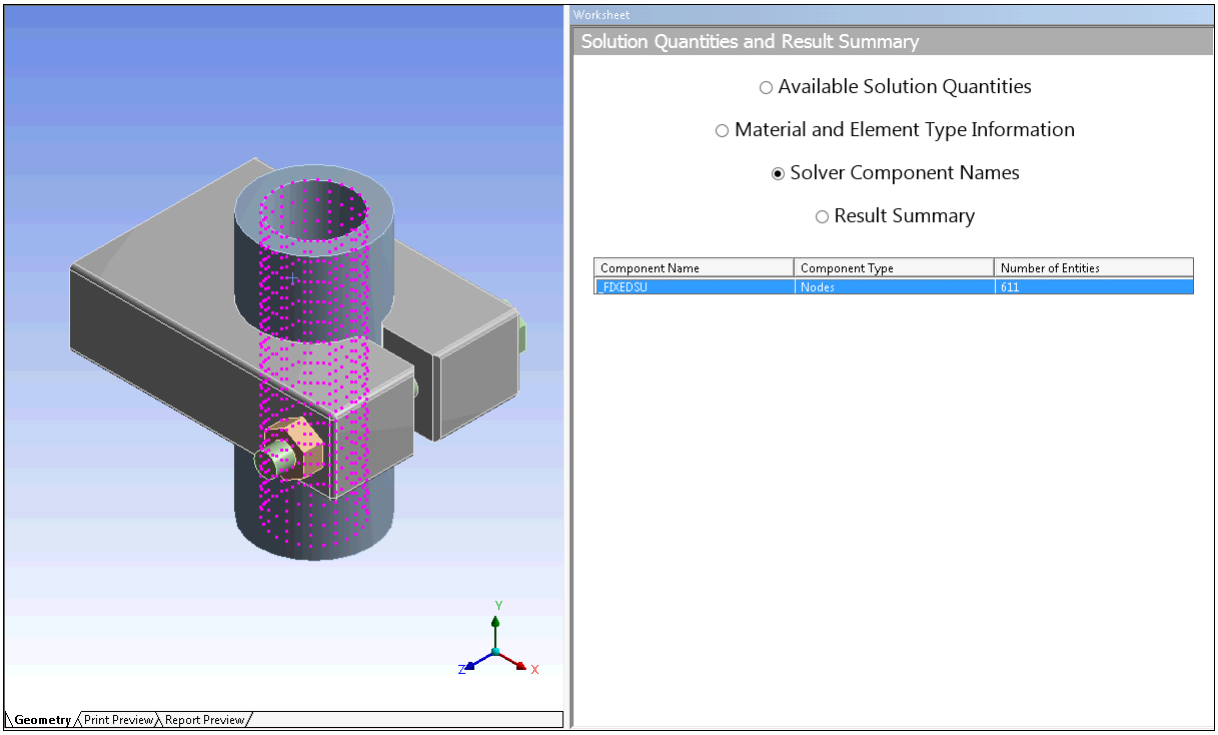
If you are performing a Fracture analyses (p. 406) that includes a crack (p. 424), the application automatically generates three Named Selections for the top and bottom face nodes as well as for the crack front nodes (if the **Crack Faces Nodes** property is set to **On**). The application displays these Named Selections in the **Solver Component Names** summary, as shown above (CRACKT_SUR_01, CRACKT_SUR_02, and CRACKT). The right-click options for these three table cells produce results that display contour colors at the scoped nodes only (as defined by the Named Selections). The remainder of the part displays as grey and translucent, as illustrated in the following example.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The contextual menu also includes the option **Plot Selection**. This option displays the elements or nodes on your model that are associated with the data (row or rows) selected in the **Worksheet**. In order to see these associated nodes and/or elements, you need to move the **Worksheet** so that you can also see the **Geometry** window. Refer to the Moving and Docking topic in the Windows Management (p. 168) section.



Note:

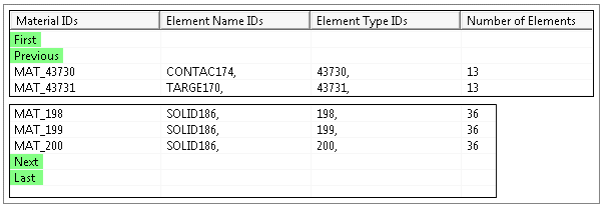
Double-clicking a single Worksheet row also plots the node selections for that row. This feature only works for a singular selection.

Previous and Next Selections

For the **Materials and Element Type Information** and the **Solver Component Names** selections, when your Worksheet includes more than 200 rows of data, the table provides **First**/**Previous** and **Next**/**Last** (double-click) options, as illustrated below. These options display at the end of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the table as well as at the beginning of the table (once you have advanced to a new page). They enable you to page through the data and/or jump to the beginning and the end of the Worksheet.



Material IDs	Element Name IDs	Element Type IDs	Number of Elements
First			
Previous			
MAT_43730	CONTAC174,	43730,	13
MAT_43731	TARGE170,	43731,	13
MAT_198	SOLID186,	198,	36
MAT_199	SOLID186,	199,	36
MAT_200	SOLID186,	200,	36
Next			
Last			

Vector Plots

Certain result items can be displayed using vectors such as the vector principal stresses or vector principal strain results. Similarly total deformation, total velocity and total acceleration can also be displayed using vectors. Using the **Graphics** button, you can display results as vectors with various options for controlling the display. See the Vector Display (p. 74) section for more information.

Result Definitions

The following topics related to result definitions are covered in this section.

Applying Results Based on Geometry

Specifying Result Coordinate Systems

Applying Decibel (dB) Weighting Filters

Defining Result Identifiers

Understanding the Material Properties Used in Postprocessing

Clearing Result Data

Understanding Averaged and Unaveraged Contour Results

Working with Multiple Result Sets

Displaying Surface Body Results (including Layered Shell Results)

Reviewing Unconverged Results

Handling of Degenerate Elements

Understanding Result Data Display Issues

Important:

Note the following application result calculations applicable to Coordinate System selection.

Quantities such as Equivalent Stress (SEQV) and Thermal Flux Sum (TFSUM) are derived from entities like tensors and vectors.

If the tensors or vectors are in the Solution Coordinate System and if the elemental coordinate systems vary from element to element (that is, shell model), then the process of averaging can affect Equivalent Stress values, as well as the values of other derived quantities.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Therefore, if you specify the Solution Coordinate System option for the Coordinate System property, you cannot expect the averaged SEQV solution to be the same as when you specify the Global Coordinate System option for the Coordinate System property.

Furthermore, for results that can display unaveraged contour results, if you specify Unaveraged for the Display Option property, the SEQV solution for Solution Coordinate System is the same as the solution for Global Coordinate System setting.

Applying Results Based on Geometry

The results available for your analysis may depend on the selected geometry type (Solid, Surface, or Line). The tables below describe how to match geometry and result types (available on the Solution Context (p. 62) tab).

Go to a section topic:

- Static Structural Analysis (p. 1969)
- Transient Analysis (p. 1970)
- Modal and Eigenvalue Buckling Analyses (p. 1971)
- Random Vibration Analysis (p. 1971)
- Response Spectrum Analysis (p. 1972)
- Steady-State Thermal and Transient Thermal Analyses (p. 1973)
- Magnetostatic Analysis (p. 1973)
- Electric Analysis (p. 1973)
- Harmonic Response Analysis (Deformation, Strain, Stress) (p. 1973)
- Harmonic Response Analysis (Frequency Response, Phase Response, User Defined Result) (p. 1974)

Static Structural Analysis

Geometry	Solution Tab Options				
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)	**Toolbox**	**User Defined Result** (p. 2192)
Solid Body	**Total**, **Directional**	All choices	All choices	**Stress** (p. 2058), **Fatigue** (p. 2181), **Contact** (p. 2079)[a]	Yes
Surface Body	**Total**, **Directional**	All choices	All choices	**Stress** (p. 2058), **Fatigue** (p. 2181),	Yes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

				Contact (p. 2079)[a]	
Line Body: Beam (p. 872)	**Total**, **Directional**	All choices, *except* **Equivalent (von-Mises)**, **Equivalent Plastic**, **Equivalent Creep**, **Equivalent Total**	All applicable choices, *except* **Error**	**Contact** (p. 2079)[a], **Beam** (p. 2087)	Yes
Line Body: Pipe	**Total**, **Directional**	None	None	**Contact** (p. 2079)[a], **Beam** (p. 2087)	Yes

[a] Contact results are *not* reported and are not applicable to the edges, MPC Contact, or the Target side of asymmetric contact.

Transient Analysis

Geometry	**Solution Tab Options**				
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)	**Toolbox**	**User Defined Result** (p. 2192)
Solid Body	All choices	All choices	All choices	**Stress** (p. 2058), **Fatigue** (p. 2181), **Contact** (p. 2079)[a]	Yes
Surface Body	All choices	All choices	All choices	**Stress** (p. 2058), **Fatigue** (p. 2181), **Contact** (p. 2079)	Yes
Line Body: Beam (p. 872)	All	All choices, *except* **Equivalent (von-Mises)**, **Equivalent Plastic**, **Equivalent Creep**, **Equivalent Total**	All applicable choices, *except* **Error**	**Contact** (p. 2079)[a], **Beam** (p. 2087)	Yes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Line Body: Pipe	All	None	None	**Contact** (p. 2079)[a], **Beam** (p. 2087)	Yes

[a] Contact results are *not* reported and are not applicable to the edges, MPC Contact, or the Target side of asymmetric contact.

Modal and Eigenvalue Buckling Analyses

Geometry	Solution Tab Options				
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)	**Toolbox**	**User Defined Result** (p. 2192)
Solid Body	**Total**, **Directional**	All applicable choices, *except* **Energy**	All choices	None	Yes
Surface Body	**Total**, **Directional**	All applicable choices, *except* **Energy**	All choices	None	Yes
Line Body: Beam (p. 872)	**Total**, **Directional**	All choices, *except* **Equivalent (von-Mises)**, **Equivalent Plastic**, **Equivalent Creep**, **Equivalent Total**	All applicable choices, *except* **Error**	None	Yes
Line Body: Pipe	**Total**, **Directional**	None	None	None	Yes

Random Vibration Analysis

Geometry	Solution Tab Options				
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)	**Toolbox**	**User Defined Result** (p. 2192)
Solid Body	**Directional**, **Directional Velocity**, **Directional Acceleration**	**Normal**, **Shear**	**Equivalent (von-Mises)**, **Normal**, **Shear**	None	No
Surface Body	**Directional**, **Directional**	**Normal**, **Shear**	**Equivalent (von-Mises)**,	None	No

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	Velocity, **Directional Acceleration**		**Normal**, **Shear**		
Line Body: Beam (p. 872)	**Directional**, **Directional Velocity**, **Directional Acceleration**	**Normal**, **Shear**	**Equivalent (von-Mises)**, **Normal**, **Shear**, **Membrane Stress**, **Bending Stress**	None	No
Line Body: Pipe	**Directional**, **Directional Velocity**, **Directional Acceleration**	None	None	None	No

Response Spectrum Analysis

Geometry	**Solution Tab Options**				
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)	**Toolbox**	**User Defined Result** (p. 2192)
Solid Body	**Total**, **Directional**, **Directional Velocity**, **Directional Acceleration**	**Normal**, **Shear**	**Equivalent (von-Mises)**, **Normal**, **Shear**	None	No
Surface Body	**Total**, **Directional**, **Directional Velocity**, **Directional Acceleration**	**Normal**, **Shear**	**Equivalent (von-Mises)**, **Normal**, **Shear**	None	No
Line Body: Beam (p. 872)	**Total**, **Directional**, **Directional Velocity**, **Directional Acceleration**	**Normal**, **Shear**	**Equivalent (von-Mises)**, **Normal**, **Shear**, **Membrane Stress**, **Bending Stress**	None	No
Line Body: Pipe	**Total**, **Directional**, **Directional Velocity**, **Directional Acceleration**	None	None	None	No

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Steady-State Thermal and Transient Thermal Analyses

Geometry	Solution Tab Options	
	Thermal (p. 2153)	**User Defined Result** (p. 2192)
Solid Body	**Temperature**, **Total Heat Flux**, **Directional Heat Flux**, **Error**	Yes
Surface Body	**Temperature**, **Total Heat Flux**, **Directional Heat Flux**, **Error**	Yes
Line Body: Beam (p. 872)	**Temperature** (p. 2153)	Yes
Line Body: Thermal Flow	All choices	No

Magnetostatic Analysis

Geometry	Solution Tab Options	
	Electromagnetic (p. 2157)	**User Defined Result** (p. 2192)
Solid Body	All choices[a]	Yes
Surface Body	Not Applicable	Yes
Line Body	None	Yes

[a] Electric Potential can only be scoped to conductor bodies.

Electric Analysis

Geometry	Solution Tab Options	
	Electric (p. 2162)	**User Defined Result** (p. 2192)
Solid Body	All choices	Yes
Surface Body	Yes	Yes
Line Body	Yes	Yes

Harmonic Response Analysis (Deformation, Strain, Stress)

Geometry	Solution Tab Options		
	Deformation (p. 2018)	**Strain** (p. 2026)	**Stress** (p. 2026)
Solid Body	All choices[a]	All choices, *except* **Energy**, **Thermal**, **Equivalent Plastic**	All choices
Surface Body	All choices[a]	All choices, *except* **Energy**, **Thermal**, **Equivalent Plastic**	All choices
Line Body: Beam	All choices	All choices, *except* **Equivalent (von-Mises)**, **Energy**, **Thermal**, **Equivalent Plastic**,	All choices

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

		Equivalent Creep, Equivalent Total	
Line Body: Pipe	All choices	None	None

[a] See the Harmonic Analysis (p. 496) section for more information.

Harmonic Response Analysis (Frequency Response, Phase Response, User Defined Result)

Geometry	**Solution Tab Options**		
	Frequency Response[a]	**Phase Response**[a]	**User Defined Result** (p. 2192)
Solid Body	All choices	All choices	Yes
Surface Body	All choices	All choices	Yes
Line Body	All choices	All choices	Yes

[a] See the Harmonic Analysis (p. 496) section for more information.

Specifying Result Coordinate Systems

The application, by default, creates a global coordinate system for your model when it is imported. A variety of environmental objects use the **Global Coordinate System** option during the analysis.

A **Coordinate System** property is available for results that change with respect to a coordinate system, such as Normal Stress. For the applicable result types, the Coordinate System property provides a drop-down list of available coordinate system options, which include:

- **Global Coordinate System** (default setting)
- User-defined local coordinate systems (p. 1192)
- **Solution Coordinate System** (p. 1975)

The **Global Coordinate System** setting is the default setting. However, you can create local coordinate systems (p. 1192) and apply them to individual result objects. Also, the **Solution Coordinate System** (p. 1975) option is available for circumstances such as examining surface and line bodies that align themselves on a per element basis. Note, in some scenarios, result contours in the **Solution Coordinate System** can appear to be a random collection of colors.

Note:

- If a given result experiences any change in position due to the loading conditions, such as a displacement or deformation, the **Geometry** window always displays this position change in the **Global Coordinate System** (per global XYZ coordinate triad). If you have specified a local coordinate system for your result, the application exposes all data contained in the **Details** view as well as the **Tabular Data** window based the local coordinate system entry. Furthermore, if applicable, the application always displays result contour colors in the **Geometry** window based on a specified local coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The Mechanical APDL solver treats a "cylindrical" coordinate system as a set of Cartesian coordinate systems aligned with the specified cylindrical system at each point in the model. See the Additional POST1 Postprocessing section in the *Mechanical APDL Basic Analysis Guide* for additional technical information.
- For **Probes**, the **Orientation** property specifies coordinate systems.

Solution Coordinate System

Solution Coordinate System is an option of the **Coordinate System** property in the Details view for most result objects. If you are familiar with the Mechanical APDL application commands, **Solution Coordinate System** is an implementation of the **RSYS**,SOLU command, where for element results, such as stress, a coordinate system is produced for each element. If these individual element coordinate systems are aligned randomly, you can re-align them to a local coordinate system to obtain a uniform alignment. Viewing results in the element solution coordinate system has value since results in a local coordinate system aligned with a certain shell direction are typically more meaningful than results in a global coordinate system. For example, seeing bending and in-plane stresses have meaning in a local coordinate system, but have no meaning in a global coordinate system. See the Additional POST1 Postprocessing section in the *Mechanical APDL Basic Analysis Guide* for additional information.

Important:

For Beam and Pipe elements:

- The following stress and strain component results are displayed in the element coordinate system:
  - Normal X, Y, and Z
  - Shear XY, YZ, and XZ
- When a Line Body (p. 872) is specified as a beam/pipe that includes cross sections (**SECDATA**), beam strains/stresses always use the **Solution Coordinate System**, even if you specify the global coordinate system. To calculate beam strains/stresses, you need to set the **Beam Section Results** property of the Solution object to **Yes**.

Application

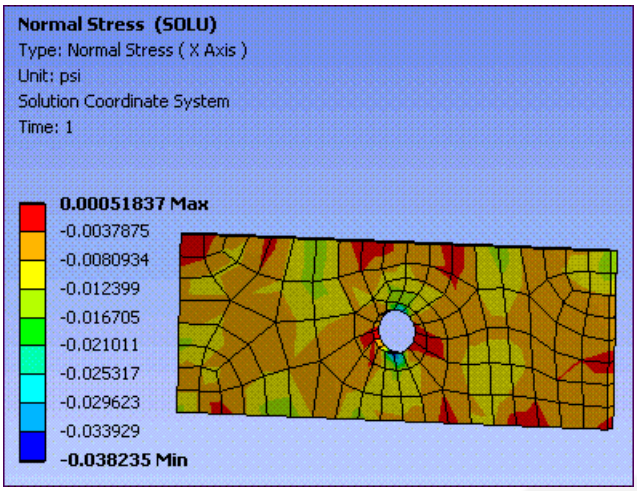
The following are typical applications for viewing results in a solution coordinate system:

- Viewing results in a particular direction for surface bodies or "solid shell" bodies, that is, solids meshed with the **Solid Shell** element option (see the Meshing Help: **Sweep** description in the Method Control section).
- Viewing results in a random vibration, spectrum, or surface bodies in an explicit dynamics analysis. Results for these analysis types only have meaning in a solution coordinate system.

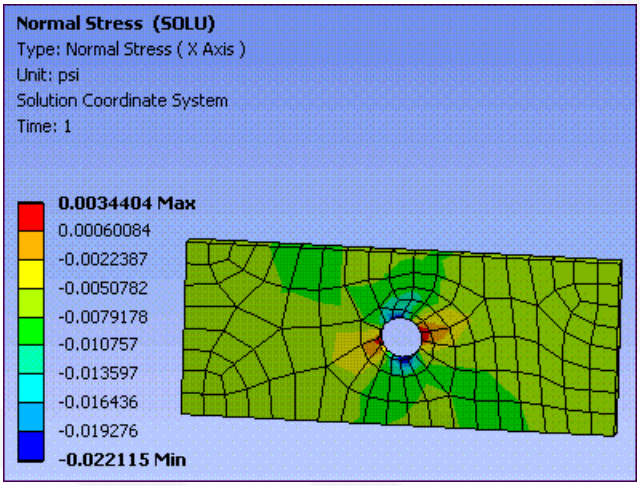
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Background

The meshing of surface bodies and solid shell bodies result in coordinate systems whose alignment is on a per element basis, in contrast to solid body element types whose coordinate systems are aligned with the global coordinate system by default. Surface body alignment on a per element basis can lead to results with totally random alignment directions as shown below.



To produce meaningful results for surface body and solid shell bodies, you can re-align the random direction of each element's solution coordinate systems to a uniform direction of a local coordinate system. An example is shown below.



Procedure

To correct for random coordinate system alignments in surface bodies and solid shell bodies, and to ensure a consistent alignment:

1. For each part, create a local coordinate system (p. 1191) to specify the alignment of the elements of the part.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2. Choose the **Solution Coordinate System** option for the result.

Note:

- The **Coordinate System** setting for result objects in a random vibration or spectrum analysis is set to **Solution Coordinate System** by default and cannot be changed because the results only have meaning when viewed in the solution coordinate system.
- In Explicit Dynamics analyses, the solution coordinate system only has meaning for results on surface bodies. For results with the Solution Coordinate System selected, all bodies scoped to the result with body types other than shell bodies will have the results displayed in the global coordinate system.
- The solution coordinate system is only supported for beam elements in LS-DYNA analyses for results.

Applying Decibel (dB) Weighting Filters

For **Harmonic Response** analyses, the application enables you to specify A-, B-, and C-frequency-weighting for the Equivalent Radiated Power Level (p. 2047) (ERPL) and Equivalent Level Waterfall Diagram (p. 2135) result options.

Application

For the supported results, set the **dB Weighting** property to either **A**, **B**, or **C** to produce results in decibels in order to calculate results perceptible to human hearing.

Weighting Calculations

Weightings are calculates as shown below.

A-Weighting

$$R_A(f) = \frac{12194^2 f^4}{(f^2+20.6^2)\sqrt{(f^2+107.7^2)(f^2+737.9^2)}\,(f^2+12194^2)},$$
$$A(f) = 20\log_{10}(R_A(f)) - 20\log_{10}(R_A(1000))$$

B-Weighting

$$R_B(f) = \frac{12194^2 f^3}{(f^2+20.6^2)\sqrt{(f^2+158.5^2)}\,(f^2+12194^2)},$$
$$B(f) = 20\log_{10}(R_B(f)) - 20\log_{10}(R_B(1000))$$

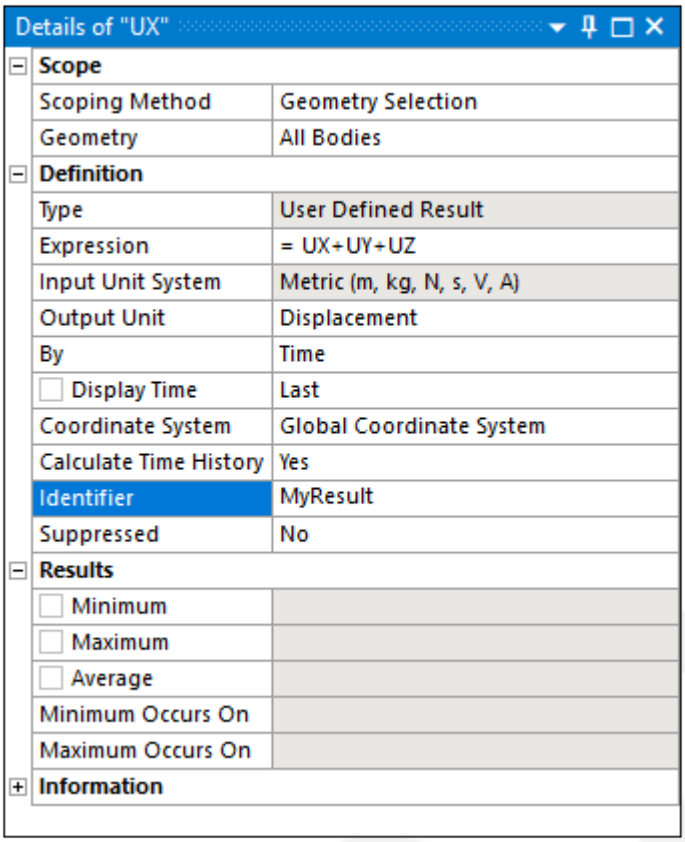
C-Weighting

$$R_C(f) = \frac{12194^2 f^2}{(f^2+20.6^2)(f^2+12194^2)},$$
$$C(f) = 20\log_{10}(R_C(f)) - 20\log_{10}(R_C(1000))$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Defining Result Identifiers

Using the **Identifier** property, you can assign a unique name to a result.



Requirements

Note the rules for creating an identifier. An Identifier entry:

- Can begin with a letter or an underscore character.
- Can contain any number of letters, digits, or underscores.
- Is not case sensitive, however, functions (p. 2200) should always use lowercase (sqrt, max, min, etc.).
- Is not affected by the order in which they are entered. For example, for Identifiers **A** and **B**, the expression for:
  - Result 1 can equal: B = 2*A, and:
  - Result 2 can equal: A = UX

  Ansys recommends that you use the proper order and try to define dependents first. For example, define A, B, C and then D = A^2+B^2+C^2
- Cyclic dependencies are blocked, such as the following:
  - Result 1: A = UX + C

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - Result 2: C = 2 * A - 1

- Correspond to an array over all nodes (or all elements):
  - Length = number of nodes (or elements)
  - Width = 1, 3, or 6 columns

Combining with Expressions

An **Identifier**, together with **Expression** content (UX, UY, etc.), can be used in combination with other user defined results. For example, using the identifier **MyResult**, you could create the **Expression**: **`sqrt(MyResult+UX+UY)`**. In addition, if an **Identifier** is used in an expression, it must be scoped to the same geometry.

Ansys recommends that when you assign an identifier to the expression, that you rename the object with the same name/identifier.

Evaluated Results and Downstream Processing

You can edit the Identifier property for an evaluated result. Doing so affects any downstream results that use the Identifier as part of their **Expression**. The application will automatically clear these results and mark them Obsolete.

Note:

The application includes a preference that will attempt to automatically update downstream Expression fields that include revised Identifiers. In the preference dialog, under the **Results** category, set the **Update Dependent Expressions on Identifier Change** to **Yes**. The default is **No**.

Important:

Be careful when using this preference. Automatically replacing expressions may cause failures or result in inadvertent errors.

Limitations of the Identifier Property

There are several problematic scenarios that can arise when you use the **Identifier** property for an existing result to create a new result. For each scenario, changing an item in the Details of the new result causes the new result to be unreliable.

For example, the **Display Time** of a result is only relevant when the expression consists of built-in identifiers. Unlike user-defined identifiers, built-in identifiers retain their time dependence through

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the evaluation of the expression. To reveal the built-in identifiers for a given solver, examine the **Worksheet** view on the **Solution** folder.

Note:

Mechanical may not necessarily issue a warning or error message for these situations.

For example, what if the **Identifier** of the original result is "Original". Further, suppose that the **Expression** of the new result is "2 * Original". Consider the following:

- *Different choices of **By Time** or **By Result Set***
- *Different choices of the value of **Display Time** or **Set***
- *Different choices of **Coordinate System***
- *Different choices of **Yes/No** for **Calculate Time History***
- *Different choices for **Use Average***

Understanding the Material Properties Used in Postprocessing

The material properties listed below are used in postprocessing calculations to produce the displays of probe and contour results. For reference, the corresponding labels (Lab argument) for the **MP** command in Mechanical APDL are included in parentheses.

- Elasticity modulus (EX, EY, EZ)
- Shear modulus (GXY, GYZ, GXZ)
- Poisson's ratio (NUXY, NUYZ, NUXZ)
- Thermal conductivities (KXX, KYY, KZZ)
- Magnetic permeability (MURX, MURY, MURZ)

The following results, together with their identifiers (see User Defined Results (p. 2192)), are directly affected by the material property values:

- Equivalent Strain (p. 2027) - uses only NUXY Poisson's ratio
  1. Plastic (EPPL) and Creep (EPCR) strain always use NUXY = 0.5.
  2. Elastic (EPEL), Thermal (EPTH) and Total (EPTO) default to 0.0.
- Structural Error (p. 2030) - uses elasticity modulus, shear modulus and Poisson's ratio.
- Thermal Error (p. 2154) - uses thermal conductivities
- Magnetic Error (p. 2161) - uses magnetic permeability

An error message is generated if an associated material property is not defined when evaluating Structural, Thermal or Magnetic Error result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If Poisson's ratio is not defined when evaluating Equivalent Strain, the Poisson's ratio will assume a zero value.

Other results affected by material property values include Stress Tool (p. 2058) and Fatigue Tool (p. 2165) results.

Note:

If a material property is temperature dependent, it is evaluated at the reference temperature of the body to be used in the computation for the result.

Clearing Result Data

You can clear results and meshing data from the database using the **Clear Generated Data** option from the **File** tab, or using the right-click (context) menu option. This reduces the size of the database file, which can be useful for archiving.

To clear all results data, simply select the **Solution** object and choose the **Clear Generated Data** menu item from the **File** tab or from a right-mouse click menu. You can clear individual results by selecting a result object before choosing the **Clear Generated Data** option.

Important:

- A geometry or mesh change clears **all** results data.
- The application clears the solution and the results if the mesh becomes obsolete. See Clearing Generated Data in the *Meshing User's Guide*.

Understanding Averaged and Unaveraged Contour Results

Go to a section topic:

- Overview (p. 1981)
- Display Option (p. 1983)
- Average Across Bodies: Using the Feature (p. 1984)
- Support Limitations (p. 1986)
- Averaging Element Nodal Results (p. 1987)
- Corner Node Calculation Examples (p. 1987)

Overview

Normally, contour results in the Mechanical application are displayed as *averaged* results. Some results can also display as *unaveraged* contours. Averaged contours distribute the average elemental nodal results across element and geometric discontinuities. A user option exists that allows you to control

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

whether results are also averaged across body boundaries that contain a conformal mesh. The default setting does not calculate an average across bodies.

Using the Mechanical APDL application terminology, unaveraged contour results display as element nodal contours that vary discontinuously even across element boundaries. These contours are determined by linear interpolation within each element and are unaffected by surrounding elements (that is, no nodal averaging is performed). The discontinuity between contours of adjacent elements is an indication of the gradient across elements.

Results that include the unaveraged contour display option are elemental-nodal quantities such as stress or strain. This option is not available for degree of freedom results such as displacements.

Nodal averaging of element quantities involves direct averaging of values at corner nodes. For higher-order elements, midside node results are then taken as the average of the corner nodes.

In Mechanical, there are two distinct techniques for calculating averaged principal nodal results. The calculation for the first technique is as follows (using stress as the result in question):

1. Average the component (X, Y, Z, XY, YZ, XZ) stress values from the elements at a common node.

2. Calculate the principal stresses from the averaged component values

The calculation for the second technique is as follows:

1. Calculate the principal stress values (from the six component strains) on a per element basis.

2. Average these values from the elements at a common node.

For principal tensor results excluding equivalent strain, the first technique is normally used to calculate the results. For equivalent stress, stress/strain intensity, max shear stress/strain, and principal stresses/strains, the first technique is used to calculate the results. For equivalent strains, which are calculated by the Mechanical APDL solver, the second technique is used. For random vibration analysis, equivalent stresses are calculated by the Mechanical APDL solver using the Segalman method, so the second technique is also used. See the Average Across Bodies: Using the Feature (p. 1984) topic below for additional discussions. Also see the **AVPRIN** command in the *Mechanical APDL Commands Reference*.

Note:

If an elemental result is scoped to a surface body, then there may be two sets of results at each node (**Top** and **Bottom**) and sometimes a third set of results (**Middle**). At release 12.0, if the solver writes **Middle** results to the result file, then Mechanical displays these results if the **Shell Face** setting in the Details view equals **Middle** (Membrane). If the solver did not write **Middle** results to the result file, then Mechanical displays the average of **Top** and **Bottom** if the **Shell Face** setting in the Details View is **Middle** (Membrane). For a given node on the shell, the application will average **Top** results, separately average **Bottom** results, and separately average **Middle** results. When you export a result in the application that is set to Top/Bottom, you may note that a node number is repeated in the Excel file. This is because both the Top and **Bottom** stresses are listed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Beginning at Release 16.0, Mechanical supports the **Top**/**Bottom**/**Middle** positions for layered solid bodies. Therefore, layered solid elements mimic expanded shell elements, which display as having a thickness between the **Top** face and **Bottom** face.

Display Option

The **Display Option** property enables you to display contour results as:

- **Unaveraged**: Displays unaveraged results.
- **Averaged** (default): Displays averaged results.
- **Nodal Difference**: Computes the maximum difference between the unaveraged computed result (for example, total heat flux, equivalent stress) for all elements that share a particular node.
- **Nodal Fraction**: Computes the ratio of the nodal difference and the nodal average.
- **Elemental Difference**: Computes the maximum difference between the unaveraged computed result (for example, total heat flux, equivalent stress) for all nodes in an element, including midside nodes.
- **Elemental Fraction**: Computes the ratio of the elemental difference and the elemental average.
- **Elemental Mean**: Computes the elemental average from the averaged component results.

Caution:

Although you can select the above options, certain result types do not support them. For example, the application will generate a **Warning** if you specify **Nodal Difference** or **Nodal Fraction** for elemental results, such as **Volume**, and the result will not be evaluated. This is also true if you select **Elemental Difference**, **Elemental Fraction**, and **Elemental Mean** for nodal results, such as displacement.

Note:

For result displays on shells, when the **Display Option** property is set to **Elemental Mean** and the **Position** property is set to **Top/Bottom**, the mean value will include values from both the top and bottom of the shell. You can view the **Elemental Mean** for either the top or bottom of the shell by using the **Position** property setting **Top** or **Bottom**.

Characteristics of unaveraged contour displays:

- Because of the added data involved in the processing of unaveraged contour results, these results take a longer time to display than averaged results.
- Occasionally, unaveraged contour result displays tend to resemble a checkerboard pattern.
- Capped Isosurface displays can have missing facets.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Average Across Bodies: Using the Feature

When you select **Averaged** as the **Display Option**, the **Average Across Bodies** property displays in the **Integration Point Results** category. Setting this property to **Yes** (the default value is **No**) allows you to averages the results across separate bodies on your model. This post-processing feature is supported for most averaged element nodal contour results (like stress, strain, and thermal flux).

If a node belongs to two different bodies, its averaged stress value of one of the bodies is typically different from the stress value of the other body. Using the **Average Across Bodies** feature, the average value at this node is the sum of all of the stress values from all "scoped" elements that contain the node (divided by the number of elements). The feature graphically renders a smoother result contour at the interfaces of bodies. If bodies do not share any nodes, then the feature has no effect.

Calculation Condition Notes

Note the following requirements for averaging results across bodies.

- A principal value is averaged when Mechanical averages results across bodies for principal and vector sums. Calculated results, such as the types listed below, are averaged according to the following definitions:

  1) Tensor or vector components (corresponding to X,Y,Z,YZ,YZ,XZ) at the node in question are averaged within each body connected to that node, and,

  2) A principal value, such as S1 or SEQV or TFSUM, is calculated at the node in question for each body connected to that node, and,

  3) A weighted average is calculated from these principal values. This is different than the usual method (except for Equivalent Strain) where components are averaged and then the principal value is computed.

  Example Averaged Result Types

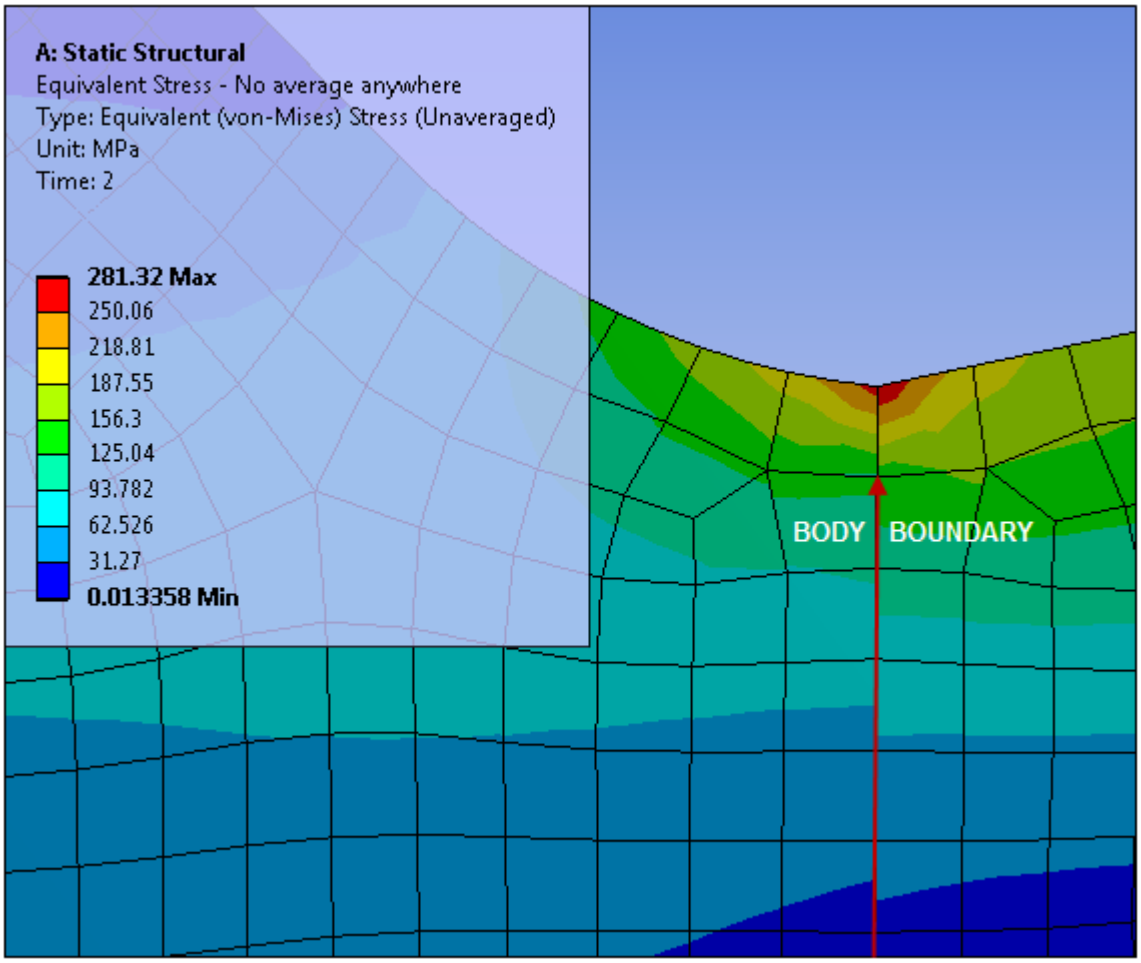
  – Principal Stresses (1, 2, 3)
  – Stress Intensity (INT)
  – Equivalent Stress (EQV)
  – Principal Strains (1, 2, 3)
  – Strain Intensity (INT)
  – Equivalent Strain (EQV)
  – Total Thermal Flux

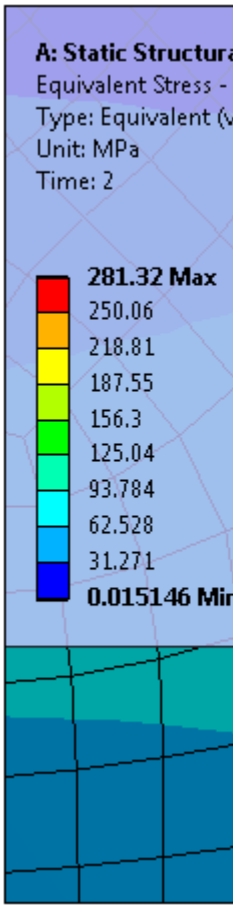
  The following result illustrations illustrate the outcomes between **not** performing an average calculation, performing an average calculation that is not across bodies, and performing an average calculation that is across bodies.

  No Averaging **No Averaging Acr**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

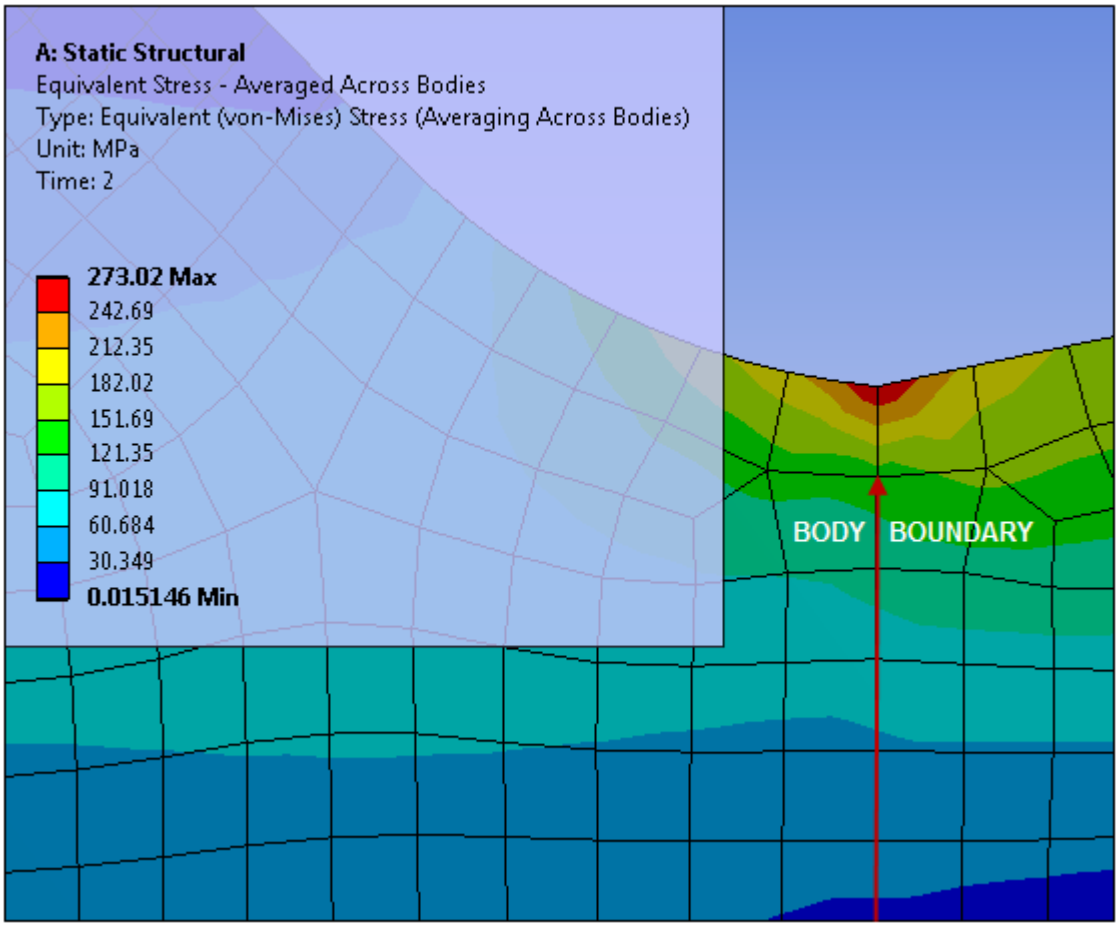
Published: 2023-10-23





Averaging Across Bodies

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- If the associated bodies have different material properties, such as Poisson's ratios, then, because this feature averages quantities like equivalent elastic strain at common nodes, you may see unexpected results at the interfaces. The Poisson Ratio employed to calculate elastic equivalent in one body may be significantly different from the Poisson Ratio employed to calculate elastic equivalent in a different body. Therefore, in this scenario, averaging across bodies at the interface is not recommended.
- If you choose to compare this feature against Mechanical APDL PowerGraphics with **AVRES**,**1**,**FULL** in effect, PowerGraphics employs the effective Poisson's ratio in the **AVPRIN**,**KEY**,**EFFNU** command. The **EFFNU** value may not match the Poisson's ratios in all bodies. PowerGraphics also calculates equivalent strain from the average component strains if **KEY** (in the **AVPRIN** command) is set to **ZERO**. As a result, there may be differences between this feature and PowerGraphics when the **AVRES**,**1**,**FULL** command is employed.

Support Limitations

The following results features are not supported:

- Probe results
- Results in cyclic symmetry analyses
- Results on line bodies

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Body Type Calculations

If the scoping of your result includes both solid and surface bodies, the application will average across the solid body first and then performs a separate averaging across the surface body. These results are averaged separately because surface body elements element includes bottom and top results for each node on the top/bottom surface. The solid body element has only one result at a node (as there is no bottom/top).

Averaging Element Nodal Results

For *most* element nodal results (like stresses and strains), the Mechanical solvers write only the un-averaged corner node values to the result file. No element nodal results are written for mid-side nodes. This difference is of critical importance.

To derive the value at a mid-side node, the Mechanical post processor must employ the values at the corner nodes. There are three distinct techniques for this process:

1. For line elements, such as unexpanded beam and pipe stress, Mechanical calculates the average of the **unaveraged** values of those corner nodes which share an edge with the mid-side node - corner nodes, node I and node J. See **Case 1** below.

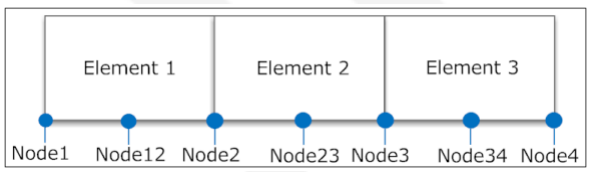
2. For solid elements (i.e. shell, hexahedron, tetrahedron, wedge, and pyramid elements), the technique calculates the mid-side node results from the **averaged** corner nodes. See **Case 2** below.

3. For contact elements, the technique calculates the mid-side node results by averaging the **un-averaged** values of adjacent corner nodes. Note: A result max may be at a mid-side node. See **Case 3** below.

Corner Node Calculation Case 1

Consider the following example consisting solely of high order line elements, where:

- Element 1 contains nodes 1, 2, and 12.
- Element 2 contains nodes 2, 3, and 23.
- Element 3 contains nodes 3, 4, and 34.
- Nodes 12, 23, and 34 are mid-side nodes.



If:

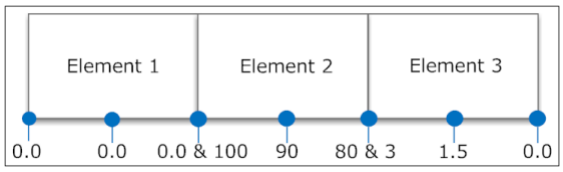
- The element nodal solution for element 1 is 0.0 and 0.0 for nodes 1 and 2.
- The element nodal solution for element 2 is 100 and 80 for nodes 2 and 3.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The element nodal solution for element 3 is 3 and 0.0 for nodes 3 and 4.

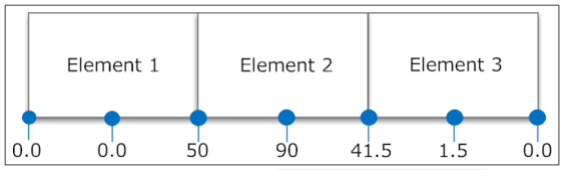
The unaveraged solution is then:

- The value at node 12 is the average of the values at the associated element's end points, namely 0.0.
- The value at node 23 is the average of the values at the associated element's end points, namely 90.
- The value at node 34 is the average of the values at the associated element's end points, namely 1.5.



For the averaged solution of nodes 2 and 3:

- The value at node 2 is 50.
- The value at node 3 is 41.5.



Note that the value at the mid-side node 23 (90) exceeds the values at the end points.

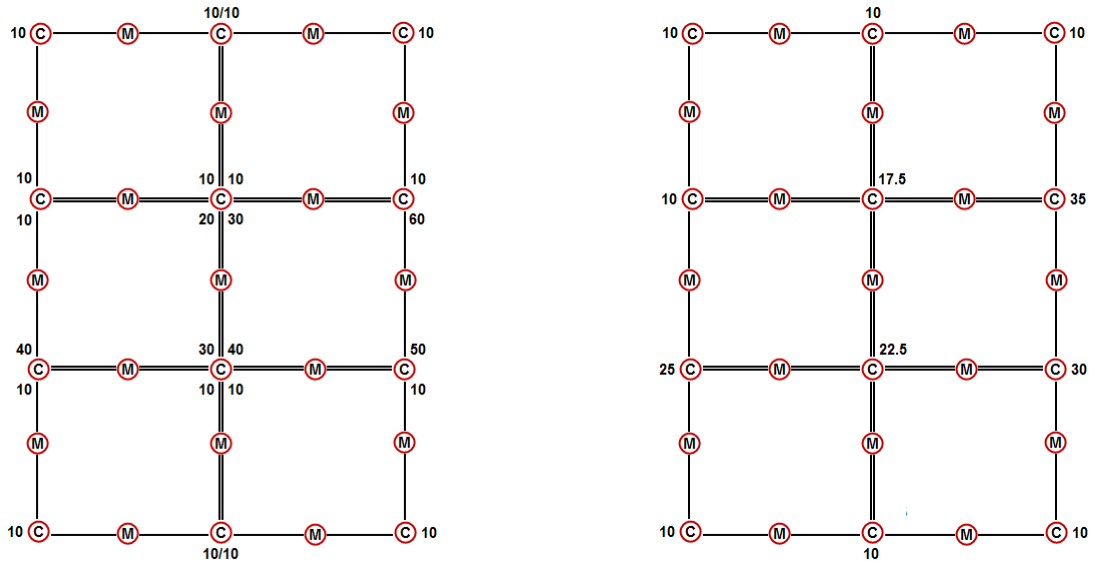
Corner Node Calculation Case 2

Result averaging for solids: mid-side node results are the average of the averaged corner node values.

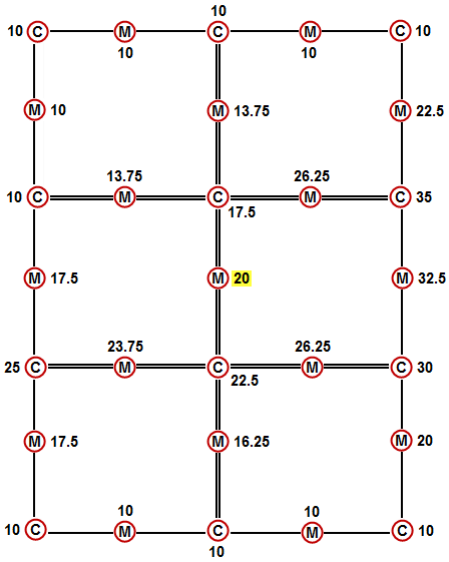
Raw Data	Averaged Corner Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Mid-Side Nodes Calculated from Averaged Corner Nodes

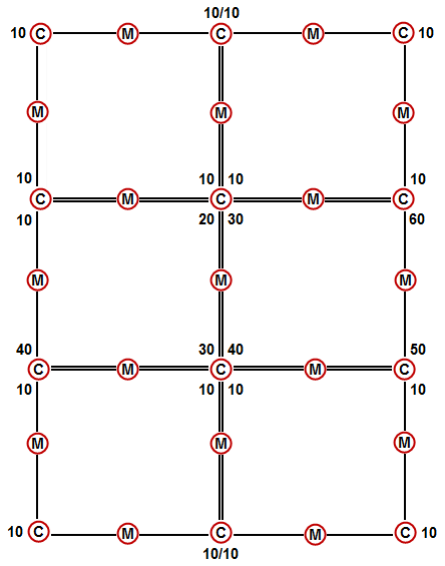


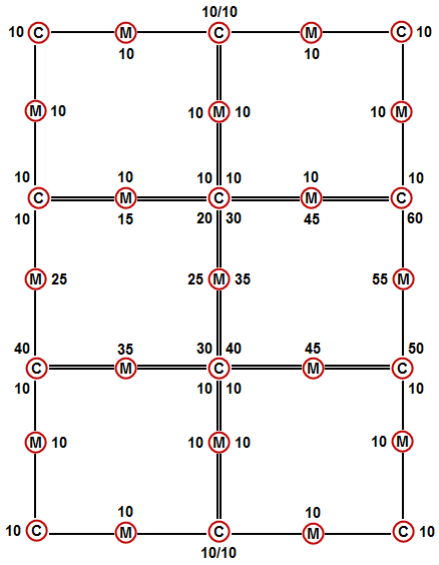
Corner Node Calculation Case 3

Result averaging for contacts: calculate the final mid-side node results by averaging intermediate mid-side values.

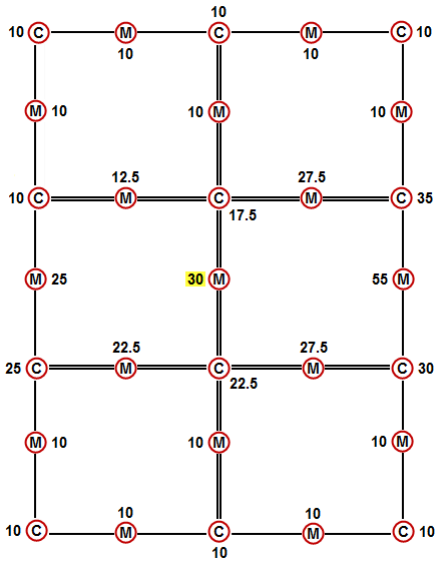
Raw Data **Averaged Corner Nodes**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23





Mid-Side Nodes Calculated from Averaged Corner Nodes



Note:

Result maximum in this case may be at a mid-side node.

Working with Multiple Result Sets

Many analyses produce multiple result sets over a domain, such as time in a static or transient structural analysis, or frequency/phase in a harmonic analysis, or cyclic phase in a cyclic modal analysis. The **By** property options enable you to specify how you wish to review result contours from multiple result sets. The options available to you depend on the type of analysis being performed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Structural Analysis

- **Time** (default): This option displays the results for a particular Time in the solution history. By default, this is the end time. For a solution that includes steps and substeps, you can use the **Display Time** property to specify a desired time value.

  If you specify a time that lies between two times that exist in the result file, the application interpolates the results, except in these cases:

  – For non-fracture tool results in an additive manufacturing analysis (with birth/death), the set with time closest to the specified time is used.

  – For non-fracture tool results in a changing mesh environment, if the mesh in the set before the specified time differs from the mesh in the set after the specified time, the set with time closest to the specified time is used.

  – For fracture tool results when there are no mesh changes between the earlier and later times **and** there is no additive manufacturing, the set before the specified time is used.

  – For fracture tool results when there are **either** mesh changes between the earlier and later times **or** there is additive manufacturing, the set with time closest to the specified time is used.

- **Result Set**: This option displays the contour result for a given **Result Set** contained in the result file. By default, this value is the last set, which is to say, if only one set is available, then that is the specified **Result Set**. For a solution that includes load steps and substeps, you specify the desired chronological set number using the Set Number property. You must enter a valid set number.

- **Maximum Over Time** or **Time of Maximum**: Each node/element/sample point is swept through the result sets to find its maximum result. Either the result itself is reported (sometimes referred to as a "peak hold") or the time at which the peak occurred is reported.

- **Minimum Over Time** or **Time of Minimum**: Each node/element/sample point is swept through the result sets to find its minimum result. Either the result itself is reported or the time at which the minimum occurred is reported.

Harmonic Analysis

- **Frequency** (default): This option displays the contour result for a Frequency in the solution history. By default, this is the final frequency value. For a solution that includes steps and substeps, you can use the **Frequency** property to specify a desired value. Like Time, if you enter a frequency that is not equal to an existing value in the result file, the application interpolates the result.

  Note:

  The application does not attempt to interpolate result values when (1) your Harmonic Response analysis is using the **Mode Superposition** method or (2) the **Multiple Steps** property of the Analysis Settings is set to **Yes**. For

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

these cases, the application selects the closest frequency value available in the result history of the file.

- **Set**: results display for specified sets.
- **Maximum Over Frequency** or **Frequency of Maximum**: With these options chosen, the phase specified in the **Sweeping Phase** property is held constant and each node/element/sample point is swept through frequency range to find its maximum result.
- **Maximum Over Phase** or **Phase of Maximum**: With these options chosen, frequency is held constant and each node/element/sample point is swept through a phase period of 0° to 360° at specified increments to find its maximum result. You can control the increment using the **Phase Increment** entry.
- **Maximum Over Cyclic Phase** or **Cyclic Phase of Maximum**: Each node/element/sample point is swept through a phase angle of 0° to 360° in 10 degree increments find its maximum result.

Note:

Maximum Over Cyclic Phase and **Cyclic Phase of Maximum** require that you specify the **Harmonic Index** property as greater than zero.

Harmonic Analysis and Calculating Derived Results

A Harmonic analysis result can be expressed using the following complex notation:

$$x = x_{real} + x_{imag} \cdot i$$

For the Harmonic analysis **By** property options above, the application calculates derived results. Derived results are calculated from component results. The above notation is the basis for each component result whose values are then combined to create a derived result.

In addition, Phase must also be considered. The phase computation is applied on each component before the derived result is calculated. The application derives phase:

- Explicitly as specified by the **Sweeping Phase** property, or:
- Implicitly as defined by the **Phase Increment** property (using the phase period of 0° to 360° at the specified increment).

For example, Equivalent (von-Mises) Stress, σ_{eqv}, is a derived result because it is evaluated from the stress components σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz}, and σ_{zx}, from different directions, respectively. Therefore, it is calculated in the following equation for the general situation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

$$\sigma_{eqv} = \sqrt{\frac{1}{2}\left[\left(\sigma_{xx} - \sigma_{yy}\right)^2 + \left(\sigma_{yy} - \sigma_{zz}\right)^2 + \left(\sigma_{zz} - \sigma_{xx}\right)^2\right] + 3\left(\sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2\right)}$$

Where each tensor component has been computed as show here (θ is the **Sweeping Phase** value):

$$\sigma_{xx} = \sigma_{xx\text{ - real}} * \cos\theta - \sigma_{xx\text{ - imag}} * \sin\theta$$

$$\sigma_{yy} = \sigma_{yy\text{ - real}} * \cos\theta - \sigma_{yy\text{ - imag}} * \sin\theta$$

$$\sigma_{zz} = \sigma_{zz\text{ - real}} * \cos\theta - \sigma_{zz\text{ - imag}} * \sin\theta$$

$$\sigma_{xy} = \sigma_{xy\text{ - real}} * \cos\theta - \sigma_{xy\text{ - imag}} * \sin\theta$$

$$\sigma_{yz} = \sigma_{yz\text{ - real}} * \cos\theta - \sigma_{yz\text{ - imag}} * \sin\theta$$

$$\sigma_{zx} = \sigma_{zx\text{ - real}} * \cos\theta - \sigma_{zx\text{ - imag}} * \sin\theta$$

Modal Analysis (Cyclic Only)

- **Cyclic Phase** (default)
- **Maximum Over Cyclic Phase** or **Cyclic Phase of Maximum**. Each node/element/sample point is swept through a phase angle of 0° to 360° in 10 degree increments find its maximum result. This result is applicable during a cyclic modal analysis only.

Displaying Surface Body Results (including Layered Shell Results)

For surface bodies, stress and strain results at the top and bottom faces are displayed simultaneously, by default. (See Understanding Surface Body Shell Offsets (p. 862) for information on identifying the top and bottom faces.) The contours vary linearly through the thickness from the top face to the bottom face. However you can choose to display only the **Top**, **Middle**, or **Bottom** stress/strains in the Details view of the result item. Selecting **Top**, **Middle**, or **Bottom** will display the result at the selected location as a uniform contour through the thickness.

Middle Stresses

- **Normal and Shear results**

  The middle stresses are calculated at the shell mid-surface or at each layer mid-surface if layers are present. The Middle option for Shell gives the actual result values at the mid-surface if the solver was directed to calculate these results. In Mechanical APDL terminology, the solver computes results at mid-surface if KEYOPT(8) for the shell element is set to **2** at the time of element creation. Otherwise, the **Middle** results are computed as the average of the **Top** and **Bottom** results, that is, (**Top** + **Bottom**) / 2. Note that these results are valid only for linear analyses.

- **Equivalent and Principal results**

  These results are derived from the **Normal** and **Shear** results. Hence the **Normal** and **Shear** component results for **Middle** are computed first, and then the **Equivalent** and **Principal** results are derived.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Element Nodal results (like stress/strain), as well as EDIR- and PNUM-type Elemental results, can be plotted on a specific layer or on a specific ply by entering the desired **Layer** number (**Layer** only) or by scoping (p. 1997) the result to a specific Ply using the Sub Scope By property in the Details view of the result object. Elemental results outputting volume or energy are calculated for the entire element, regardless of the requested layer.

If the **Layer** specified does not exist for a particular surface body, the display of the result will be translucent with zero values for minimums and maximums on that body. If you enter **0** for **Layer**, it defaults to the Entire Section.

Note:

- A **Layer** number must be specified to calculate the **Middle** stresses and strains. If you set Layer to 0 (Entire Section) while Shell is Middle, the Shell option will become invalid. Similarly, if you have Layer set to Entire Section and you try to set Shell to Middle, Shell will become invalid.
- If there is a Layered Section in the model, convergence is not supported for results.
- If **Layer** is **Entire Section**, **Top** stresses and strains are for the top surface of the topmost layer and the **Bottom** stresses and strains are for the outer surface of the bottom layer.
- If a Layered Section is present in the model and you enter a number larger than the maximum number of layers that exists in the model, the **Layer** field will become invalid.
- All stress tool results and all fatigue tool results are unsupported if Layered Sections are present in the model.
- Only results from the section top and bottom are available on hyperelastic layered shells. Thus no results will be reported on such bodies if the layer is not set to 0 "Entire Section".
- Imported Plies are not supported for an Explicit Dynamics analysis.

Layered Solid Body Results

Certain solid elements in the Mechanical APDL element library (including SOLID185, SOLID186, and SOLSH190) support stress and strain results on layers. These layered solid elements are created on solid bodies.

At Release 16.0, these elements support the **Layer** property, which specifies the layer to calculate Shell result values. This property specifies a Layer number. In addition, the **Position** property appears only for stress and strain results scoped to surface bodies or to a layer. This property is specified as either **Top/Bottom**, **Top**, **Middle**, or **Bottom**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For the elements, the convention is that the **Bottom** corresponds to nodes I-J-K-L and the **Top** corresponds to nodes M-N-O-P.

Note:

If the model does not include surface bodies or layered sections, the **Layer** and **Position** properties do not display in the Details view and the application displays layer results in **Top/Bottom** format.

For Explicit Dynamics Layer Results

Normal/shear stresses and strains are available in global and solution coordinate systems.

Stress and strain results for individual layers may be selected by using the **Layer** property in the result's Details view. Only a single result is available per layer.

Reviewing Unconverged Results

A nonlinear analysis may fail to converge due to a number of reasons. Some examples may be initially open contact surfaces causing rigid body motion, large load increments, material instabilities, or large deformations that distort the mesh resulting in element shape errors.

In the Mechanical application, you can review this unconverged result as well as any converged results at previous time points. These results are marked in the legend of contour/vector plots as 'Unconverged' indicating that these results must be used only for debugging purposes. Note that a plot of Newton-Raphson residuals (p. 1833) is a very useful tool to identify regions of your structure that led to the convergence difficulty.

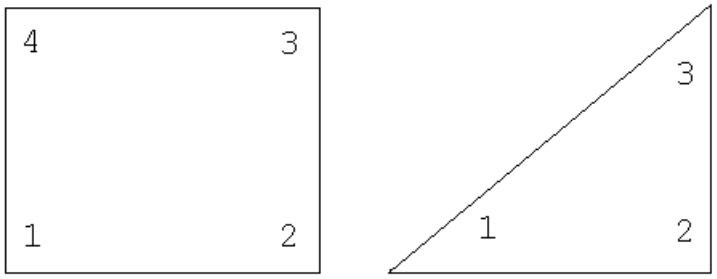
Note:

- Results in **Solution Combination** objects that use partial solutions will not be solved. You can view partial results but cannot use them in further post/solution work.
- Newton-Raphson residuals (p. 1833) is a very useful tool to identify regions of your structure that led to the convergence difficulty.
- The handling of unconverged solutions is the same for both probes and results, with the following exception: reaction probes scoped to a Compression Only boundary condition cannot display results if the solver did not converge.

Handling of Degenerate Elements

The finite element method can create elements that are classified as degenerate. For example, a quad element, with four nodes 1, 2, 3, and 4, may contain duplicate nodes 3 and 4. In this case, node 3 and 4 are located at the same (x, y, z).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The degenerate quad element (above, right) contains three distinct nodes and four distinct integration (Gauss) points. Mechanical APDL's solver calculates element nodal results (like stress and strain and flux, et. al.) at each of the integration points. Hence, element nodal results in the MAPDL result file are stored as though an element is *not* degenerate (even when it is degenerate). For the element (above, right), the file would contain stress and strain and flux listings for four nodes, 1, 2, 3, and 4.

At nodes that share the same (x,y,z) in an element, it is not necessarily true that the element nodal results are equal for each coincident node. Depending upon the analysis, the element nodal results for the element (above, right) at node 3 may not equal the element nodal results at node 4. During the post processing phase, Mechanical drops the values of all but the first duplicate node at an (x,y,z). The element (above, right) would display the stress and strain and flux contours for nodes 1, 2, and 3 (but not 4).

Understanding Result Data Display Issues

When defining a result, it is necessary to make sure that you have made selections that are supported by the application. Normally, the application generates a message indicating that the result type is not available but there are instances when result information is not provided, that is, the application will not display contour information or it will display generically colored contour information.

Note that:

- Result objects must be scoped to geometry that supports the elements and nodes used by the body; failing to do so will cause Mechanical to display result contours in grayscale. For example, stress results do not exist on gasket elements. Therefore, if you scope a stress result to a gasket body, the result displays in grayscale.
- All possible result data is not provided by a result file. If you create an expression in a user-defined result that is not applicable to the analysis, such as **Total Deformation** during in a thermal analysis, then the result display will have no contour colors.
- The solver can create elements such as surface effect as well as interface elements. Results cannot be scoped to these elements. For example, because of software limitations, contour colors for surface effect results will not be shown, even though the surface effect elements cover meshed elements on the body.

Result Scoping

The topics of this section examine the methods to scope your results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Geometry and Mesh
Path Construction Geometry
Surface Construction Geometry
Result File Items
Surface Coatings
Composite Analysis
Imported Condensed Parts

Geometry and Mesh

Go to a section topic:

- Supported Scoping Types (p. 1997)
- Result Object Scoping Requirements (p. 1997)
- Node-Based Scoping Characteristics (p. 1998)
- Element-Based Scoping Result Evaluation (p. 1998)
- Separating Result Data by Entity (p. 1999)
- Geometry Selection Limitation for Node-Based Results (p. 2001)

Supported Scoping Types

All result objects can be scoped to:

- Geometry selections - edges, a single vertex, faces, parts, bodies, or the entire assembly (**All Bodies**).
- Geometry-based Named Selections (p. 1057).
- Node selections (p. 269) of the underlying mesh.
- Node-based Named Selections (p. 275)
- Element or Element Face selections (p. 276) of the underlying mesh.
- Element or Element Face-based Named Selections (p. 276).

Result Object Scoping Requirements

Make sure that your result objects conform to the following:

- If you scope a result to one or more geometric entities (body, face, edge, or vertex) and subsequently suppress all of the bodies associated with the scoped entities, the application, by default, sets the scoping to **All Bodies**. However, the application still stores the suppressed geometric entity set for future reference. Therefore, if any of the bodies associated with the scoped entities are later unsuppressed, the result scoping is set back to the originally defined (unsuppressed) geometric entities. If this behavior is undesirable, it is recommended that you use a Named Selection to define the scoping of the result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Once a solution is computed, the scope of the result object cannot change. You must either add a new result object with the desired scope, or you can right-click that result item, and choose **Clear Generated Data** to change its scope.
- Result scoping has an effect on convergence. Refinement does not happen outside the scope for a given convergence control. Multiple convergence controls are possible, however.

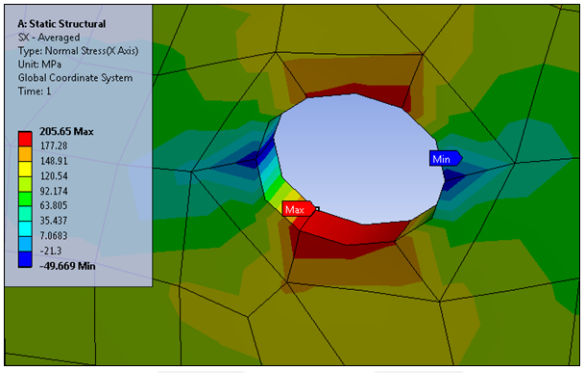
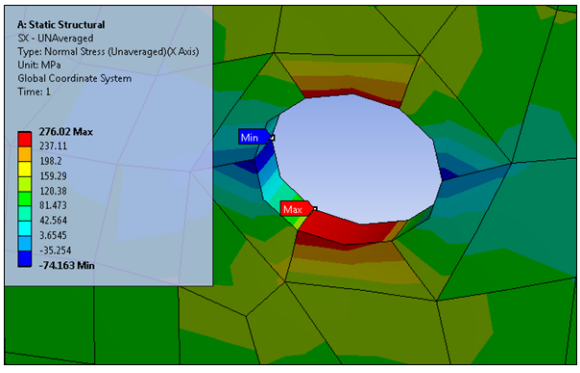
Node-Based Scoping

The following are known characteristics related to node-based scoping:

- If all nodes of an element face are scoped, then Mechanical will draw contour bands on the entire face.
- If some nodes of an element face are *not* scoped, then Mechanical will draw the face as transparent and draw the scoped nodes in contour colors.
- As is the case with other scoping that occurs within a body (such as vertex or edge), any applicable averaging is done considering all of the nodes on a body.

Element-Based Scoping Result Evaluation

Unlike results scoped to geometries or nodes, results scoped to elements evaluate only the scoped elements. No adjacent elements are considered. The example results shown below illustrate this behavior. Refer to the Averaged vs. Unaveraged Contour Results (p. 1981) section of the Help for additional information on this topic. The following results illustrate contour bands for all nodes.

Global Averaged Result	**Global Unaveraged Result**
Max. = 205 and Min. = -50	Max. = 276 and Min. = -74
	

The following results illustrate contour bands for elements only.

Result Scoped to One Element

Max. = 276: Matches the Global Unaveraged Result

(Min. Value = 127)

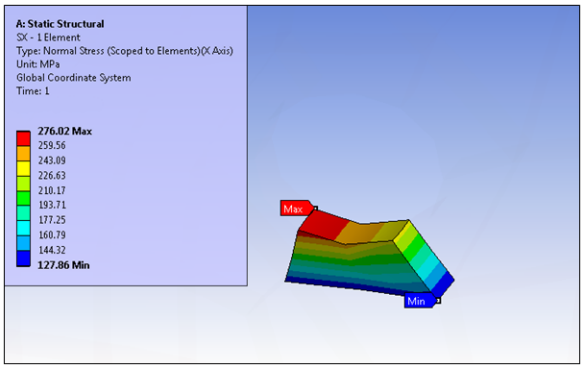
Result Scoped to Three Elements

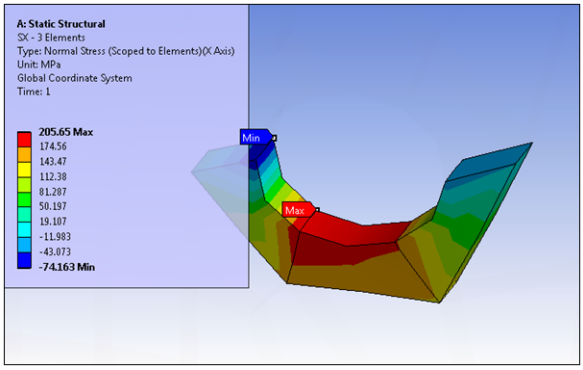
Max. = 205: Matches the Global Averaged Result

Min. = -74: Matches the Global Unaveraged Result

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

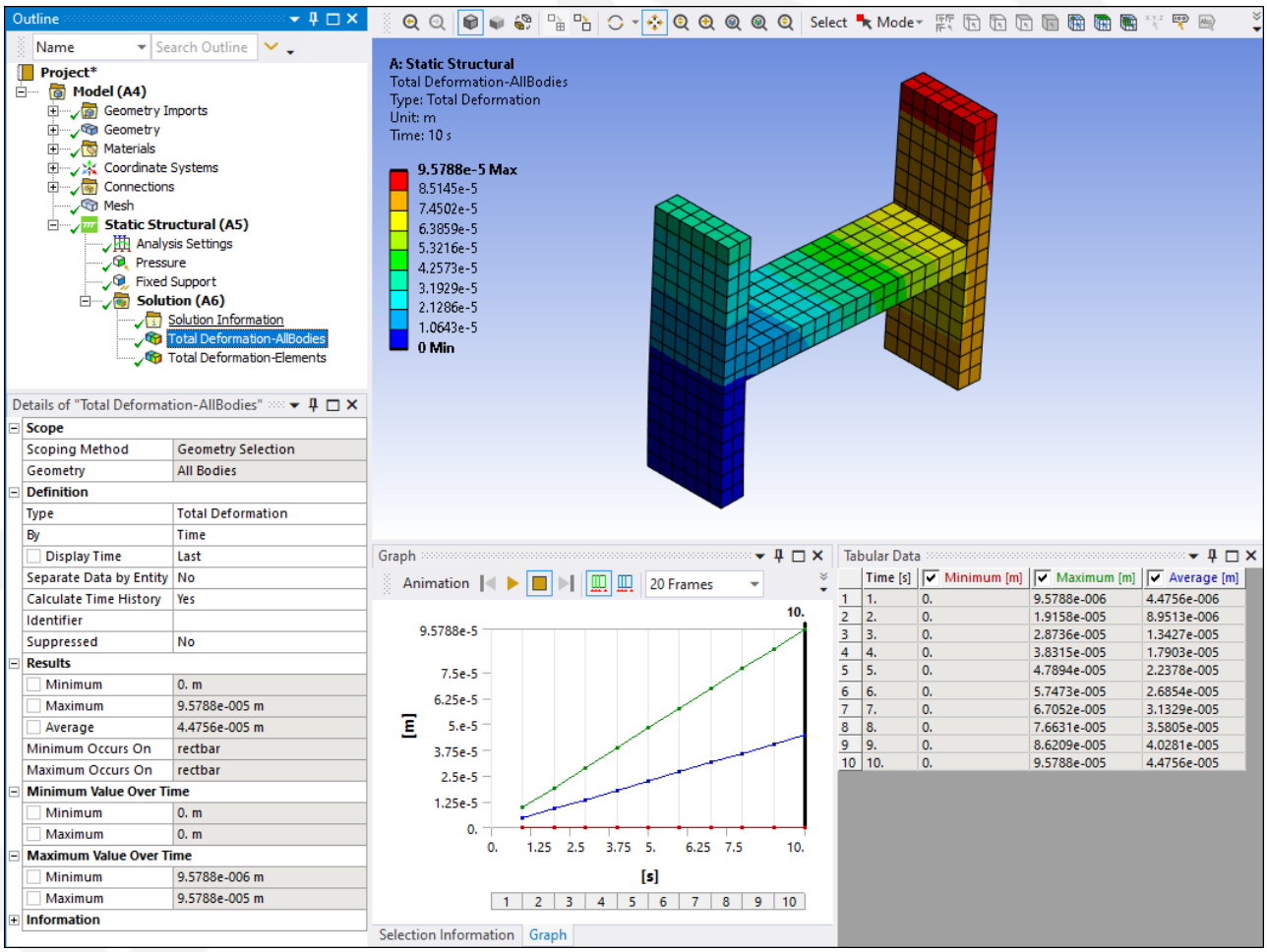




Separating Result Data by Entity

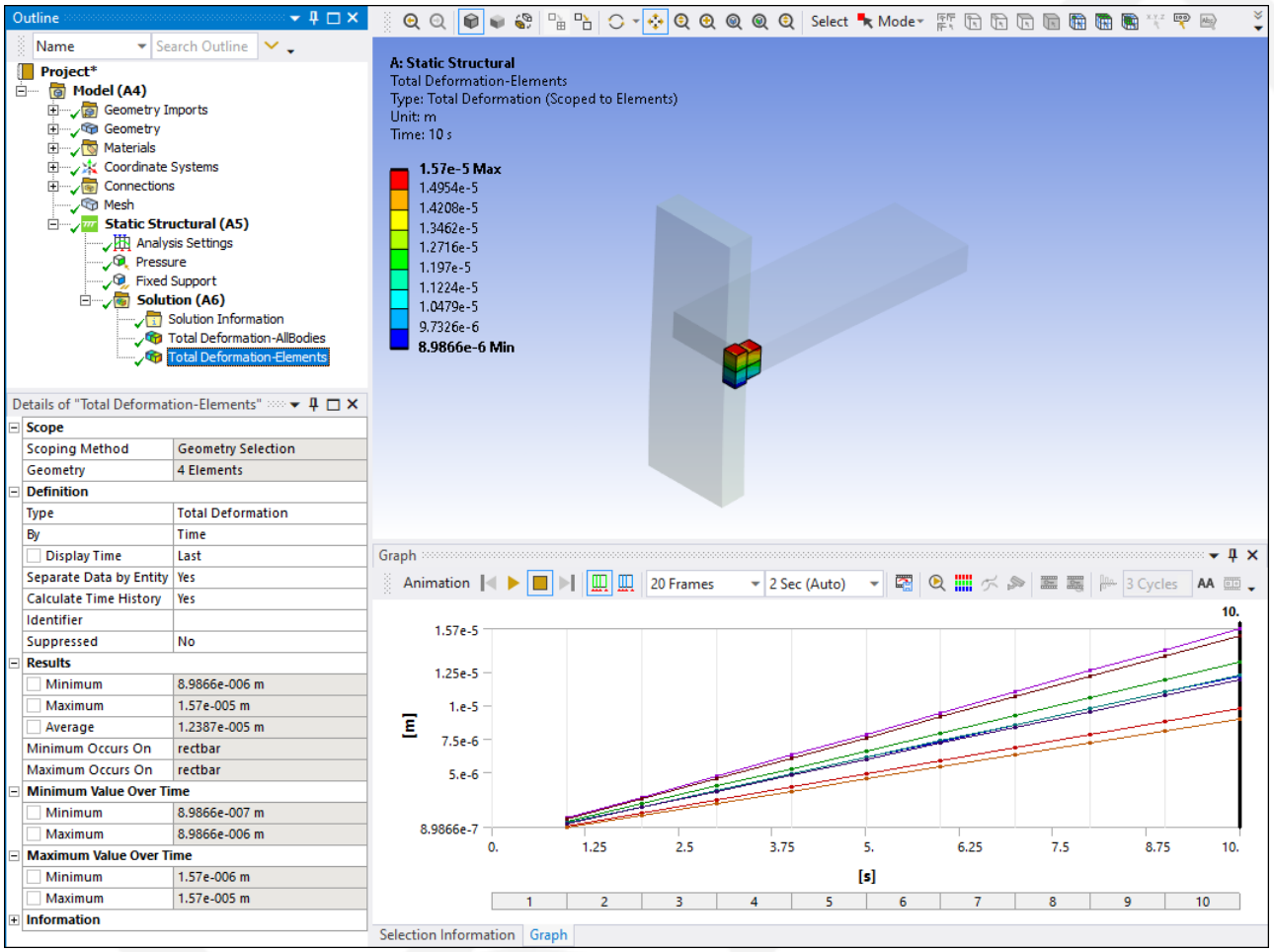
In the **Definition** category of the **Details** pane, result objects include the property: **Separate Data by Entity**. Options include **Yes** and **No** (default). This property enables you to display individual minimum and maximum values for each scoped entity of the result, such as 2 bodies, 3 Faces, 5 elements, etc. Be sure to review the limitations (p. 2000) listed below before using this property.

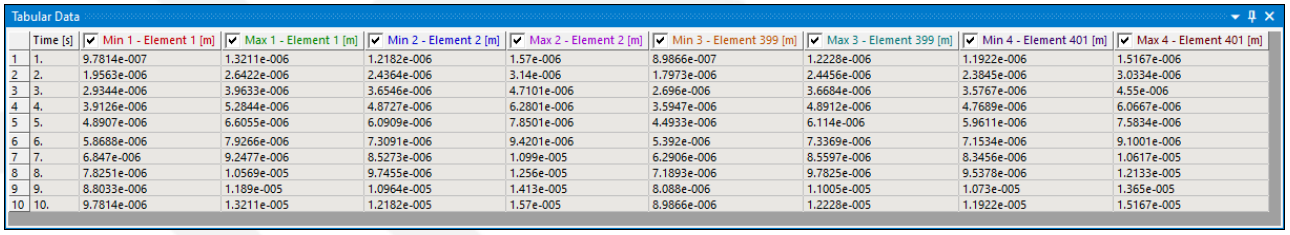
When set to **No**, result data displays normally, and the **Tabular Data** window displays one column for the global **Minimum** and one column for the global **Maximum** value for each time step.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

When you set this property to **Yes**, the application calculates and displays local **Minimum** and **Maximum** values for each scoped entities and displays each separately in the **Tabular Data** window. For the scoped entities, each column header in the window displays either a Node ID, Element ID, or the title "Reference" for geometric entities. Note that the global minimum and maximum values are not displayed when the property is set to **Yes**.





Tabular Data

	Time [s]	Min 1 - Element 1 [m]	Max 1 - Element 1 [m]	Min 2 - Element 2 [m]	Max 2 - Element 2 [m]	Min 3 - Element 399 [m]	Max 3 - Element 399 [m]	Min 4 - Element 401 [m]	Max 4 - Element 401 [m]
1	1.	9.7814e-007	1.3211e-006	1.2182e-006	1.57e-006	8.9866e-007	1.2228e-006	1.1922e-006	1.5167e-006
2	2.	1.9563e-006	2.6422e-006	2.4364e-006	3.14e-006	1.7973e-006	2.4456e-006	2.3845e-006	3.0334e-006
3	3.	2.9344e-006	3.9633e-006	3.6546e-006	4.7101e-006	2.696e-006	3.6684e-006	3.5767e-006	4.55e-006
4	4.	3.9126e-006	5.2844e-006	4.8727e-006	6.2801e-006	3.5947e-006	4.8912e-006	4.7689e-006	6.0667e-006
5	5.	4.8907e-006	6.6055e-006	6.0909e-006	7.8501e-006	4.4933e-006	6.114e-006	5.9611e-006	7.5834e-006
6	6.	5.8688e-006	7.9266e-006	7.3091e-006	9.4201e-006	5.392e-006	7.3369e-006	7.1534e-006	9.1001e-006
7	7.	6.847e-006	9.2477e-006	8.5273e-006	1.099e-005	6.2906e-006	8.5597e-006	8.3456e-006	1.0617e-005
8	8.	7.8251e-006	1.0569e-005	9.7455e-006	1.256e-005	7.1893e-006	9.7825e-006	9.5378e-006	1.2133e-005
9	9.	8.8033e-006	1.189e-005	1.0964e-005	1.413e-005	8.088e-006	1.1005e-005	1.073e-005	1.365e-005
10	10.	9.7814e-006	1.3211e-005	1.2182e-005	1.57e-005	8.9866e-006	1.2228e-005	1.1922e-005	1.5167e-005

Limitations

Note that the **Separating Result Data by Entity** property does not support:

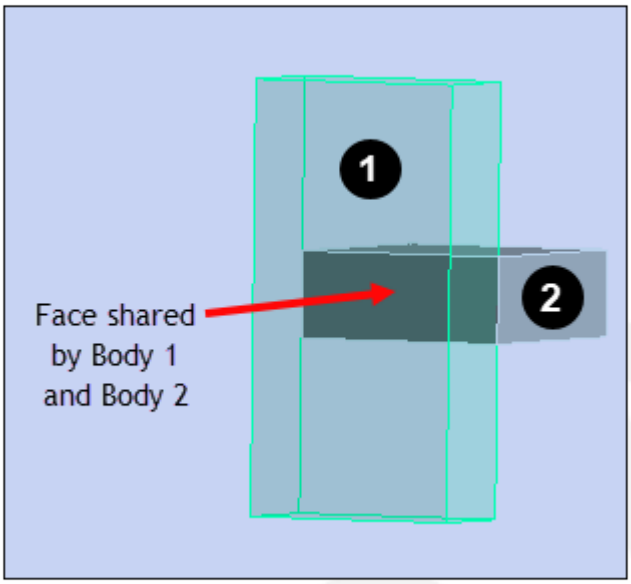
- **Vector Principal Stress** and **Vector Principal Strain** results.
- Fatigue Results (**Fatigue Tool**), Contact Results (**Contact Tool**).
- Solution Combination results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

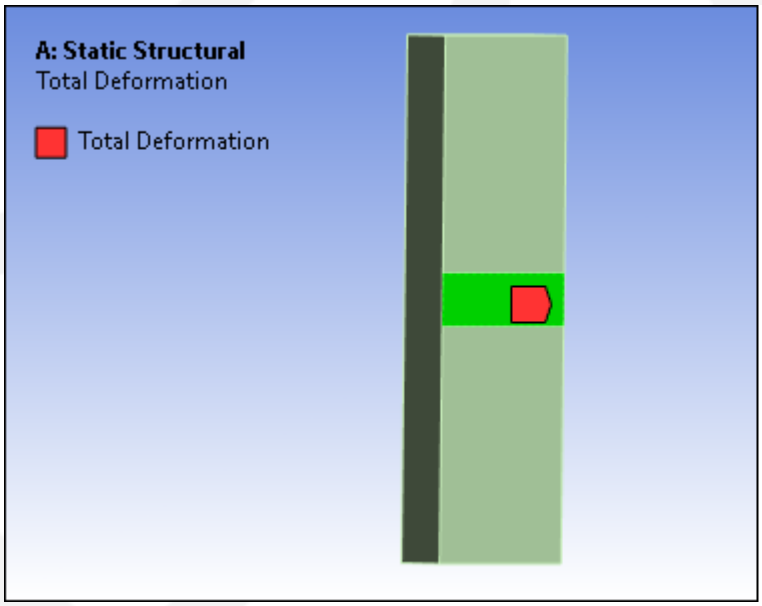
- Results scoped to a Construction Geometry **Path** or **Surface**.
- Results with the **Display Option** property set to **Ply Maximum**.
- Coordinate System Results: **Nodal Triads**, **Nodal Euler XY/YZ/XZ Angle**, **Elemental Triads**, **Elemental Euler XY/YZ/XZ Angle**.

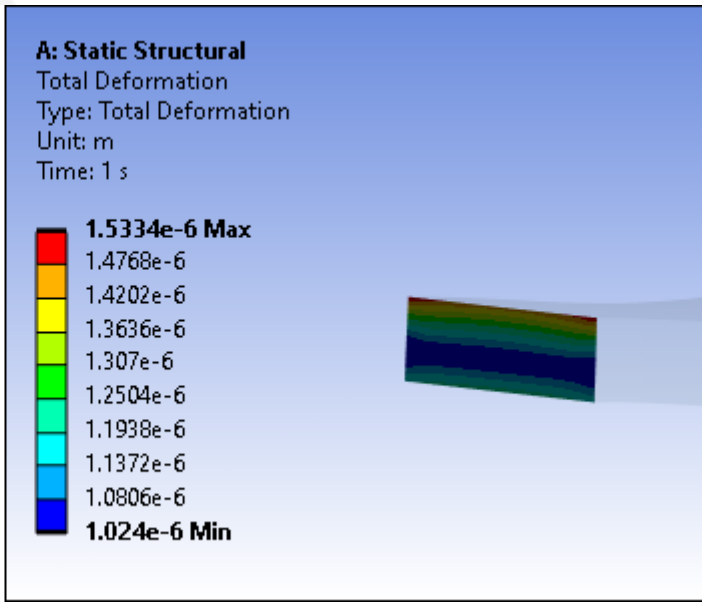
Geometry Selection Limitation for Node-Based Results

Illustrated below are two bodies that share a singular face but that are not in contact.



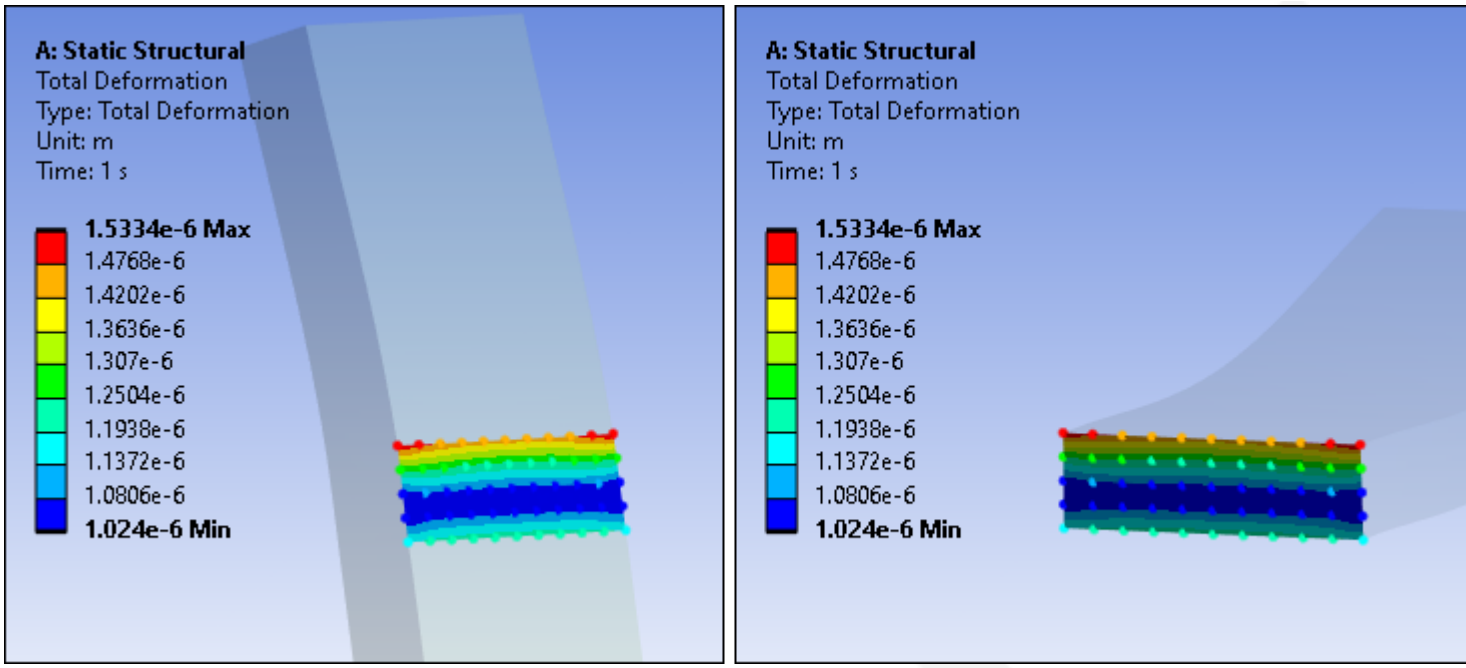
When you scope a node-based result type, such as deformation, to either of the faces, contour results are displayed on only one of the faces. As shown, this face may not be the one you selected.





To display contour results on both bodies with a shared face for a node-based result type (only), create a Named Selection for the shared face, and scope to the Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Tip:

To display contour results on both bodies of a shared face for *any* result type, clear the result, create a Named Selection on one of the faces, and solve the simulation. Open the Worksheet (p. 1961) from the **Solution** object and select **Solver Component Names**. On this page of the Worksheet, you will see a node-based component for the Named Selection. You can now scope your node-based result to this component to produce the desired results for both faces of the associated bodies.

Path Construction Geometry

When you defined a path (p. 883) in your analysis, you may wish to scope results directly on this defined path. In order to do so, you can set the **Scoping Method** of the result object to **Path** and then choose the name of the particular path that you defined. This enables you to produce Path Results (p. 1940) as a result output.

Surface Construction Geometry

When you defined a surface (p. 890) in your analysis, you may wish to scope results directly on this defined surface. In order to do so, you can set the **Scoping Method** of the result object to **Surface** and then choose the name of the particular surface that you defined. This enables you to produce Surface Results (p. 1959) as a result output.

Result File Items

The result object **Scoping Method** option, **Result File Item**, enables you to scope a result to the following Mechanical APDL Solver generated data items generated in the result (.rst) file, once you have performed a solution:

- **Material IDs**: The material ID assigned to the element by Mechanical APDL.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Element Name IDs**: Mechanical APDL element name string such as BEAM188, SOLID185, and SURF154. In addition, for this option, you can enter an element group label, such as "beam", to include all beam element types. Supported group labels include:

  - COMBIN
  - MASS
  - LINK
  - PLANE
  - SOLID
  - SHELL
  - SURF
  - TARGE
  - CONTAC
  - BEAM
  - PIPE
  - ELBOW
  - FOLLW
  - FLUID
  - PRETS
  - MPC
  - INTER
  - SOLSH

  See the Element Library section of the *Mechanical APDL Element Reference* for a complete listing of all available elements.

- **Element Type IDs**: The element type ID assigned to the element by Mechanical APDL.

- **Component Name**: Component names are created by the Mechanical APDL Solver. Components are node- or element-based groupings, node- and element-based Name Selections that were written to the result file from Mechanical, or components that the solver needed to generate in order to properly solve the analysis. Solver generated components typically contain an underscore (i.e. "_") at the beginning of the Component Name.

- **Element IDs** and **Node IDs**: During the mesh process, the application assigns each element and node of the model an ID. The application sends the ID values to the solver during the solution process. Once complete, you can scope one or more element or node IDs to a result. In addition, the solver may generate new elements, not included in the original mesh, in order to process loads, contact conditions, or support conditions. The application also assigns an ID to these elements as well as the elements corresponding nodes. You can also use these system generated element and node IDs for further post processing.

This scoping option is unique in that it obtains data from the result file following the solution. This section examines the following topics for this feature. Select a link to jump to that topic.

- Background (p. 2003)
- Application (p. 2005)
- Examples (p. 2006)

Background

During the solution process, Mechanical identifies nodes and elements contained in the mesh and sends this data to the solver for analysis processing. An element is identified by an element type number, an element name and a material number. A typical element name, such as SOLID285, consisting of a group label (SOLID) and a unique, identifying number (285).

Additional elements, nodes, and groups of nodes or elements, not included in the original mesh, may also be created to process loads, contact conditions, or support conditions, that are required to

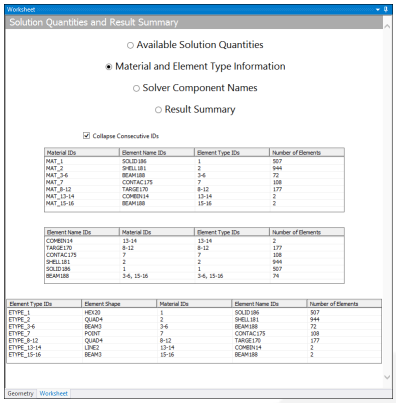
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

properly solve the analysis. Once the solution is complete, the data is written to the result file, and the element data, as well as material data, becomes available to you.

In addition, during the solution process, node and element Name Selections are also written to the result file and may be selected for scoping. The Mechanical solve process may also create additional components for use in solving the analysis. The components typically contain an underscore (i.e. "_") at the beginning of the Component Name.

The application presents this information in the **Solution Summary Worksheet** (p. 1961), using the **Material and Element Type Information** and **Solver Component Names** options. As illustrated below, the Worksheet option **Material and Element Type Information** lists **Material IDs**, **Element Name IDs**, and **Element Type IDs** generated during the solution as well as other appropriate information such as Element Shape. The **Solver Component Names** option lists the solver generated **Component Name** as well as the **Component Type** (nodes or elements) and the **Number of Entities**. The Component Name always includes an underscore prefix.

Material and Element Type Information



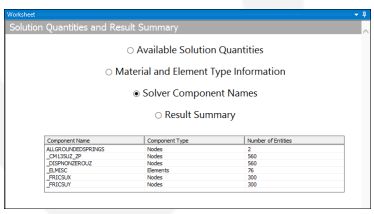
Note:

The **Material and Element Type Information** option does not display elements for a Material ID value of 0.

Important:

If your analysis includes a Condensed Part and a result is using the **Result File Item** option for the **Scoping Method**, **Material IDs** and **Element Name IDs** are not available for the **Material and Element Type Information** selection on the **Solution Quantities and Result Summary** page.

Solver Component Names

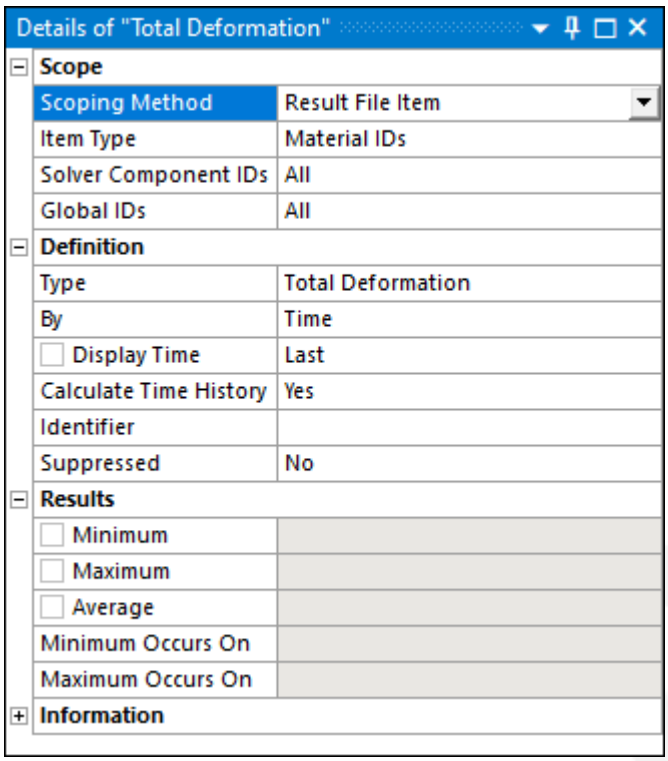


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Application

Once you solve your analysis, you can select the **Result File Item** option for the **Scoping Method** of the desired result object in the Details view, as illustrated below for a total Deformation result.



When you specify the **Scoping Method** as **Result File Item**, the following additional properties display in the Details view:

- **Item Type**: The options for this property include **Materials IDs** (default), **Element Name IDs**, **Element Type IDs**, and **Component Name**.
- **Solver Component IDs**: Based on the option selected in the **Item Type** property, you enter the appropriate ID or **Component Name** in this property as listed in the **Worksheet**. The default value is **All**. You can enter a number or a number range (that is, 1, 2, 3, or 2-5). For **Element Name IDs**, you enter a element name, such as BEAM188, or you can simply enter "beam" to include all beam element types.
- **Global IDs**: This property is used to scope results to bodies that have the **Model Type** property set to **Reinforcement**. This property is not applicable to bodies with any other **Model Type** setting. The application assigns a global ID to each body contained under the **Geometry** object, in the order that they are listed. That is, the first body in the tree has a global ID of 1 and so on in descending order for each object. As a result, you must determine the Global ID of your reinforcement bodies based on their position in the tree. The default setting for the **Global IDs** property is **All**. You can enter an individual ID, or a range of IDs based on the number of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

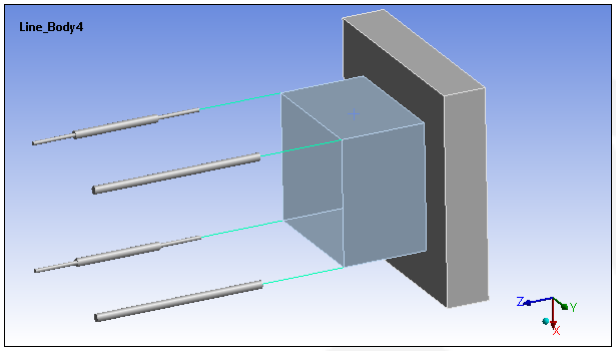
reinforcement bodies. See the Reinforcement Specification Using Mesh-Independent Method (p. 775) for additional information about performing reinforcement analyses.

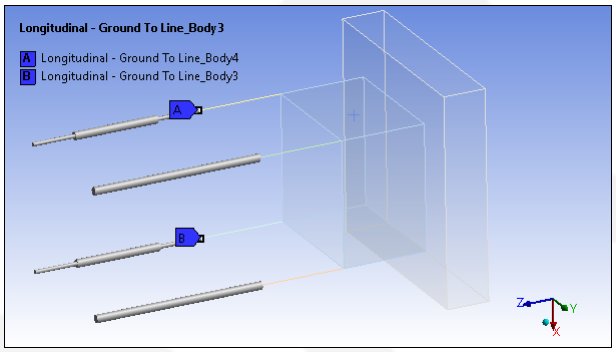
Note:

- The assignment of a Global ID is not affected (changed) if you Group (p. 208) body objects.
- You may encounter a rare case when you are scoping a result to a solver component for a body that contains multiple materials. In this instance, set the **Mesh Source** property (**Solution** object) to the option **Result File**.

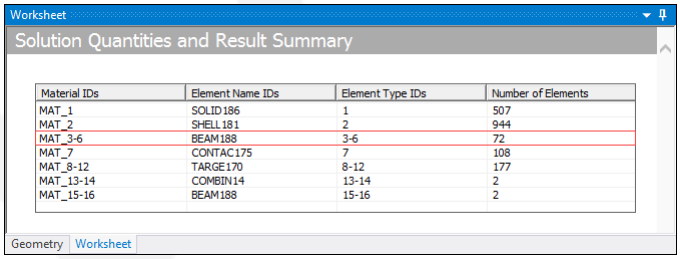
Examples

Here is an example model. It includes four line bodies as well as two user-created beam connections.





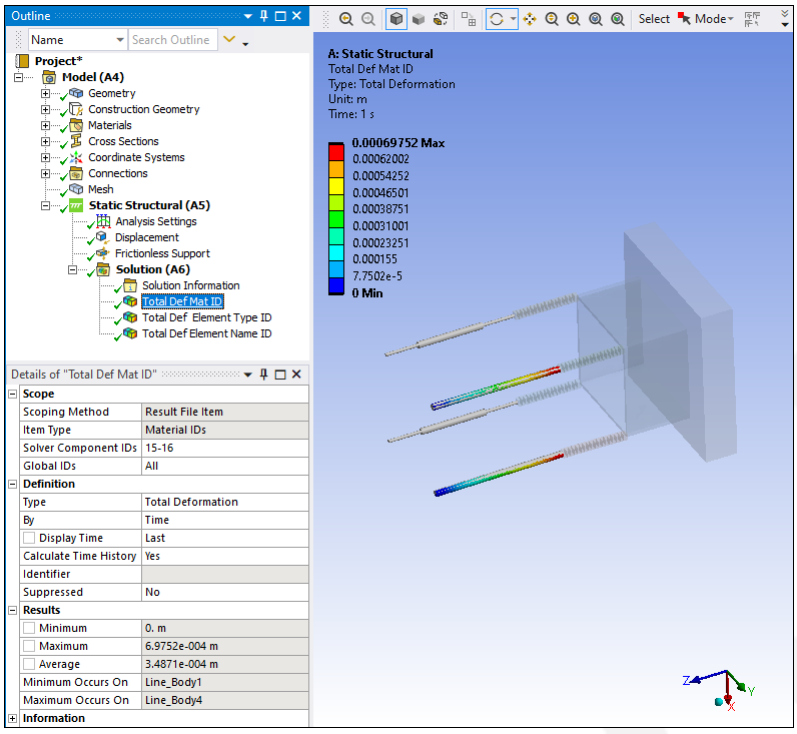
Examining the **Worksheet** following a solution process for the material and element data, we have the following. We are going to further examine the beam connections that correspond with Materials 15 and 16 (Element Type ID as well).

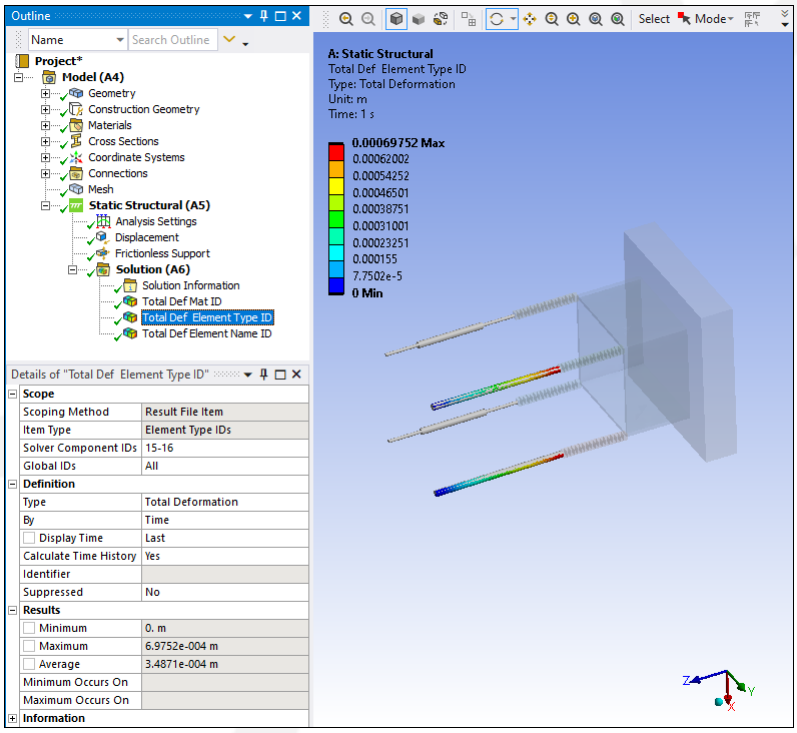


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

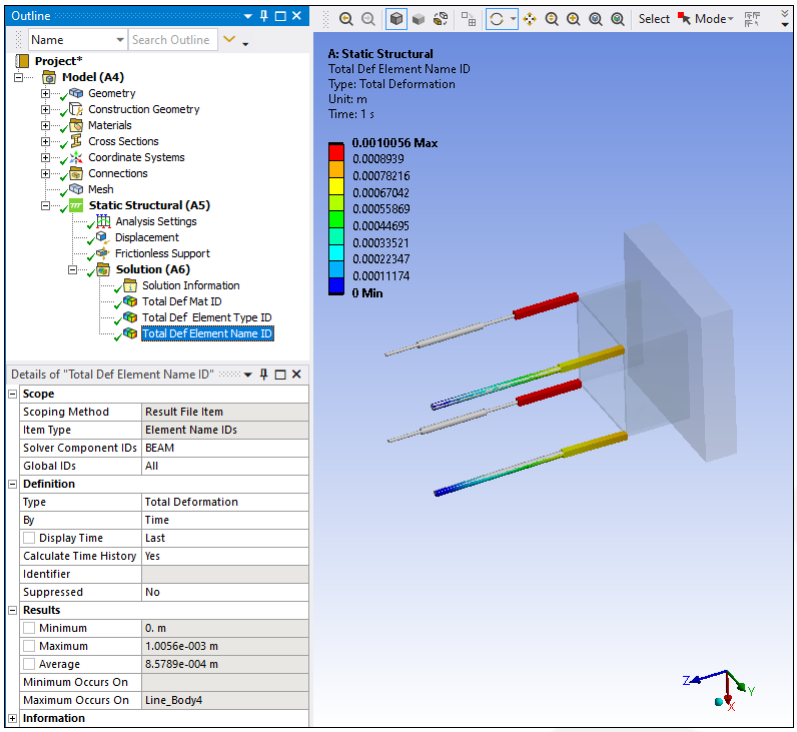
The following sequence of Total Deformation results illustrate the Result File Type scoping options, Material ID, Element Type ID, and Element Name ID. Note that the Material ID and Element Type ID present the same result data using different scoping options.





For this example, we generalized the Element Name and used "Beam." This generates results for the additional (four) line bodies of the model as well as the beam connections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Surface Coatings

You can apply a **Surface Coating** (p. 903) to your model using the option from the **Modify** group on the Geometry Context Tab (p. 56). You can apply one or more of these coatings, or layers, to the faces of your model. This feature places shell elements of a specified material and thickness on the selected face or faces of your model.

Once properly defined, you can use these surface coating definitions as a **Scoping Method** for results. The **Surface Coating** scoping option is supported by most result types, including User Defined (p. 2192) results. During the solution process, the Mechanical APDL solver specifies the elements as either SHELL181 (low order) or SHELL281 (high order).

Important:

To retrieve normal stress component **Sz**, where z is the shell's normal direction, you must make sure of the following:

1. KEYOPT(10) = 1 and,
2. Pressure loads are applied directly to the shell structure.

If these settings are not specified, the application reports the **Sz** results as zero in the element coordinate system. Furthermore, applying loads such as displacements, forces, or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

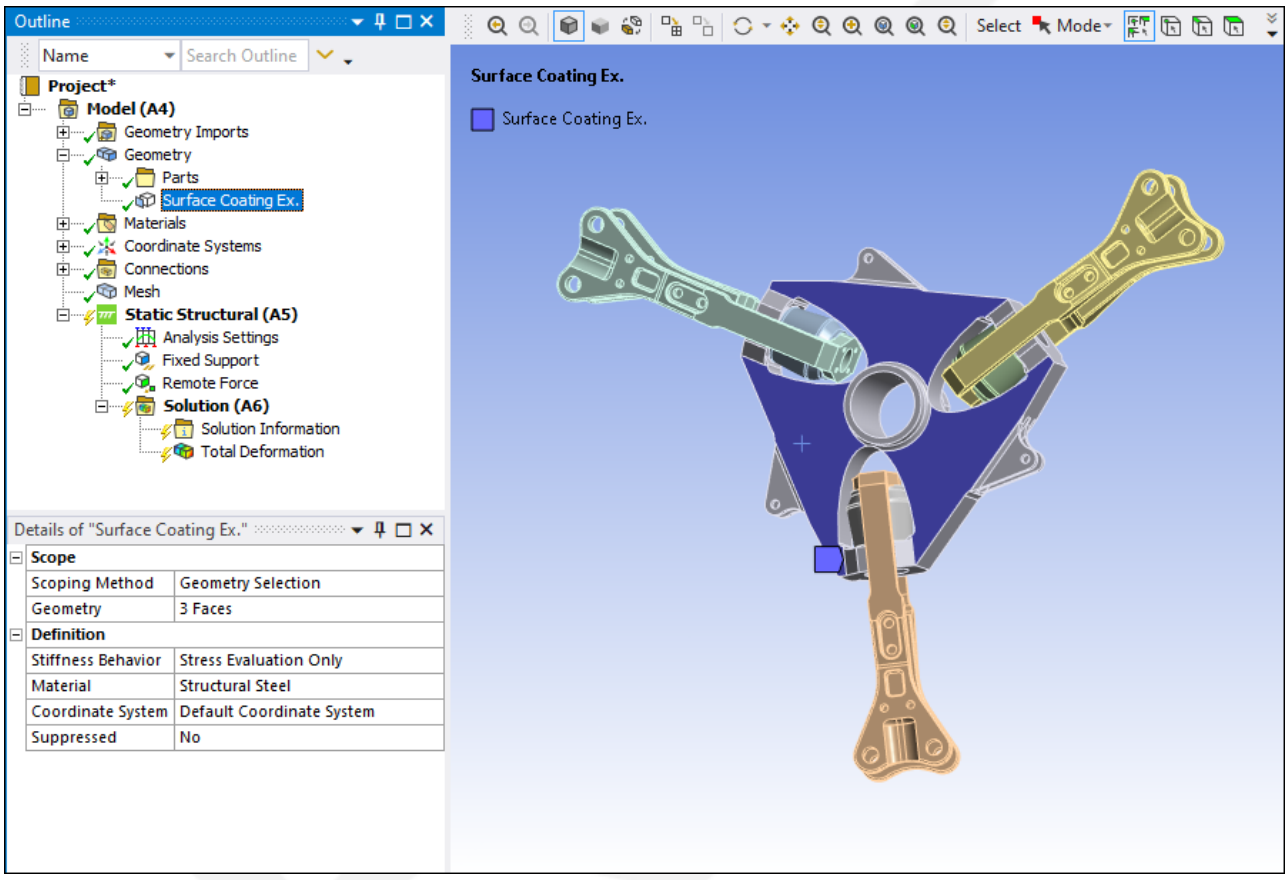
Published: 2023-10-23

pressures using the **Surface Effect** option of the **Applied By** property, also does not retrieve **Sz** results. See the SHELL181 and SHELL281 *Element Reference* sections for more information.

Important:

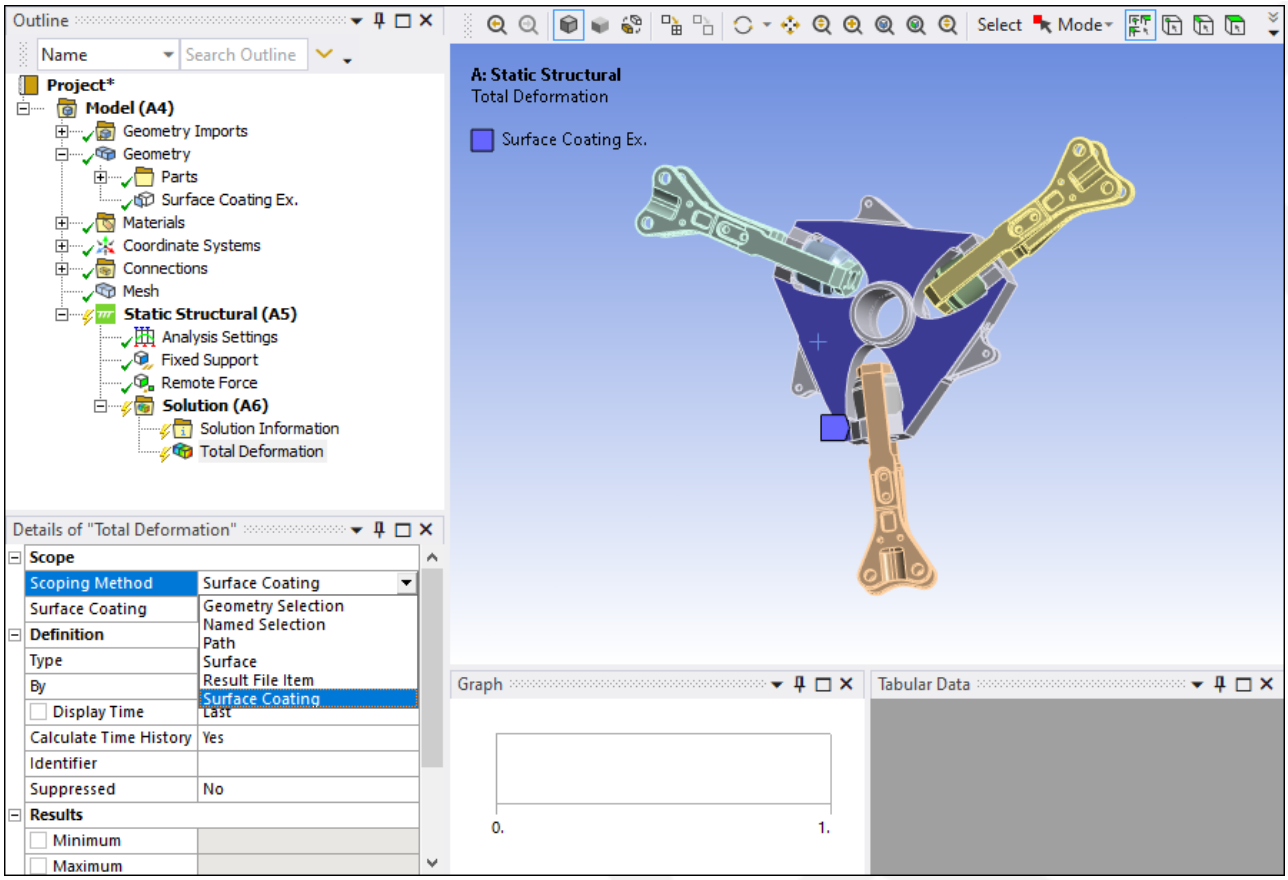
During a Cyclic Symmetry analysis, the application does not expand Surface Coating results.

A **Surface Coating** is defined in the following example.



Surface Coating is an available option for the **Scoping Method** property.

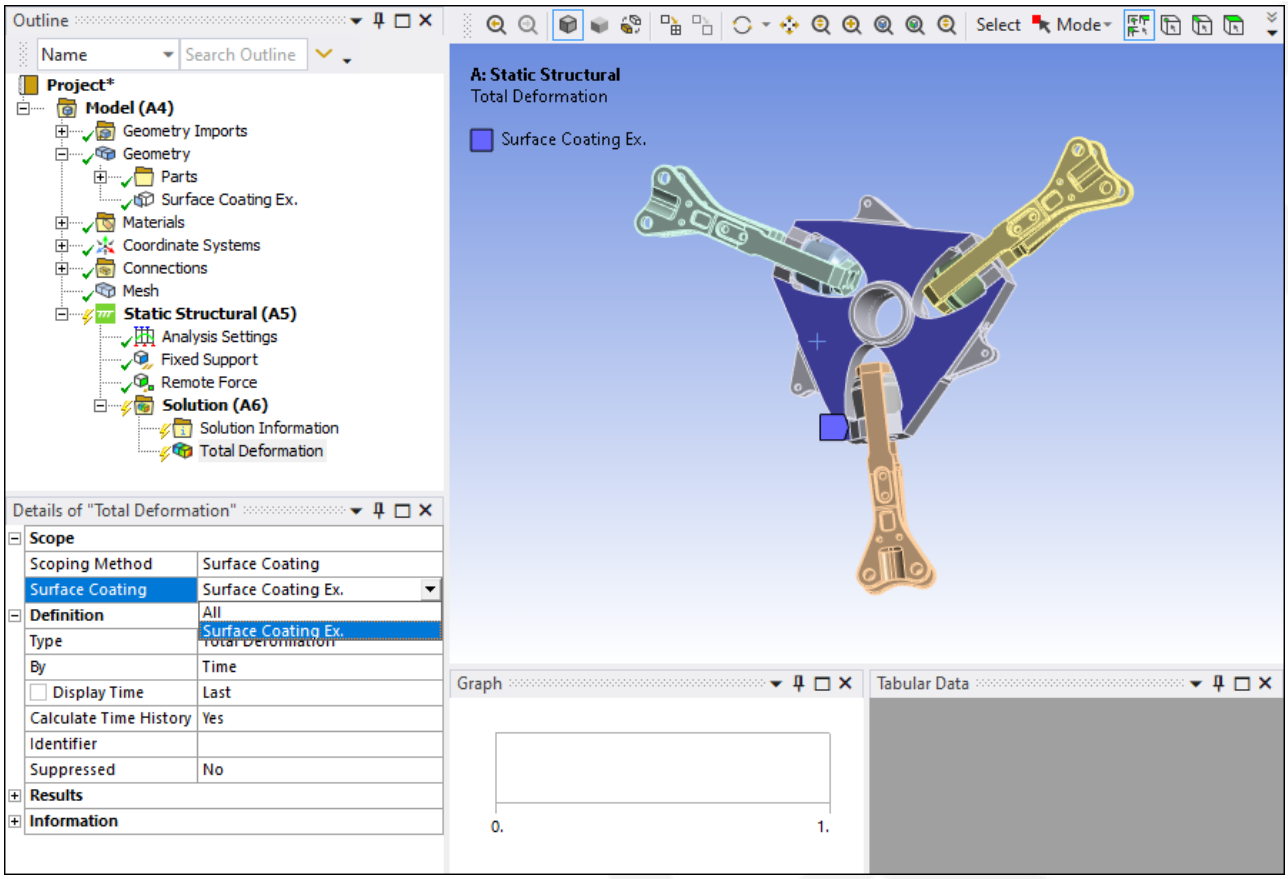
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



When the **Scoping Method** is set to **Surface Coating**, an associated **Surface Coating** property displays and is used to select the coating defined under the **Geometry** object (**Surface Coating Ex.**). If multiple coatings were specified, they would display in the drop-down list of this **Surface Coating** property. The default option for this property is **All**.

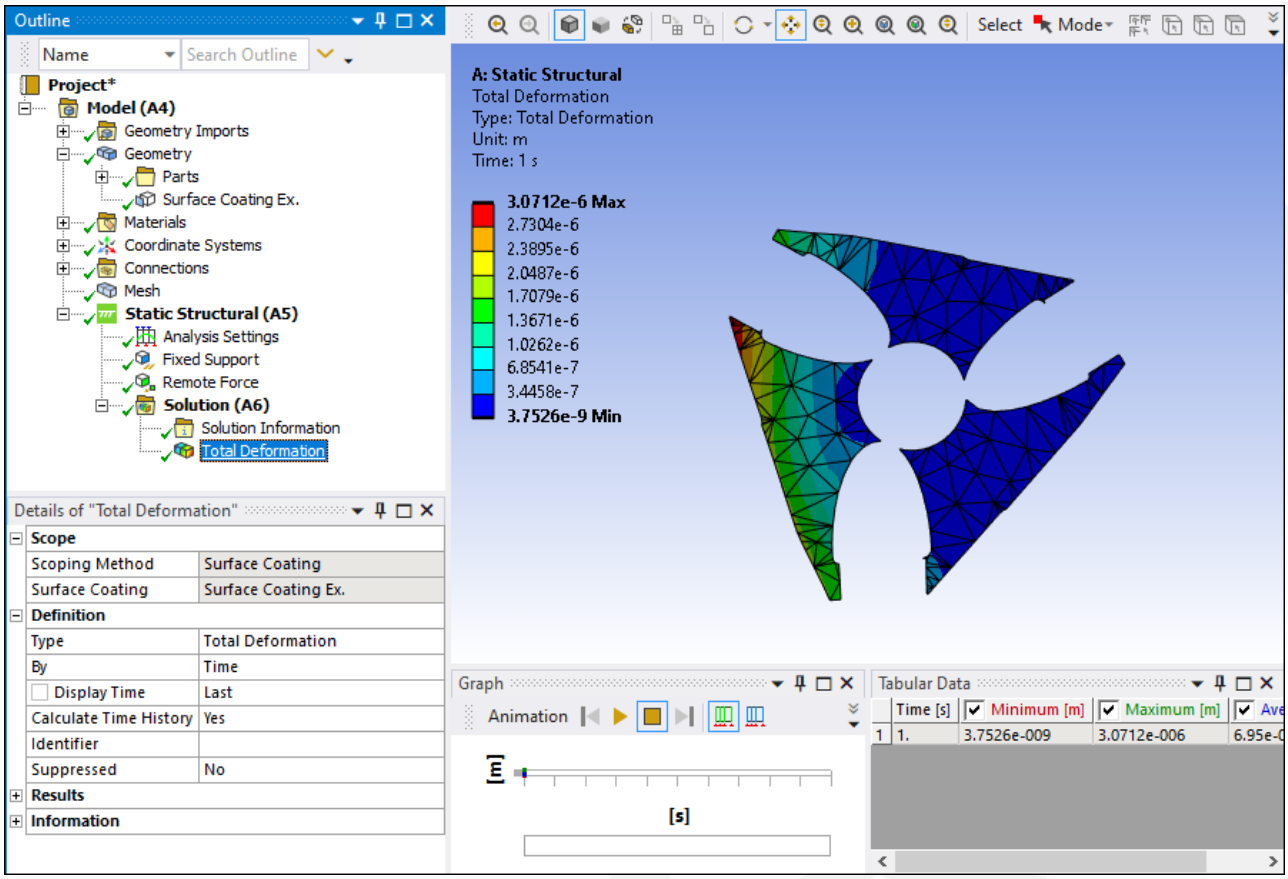
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Here is the evaluated result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Composite Analysis

For a Composite Analysis (p. 685), elemental strain and elemental stress results, elemental coordinate system (p. 1936) results, User Defined Results (p. 2192) (for Structural analyses), and heat flux results (for Thermal Analyses), can be scoped to Analysis Ply objects using the Sub Scope By property and the corresponding **Ply** property. The result calculation uses the intersection of the specified **Ply** and the **Geometry** property scoping. When scoping ply objects, the associated geometry selection needs to be based on either bodies or elements (p. 276). Body- and element-based Named Selections (p. 276) are supported. In addition, for shell topologies, you can scope to a face (which represents an entire element).

Note:

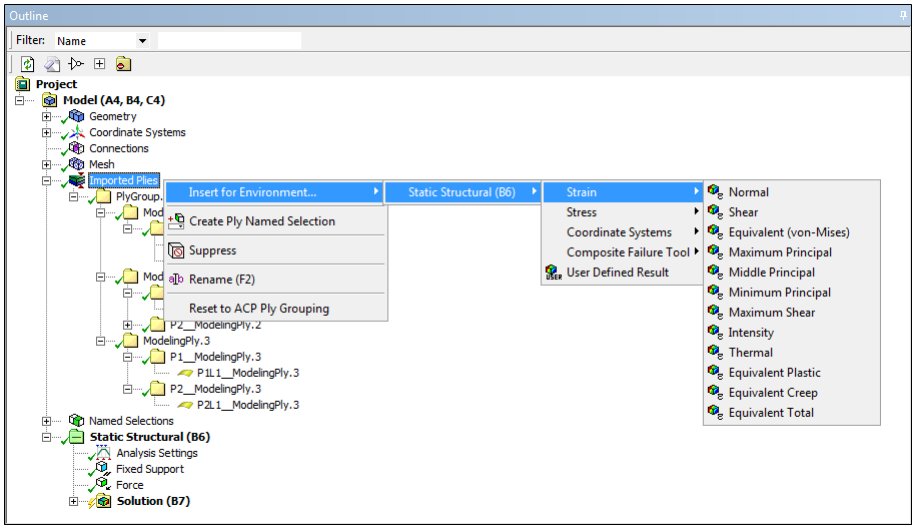
Direct graphical node or element selection requires you to generate the mesh and have the **Show Mesh** tool turned on.

Automated Result Scoping for Composites

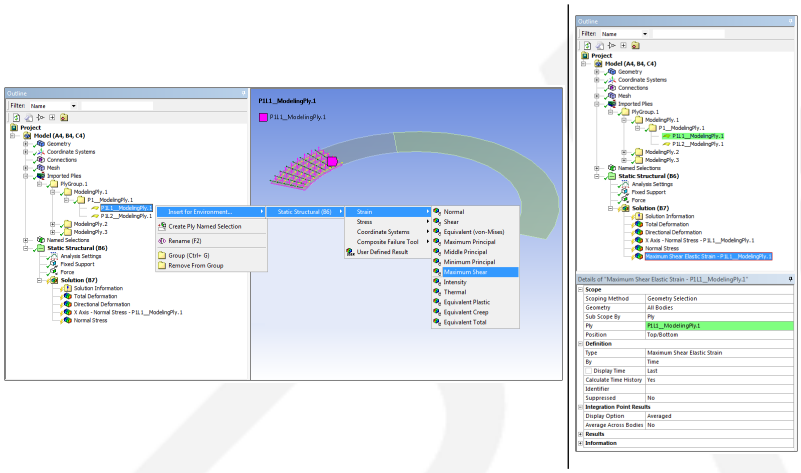
The Imported Plies object, as well as all of its child objects, offer an automated option for creating results, including User Defined Results (p. 2192). Right-clicking on a ply object provides the following dynamic menu options that enable you to create and scope a result at the same time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

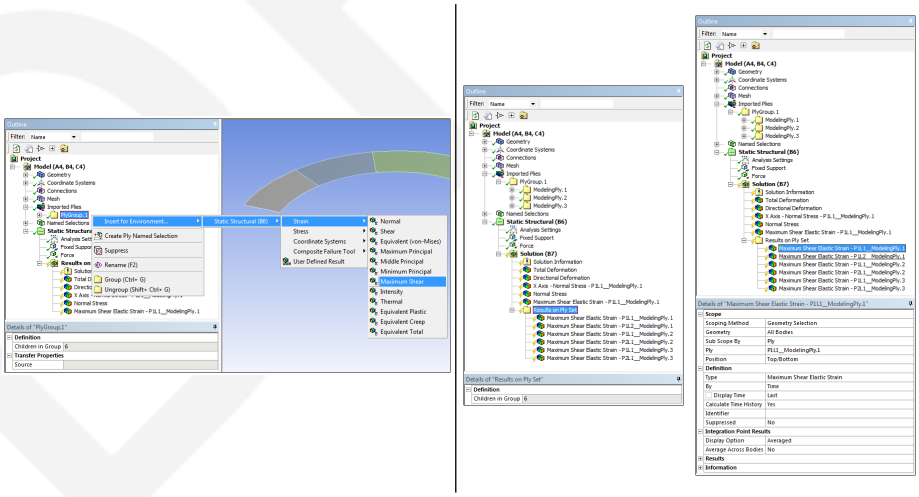
Published: 2023-10-23



Depending upon the level within the **Imported Plies** object that you select, you can create individual results, as shown below, or you can create a group of results.

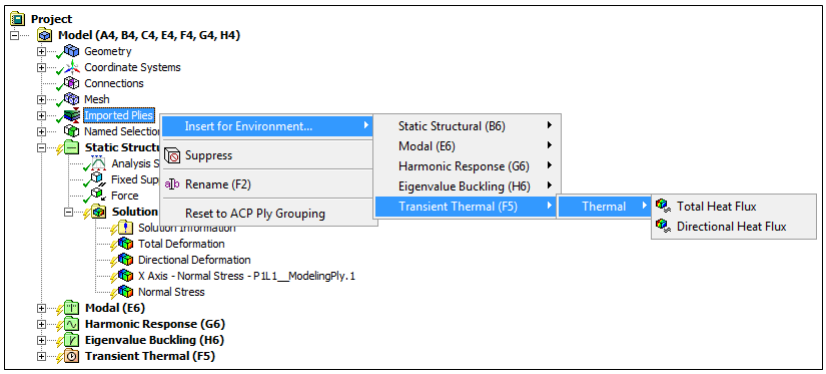


This example shows the creation of results for all of the individual plies within a ply group. The feature created a group of six results and scoped them accordingly. Once you create a group, the rules for Group Tree Objects (p. 208) apply.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

If your analysis includes multiple environments, only supported environments display on the menu and, only result objects supported by the given scoping display as the menu extends. As illustrated in the following example, multiple supported environments are selected and the supported result types for the Transient Thermal selection are the only options available. See the Result Context Tab (p. 62) section for additional information about how results are graphically represented following a solution.



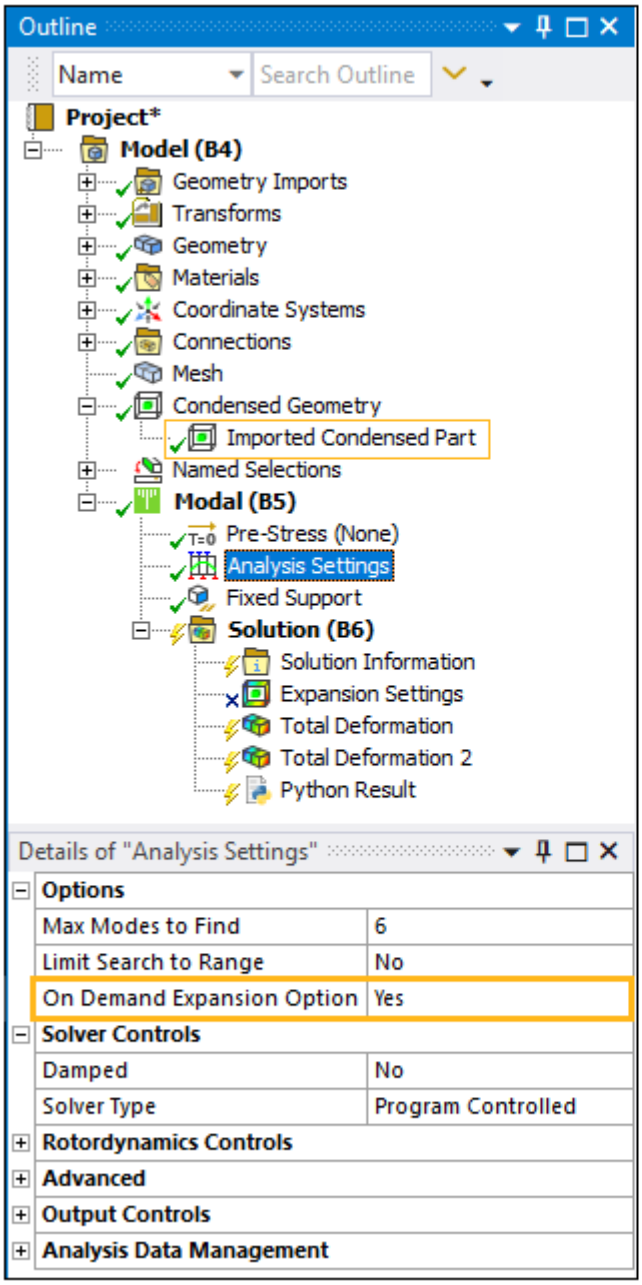
Imported Condensed Parts

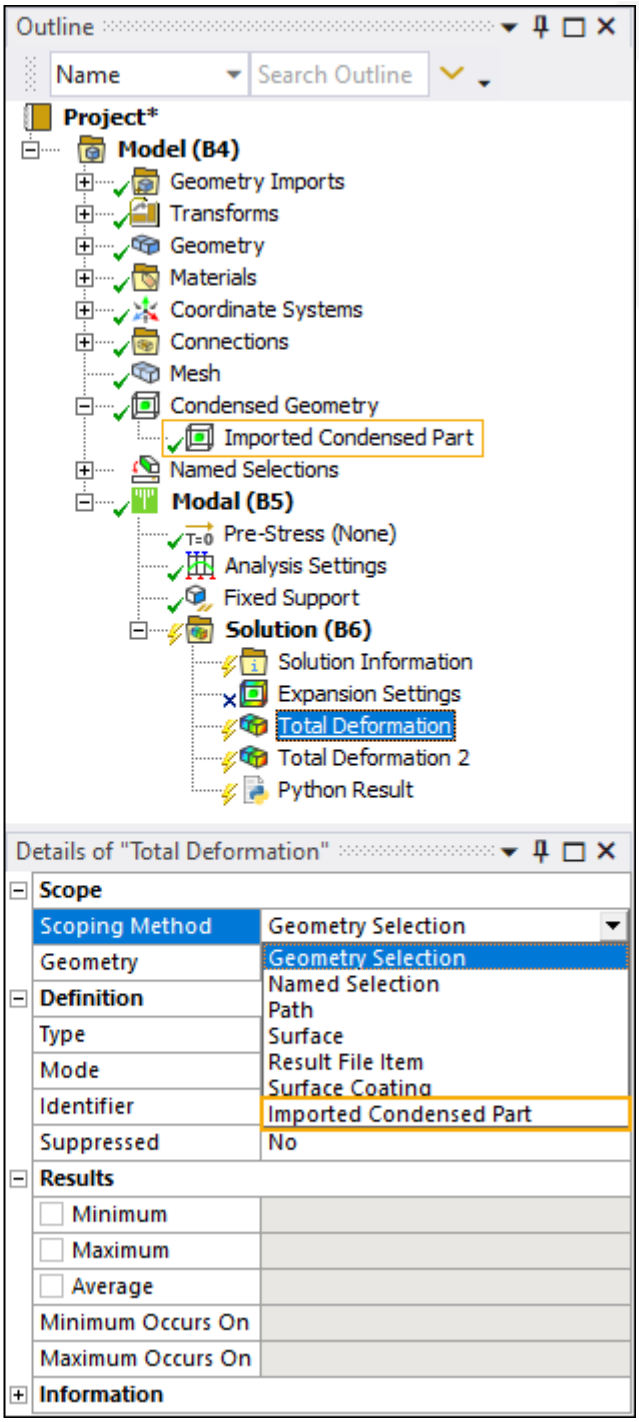
When your analysis includes an Imported Condensed Part, the **Scoping Method** option **Imported Condensed Part** becomes available for **Total Deformation** results that enables you to specify the scoping of the result on all bodies of the model as well as the expansion of the imported condensed parts. An example of this scenario is shown below.

This capability requires that you export the solution of a Substructure (p. 540) analysis using the **Use Pass and On Demand Expansion** option and that your current analysis have the On Demand Expansion Option (p. 1447) property set to **Yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

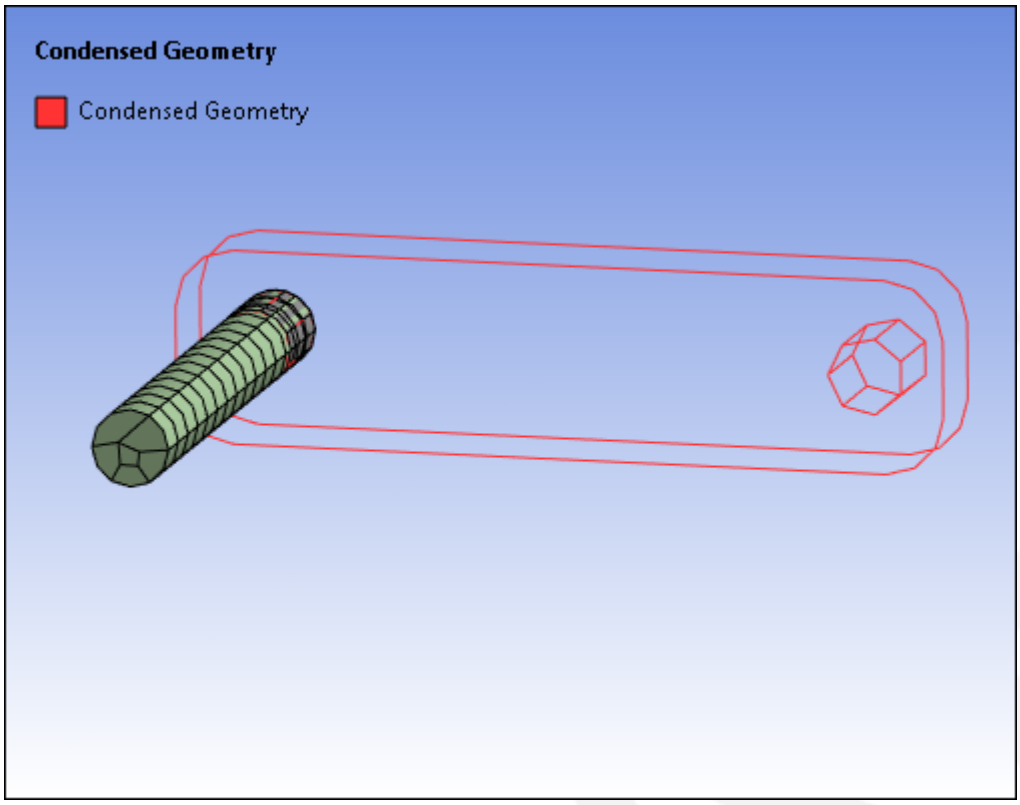




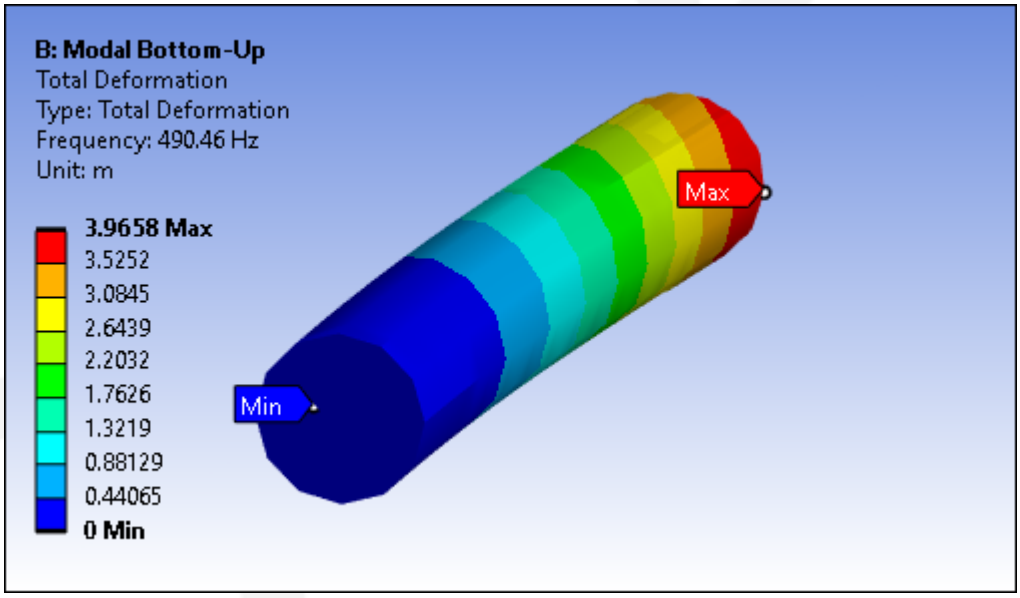
Example Scoping Scenario

Here is an example of a model that includes an **Imported Condensed Part**. The imported condensed part is shown in red wireframe form.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



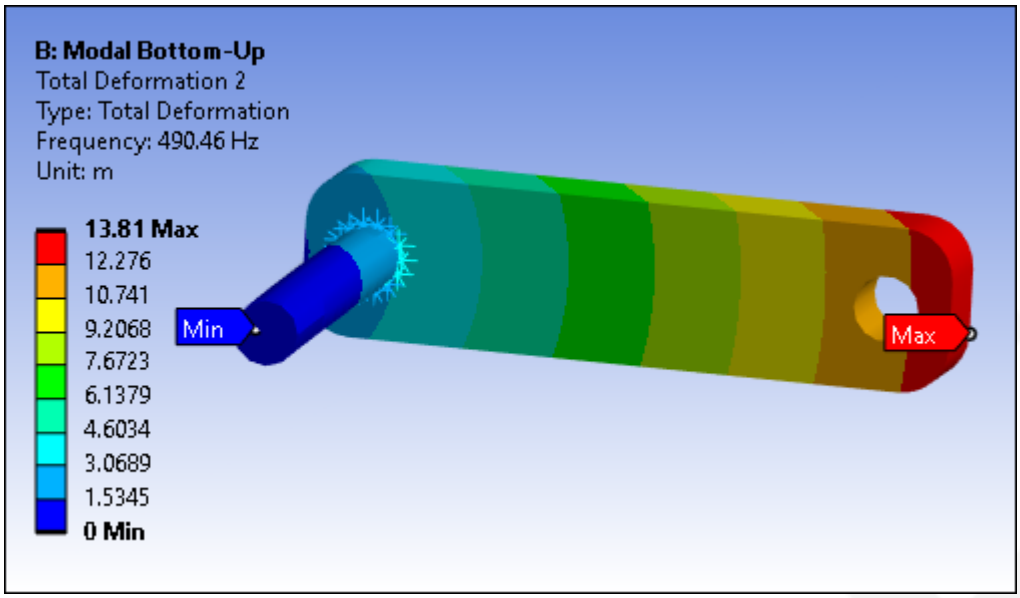
Here is a deformation result for the **Scoping Method** set to **Geometry Selection** and the **Geometry** property set to **All Bodies**. The imported condensed part not expanded.



Here is a deformation result for the **Scoping Method** set to **Imported Condensed Part** option for **All Bodies**. The expanded imported condensed part is now included.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Structural Results

The following structural result topics are addressed in this section:

- Deformation
- Stress and Strain
- Stabilization Energy
- Strain Energy
- Damage Results
- Linearized Stress
- Contact Results
- Frequency Response
- Phase Response
- Stress Tools
- Fatigue (Fatigue Tool)
- Fracture Results (Fracture Tool)
- Composite Failure Tool
- Composite Sampling Point Tool
- Contact Tool
- Bolt Tool
- Beam Tool
- Beam Results
- Structural Probes
- Response PSD Tool
- Gasket Results
- Campbell Diagram Chart Results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Waterfall Diagrams

Stress Tools (p. 2058) are used to determine the following results:

- Maximum Equivalent Stress Safety Tool (p. 2059)
- Maximum Shear Stress Safety Tool (p. 2061)
- Mohr-Coulomb Stress Safety Tool (p. 2062)
- Maximum Tensile Stress Safety Tool (p. 2064)

Structural Probes (p. 2092) can be used to determine the following results:

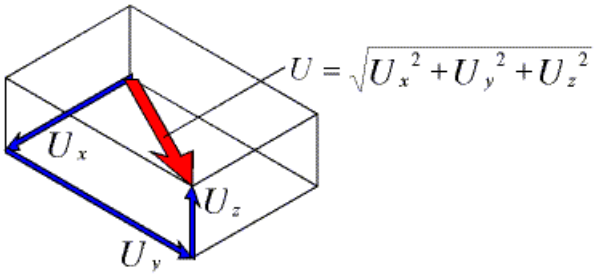
- Deformation
- Strain
- Position (p. 2102)
- Velocity
- Angular Velocity
- Acceleration
- Angular Acceleration
- Energy (p. 2105)
- Flexible Rotation Probe
- Force Reaction (p. 2107)
- Moment Reaction (p. 2107)
- Joint (p. 2118)
- Response PSD (p. 2121)
- Spring (p. 2123)
- Bearings (p. 2124)
- Beam (p. 2125)
- Bolt Pretension (p. 2125)
- Generalized Plane Strain (p. 2125)

Deformation

Physical deformations can be calculated on and inside a part or an assembly. Fixed supports prevent deformation: locations without a fixed support usually experience deformation relative to the original location. Deformations are calculated relative to the part or assembly *world coordinate system*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



- Component deformations (**Directional Deformation**)
- Deformed shape (**Total Deformation** vector)

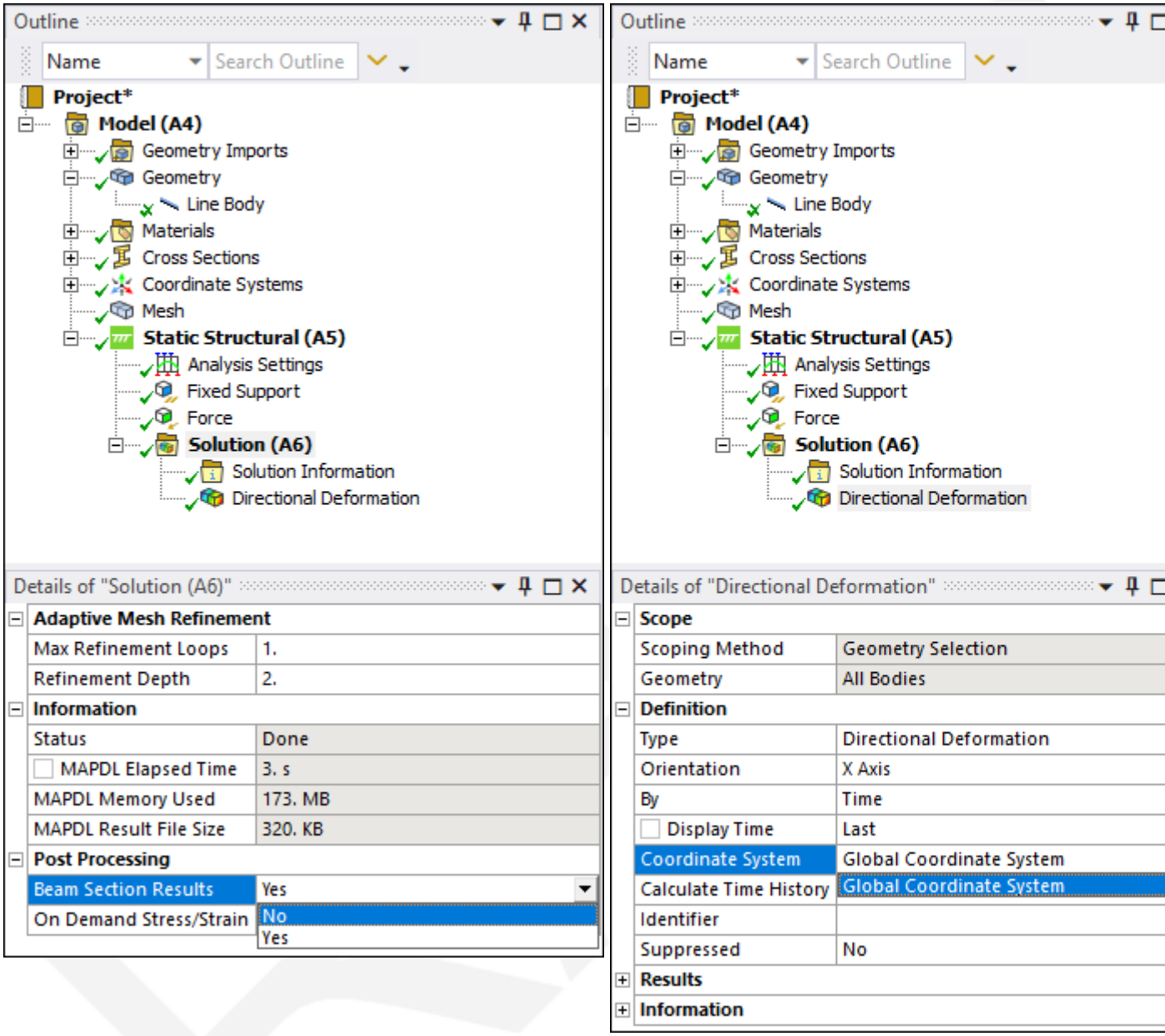
The three component deformations U_x, U_y, and U_z, and the deformed shape U are available as individual results.

Scoping (p. 1497) is also possible to both geometric entities and to underlying meshing entities (see example below). Numerical data is for deformation in the global X, Y, and Z directions. These results can be viewed with the model under wireframe (p. 68) display, facilitating their visibility at interior nodes.

Note:

The application currently has a limitation. If you have a Line Body model and the **Model Type** property is set to either **Beam** or **Pipe** (elements), and the **Beam Section Results** property of the **Solution** object is set to **Yes**, the application does not display result contours in the **Solution Coordinate System**. This is true even if you set the **Coordinate System** property of a result object to **Solution Coordinate System**. The contours will display in the **Global Coordinate System**.

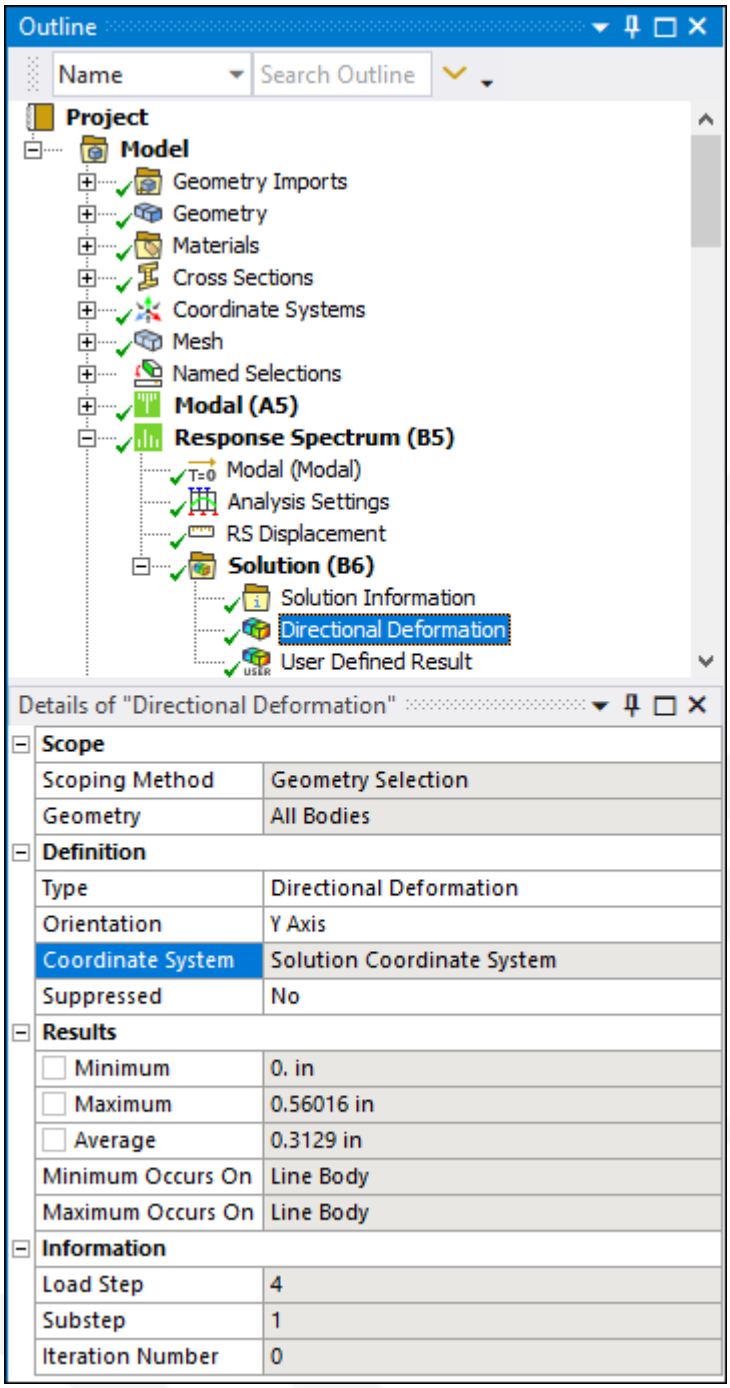
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



This limitation is also applicable if you are performing a **Response Spectrum** analysis where the **Coordinate System** property is read-only and set to **Solution Coordinate System**. Result contours display in the **Global Coordinate System**. In this scenario, you can display the contours in the **Solution Coordinate System** by setting the **Beam Section Results** property to **No**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Example: Scoping Deformation Results to Mesh Nodes

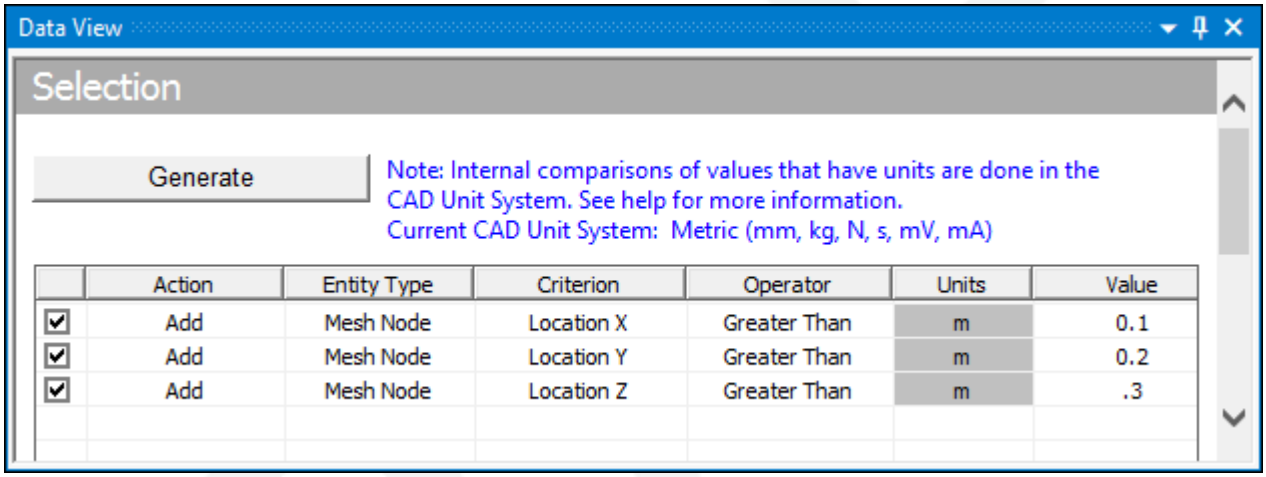
The following example illustrates how to obtain deformation results for individual nodes in a model. The nodes are specified using criteria based named selections.

1. Create a named selection by highlighting the **Model** object and select the **Named Selection** option from the **Insert** (p. 42) group of the **Model** Context tab.

2. Highlight the **Selection** object and in the Details, set the **Scoping Method** property to **Worksheet**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. In the **Worksheet**, add a row and specify the following. Refer to Specifying Named Selections using Worksheet Criteria (p. 1061) for assistance, if needed.
   - **Entity Type** = **Mesh Node**.
   - **Criterion** = **Location X**.
   - **Operator** = **Greater Than**.
   - **Value** = **0.1**.
4. Add a second row with **Criterion** = **Location Y**, **Value** = **0.2**, and all remaining items set the same as the first row.
5. Add a third row with **Criterion** = **Location Z**, **Value** = **0.3**, and all remaining items set the same as the first row. The table should appear as follows:

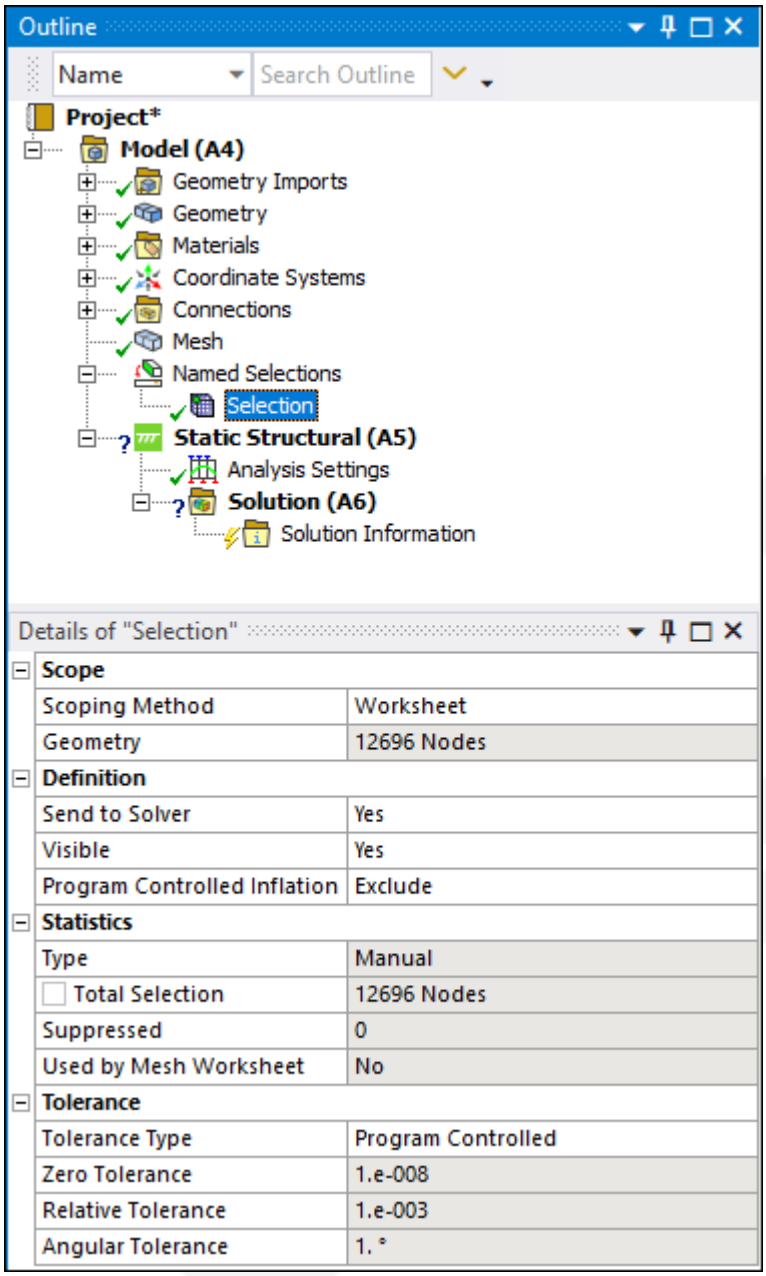


	Action	Entity Type	Criterion	Operator	Units	Value
☑	Add	Mesh Node	Location X	Greater Than	m	0.1
☑	Add	Mesh Node	Location Y	Greater Than	m	0.2
☑	Add	Mesh Node	Location Z	Greater Than	m	.3

6. Click the **Generate** button. The **Geometry** property displays the number of nodes that meet the criteria defined in the **Worksheet**.

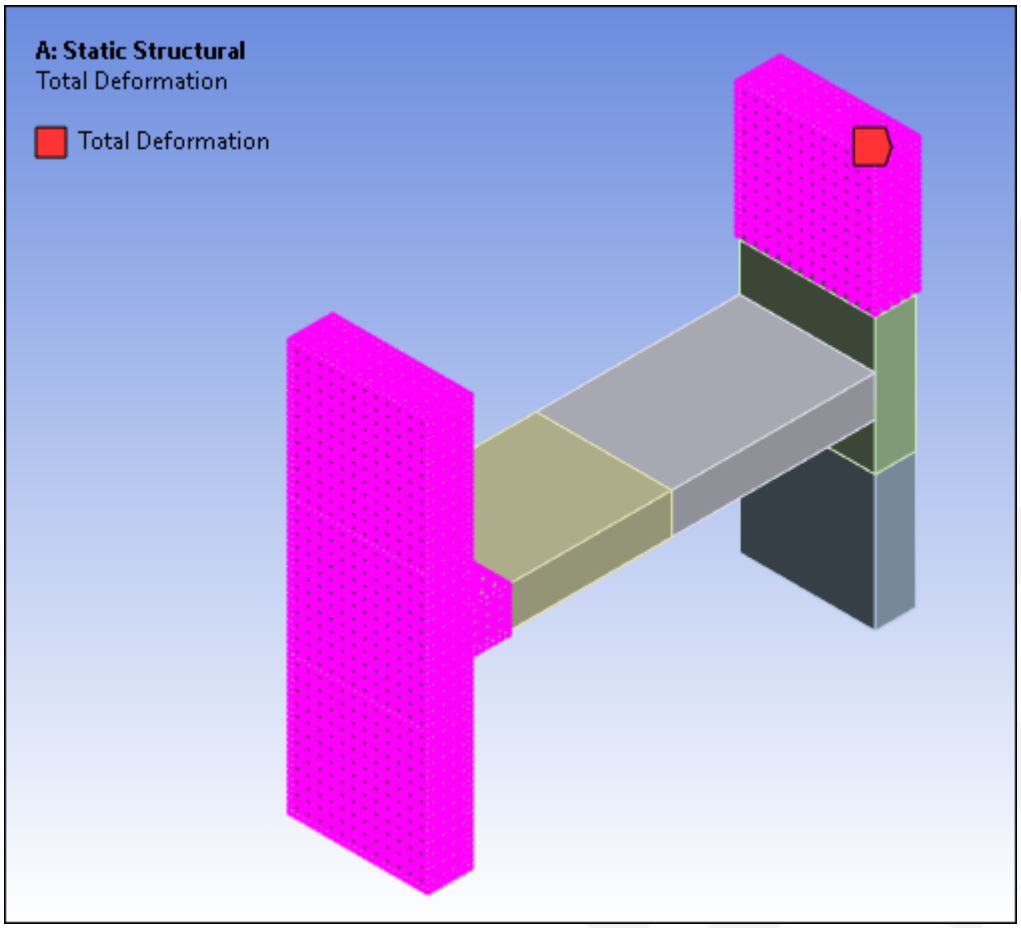
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



7. After applying loads and supports to the model, add a **Total Deformation** result object, highlight the object, set **Scoping Method** to **Named Selection**, and set **Named Selection** to the **Selection** object defined above that includes the mesh node criteria. Before solving, annotations are displayed at each selected node as shown below.

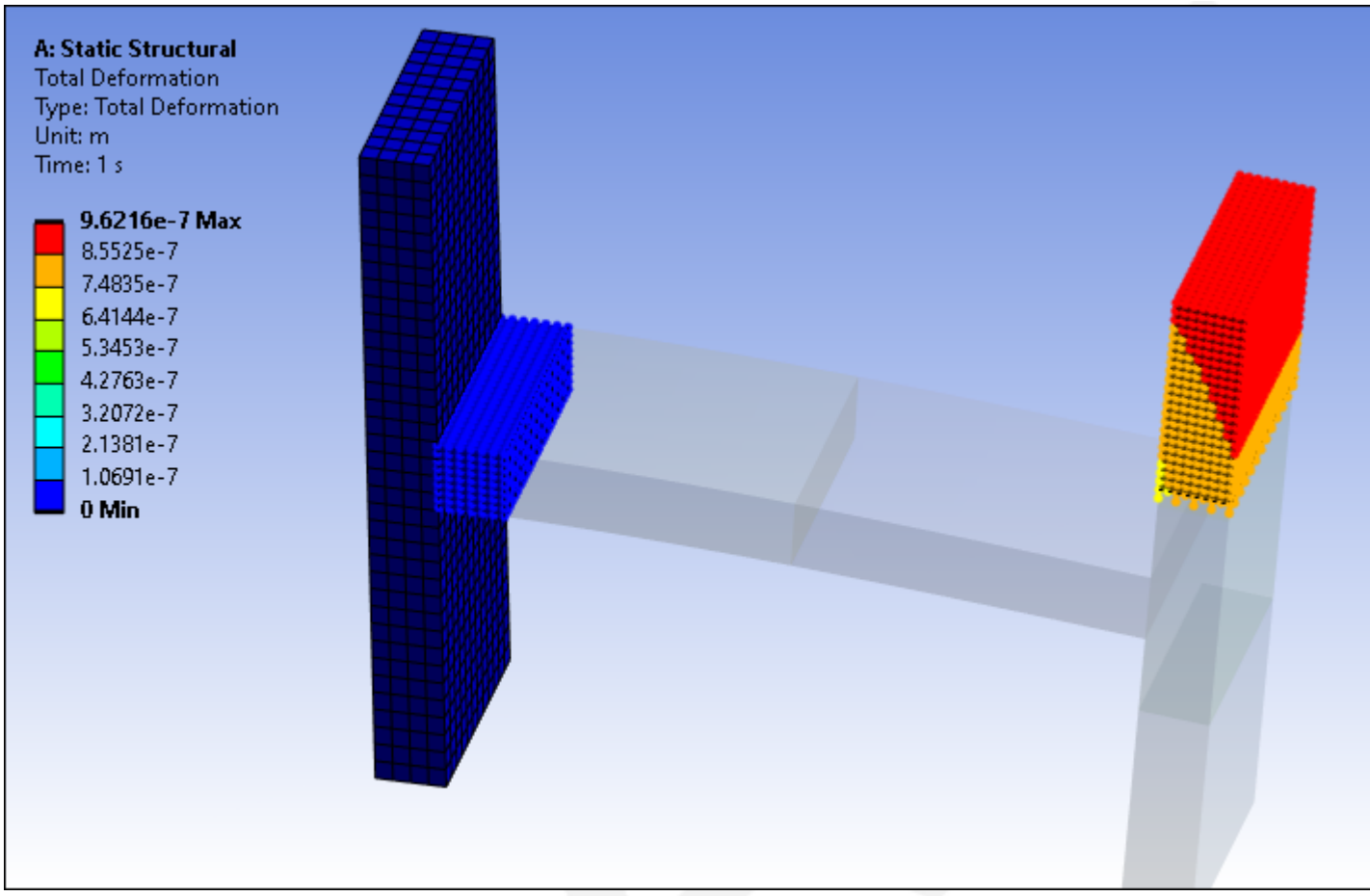
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



8. Solve the analysis. Any element containing a selected node will display a contour color at the node. If all nodes on the element are selected, the element will display contour colors on all facets. Element facets that contain unselected nodes will be transparent. An example is shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Note that all element facets are drawn, not just the facets on the surface or skin of the model.

To possibly reduce clutter for complex models, the size of the dots representing the nodes can be changed by selecting the **Large Vertex Contours** option from the **Display** group of Result Context Tab (p. 62).

Working with Deformations

Deformations can be used to:

- Set Alert objects.
- Control accuracy and convergence and to view converged results.
- Study deformations in a selected or scoped area of a part or an assembly.

Important:

The deformation result can exhibit a node-based display limitation. If a node represents a remote point, the application does not process result data for it and as a result, Mechanical does not display result data. Deformed shapes, deformation contour colors, and deformation MIN/MAX values can differ from the displays (and listings) of Mechanical APDL commands, such as **PRNSOL**, **PLNSOL**, and **MONITOR**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Velocity and Acceleration

In addition to deformation results, velocity and acceleration results are also available for Transient Structural (p. 665), Harmonic Response (p. 496) (Full and MSUP), Rigid Dynamics (p. 550), Random Vibration (p. 524), and Response Spectrum (p. 533) analyses. Both total and directional components are available for the Transient Structural and Harmonic Response analyses but only directional components are available for Random Vibration and Response Spectrum (**Total Deformation** is available).

Considerations for Random Vibration

For Random Vibration analyses, only component directional deformations are available because the directional results from the solver are statistical in nature. The X, Y, and Z displacements cannot be combined to get the magnitude of the total displacement. The same holds true for other derived quantities such as principal stresses.

Directional Deformation, **Directional Velocity**, and **Directional Acceleration** result objects in Random Vibration analyses also include the following additional items in the Details:

- **Reference** - Read-only reference indication that depends on the directional result. Possible indications are:
  - **Relative to base motion** for a **Directional Deformation** result.
  - **Absolute (including base motion)** for a **Directional Velocity** or **Directional Acceleration** result.
- **Scale Factor** - A multiple of standard deviation values (with zero mean value) that you can enter which determines the probability of the time the response will be less than the standard deviation value. By default, the results output by the solver are **1 Sigma**, or one standard deviation value. You can set the **Scale Factor** to **2 Sigma**, **3 Sigma**, or to **User Input**, in which case you can enter a custom scale factor in the **Scale Factor Value** field.
- **Probability** - Read-only indication of the percentage of the time the response will be less than the standard deviation value as determined by your entry in the **Scale Factor** field. A **Scale Factor** of **1 Sigma** = a **Probability** of **68.3 %**. **2 Sigma** = **95.951 %**. **3 Sigma** = **99.737 %**.

Stress and Strain

Stress solutions allow you to predict safety factors, stresses, strains, and displacements given the model and material of a part or an entire assembly and for a particular structural loading environment.

A general three-dimensional stress state is calculated in terms of three normal and three shear stress components aligned to the part or assembly world coordinate system.

The principal stresses and the maximum shear stress are called invariants: that is, their value does not depend on the orientation of the part or assembly with respect to its world coordinate system. The principal stresses and maximum shear stress are available as individual results.

The principal strains ε_1, ε_2, and ε_3 and the maximum shear strain γ_{max} are also available. The principal strains are always ordered such that $\varepsilon_1 > \varepsilon_2 > \varepsilon_3$. As with principal stresses and the maximum shear stress, the principal strains and maximum shear strain are invariants.

You can choose from the following stress/strain results:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Equivalent (von Mises)
Maximum, Middle, and Minimum Principal
Maximum Shear
Intensity
Vector Principals
Error (Structural)
Thermal Strain
Equivalent Plastic Strain
Accumulated Equivalent Plastic Strain
Equivalent Creep Strain
Equivalent Total Strain
Membrane Stress
Bending Stress

Normal (X, Y, Z) and **Shear** (XY, YZ, XZ) stress and strain results are also available.

It is assumed that whatever holds true for stress applies to strain as well. However, the relationship between maximum shear stress and stress intensity does not hold true for an equivalent relationship between maximum shear strain and strain intensity.

For more information about Stress/Strain, see the *Mechanical APDL Theory Reference*.

Considerations

Note the following important aspects of stress and strain results:

- The degree of uncertainty in the numerical calculation of Stress answers depends on your accuracy preference. See Using Adaptive Convergence (p. 1852) for information on available options and their effect on Stress answers.
- Although a rare occurrence, for results defined by the expressions EPTO and EPTT (total mechanical strains and total mechanical strains and thermal strains), Mechanical will not create contours for these expressions for an element if the element records lengths of the individual strains EPEL/EPPL/EPCR/EPTH in the result file. See the **PRESOL** command for complete definitions.
- Mechanical does not currently support swelling strain (EPSW) and does not use it in the calculation of total strains.
- Mechanical does not support standard stress and strain results for gasket results (p. 2131). However, if you specify a body as a gasket and the Mechanical APDL key option KEYOPT(8) is set to 1, (KEYOPT(8)=1), for the gasket element, you can view regular stress/strain result contours using a post commands snippet.

Equivalent (von Mises)

Equivalent stress is related to the principal stresses by the equation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

$$\varepsilon_e = \frac{1}{1+\nu'} \left(\frac{1}{2} \left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2 \right] \right)^{\frac{1}{2}}$$

where:

ν' = effective Poisson's ratio, which is defined as follows:

- Material Poisson's ratio for elastic and thermal strains computed at the reference temperature of the body.
- 0.5 for plastic strains.

Note:

Currently, for Linked MSUP analyses with the **Expand Results From** detail under **Output Controls** set to **Modal Solution**, the Mechanical APDL solver does not calculate equivalent strains. If you choose to display equivalent strain results, you will see zero contours.

Maximum, Middle, and Minimum Principal

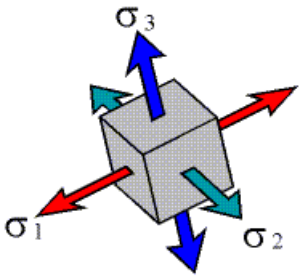
From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses:

σ_1 - Maximum

σ_2 - Middle

σ_3 - Minimum

The principal stresses are always ordered such that $\sigma_1 > \sigma_2 > \sigma_3$.

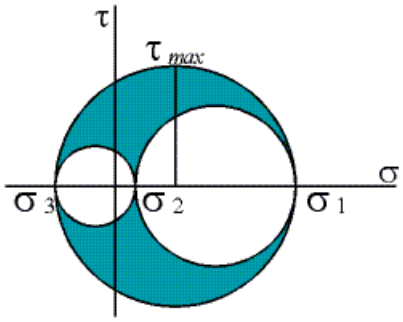


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Maximum Shear

The maximum shear stress τ_{max}, also referred to as the maximum shear stress, is found by plotting Mohr's circles using the principal stresses:



or mathematically through:

$$\tau_{\max}=\frac{\sigma_1-\sigma_3}{2}$$

For elastic strain, the maximum shear elastic strain γ_{max} is found through:

$$\gamma_{max} = \varepsilon_1 - \varepsilon_3$$

since the shear elastic strain reported is an engineering shear elastic strain.

Intensity

Stress intensity is defined as the largest of the absolute values of $\sigma_1 - \sigma_2$, $\sigma_2 - \sigma_3$, or $\sigma_3 - \sigma_1$:

$$\sigma_I=MAX\left(\left|\sigma_1-\sigma_2\right|,\left|\sigma_2-\sigma_3\right|,\left|\sigma_3-\sigma_1\right|\right)$$

Stress intensity is related to the maximum shear stress:

$$\sigma_I = 2\tau_{max}$$

Elastic Strain intensity is defined as the largest of the absolute values of $\varepsilon_1 - \varepsilon_2$, $\varepsilon_2 - \varepsilon_3$, or $\varepsilon_3 - \varepsilon_1$:

$$\varepsilon_I=MAX\left(\left|\varepsilon_1-\varepsilon_2\right|,\left|\varepsilon_2-\varepsilon_3\right|,\left|\varepsilon_3-\varepsilon_1\right|\right)$$

Elastic Strain intensity is equal to the maximum shear elastic strain:

$$\varepsilon_I = \gamma_{max}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Equivalent Stress (and **Equivalent Elastic Strain**) and **Stress Intensity** are available as individual results.

Note:

Computation of **Equivalent Elastic Strain** uses Poisson's ratio. If Poisson's ratio is temperature dependent then the Poisson's ratio value at the reference temperature of the body is used to compute the **Equivalent Elastic Strain**.

Vector Principals

A Vector Principals plot provides a three-dimensional display of the relative size of the principal quantities (stresses or elastic strains), and the directions along which they occur. Positive principals point outwards and negative ones inwards.

Plots of Vector Principals help depict the directions that experience the greatest amount of normal stress or elastic strain at any point in the body in response to the loading condition. The locus of directions of maximum principal stresses, for example, suggests paths of maximum load transfer throughout a body.

You specify this result as you would any other result. The default scoping is **All Bodies**, but supported options include geometry, named selection, Path, Surface, Result File Item and Surface Coating.

Once evaluated, the **Tabular Data** window displays the Minimum and Maximum result values for each principal component at each step.

In addition, you can export the result to a Microsoft Excel (.xls) file. This file provides six fields of data, including the maximum, middle, and minimum principal quantities (stresses or elastic strains) as well as the Mechanical APDL application Euler angle sequence. The Euler angle sequence (the **CLOCAL** command in MAPDL) produces a coordinate system whose X, Y and Z-axis are the directions of maximum, middle and minimum principal quantities, respectively. Denoted as ThetaXY, ThetaYZ, and ThetaZX, the sequence orients the principal coordinate system relative to the global system.

You can graphically review these results in the **Geometry** window using the options of the Vector Display (p. 74) group on the **Result** Context Tab.

Note:

Vector principal results do not support amplitude data.

Error (Structural)

You can insert an **Error** result based on stresses to help you identify regions of high error and therefore show where the model would benefit from a more refined mesh in order to get a more accurate answer. You can also use the **Error** result to help determine where Mechanical will be refining elements if Convergence (p. 1852) is active. The **Error** result is based on the same errors used

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

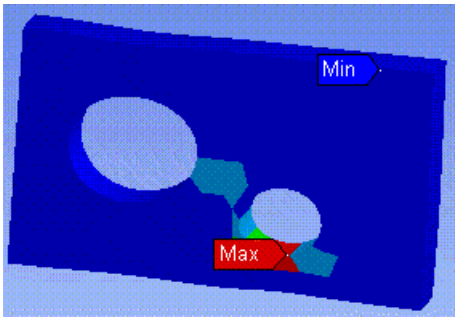
in adaptive refinement (p. 1853). Information on how these errors are calculated is included in POST1 - Error Approximation Technique, in the Theory Reference.

Note:

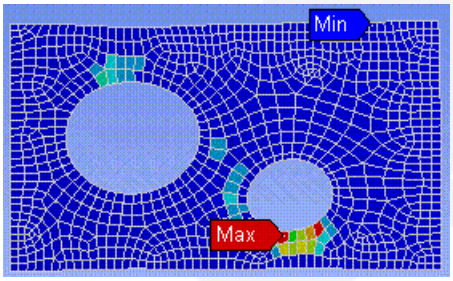
- The **Error** result is based on linear stresses and as such may be inaccurate in certain nonlinear analyses (for example, when plasticity is active). Furthermore, the **Error** result is currently restricted to isotropic materials. You may wish to refer to the *Structural Material Properties* section of the Engineering Data help for additional information.
- A **Error** result does not support being scoped to a reinforcement element (REINF263, REINF264, or REINF265) and if selected, these elements would report a value of zero.

Presented below are example applications of using the **Error** result in a Structural simulation.

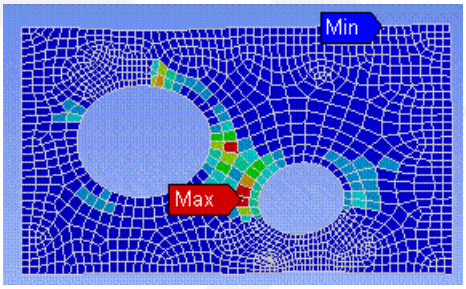
3D Model:



2D Model, Base Mesh:

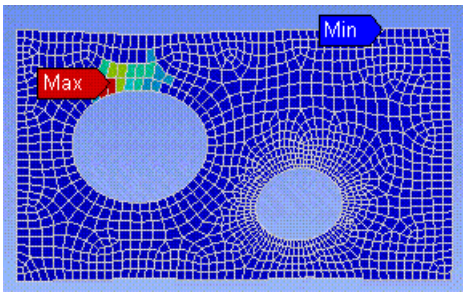


2D Model, Adaptive Refinement (Convergence Added):



2D Model, With Mesh Control:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Thermal Strain

Thermal strain is computed when coefficient of thermal expansion is specified and a temperature load is applied in a structural analysis. To specify the coefficient of thermal expansion, you must set **Thermal Strain Effects** (p. 342) to **Yes** in the Details of the part or body objects before initiating a solve.

Each component of thermal strain is computed as:

$$\varepsilon^{th} = \alpha^{se}(T - T_{ref})$$

Where:

ε^{th} - thermal strain in one of the directions x, y, or z.

α^{se} - Secant coefficient of thermal expansion defined as a material property in Engineering Data (see "Chapter 2.4 Linear Material Properties" in the *Element Reference* of the Mechanical APDL application Help for more information about the secant function).

T_{ref} - reference temperature or the "stress-free" temperature. This can be specified globally for the model using the Reference Temperature field of Static Structural or Transient Structural analysis types. Optionally you can also specify the reference temperature as a material property for cases such as the analysis for cooling of a weld or solder joint where each material has a different stress-free temperature.

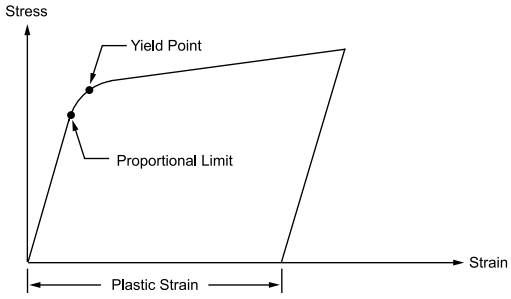
Equivalent Plastic Strain

The equivalent plastic strain gives a measure of the amount of permanent strain in an engineering body. The equivalent plastic strain is calculated from the component plastic strain as defined in the Equivalent (p. 2027) stress/strain section.

Most common engineering materials exhibit a linear stress-strain relationship up to a stress level known as the *proportional limit*. Beyond this limit, the stress-strain relationship will become nonlinear, but will not necessarily become inelastic. Plastic behavior, characterized by nonrecoverable strain or plastic strain, begins when stresses exceed the material's *yield point*. Because there is usually little difference between the yield point and the proportional limit, the Mechanical APDL application assumes that these two points are coincident in plasticity analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



In order to develop plastic strain, plastic material properties must be defined. You may define plastic material properties by defining either of the following in the Engineering Data:

- Bilinear Stress/Strain curve.
- Multilinear Stress/Strain curve.

Note:

Yield stresses defined under the **Stress Limits** section in the Engineering Data are used for the post tools only (that is, Stress Safety Tools and Fatigue tools), and do not imply plastic behavior.

Accumulated Equivalent Plastic Strain

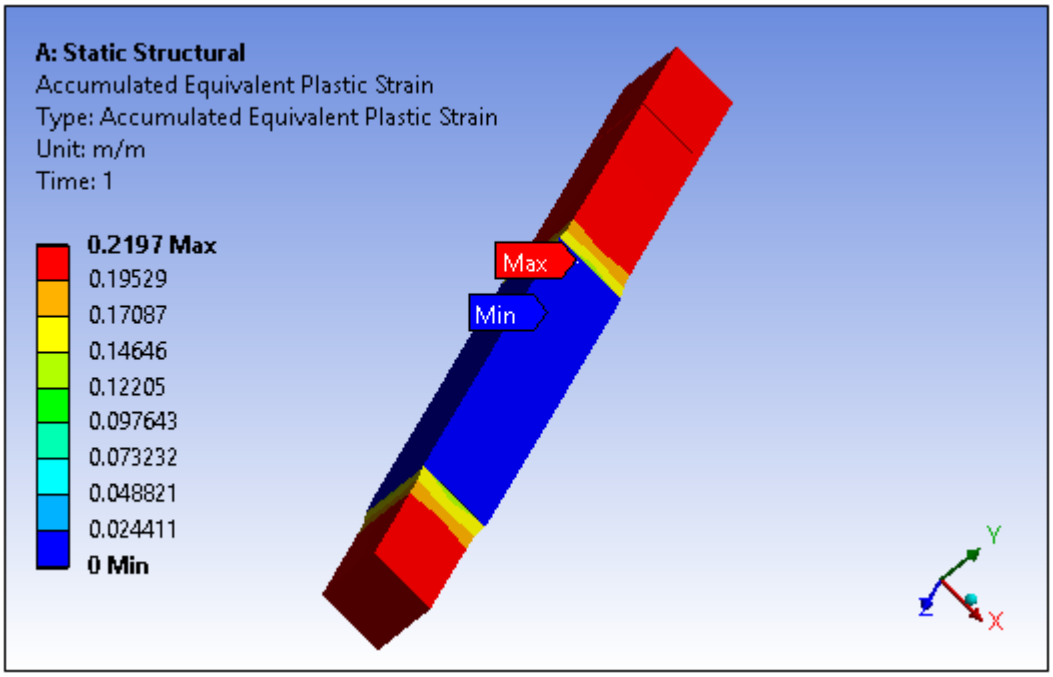
The **Accumulated Equivalent Plastic Strain** result is the summation of the **Equivalent Plastic Strain** (p. 2032) result increment for each step. **Equivalent Plastic Strain** is calculated from plastic strain components directly and **Accumulated Equivalent Plastic Strain** is integrated from the equivalent incremental plastic strain along the deformation path. This result is an indicator for the hardening of an engineering body. **Accumulated Equivalent Plastic Strain** is based on the alternating loading and the values are always increasing (like plastic work).

Accumulated Equivalent Plastic Strain and **Equivalent Plastic Strain** are equal for proportional, monotonic loading, only.

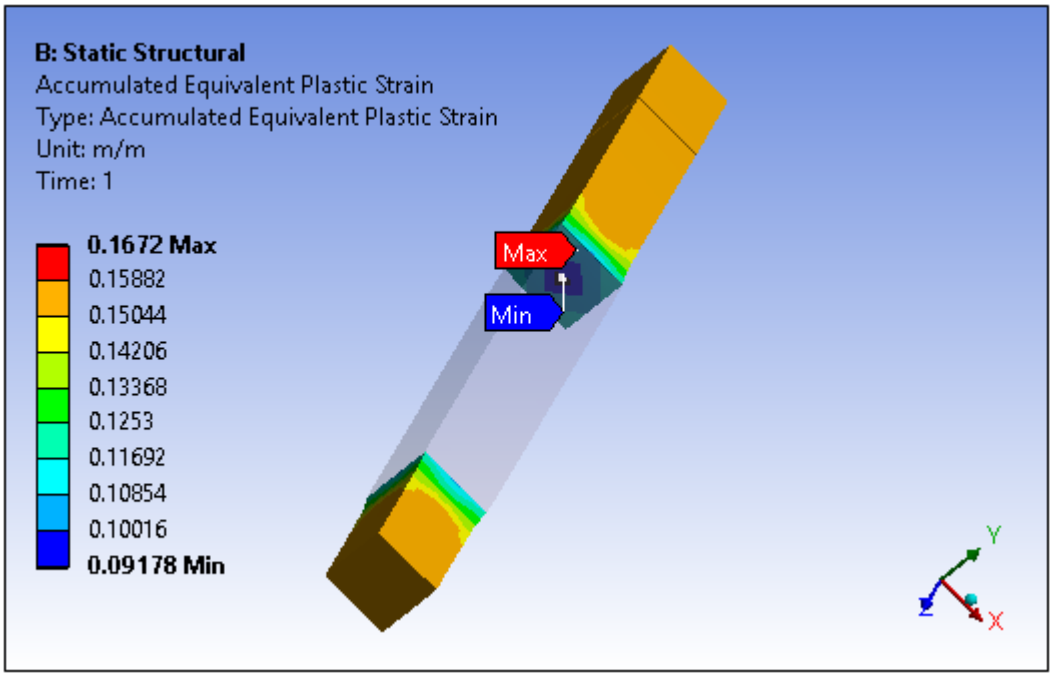
Large Deflection Effects

Note that the display of this result changes based on the setting of the **Large Deflection** property. When the property is set to **On**, the result displays zero values for any body with non-linear materials, as illustrated for the three body part shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Setting the property to **Off**, the same body with non-linear materials displays transparently. This display is like the Equivalent Plastic Strain result that displays non-linear materials transparently regardless of the **Large Deflection** property setting.



Equivalent Creep Strain

Creep is a rate-dependent material nonlinearity in which the material continues to deform under a constant load. The material deforms under an initial applied load and the load diminishes over time with an increase in deformation or creep strain. The equivalent creep strain gives a measure

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

of the amount of the creep strain in an engineering body. The equivalent creep strain is calculated from component creep strains.

In order to develop creep strain, creep material properties must be defined. You may define creep material properties by choosing one of the available 13 creep models in Engineering Data. This result type is available in Mechanical only after you have selected a creep material for at least one prototype in the analysis.

Equivalent Total Strain

The equivalent total strain gives a total value of strain in any engineering body. The total strain components are calculated by addition of components of elastic, plastic, thermal, and creep strains and then equivalent total strain is calculated from total strain components. This result type is available in Mechanical only if at least one of the other three strain results is available for post processing. In Mechanical APDL this strain in called Total Mechanical and Thermal Strain.

Membrane Stress

Membrane stress calculates the stresses along the thickness of the shell in longitudinal direction, in transverse direction, and in plane shear. The result is available only for shell bodies and solids that are meshed using the thin-solid meshing option. Each element of the body can display individual stress values and give a checkboard appearance to the result contours. The results are calculated in the element coordinate system.

Shell membrane stress tensor (s11m, s22m, s12m) is the average of the in-plane stress tensor (s11(z), s22(z), s12(z)) along the shell thickness direction:

$$\sigma_{11}^{m}=\frac{1}{t}\int_{0}^{t}\sigma_{11}(z)dz$$

$$\sigma_{22}^{m}=\frac{1}{t}\int_{0}^{t}\sigma_{22}(z)dz$$

$$\sigma_{12}^{m}=\frac{1}{t}\int_{0}^{t}\sigma_{12}(z)dz$$

Where:

t is the total shell thickness,

z is the thickness location where the in-plane stress is evaluated.

Unlike linearized stress in other elements, a pre-defined path through the shell thickness is not required in order to compute shell membrane stress.

Note:

Make sure that the Output Control (p. 1473), **General Miscellaneous** is set to **Yes** or your results may be under-defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Bending Stress

The result is available only for shell bodies and solids that are meshed using the thin-solid meshing option and are calculated in the element coordinate system. Each element of the body can display individual stress values and give a check-board appearance to the result contours.

Shell bending stress tensor (s11b, s22b, s12b) represents the linear variation portion of the in-plane stress tensor (s11(z), s22(z), s12(z)) along the shell thickness direction:

$$\sigma_{11}^{b}=\frac{6}{t^{2}}\int_{0}^{t}\sigma_{11(z)}\left(\frac{t}{2}-z\right)dz$$

$$\sigma_{22}^{b}=\frac{6}{t^{2}}\int_{0}^{t}\sigma_{22(z)}\left(\frac{t}{2}-z\right)dz$$

$$\sigma_{12}^{b}=\frac{6}{t^{2}}\int_{0}^{t}\sigma_{12(z)}\left(\frac{t}{2}-z\right)dz$$

Where:

t is the total shell thickness,

z is the thickness location where the in-plane stress is evaluated.

Note:

Make sure that the Output Control (p. 1473), **General Miscellaneous** is set to **Yes** or your results may be under-defined.

Stabilization Energy

Stabilization can help with convergence problems, but it can also affect accuracy if the stabilization energy or forces are too large. Although Ansys automatically reports the stabilization force norms and compares them to internal force norms, it is still very important to check the stabilization energy and forces to determine whether or not they are excessive. If the stabilization energy is much less than the potential energy (for example, within a 1.0 percent tolerance), the result should be acceptable. Stabilization energy is not available to the Samcef or ABAQUS solver.

When stabilization energy is large, check the stabilization forces at each DOF for all substeps. If the stabilization forces are much smaller than the applied loads and reaction forces (for example, within a 0.5 percent tolerance), the results are still acceptable. Such a case could occur when an elastic system is loaded first, then unloaded significantly. It is possible that the final element potential energy is small and stabilization energy is relatively large, but all stabilization forces are small. Currently, stabilization forces are accessible in the .OUT file.

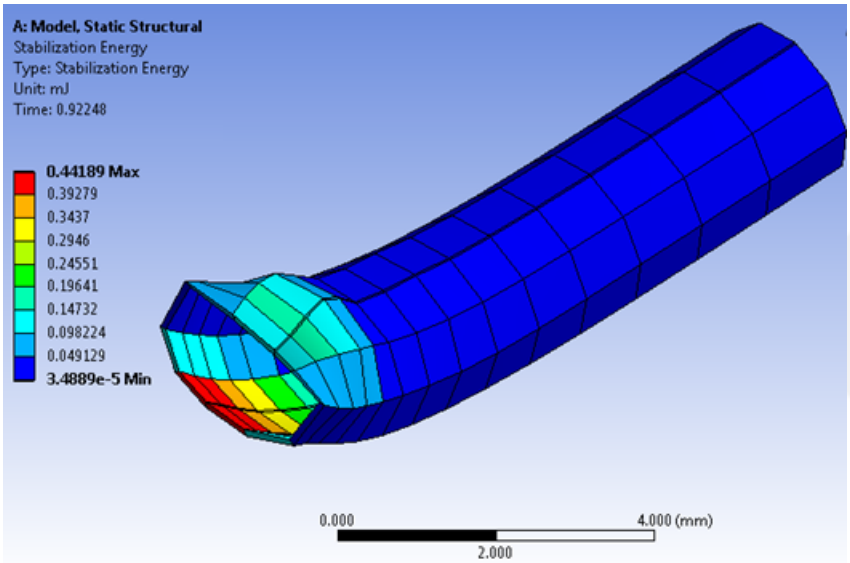
Even when both stabilization energy and forces are too large, the results could still be valid. Such a scenario is possible when a large part of an elastic structure undergoes large rigid body motion (as in a snap-through simulation). In such a case, the stabilization energy could be large as well as the stabilization force for some DOFs at some substeps, but the results could still be acceptably accurate. Nevertheless, consider the results along with other support data and use your own discretion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To insert a **Stabilization Energy** result, highlight the **Solution** object in the tree, then select **Stabilization Energy** from the Solution Context Tab (p. 62) or right-click the object and choose **Insert> Energy> Stabilization Energy**.

The following figure shows an example stabilization energy contour plot:



Strain Energy

Energy stored in bodies due to deformation. This value is computed from stress and strain results. It includes plastic strain energy as a result of material plasticity.

The **Strain Energy** result is available in the **Energy** drop-down menu of the **Results** group on the Solution Context (p. 62) tab. The option is also available from the context (right-click) menu of the **Solution** object (**Insert> Energy> Strain Energy**).

Damage Results

Mechanical supports a number of damage results using non-linear material models, including the Mullins Effect, Progressive Damage, and Physical Failure Criteria.

Mullins Effect

The Mullins effect is a phenomenon resulting from load-induced changes to constitutive response exhibited by some hyper elastic materials, especially filled polymers. The effect is most evident during cyclic loading, where the unloading response is more compliant than the loading behavior. During the process of cyclic loading, stress-strain curve for these materials is dependent on the maximum previous load, where the load is the strain energy of the virgin hyper elastic material. As the maximum previous load increases, changes to the virgin hyper elastic constitutive model also increase, due to the Mullins effect. Below the maximum previous load, the Mullins effect changes are not evolving: however, the Mullins effect still modifies the hyper elastic constitutive response based on the maximum previous load. If the load increases beyond the maximum previous all time value, the result is an irreversible and instantaneous softening of the material, which causes a hysteresis in the stress-strain response.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The Mullins effect is modeled with the modified Ogden-Roxburgh pseudo-elastic model (**TB**,CDM,,,,PSE2) and is applicable to any nearly or purely incompressible hyperelastic model (**TB**,HYPER). For more information on the Mullins effect, see Mullins Effect Model.

Mechanical supports two results for the Mullins Effect: Mullins Damage Variable and Mullins Max. Previous Strain Energy.

The Mullins Damage Variable is a unitless scale range from 0, at which the material is completely damaged without any stiffness, to 1, at which the material is intact, without any loss of stiffness.

At a given time step, the Mullins Max. Previous Strain Energy result is the maximum value of strain energy of the virgin material in the time interval [0, t0], where t0 is the beginning of a time step. Depending on the unit system you choose, this result chooses the appropriate unit of energy. A typical unit is the Joules (J) unit.

Progressive Damage

Progressive Damage is associated with the damage phenomenon that occurs in composite materials. When a composite material is subjected to loading, the matrix and fiber controlled types of failure can occur both separately or sequentially. After a certain point, the material experiences enough damage in the form of the local failures that the material can no longer sustain the load. These local failures govern the ultimate load that the material can withstand.

Progressive Damage uses material damage initiation (**TB**, DMGI) and evolution criteria (**TB**, DMGE) to analyze the progressive damage in composites. While Physical Failure Criteria analyzes the failure criteria, Progressive Damage analyzes the progression of the damage.

Damage Initiation Criteria defines the criteria type for determining the onset of material damage under loading. Depending upon the failure mode selected here, the respective failure criteria will be computed for "Physical Failure Criteria". The available failure modes for damage are:

- Maximum Strain
- Maximum Stress
- Puck
- Hashin
- LaRc03
- LaRc04

The Damage Evolution Law defines the material damage evolution law (or the way a material degrades) following the initiation of damage. The stiffness reduction takes a value of 0 to 1, where 0 is no damage and 1 is completely damaged.

For more information, see Damage-Evolution Law and Fiber-Reinforced Material Damage in the Mechanical APDL documentation.

The Progressive Damage model supports the following results:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Result	Description
Damage Status	The Damage Status result will be an enum type with values of 0, 1, or 2, where • 0 -- undamaged • 1 -- partially damaged • 2 -- completely damaged
Fiber Tensile Damage Variable	The Fiber Tensile Damage Variable result value will be in the range of 0 to the "Tensile Fiber Stiffness Reduction" value set in the Damage Evolution Law. That is, if you set the Tensile Fiber Stiffness Reduction to 0.6, the range of Fiber Tensile damage variable result will be in the range of 0 to 0.6. A value of 0 for this result means undamaged and a value of 1 means completely damaged. The result has no units.
Fiber Compressive Damage Variable	The Fiber Compressive Damage Variable result value will be in the range of 0 to the "Compressive Fiber Stiffness Reduction" value set in the Damage Evolution Law. That is, if you set the Compressive Fiber Stiffness Reduction to 0.6, the range of Fiber Tensile damage variable result will be in the range of 0 to 0.6. A value of 0 for this result means undamaged and a value of 1 means completely damaged. The result has no units.
Matrix Tensile Damage Variable	The Matrix Tensile Damage Variable result value will be in the range of 0 to the "Tensile Matrix Stiffness Reduction" value set in the Damage Evolution Law, that is, if you set the Tensile Matrix Stiffness Reduction to 0.6, the range of Fiber Tensile damage variable result will be in the range of 0 to 0.6. A value of 0 for this result means undamaged and a value of 1 means completely damaged. The result has no units.
Matrix Compressive Damage Variable	The Matrix Compressive Damage Variable result value will be in the range of 0 to the "Compressive Fiber Stiffness Reduction" value set in the Damage Evolution Law, that is, if you set the Compressive Fiber Stiffness Reduction to 0.6, the range of Fiber Tensile damage variable result will be in the range of 0 to 0.6. A value of 0 for this result means undamaged and a value of 1 means completely damaged. The result has no units.
Shear Damage Variable	The Shear Damage Variable result value will be in the range of 0 to 1. This value is computed using the results of Fiber Tensile Damage Variable, Fiber Compressive Damage Variable, Matrix Tensile Damage Variable, and Matrix Compressive Damage Variable. The result has no units.
Energy Dissipated Per Volume	The Energy Dissipated Per Volume result value will be a positive real number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Result	Description
	This result uses a unit of "Energy/Volume" in the unit system you choose.

Physical Failure Criteria

The respective failure criteria are computed for the failure modes chosen in the damage initiation criteria. While the damage variables give you an idea where the damage is located and its likely direction of propagation, the Physical Failure Criteria helps you determine how much more load the material can handle.

These failure criteria are computed based on the parameters given using the material damage initiation (**TB**, DMGI) and evolution criteria (**TB**, DMGE). For more information, see Progressive Damage (p. 2038), above, as well as Damage-Evolution Law, Fiber-Reinforced Material Damage, and Physical Failure Criteria in the Mechanical APDL documentation.

The Physical Failure Criteria model supports the following results:

Result	Description
Max Failure Criteria	The Max Failure Criteria is computed based on the maximum of Fiber Tensile Failure Criterion, Fiber Compressive Failure Criterion, Matrix Tensile Failure Criterion, and Matrix Compressive Failure Criterion.
Fiber Tensile Failure Criterion	The Fiber Tensile Failure Criterion result value will be a positive integer. A value of 0 indicates no failure, while 1 is a complete failure. A value above 1 indicates the material has completely failed. The higher this number, the higher the load above the prescribed limits, although specifics are dependent on the stress limits you set and the amount of loading applied.
Fiber Compressive Failure Criterion	The Fiber Compressive Failure Criterion result value will be a positive integer. A value of 0 indicates no failure, while 1 is a complete failure. A value above 1 indicates the material has completely failed. The higher this number, the higher the load above the prescribed limits, although specifics are dependent on the stress limits you set and the amount of loading applied.
Matrix Tensile Failure Criterion	The Matrix Tensile Failure Criterion result value will be a positive integer. A value of 0 indicates no failure, while 1 is a complete failure. A value above 1 indicates the material has completely failed. The higher this number, the higher the load above the prescribed limits, although specifics are dependent on the stress limits you set and the amount of loading applied.
Matrix Compressive Failure Criterion	The Matrix Compressive Failure Criterion result value will be a positive integer. A value of 0 indicates no failure, while 1 is a complete failure. A value above 1 indicates the material has completely failed. The higher this number, the higher the load above the prescribed limits, although specifics are dependent on the stress limits you set and the amount of loading applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Linearized Stress

The Linearized Stress results calculate membrane, bending, peak, and total stress along a straight line path in the Mechanical application. To calculate linearized stress, you must first define a straight line path object using **Construction Geometry** under **Model**. A path you define for linearized stress can be of type **Two Points** or of type **X axis Intersection** and should have at least 47 sample points. The number of points must be an odd number: otherwise the result will not solve and an error message will be issued. The path must be straight and entirely within the model's elements. The **X axis Intersection** option is recommend as it ensures that the start and end points are inside the mesh and that the path is straight. Note that the **Two Points** method obtains the points from the tessellation of the geometric model, and if the geometry faces are curved, the points might not be inside the mesh. For these situations, you can use the **Snap to mesh nodes** feature (see Path (p. 883)) to ensure that the two points are contained within the mesh.

Linearized stress does not support the **Edge** path type. To calculate linearized stresses:

1. In the object tree, select the **Solution** object.
2. On the **Solution** Context Tab (p. 62), open the **Linearized Stress** drop-down menu and select your desired stress.
3. In the Details, select the **Path** (p. 883) you have defined to calculate the linearized stress.
4. Select the coordinate system you have used for the model.
5. As desired, for 3D analyses (only), set the **Zero Through-Thickness Bending Stress** property to **Yes** to ignore out-of-plane bending stresses (SX, SXY, SXZ) in the linearized bending stress calculations.
6. Click **Solve** to calculate linearized stress along the path.

Geometry	Select bodies that contribute toward stress calculation
Path	The path you define to calculate the linearized stresses
Type	Types of linearized stresses available
Coordinate System	Coordinate systems you can select for stress calculation

About Linearized Stress

When the result is evaluated, component stress values at the path points are interpolated from the appropriate element's average corner nodal values. Stress components through the section are linearized by a line integral method and are separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the actual (total) stress and the membrane plus bending combination).

The Details shows **Membrane**, **Bending**, **Membrane + Bending**, **Peak**, and **Total** stresses. The bending stresses are calculated such that the neutral axis is at the midpoint of the path.

Principal stresses are recalculated from the component stresses and are invariant with the coordinate system as long as stress is in the same direction at all points along the defined path. It is generally recommended that calculations be performed in a rectangular coordinate system (global Cartesian).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The Details also includes the following three choices for the **2D Behavior** (2D analysis (p. 878) only) property: **Planar**, **Axisymmetric Straight**, and **Axisymmetric Curve**. These choices are available only for 2D geometries (for example, plane stress).

For **Axisymmetric Straight** and **Axisymmetric Curve**, the Details includes entries for **Average Radius of Curvature** and **Through-Thickness Bending Stress**.

The **Average Radius of Curvature** represents the in-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If the radius is zero, a plane or 3D structure is assumed. The curve radius is in the current units.

An **Axisymmetric Straight** analysis always has an infinite radius of curvature (which is denoted by a value of -1).

The choices for **Through-Thickness Bending Stress** are:

- **Include**: Include the thickness-direction bending stresses.
- **Ignore**: Ignore the thickness-direction bending stresses.
- **Include Using Y Dir. Formula**: Include the thickness-direction bending stress using the same formula as the Y (axial direction) bending stress. Also use the same formula for the shear stress.

If the **Average Radius of Curvature** is non-zero, Mechanical reports the linearized stresses in the section coordinates (SX – along the path, SY – normal to the path, and SZ – hoop direction). In this case, the choice of **Coordinate System** in the Details is ignored.

If the **Average Radius of Curvature** is zero, Mechanical reports the linearized stresses in the active results coordinate system.

For 3D geometries only, the property **Zero Through-Thickness Bending Stress** is displayed. It includes the following options:

- **No** (default): The application performs calculations as in previous revisions.
- **Yes**: When this option is selected, the bending stresses SX, SXY, SXZ are set to zero and the principal bending stress calculation for S1, S2, S3, SINT, and SEQV are performed using these zeroed components. The direction of the positive X-axis of the result's Coordinate System must be the same as the direction of the specified Path (Start to End).

Notes on Linearized Stress

- The line integral method is the same as that used in the Mechanical APDL command **PRSECT**, RHO, KBR, and KBR3D.
- Mechanical does not support the **Solution Coordinate System** for this result.
- The Worksheet reports the linearized component and principal stresses for each stress category at the beginning, mid-length, and end of the section path.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Contact Results

If your model contains **Contact Regions**, you can define the contact results listed below under the **Solution** object by inserting a **Contact Tool** (p. 2079). See the Reviewing the Results section of the *Contact Technology Guide* for additional information.

- **Gap**
- **Penetration**
- **Pressure**

Important:

The application only reports contact pressure results for corner nodes. However, the possibility exists that the contact element has contact with at least one mid-side node while the corner nodes are not in contact. As a result, the element may have a closed contact status even though the reported contact pressure is zero. To verify the contact status for contact elements in this situation, list the following **ETABLE** quantities: SMISC,13 (PRES): NMISC,41 (STAT): NMISC,43 (CNFX): NMISC,44 (CNFY): and NIMSC,45 (CNFZ). Also note you can specify these quantities, such as **SMISC,13**, in a User Defined (p. 2194) result using an appropriate Expression (SMISC13).

- **Frictional Stress**: Available only for evaluating contact conditions after solution.

Note:

– To reflect total contact pressures or frictional stress, you must either set the **Behavior** (p. 1236) option to **Asymmetric** or **Auto Asymmetric**, or manually create an asymmetric contact pair (p. 1266).

– For node-to-surface contact, **Pressure** will display zero results. To display the associated contact force, you must insert a user defined result (p. 2203) called CONTFORC. This is also the case for the General Axisymmetric feature, which employs node-to-surface contact.

- **Sliding Distance**: Available only for evaluating contact conditions after solution. The total sliding distance (SLIDE) is the maximum total sliding distance (algebraic sum) when the contact status is sticking or sliding (STAT = 2, 3). It contains contributions from the elastic slip and the frictional slip. Elastic slip due to sticking represents the reversible tangential motion from the point of zero tangential stresses. Ideally, the equivalent elastic slip does not exceed the user-defined absolute limit. The higher the tangent stiffness, the smaller the resulting elastic slip. The pair-based elastic slip can be monitored using the Contact Result Tracker (p. 1848).
- **Fluid Pressure**: Fluid penetration pressure (surface-to-surface contact only). Note that command snippets are required to apply the loading to create this result. For more information, see Applying Fluid-Pressure-Penetration Loads in the *Contact Technology Guide*.
- **Status**. Status codes include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **-3** - MPC bonded contact.
- **-2** - MPC no-separation contact.
- **0** - open and not near contact.
- **1** - open but near contact.
- **2** - closed and sliding.
- **3** - closed and sticking.

The labels **Far**, **Near**, **Sliding**, and **Sticking** are included in the legend for Status.

Note:

- MPC-based contact definitions use negative values. They indicate the intentional removal of one or more contact constraints to prevent over-constraint.
- Contact that has been deactivated via **Auto Asymmetric** behavior will be displayed with a status of **Far-Open**. Results for deactivated pairs can be suppressed in the **Contact Tool** by changing **Both** to either **Contact** or **Target** as necessary.

If you choose to display contact results with a display option other than `Unaveraged`, then Mechanical uses all elements in the selected regions to calculate the result. That is, Mechanical averages contact across regions regardless of whether you scoped the result via Geometry Selection or via the Worksheet.

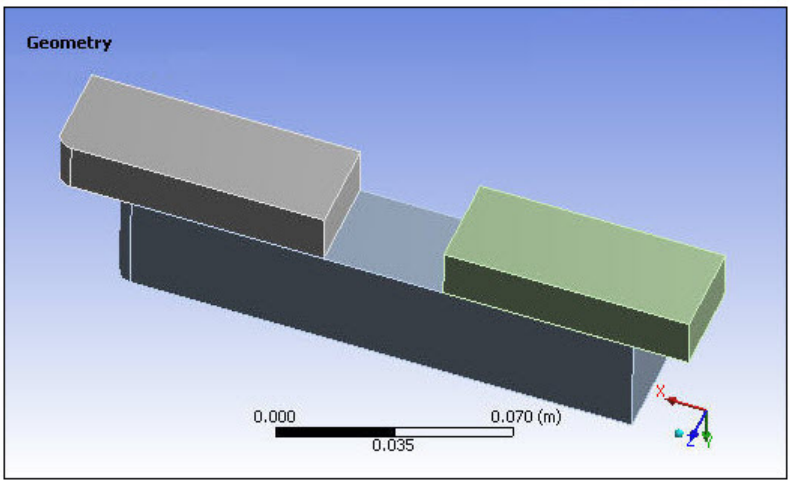
For example, if you set the display option to `Averaged`, then the displayed result for a node is the average of all values (from all selected elements) at that node. Contact elements can be coincident, which may be difficult to discern visually, and Mechanical does not display unaveraged contact results if it detects coincident elements in the scoping. However, Mechanical calculates and displays averaged contact results for coincident elements.

In addition, if more than one face on a non-contact element (such as a solid element) includes contact elements, Mechanical does not display unaveraged contact results because the application cannot assign multiple contact values to a node.

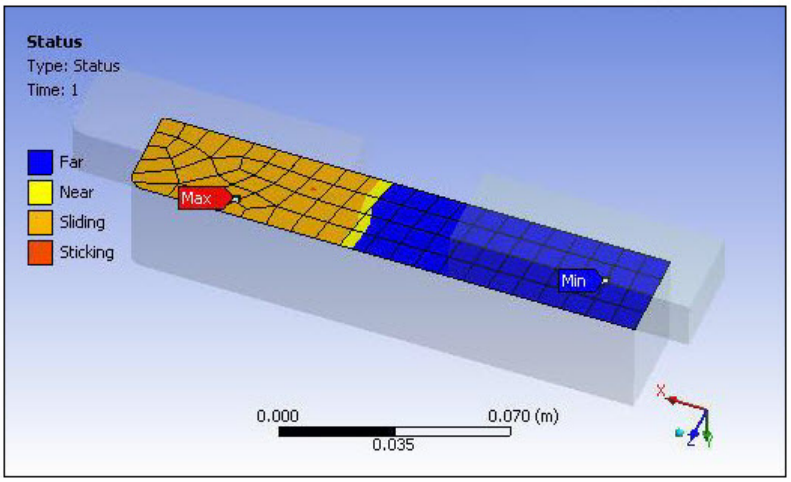
The images below illustrate how contact results are affected by the different scoping types. The model consists of two blocks contacting a third block.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

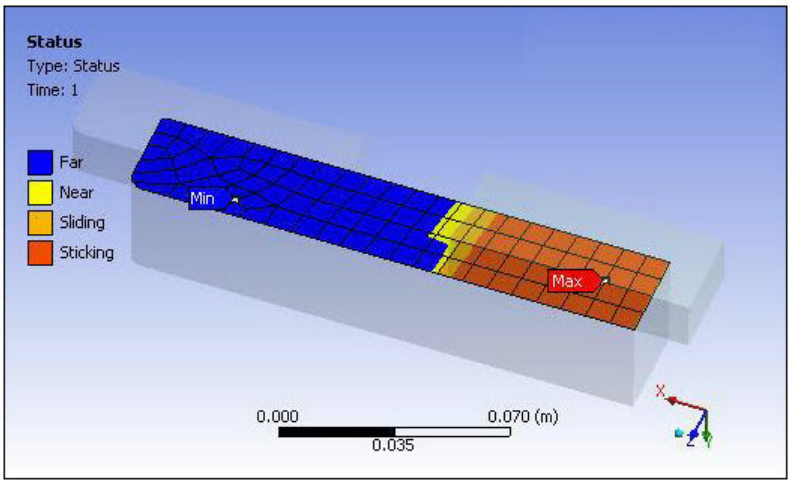
Published: 2023-10-23



Using the Worksheet method, one **Contact Tool** was scoped to the contact pair on the left, and another one was scoped to the contact pair on the right. This allows you to view the contact results for each contact pair individually. The contact status for the contact pair on the left is shown below.

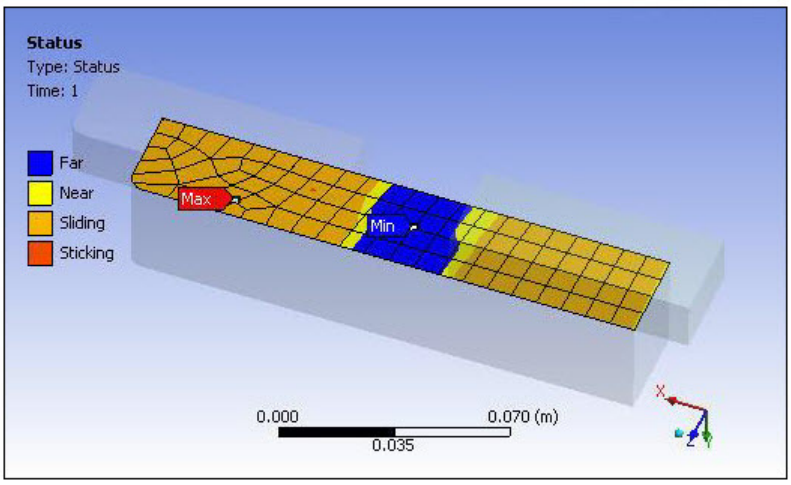


The contact status for the contact pair on the right is shown below.



A third Contact Tool scoped to the surface of the large block (using the Geometry Selection method) enables you to view the contact status averaged over that surface, as shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Contact Results Restrictions

Note the following restrictions regarding contact results:

- When a contact result is scoped to a face of an assembly, a contact result may not be obtained in certain cases, especially if the scoped face is not a part of any contact region.
- Contour contact results are not reported for 3D edge contact.
- If you use the **Generate Initial Contact Results** option on a **Contact Tool** contained in the **Connections** folder, and use the Worksheet to scope these Initial Contact results, the application displays the contour results for the geometry scoping of the contact pairs, not the individual contact elements. Therefore, and as illustrated above by the third Contact Tool scoped to a surface, the results appear as if the scoping is to the geometric faces.
- If the contact status of all nodes of a contact element is FAR (or 0), then 16.0 will report **all** contact results as undefined.
- For any contact result, the averaged contact results at 16.0 can display values that differ from values of previous revisions, if:
  - (a) two contact elements share nodes, and,
  - (b) one of the elements has a FAR contact status, and,
  - (c) one of the elements does **not** have a FAR status.

  This is because, prior to 16.0, the elements with FAR status reported values that were ZERO. At 16.0, these elements have **no** values and are **not** involved in the averaging process.

Frequency Response

Frequency response results enable you to plot graphs that display how the given response (stress, strain, etc.) varies with frequency.

Go to a section topic:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Result Options (p. 2047)
- Scoping (p. 2048)
- Limitations (p. 2048)
- Result Calculations (p. 2049)
- Display Options (p. 2051)
- Creating Contour Result from Frequency Response Results (p. 2053)

Result Options

The **Frequency Response** menu of options is accessed from the Solution object context tab, or by right-clicking the **Solution** object, or an existing result object, and selecting **Insert** > **Frequency Response**. Menu options include:

- **Stress**
- **Strain** (elastic)
- **Deformation**
- **Velocity**
- **Acceleration** (frequency only)
- **Force Reaction**
- **Equivalent Radiated Power**
- **Equivalent Radiated Power Level**

You can view these options as a value graphed along a specified frequency range. In addition, you can parameterize these frequency results. The plot will include all the frequency points at which a solution was obtained. When you generate frequency response results, the default plot (Bode) shows the amplitude and phase angle.

Equivalent radiated power results estimate the radiated structure-borne sound power from a vibrating structural surface. These results are available in 3D Harmonic Response (p. 496) and Harmonic Acoustics (if you define a structural Physics Region) analyses. The result is evaluated on faces that you select, either using face geometry section or face-based Named Selections, and the result is displayed on a graph over the specified frequency range. You specify the frequency range using the **Specified** setting for the **Frequency Range** property. When you generate frequency response results, the plot shows **Equivalent Radiated Power** (**ERP**) or **Equivalent Radiated Power Level** (**ERPL**) as a function of the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

frequency. See the expressions described below as well as the **PLAS** command for the **ERP** and **ERPL** acoustic quantities.

Note:

You can specify a weighting filter (p. 1977) (A, B, and C) for the **Equivalent Radiated Power Level** result.

Important:

For **Equivalent Radiated Power** and **Equivalent Radiated Power Level** results, if the **Multiple Step Type** (p. 1426) property is set to **RPM** in the **Analysis Settings**, the result file must exist for the result to be displayed.

Scoping

These results support the following scoping options:

- Geometry selection (vertex, face, edge, or nodes)
- Named Selections (p. 1057)
- Contact Regions
- Springs (p. 1395)
- Beam Connections (p. 1404)
- Support Boundary Conditions (p. 1663) (Fixed Support, Displacement, Frictionless Support, Cylindrical Support)
- Remote Points (p. 1089)

Note:

Equivalent Radiated Power and **Equivalent Radiated Power Level** results must be scoped to surfaces.

Note:

Direct graphical node selection requires you to generate the mesh and have the **Node selection option** (p. 97) active.

Limitations

For a Mode Superposition Harmonic Response analysis (standalone or linked), if the **Store Results At All Frequencies** property in the **Options** category (p. 1447) of the **Analysis Settings** is set to **No**, the Frequency Response results can be scoped to geometric entities *only* (node-based scoping is not allowed).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If the **Store Results At All Frequencies** property in the **Options** category (p. 1447) of the **Analysis Settings** is set to **No**, the Frequency Response results for force reactions cannot be extracted.

Result Calculations

The following equations describe how frequency graphs are defined and plotted for the supported types.

Stress and Strain Results

The strain result is calculated using the displacement result. Using the Young's Modulus and strain result, the stress result can be evaluated. Because of this reason, the stress and strain results are in phase with the displacement result.

Deformation Result

The displacement vector on a structure subjected to harmonic loading may be expressed as:

EQUATION 1

$$\{u(t)\} = \{u_{\max}\} e^{i\phi} e^{i\Omega t}$$

The Frequency Response chart for displacement is calculated by expressing Equation (1) in time domain as follows:

EQUATION 2

$$\{u(t)\} = \{u_{real}\}\cos(\Omega t) - \{u_{imag}\}\sin(\Omega t)$$

where:

$u_{real} = u_{\max}\cos\phi$

$u_{imag} = u_{\max}\sin\phi$

$u_{\max} = \sqrt{u_{real}^2 + u_{imag}^2}$

$\phi\,(\text{in degrees}) = \tan^{-1}\dfrac{u_{imag}}{u_{real}}$

Velocity Result

The equation for velocity $\dot{u}$ can be obtained by taking a time derivative of Equation (1). The frequency response for velocity in time domain is calculated as follows:

EQUATION 3

$$\{\dot{u}(t)\} = \{\dot{u}_{real}\}\cos(\Omega t) - \{\dot{u}_{imag}\}\sin(\Omega t)$$

where:

$\dot{u}_{real} = -\Omega \cdot u_{imag}$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\dot{u}_{imag} = -\Omega \cdot u_{real}$$

$$\dot{u}_{\max} = \sqrt{\dot{u}_{real}^2 + \dot{u}_{imag}^2}$$

$$\dot{\phi}\left(\text{in degrees}\right) = \tan^{-1}\frac{\dot{u}_{imag}}{\dot{u}_{real}}$$

Acceleration Result

The equation for acceleration $\ddot{u}$ can be obtained by taking a double time derivative of Equation (1). The frequency response for acceleration in time domain is calculated as follows:

EQUATION 4

$$\ddot{u}(t) = \{\ddot{u}_{real}\}\cos(\Omega t) - \{\ddot{u}_{imag}\}\sin(\Omega t)$$

where:

$$\ddot{u}_{real} = \Omega^2 \cdot u_{real}$$

$$\ddot{u}_{imag} = \Omega^2 \cdot u_{imag}$$

$$\ddot{u}_{\max} = \sqrt{\ddot{u}_{real}^2 + \ddot{u}_{imag}^2}$$

$$\ddot{\phi}\left(\text{in degrees}\right) = \tan^{-1}\frac{\ddot{u}_{imag}}{\ddot{u}_{real}}$$

Force Reaction

The Frequency Response for Force Reaction is calculated by replacing displacement with force in Equation (2) as shown below.

EQUATION 5

$$\{f(t)\} = \{f_{real}\}\cos(\Omega t) - \{f_{img}\}\sin(\Omega t)$$

where:

$$f_{real} = f_{\max}\cos\text{Ø}$$

$$f_{img} = f_{\max}\sin\text{Ø}$$

$$f_{\max} = \sqrt{\left(f_{real}^2 + f_{imag}^2\right)}$$ (Amplitude)

$$\text{Ø}\left(\text{in degrees}\right) = \tan^{-1}\left(f_{imag}/f_{real}\right)$$ (Phase Angle)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Equivalent Radiated Power

Equivalent Radiated Power (ERP) is expressed as:

$$ERP=\frac{1}{2}\rho_0 c\sigma(f)\iint_s|v_n|^2 ds$$

Where:

c = speed of sound (equal to 343.25 m/s).

ρ_0 = mass density (equal to 1.2041 kg/m3).

σ = radiation factor (equal to 1).

v_n = normal velocity of vibrating structural surface

Equivalent Radiated Power Level

The **Equivalent Radiated Power Level** (ERPL) is expressed as:

ERPL = 10 Log (ERP/Wref)

Where Wref is the reference sound power (equal to 10-12W).

Display Options

Display Property

The property of the frequency response results provides the following results values for graphs:

- **Real**
- **Imaginary**
- **Real and Imaginary**
- **Amplitude**
- **Phase Angle**
- **Bode** (default setting - plots both Amplitude and Phase Angle)

Chart Viewing Style Property

The **Chart Viewing Style** property provides the following options to plot results for a scale of an axis:

- **Linear**: this option plots the result values linearly.
- **Log X**: this option plots the X-Axis logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.
- **Log Y** (default when graph has Amplitude): this option plots the Y-Axis is plotted logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Log-Log**: this option plots the X-Axis and Y-Axis logarithmically. If negative axis values or a zero value exists, this option is not supported and the graph plots linearly.

For edges, faces, surface bodies, and multiple vertex selections (which contain multiple nodes), the results can be scoped as minimum, maximum, or average using the **Spatial Resolution** option. This option is also available for frequency and phase response results scoped on a single vertex.

Note:

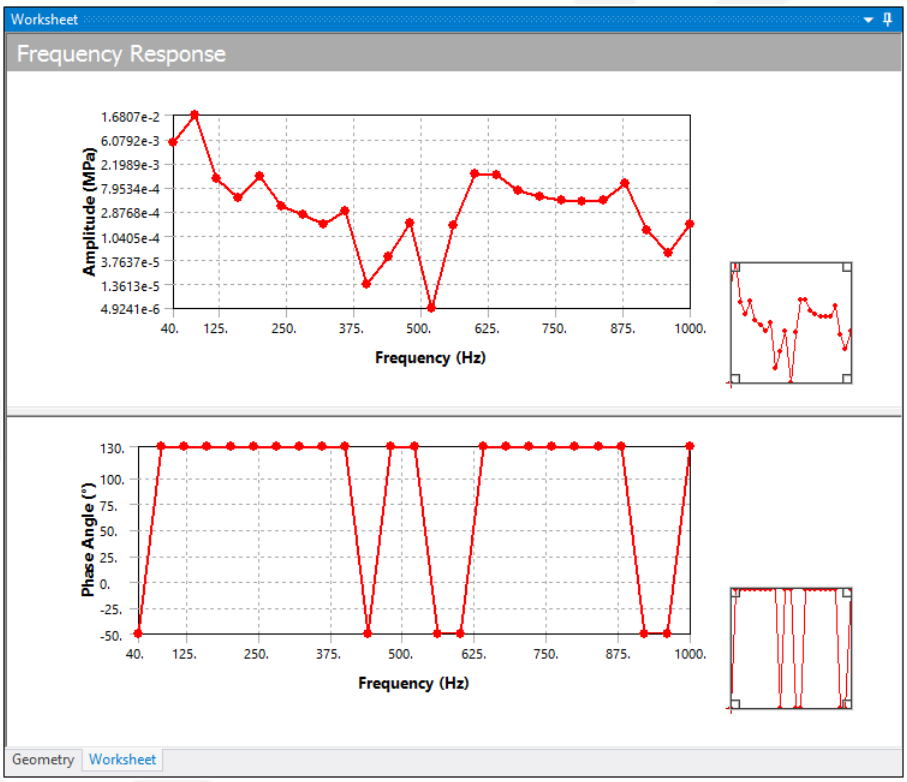
The **Spatial Resolution** option is especially important for results scoped to a shell vertex, where the default option, **Use Average**, may yield unexpected results.

The **Use Minimum** and **Use Maximum** settings of the **Spatial Resolution** option are based on the amplitude and are therefore reported from the location with either the largest or smallest amplitude. The **Use Average** setting calculates the average by calculating the real and imaginary components separately.

Note:

You cannot use the Mechanical application convergence capabilities for any results item under a harmonic analysis. Instead, you can first do a convergence (p. 1852) study on a modal analysis (p. 515) and reuse the mesh from that analysis.

Presented below is an example of a **Frequency Response** plot:



The average, minimum, or maximum value can be chosen for selected entities. **Stress**, **Strain**, **Deformation**, **Velocity**, and **Acceleration** components vary sinusoidally, so these are the only

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

result types that can be reviewed in this manner. (Note that items such as **Principal Stress** or **Equivalent Stress** do not behave in a sinusoidal manner since these are derived quantities.)

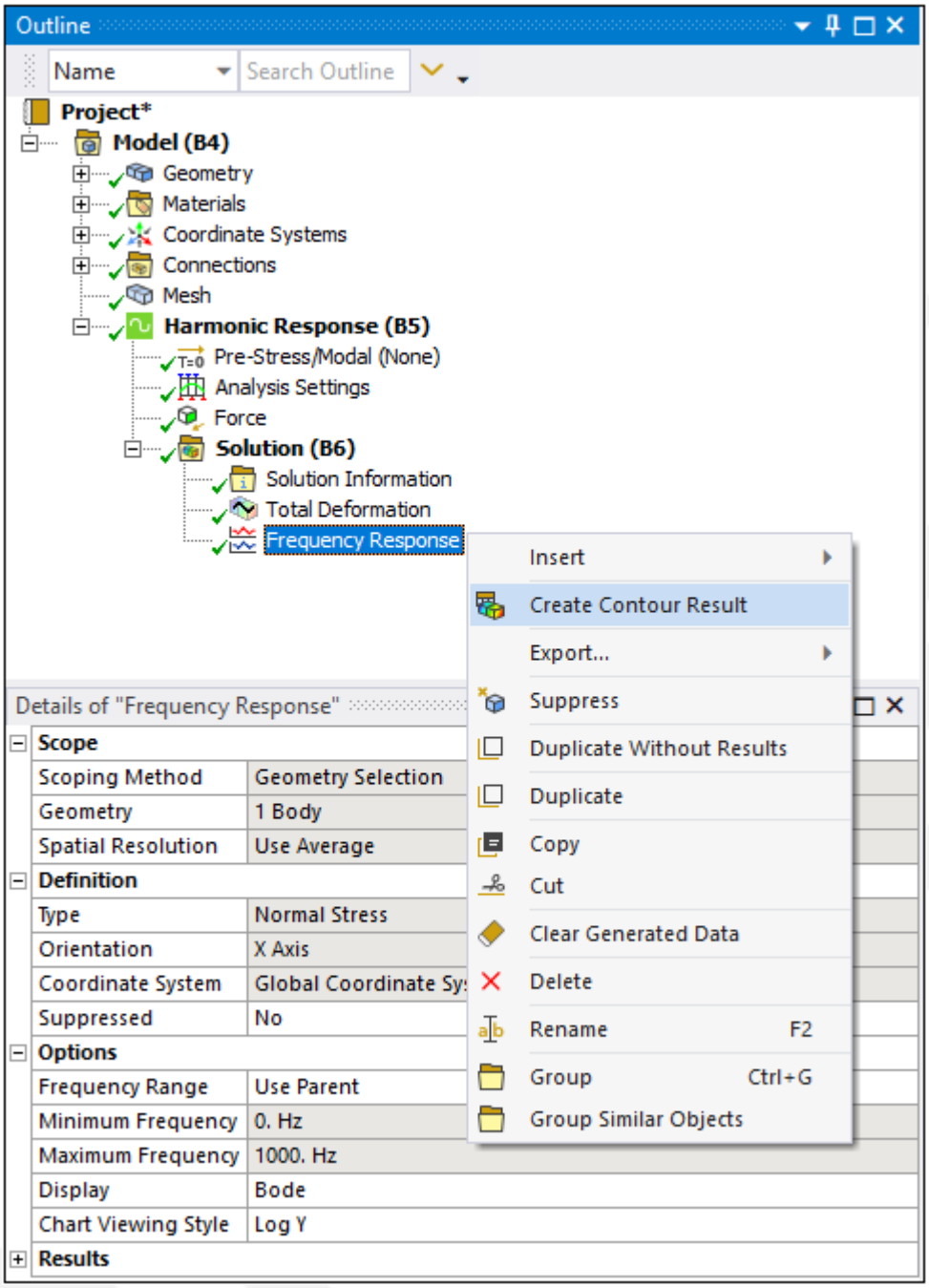
Creating Contour Result from Frequency Response Results

You can use **Frequency Response** result types (not including Velocity and Acceleration) to generate new result objects of the same type, orientation, frequency. The phase angle of the contour result will be opposite in sign with the same magnitude as the frequency response result type. The sign of the phase in the **Sweeping Phase** property of the contour result is reversed so that the response amplitude of the frequency response plot for that frequency and phase defined by the **Duration** property matches with the contour results. To create a Contour Result in a Harmonic Analysis:

1. Select and right-click the desired Harmonic result in the solution tree.
2. Choose **Create Contour Result**.

DRAFT

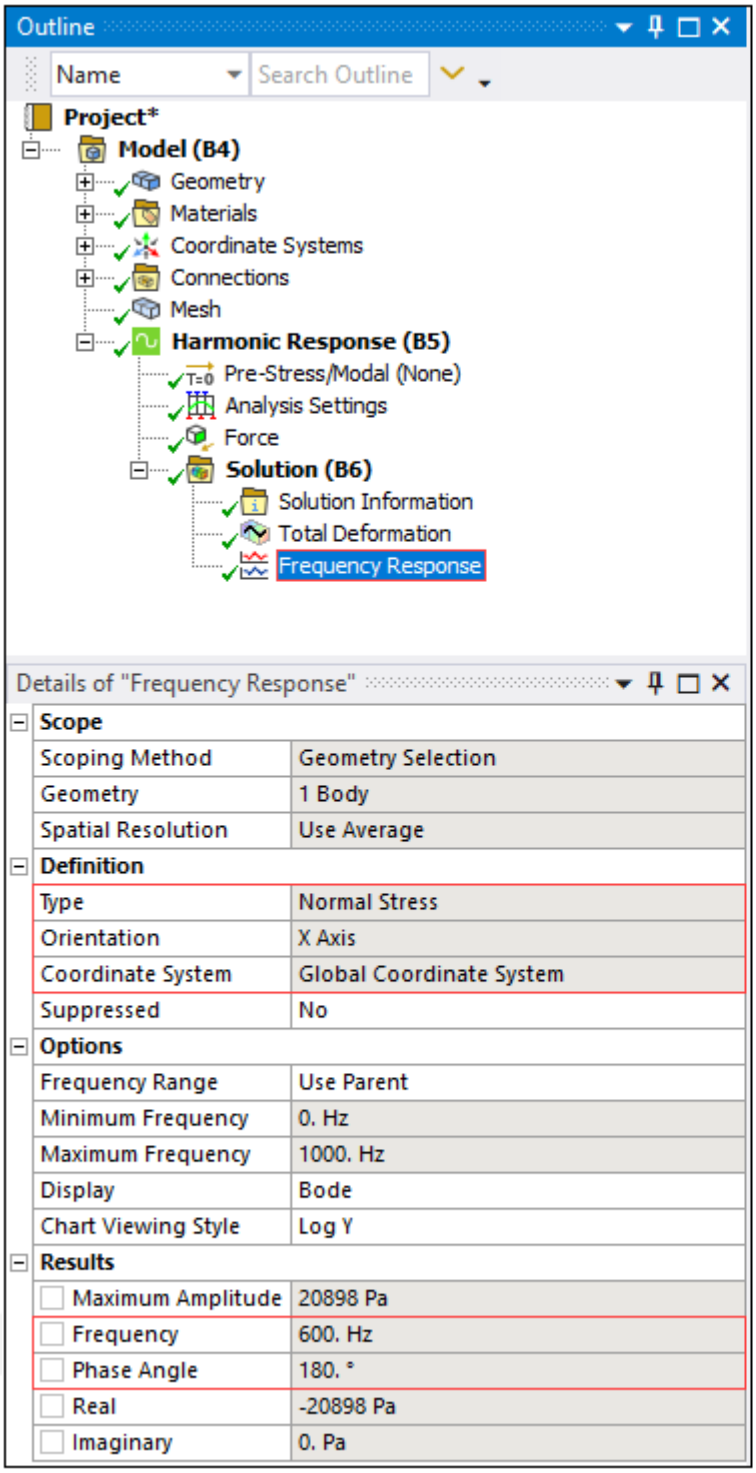
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

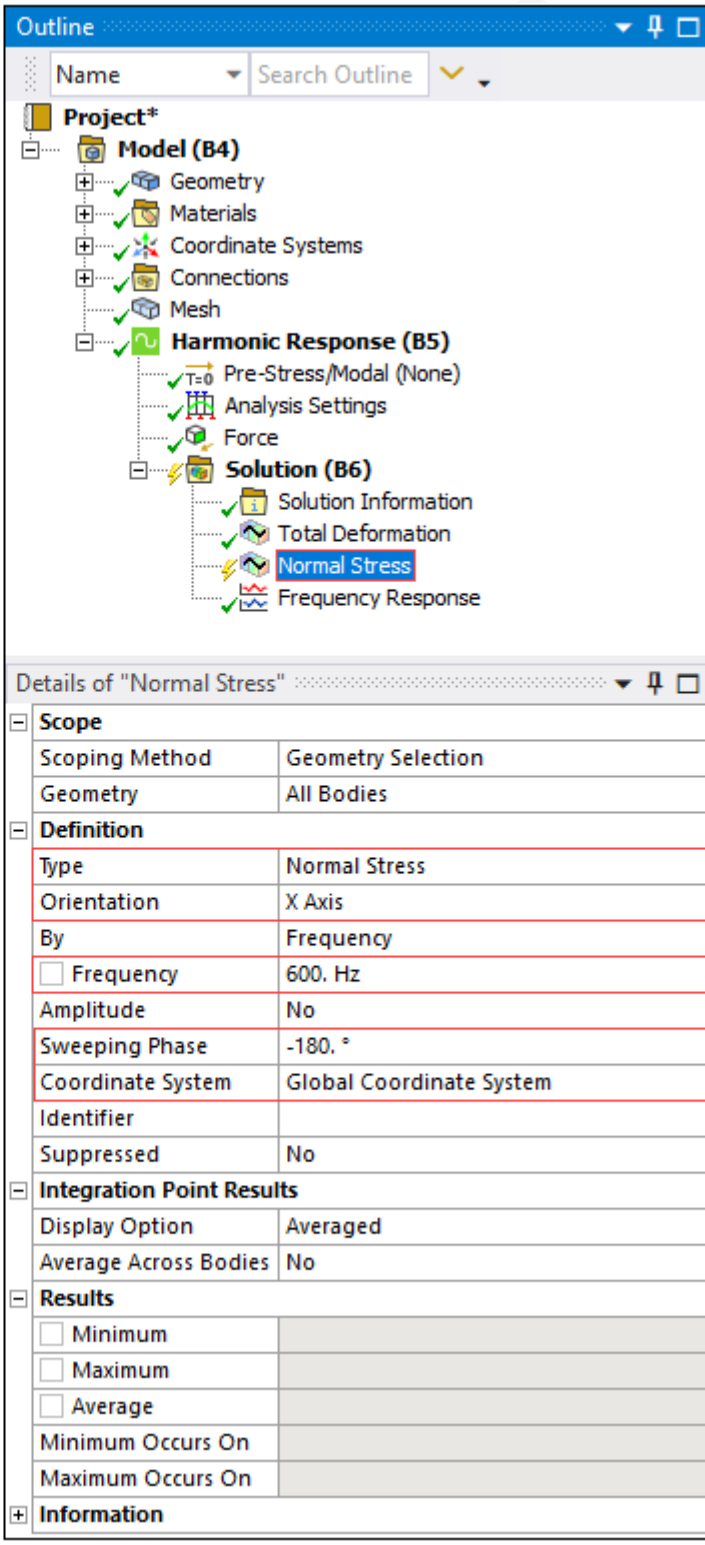


As illustrated here, you can see how the feature automatically scopes the **Type**, **Orientation**, **Coordinate System**, **Frequency**, and **Sweeping Phase**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



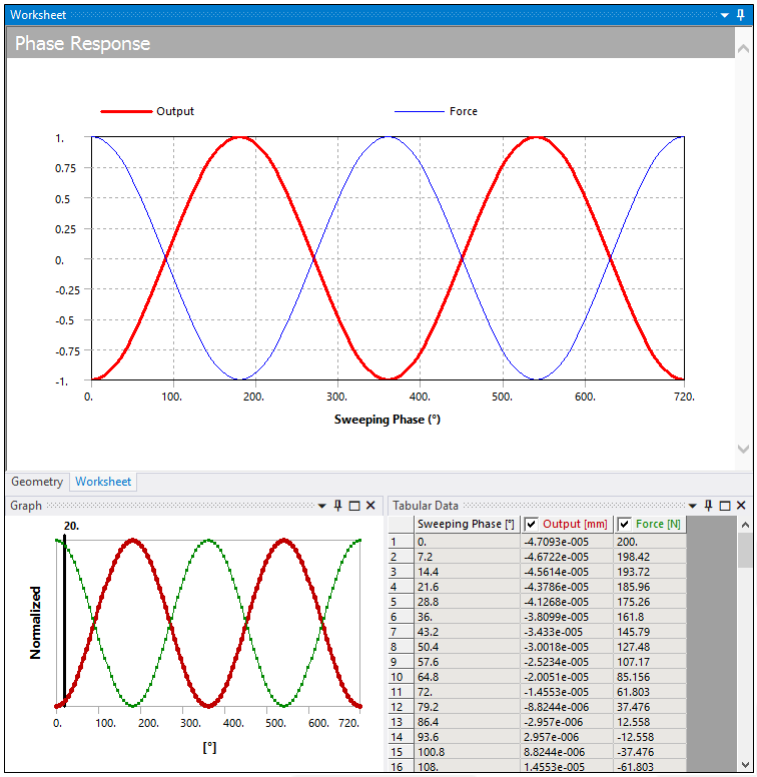


The **Reported Frequency** in the **Information** category is the frequency at which contour results were found and plotted. This frequency can be potentially different from the frequency you requested.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Phase Response

Like **Frequency Response** results, **Phase Response** results plot the minimum, average, or maximum **Stress**, **Strain**, or **Deformation** for selected graphical entities (vertex, face, edge, or nodes) or a Named Selection (p. 1057). An example of a **Phase Response** plot is illustrated below.



However, unlike **Frequency Response** plots that show a response amplitude over a frequency range, **Phase Response** plots show a response over a phase period, so you can determine how much a response lags behind the applied load.

The following functions outline the relationships of response amplitude, phase response graphs, and result contours (with associated caveats):

Response Amplitude

Response Amplitude is defined as the maximum value of the following expression:

```
value = sqrt(real*real + imag*imag)
```

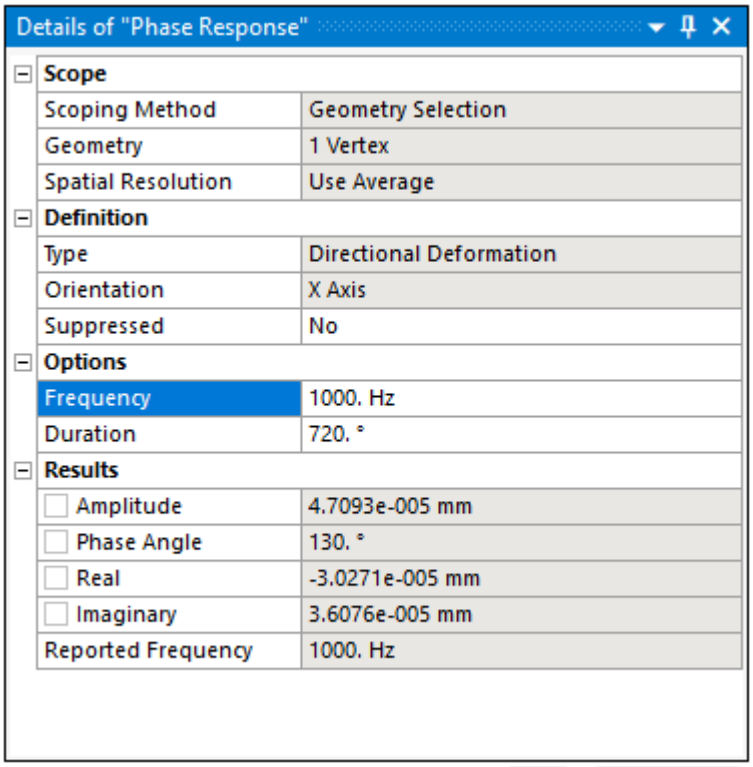
Where *real* and *imag* represent all real and imaginary result values from the result file for the selected frequency.

Phase Response Graph

Phase Response results report the following quantities at the nearest frequency requested from the **Frequency** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



For the properties contained in the **Results** category:

Amplitude = `sqrt(real*real + imag*imag)`

Phase Angle = $\tan^{-1}\frac{imag}{real}$

Real = real

Imaginary = imag

And the graph is the image of the following, where Sweeping Phase is allowed to vary across a user specified **Duration**:

```
value = Amplitude * cos( phase)
```

Where:

phase = (Phase Angle + Sweeping Phase)

Note:

Take caution when comparing the values in the **Output** column of the **Tabular Data** for a Phase Response against maximum values of contour displays.

Result Contour

Drawing contour displays in a Harmonic Response analysis, Mechanical uses the phase specified by the Sweeping Phase property defined by the user to evaluate the expression:

```
value = real*cos(phase) - imag*sin(phase)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Where *real* and *imag* represent all real and imaginary result values from the result file for the selected frequency.

Because the formula for the Phase Response graph differs from the formula for the contour, an **Output** value for the graph does not necessarily equal a maximum for a contour result at the same frequency.

General Approach to Harmonic Response Analysis Postprocessing

Generally speaking, you would look at **Frequency Response** plots at critical regions to ascertain what the frequency of interest may be. In conjunction with **Phase Response** plots, the phase of interest is also determined. Then, you can request **Stress**, **Strain**, or **Deformation** contour plots to evaluate the response of the entire structure at that frequency and phase of interest.

Limitations

For a Mode Superposition Harmonic Response analysis (standalone or linked), if the **Store Results At All Frequencies** property in the **Options** category (p. 1447) of the **Analysis Settings** is set to **No**, the Phase Response results can be scoped to geometric entities *only* (node-based scoping is not allowed).

If the **Store Results At All Frequencies** property in the **Options** category (p. 1447) of the **Analysis Settings** is set to **No**, the Frequency Response results for force reactions cannot be extracted.

Stress Tools

You can insert any of the following stress tools in a **Solution** object by choosing **Stress Tool** in the **Toolbox** drop-down menu of the **Solution** Context Tab (p. 62) or by right-clicking on the **Solution** object and selecting **Insert** > **Stress Tool**:

- Maximum Equivalent Stress Safety Tool (p. 2059)
- Maximum Shear Stress Safety Tool (p. 2061)
- Mohr-Coulomb Stress Safety Tool (p. 2062)
- Maximum Tensile Stress Safety Tool (p. 2064)

Note:

After adding a **Stress Tool** object, you can change the specific stress tool using the **Theory** property.

The Stress Tools make use of the following material properties:

- Tensile Yield Strength
- Compressive Yield Strength
- Tensile Ultimate Strength
- Compressive Ultimate Strength

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Safety Tools in the Ansys Workbench Product

The Ansys Workbench product uses safety tools that are based on four different stress quantities:

1. Equivalent stress (σ_e).
2. Maximum tensile stress (σ_1).
3. Maximum shear stress (τ_{MAX})

   This uses Mohr's circle:

   $$\tau_{MAX}=\frac{\sigma_1-\sigma_3}{2}$$

   where:

   σ_1 and σ_3 = principal stresses.
4. Mohr-Coulomb stress

   This theory uses a stress limit based on

   $$\frac{\sigma_1}{\sigma_t^f}+\frac{\sigma_3}{\sigma_c^f}$$

   where:

   σ_t^f = input tensile stress limit

   σ_c^f = input compression stress limit

Maximum Equivalent Stress Safety Tool

The Maximum Equivalent Stress Safety tool is based on the maximum equivalent stress failure theory for ductile materials, also referred to as the von Mises-Hencky theory, octahedral shear stress theory, or maximum distortion (or shear) energy theory. Of the four failure theories supported by the Mechanical application, this theory is generally considered as the most appropriate for ductile materials such as aluminum, brass and steel.

The theory states that a particular combination of principal stresses causes failure if the maximum equivalent stress in a structure equals or exceeds a specific stress limit:

$$\sigma_e \geq S_{limit}$$

Expressing the theory as a design goal:

$$\frac{\sigma_e}{S_{limit}}<1$$

If failure is defined by material yielding, it follows that the design goal is to limit the maximum equivalent stress to be less than the yield strength of the material:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\frac{\sigma_e}{S_y} < 1$$

An alternate but less common definition states that fracturing occurs when the maximum equivalent stress reaches or exceeds the ultimate strength of the material:

$$\frac{\sigma_e}{S_u} < 1$$

Options

Define the stress limit in the Details under **Stress Limit Type**. Use either **Tensile Yield Per Material**, or **Tensile Ultimate Per Material**, or enter a **Custom Value**. By default, **Stress Limit Type** equals **Tensile Yield Per Material**.

Select one of the following **Stress Tool** results from the **Result** group of the **Stress Tool** tab or by right-clicking and selecting **Insert** > [result type]:

Safety Factor

$$F_s = \frac{S_{limit}}{\sigma_e}$$

Safety Margin

$$M_s = F_s - 1 = \frac{S_{limit}}{\sigma_e} - 1$$

Stress Ratio

$$\sigma_e^* = \frac{\sigma_e}{S_{limit}}$$

Notes

- The reliability of this failure theory depends on the accuracy of calculated results (p. 1852) and the representation of stress risers (peak stresses). Stress risers play an important role if, for example, yielding at local discontinuities (for example, notches, holes, fillets) and fatigue loading are of concern. If calculated results are suspect, consider the calculated stresses to be nominal stresses, and amplify the nominal stresses by an appropriate stress concentration factor K_t. Values for K_t are available in many strength of materials handbooks.
- If fatigue is not a concern, localized yielding will lead to a slight redistribution of stress, and no real failure will occur. According to J. E. Shigley (*Mechanical Engineering Design*, McGraw-Hill, 1973), "We conclude, then, that yielding in the vicinity of a stress riser is beneficial in improving the strength of a part and that stress-concentration factors need not be employed when the material is ductile and the loads are static."
- Alternatively, localized yielding is potentially important if the material is marginally ductile, or if low temperatures or other environmental conditions induce brittle behavior.
- Yielding of ductile materials may also be important if the yielding is widespread. For example, failure is most often declared if yielding occurs across a complete section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- The proper selection and use of a failure theory relies on your engineering judgment. Refer to engineering texts such as *Engineering Considerations of Stress, Strain, and Strength* by R. C. Juvinall (McGraw-Hill) and *Mechanical Engineering Design* by J. E. Shigley (McGraw-Hill) for in-depth discussions on the applied theories.

Maximum Shear Stress Safety Tool

The Maximum Shear Stress Safety tool is based on the maximum shear stress failure theory for ductile materials.

The theory states that a particular combination of principal stresses causes failure if the Maximum Shear (p. 2029) equals or exceeds a specific shear limit:

$$\tau_{\max} \geq f\, S_{limit}$$

Where the limit strength is generally the yield or ultimate strength of the material. That is, the shear strength of the material is typically defined as a fraction (f < 1) of the yield or ultimate strength:

$$M_s = F_s - 1 = \frac{f\, S_{limit}}{\tau_{\max}} - 1$$

In a strict application of the theory, f = 0.5. Expressing the theory as a design goal:

$$\frac{\tau_{\max}}{f\, S_{limit}} < 1$$

If failure is defined by material yielding, it follows that the design goal is to limit the shear stress to be less than a fraction of the yield strength of the material:

$$\frac{\tau_{\max}}{f\, S_y} < 1$$

An alternate but less common definition states that fracturing occurs when the shear stress reaches or exceeds a fraction of the ultimate strength of the material:

$$\frac{\tau_{\max}}{f\, S_u} < 1$$

Options

Define the stress limit in the Details under **Stress Limit Type**. Use either **Tensile Yield Per Material**, or **Tensile Ultimate Per Material**, or enter a **Custom Value**. By default, **Stress Limit Type** equals **Tensile Yield Per Material**.

Define coefficient f under **Factor** in the Details. By default, the coefficient f equals 0.5.

Select one of the following **Stress Tool** results from the **Result** group of the **Stress Tool** tab or by right-clicking and selecting **Insert** > [result type]:

Safety Factor

$$F_s = \frac{f\, S_{limit}}{\tau_{\max}}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Safety Margin

$$M_s = F_s - 1 = \frac{f\,S_{limit}}{\tau_{max}} - 1$$

Stress Ratio

$$\tau^*_{max} = \frac{\tau_{max}}{f\,S_{limit}}$$

Notes

- The reliability of this failure theory depends on the accuracy of calculated results and the representation of stress risers (peak stresses). Stress risers play an important role if, for example, yielding at local discontinuities (notches, holes, fillets, etc.) and fatigue loading are of concern. If calculated results are suspect, consider the calculated stresses to be nominal stresses, and amplify the nominal stresses by an appropriate stress concentration factor K_t. Values for K_t are available in many strength of materials handbooks.
- If fatigue is not a concern, localized yielding will lead to a slight redistribution of stress, and no real failure will occur. According to J. E. Shigley (*Mechanical Engineering Design*, McGraw-Hill, 1973), "We conclude, then, that yielding in the vicinity of a stress riser is beneficial in improving the strength of the part and that stress-concentration factors need not be employed when the material is ductile and the loads are static."
- Alternatively, localized yielding is potentially important if the material is marginally ductile, or if low temperatures or other environmental conditions induce brittle behavior.
- Yielding of ductile materials may also be important if the yielding is widespread. For example, failure is most often declared if yielding occurs across a complete section.
- The proper selection and use of a failure theory relies on your engineering judgment. Refer to engineering texts such as *Engineering Considerations of Stress, Strain, and Strength* by R. C. Juvinall (McGraw-Hill) and *Mechanical Engineering Design* by J. E. Shigley (McGraw-Hill) for in-depth discussions on the applied theories.

Mohr-Coulomb Stress Safety Tool

The Mohr-Coulomb Stress Safety Tool is based on the Mohr-Coulomb theory for brittle materials, also known as the internal friction theory.

The theory states that failure occurs when the combination of the Maximum, Middle, and Minimum Principal (p. 2028) equal or exceed their respective stress limits. The theory compares the maximum tensile stress to the material's tensile limit and the minimum compressive stress to the material's compressive limit. Expressing the theory as a design goal:

$$\frac{\sigma_1}{S_{tensile\,limit}} + \frac{\sigma_3}{S_{compressive\,limit}} < 1$$

where $\sigma_1 > \sigma_2 > \sigma_3$: σ_3 and the compressive strength limit assume negative values even though you must actually enter *positive* values for these quantities. Also, a given term is only used if it in-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

cludes the correct sign. For example, σ_1 must be positive and σ_3 must be negative. Otherwise, the invalid term is assumed to be negligible.

Note that the Mohr-Coulomb Stress Safety tool evaluates maximum and minimum principal stresses at the same locations. That is, this tool does not base its calculations on the absolute maximum principal stress and the absolute minimum principal stress occurring (most likely) at two different locations in the body. The tool bases its calculations on the independent distributions of maximum and minimum principal stress. Consequently, this tool provides a distribution of factor or margin of safety throughout the part or assembly. The minimum factor or margin of safety is the minimum value found in this distribution.

For common brittle materials such as glass, cast iron, concrete and certain types of hardened steels, the compressive strength is usually much greater than the tensile strength, of which this theory takes direct account.

The design goal is to limit the maximum and minimum principal stresses to their ultimate strength values by means of the brittle failure relationship:

$$\frac{\sigma_1}{S_{ut}} + \frac{\sigma_3}{S_{uc}} < 1$$

An alternative but less common definition compares the greatest principal stresses to the yield strengths of the material:

$$\frac{\sigma_1}{S_{yt}} + \frac{\sigma_3}{S_{yc}} < 1$$

The theory is known to be more accurate than the maximum tensile stress failure theory used in the Maximum Tensile Stress Safety tool, and when properly applied with a reasonable factor of safety the theory is often considered to be conservative.

Options

Define the tensile stress limit in the Details under **Tensile Limit Type**. Use either **Tensile Yield Per Material**, or **Tensile Ultimate Per Material**, or enter a **Custom Value**. By default, **Tensile Limit Type** equals **Tensile Yield Per Material**.

Define the compressive stress limit in the Details under **Compressive Limit Type**. Use either **Comp. Yield Per Material**, or **Comp. Ultimate Per Material**, or enter a **Custom Value**. By default, **Compressive Limit Type** equals **Comp. Yield Per Material**.

Select one of the following **Stress Tool** results from the **Result** group of the **Stress Tool** tab or by right-clicking and selecting **Insert** > [result type]:

Safety Factor

$$F_s = \left[\frac{\sigma_1}{S_{tensile\,limit}} + \frac{\sigma_3}{S_{compressive\,limit}} \right]^{-1}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Safety Margin

$$M_s = F_s - 1 = \left[\frac{\sigma_1}{S_{tensile\,limit}} + \frac{\sigma_3}{S_{compressive\,limit}} \right]^{-1} - 1$$

Stress Ratio

$$\sigma^* = \frac{\sigma_1}{S_{tensile\,limit}} + \frac{\sigma_3}{S_{compressive\,limit}}$$

Notes

- The use of a yield strength limit with brittle materials is not recommended since most brittle materials do not exhibit a well-defined yield strength.
- For ductile and some other types of materials, experiments have shown that brittle failure theories may be inaccurate and unsafe to use. The brittle failure theories may also be inaccurate for certain brittle materials. Potential inaccuracies are of particular concern if the accuracy of calculated answers (p. 1852) is suspect.
- The reliability of this failure criterion is directly related to treatment of stress risers (peak stresses). For brittle homogeneous materials such as glass, stress risers are very important, and it follows that the calculated stresses should have the highest possible accuracy or significant factors of safety should be expected or employed. If the calculated results are suspect, consider the calculated stresses to be nominal stresses, and amplify the nominal stresses by an appropriate stress concentration factor K_t. Values for K_t are available in many strength of materials handbooks. For brittle nonhomogeneous materials such as gray cast iron, stress risers may be of minimal importance.
- If a part or structure is known or suspected to contain cracks, flaws, or is designed with sharp notches or re-entrant corners, a more advanced analysis may be required to confirm its structural integrity. Such discontinuities are known to produce singular (that is, infinite) elastic stresses: if the possibility exists that the material might behave in a brittle manner, a more rigorous fracture mechanics evaluation must be performed. An analyst skilled in fracture analysis can use the Mechanical APDL application to determine fracture mechanics information.
- The proper selection and use of a failure theory relies on your engineering judgment. Refer to engineering texts such as *Engineering Considerations of Stress, Strain, and Strength* by R. C. Juvinall (McGraw-Hill) and *Mechanical Engineering Design* by J. E. Shigley (McGraw-Hill) for in-depth discussions on the applied theories.

Maximum Tensile Stress Safety Tool

The Maximum Tensile Stress Safety tool is based on the maximum tensile stress failure theory for brittle materials.

The theory states that failure occurs when the maximum principal stress (p. 2028) equals or exceeds a tensile stress limit. Expressing the theory as a design goal:

$$\frac{\sigma_1}{S_{limit}} < 1$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The maximum tensile stress failure theory is typically used to predict fracture in brittle materials with static loads. Brittle materials include glass, cast iron, concrete, porcelain and certain hardened steels.

The design goal is to limit the greatest principal stress to be less than the material's ultimate strength in tension:

$$\frac{\sigma_1}{S_{ut}} < 1$$

An alternate definition compares the greatest principal stress to the yield strength of the material:

$$\frac{\sigma_1}{S_{yt}} < 1$$

For many materials (usually ductile materials), strength in compression and in tension are roughly equal. For brittle materials, the compressive strength is usually much greater than the tensile strength.

The Mohr-Coulomb theory used in the Mohr-Coulomb Stress Safety tool is generally regarded as more reliable for a broader range of brittle materials. However, as pointed out by R. C. Juvinall (*Engineering Considerations of Stress, Strain, and Strength*, McGraw-Hill, 1967), "There is some evidence to support its use with porcelain and concrete. Also, it has been used in the design of guns, as some test results on thick-walled cylinders tend to agree with this theory."

Options

Define the stress limit in the Details under **Stress Limit Type**. Use either **Tensile Yield Per Material**, or **Tensile Ultimate Per Material**, or enter a **Custom Value**. By default, **Stress Limit Type** equals **Tensile Yield Per Material**.

Select one of the following **Stress Tool** results from the **Result** group of the **Stress Tool** tab or by right-clicking and selecting **Insert** > [result type]:

Safety Factor

$$F_s = \frac{S_{limit}}{\sigma_1}$$

Safety Margin

$$M_s = F_s - 1 = \frac{S_{limit}}{\sigma_1} - 1$$

Stress Ratio

$$\sigma_1^* = \frac{\sigma_1}{S_{limit}}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Notes

- The use of a yield strength limit with brittle materials is not recommended since most brittle materials do not exhibit a well-defined yield strength.
- For ductile and some other types of materials, experiments have shown that brittle failure theories may be inaccurate and unsafe to use. The brittle failure theories may also be inaccurate for certain brittle materials. Potential inaccuracies are of particular concern if the accuracy of calculated answers is suspect.
- The reliability of this failure criterion is directly related to treatment of stress risers (peak stresses). For brittle homogeneous materials such as glass, stress risers are very important, and it follows that the calculated stresses should have the highest possible accuracy or significant factors of safety should be expected or employed. If the calculated results are suspect, consider the calculated stresses to be nominal stresses, and amplify the nominal stresses by an appropriate stress concentration factor K_t. Values for K_t are available in many strength of materials handbooks. For brittle nonhomogeneous materials such as gray cast iron, stress risers may be of minimal importance.
- If a part or structure is known or suspected to contain cracks, flaws, or is designed with sharp notches or re-entrant corners, a more advanced analysis may be required to confirm its structural integrity. Such discontinuities are known to produce singular (that is, infinite) elastic stresses: if the possibility exists that the material might behave in a brittle manner, a more rigorous fracture mechanics evaluation must be performed. An analyst skilled in fracture analysis can use the Mechanical APDL application program to determine fracture mechanics information.
- The proper selection and use of a failure theory relies on your engineering judgment. Refer to engineering texts such as *Engineering Considerations of Stress, Strain, and Strength* by R. C. Juvinall (McGraw-Hill) and *Mechanical Engineering Design* by J. E. Shigley (McGraw-Hill) for in-depth discussions on the applied theories.

Fatigue (Fatigue Tool)

See the Fatigue Results (p. 2165) section.

Fracture Results (Fracture Tool)

Use the Fracture Tool object to insert and then define fracture results for the crack types (p. 424) specified in your analysis. All fracture results are child objects of the **Fracture Tool**. Use its properties to evaluate result data.

Go to a section topic:

- Define a Fracture Tool for Crack Selection (p. 2066)
- Define Fracture Results for Crack Selection (p. 2068)
- Extract Fracture Results from a Result File (p. 2072)

Define a Fracture Tool

To define a **Fracture Tool**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

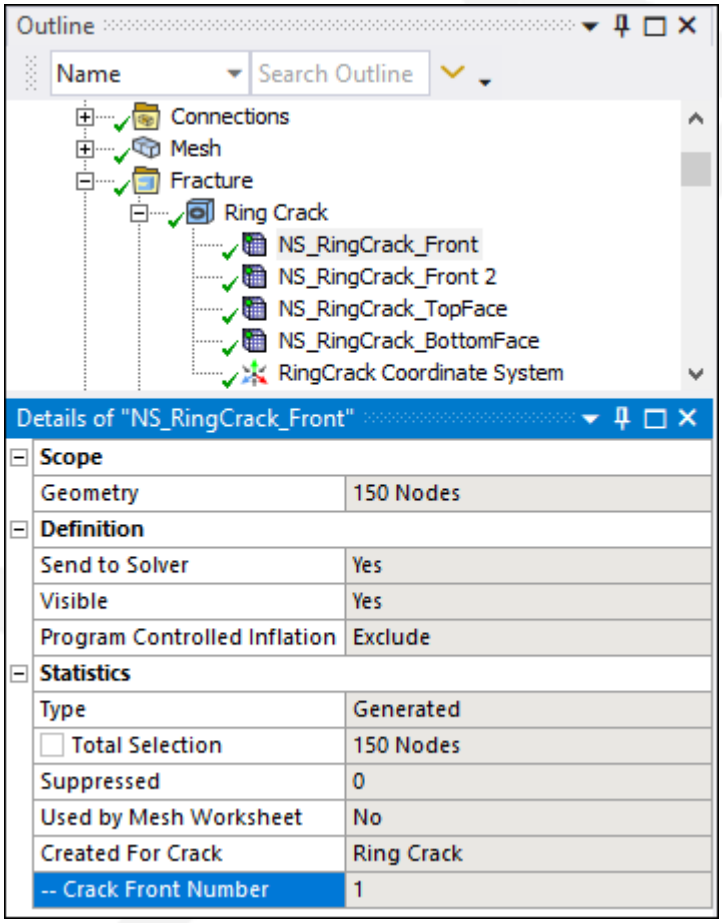
1. Select the **Solution** object, open the **Toolbox** drop-down menu from the **Solution** Context tab, and select the **Fracture Tool** option. Based on the settings of the Fracture Controls (p. 1445) of the **Analysis Settings** object, a result object, such as **SIFS (K1)**, is automatically included under the **Fracture Tool**.

2. Set the **Scoping Method** property to **Crack Selection** (default). To read results into the application using the **Result File Item** option, see the Extract Fracture Results from a Result File (p. 2072) topic below.

3. Specify the **Crack Selection Mode** property as either **Single Crack** (default) or **All Cracks**. When you specify **All Cracks**, the child result objects of the tool display the **Tabular and Graph Data** category. Use this to further select specific crack(s) of interest that are available in your model. See the **Defining a Fracture Result** topic below.

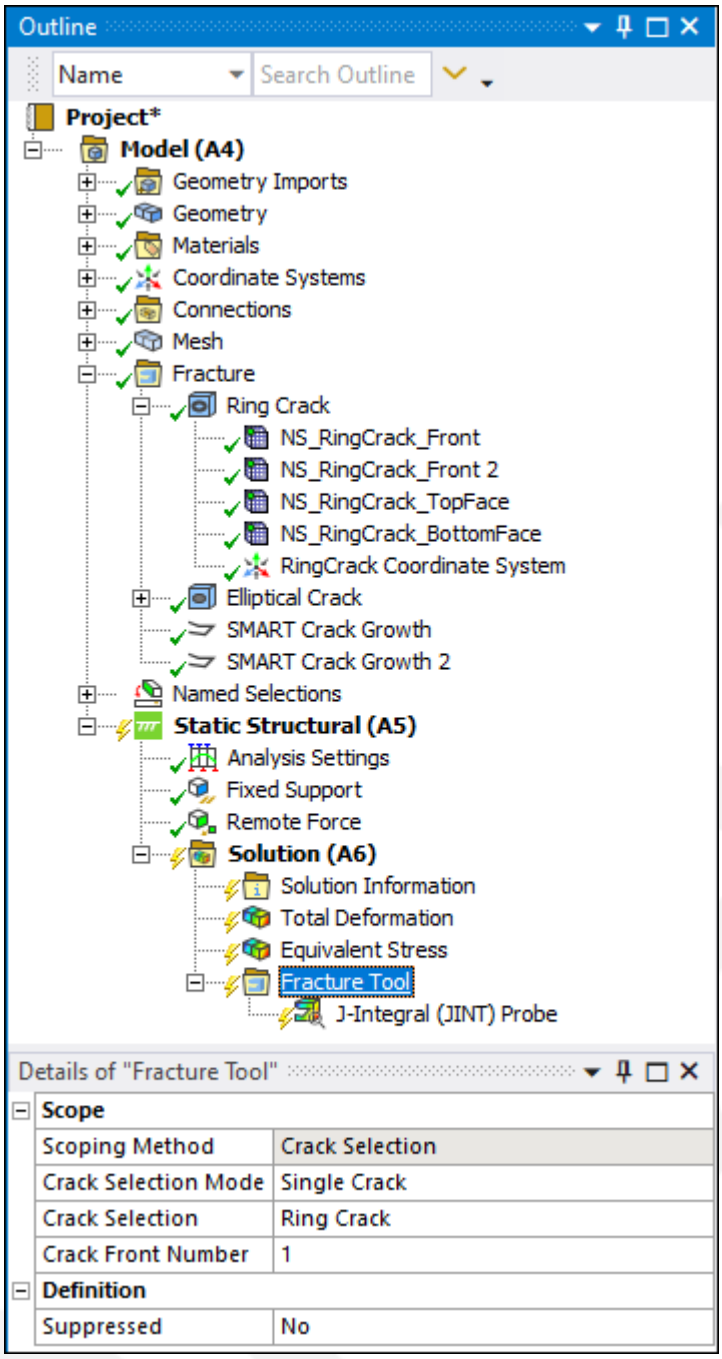
   For the **Single Crack** option, the following additional properties display:

   - **Crack Selection**: Specify the crack on which to evaluate results.
   - **Crack Front Number**: Specify the crack front(s) on which to evaluate results. Options include **All Crack Fronts** (default), which is equal to setting the property to zero (0) or you can enter a specific Crack Front Number (contained in the Details of the crack front named selection object) to evaluate results for that crack front only.



   For the above crack front, the **Fracture Tool** could be specified as shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Important:

If you specify the **All Cracks** option, the use of Fracture Probes is not supported.

Define Fracture Results

Review the **Methods Used for Calculating the Fracture Parameters** topic in the Understanding the Fracture Parameters section of the *Mechanical APDL Fracture Analysis Guide* for more information about fracture results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

To define a fracture result:

1. From the **Fracture Tool** Context tab, select desired results. Options include:

   - **SIFS Results>SIFS(K1)** inserts a Mode 1 Stress Intensity Factor result.
   - **SIFS Results>SIFS(K2)** inserts Mode 2 Stress Intensity Factor result.
   - **SIFS Results>SIFS(K3)** inserts Mode 3 Stress Intensity Factor result.
   - **J-Integral (JINT)** inserts a J-Integral result.

     Note:

     – The computation for the **J-Integral** result includes the effect of body forces.

     – The application enables you to compute the **Equivalent SIFS Range** result in addition to one fracture parameter result (**SIFS results** or **J-Integral**) when the **SMART Crack Growth** associated to the crack is of type **Fatigue** (**Crack Growth Option** = **Fatigue**).

   - **VCCT Results>VCCT(G1)** inserts Mode 1 Energy Release Rate result.
   - **VCCT Results>VCCT(G2)** inserts Mode 2 Energy Release Rate result.
   - **VCCT Results>VCCT(G3)** inserts Mode 3 Energy Release Rate result.
   - **VCCT Results>VCCT(GT)** inserts Total Energy Release Rate result.
   - **Material Force>Material Force (X Axis)** inserts Total Material Force result in the X Axis of the local crack front node coordinates.
   - **Material Force>Material Force (Y Axis)** inserts Total Material Force result in the Y Axis of the local crack front node coordinates.
   - **Material Force>Material Force (Z Axis)** inserts Total Material Force result in the Z Axis of the local crack front node coordinates.
   - **T-Stress**
   - **C*-Integral**

     Note:

     The **C*-Integral** result is only valid when the analysis includes a creep material, and the **Creep Effect** property of the Creep Controls (p. 1445) is set to **On**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Equivalent SIFS Range** (available when specified crack is associated with Fatigue **SMART Crack Growth**)

Tip:

You can use the **Type** and **Subtype** properties of a result to change result parameters as desired.

---

2. Define the properties of each result. Options specific to fracture results include:

   - **Contour Start**: Specifies the first contour number for which the result will be plotted in the graph and displayed in the tabular data. The value must not be greater than the value of **Contour End**. This option is applicable to SIFS, JINT, Material Force, T-Stress and C*-Integral types of fracture result.

   - **Contour End**: Specifies the last contour number for which the result will be plotted in the graph and displayed in the tabular data. The value must not be greater than value of the Solution Contours option specified for the associated crack object. Since the maximum of 10 contours can be plotted in **Graph** window at one point of time, the difference between **Contour End** and **Contour Start** must not be greater than 9. This option is applicable to SIFS, JINT, Material Force, T-Stress and C*-Integral types of fracture result.

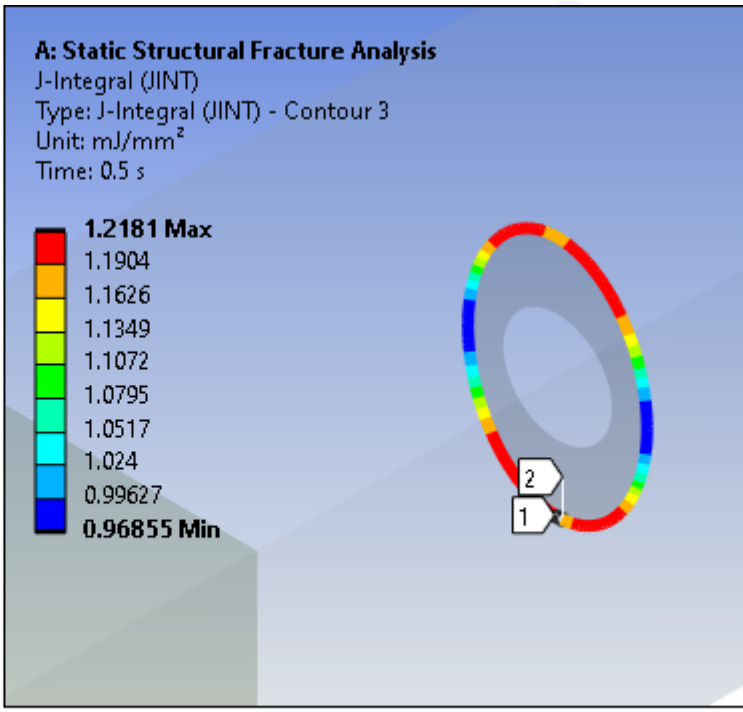
   - **Active Contour**: Specifies the contour number for which the results are plotted in the **Geometry** window. By default, it takes the "Last" value as in the **Contour End** property. This option is applicable to the SIFS, JINT, Material Force, T-Stress and C*-Integral result types. The **Geometry** window displays the graphical result for the active contour. The "1" in the **Geometry** window indicates the start of the crack front, while "3" indicates the end of the crack front.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

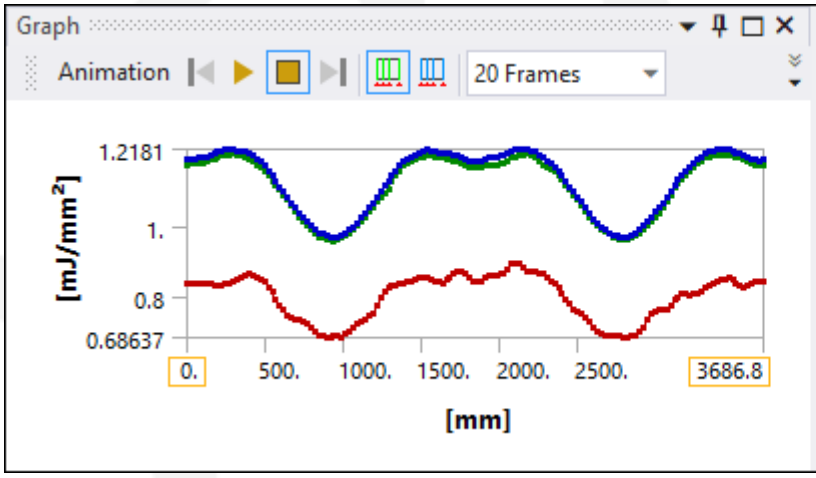
Published: 2023-10-23

Details of "J-Integral (JINT)"

Definition	
Type	J-Integral (JINT)
Contour Start	1
Contour End	3
Active Contour	Last
By	Result Set
Set Number	1.
Separate Data by Entity	No
Calculate Time History	Yes
Suppressed	No
Results	
Minimum	0.96855 mJ/mm²
Maximum	1.2181 mJ/mm²
Minimum Occurs On	Part2
Maximum Occurs On	Part2
Information	
Time	0.5 s
Load Step	1
Substep	1
Iteration Number	1



The results are plotted in the **Graph** window for all contours, starting from the **Contour Start** value and ending at the **Contour End** value. The X axis in the **Graph** window indicates the distance along the crack front. The start of the crack front has a value of zero, and the end of the crack front has the maximum value. The **Tabular Data** window displays the data points in a table format.



Tabular Data

	Length [mm]	J-Integ...	J-Integr...	J-
1	0.	0.8391	1.1744	1.19
2	24.741	0.83782	1.1758	1.191
3	49.482	0.83654	1.1772	1.192
4	74.224	0.83771	1.1787	1.193
5	98.965	0.83888	1.1803	1.194
6	123.71	0.83798	1.1845	1.198
7	148.45	0.83708	1.1887	1.202
8	173.19	0.83591	1.1921	1.206
9	197.93	0.83474	1.1956	1.210
10	222.68	0.83625	1.1985	1.213
11	247.42	0.83776	1.2015	1.215
12	272.16	0.84031	1.2031	1.215
13	296.91	0.84285	1.2046	1.215

Note:

When the **Crack Selection Mode** property of the **Fracture Tool** is set to **All Cracks**, and more than one crack is defined in the model:

- The **Contour Start** and **Contour End** properties are not available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If the **Active Contour** property is set to **Last**, it uses the lowest solution contour value of all the cracks to compute the result.

For information on other Details options, see Results and Result Tools (Group) object reference section.

Extract Fracture Results from a Result File

When you have saved a results file (`file.rst`) from another fracture analysis, you can read this file into a different analysis and extract the result data with a **Fracture Tool**. Note the following requirements:

- For a Pre-Meshed Crack (p. 465), you can evaluate the fracture results and probes without defining the crack in the new analysis, but with similar boundary conditions.
- For all other crack types (p. 424), your analysis must include a defined Fracture folder, a matching mesh (including crack mesh), as well as defined boundary conditions.

Use the following steps to specify a **Fracture Tool** and to extract the fracture results and probes from the result file.

1. Read the results file into your analysis. As needed, review the Writing and Reading the Mechanical APDL Application Files (p. 1861) section.
2. Select the **Solution** object, open the **Toolbox** drop-down menu from the **Solution** Context tab, and select the **Fracture Tool** option. Based on the settings of the Fracture Controls (p. 1445) of the **Analysis Settings** object, a result object, such as **SIFS (K1)**, is automatically included under the **Fracture Tool**.
3. Set the **Scoping Method** property to **Result File Item**.
4. From the **Fracture Tool** Context tab, select and insert desired results.
5. From the drop-down list of the **Solver ID** property, select the appropriate solver ID value for each result. You can now evaluate the specific results.

Composite Failure Tool

The **Composite Failure Tool** is a post-processing option that you can use for composite ply structures that were modeled using Ansys Composite PrepPost (ACP).

Important:

- The application supports the **Composite Failure Tool** on the same operating systems as Ansys Composite PrepPost. See the Supported Platforms section of the *Ansys Composite PrepPost User's Guide* for specific platform information.
- For a Static Structural system linked to a ACP-Pre system, created prior to 2024 R1, Mechanical does not support the **Composite Failure Tool**. You can resolve this issue

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

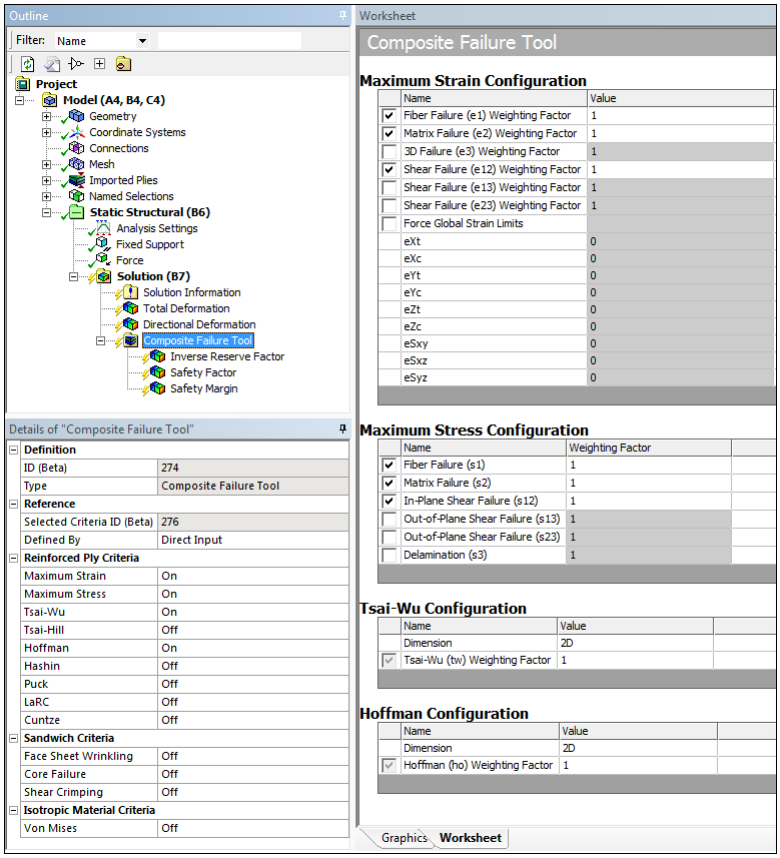
by cleaning and updating the ACP-Pre system and then refreshing the inputs of the Static Structural system.

This section addresses the following topics:

- Tool Overview (p. 2073)
- Result Types (p. 2074)
- Composite Failure Tool Promotion (p. 2074)
- Application (p. 2076)

Tool Overview

This tool provides result data that corresponds to that of the failure definition in ACP Post. In addition, and as illustrated below, the Details properties for the object provide a set of failure criteria and functionality. You enable these failure criteria using the properties of the **Reinforced Ply Criteria** category (see Composite Failure Tool Result Objects). The Worksheet provides additional details for each criteria enabling you to make further specifications.



The properties of the **Composite Failure Tool** enable you to configure your own combined failure criteria for the strength assessment of composites. Both established and basic as well as recent and advanced failure criteria are included in the program and you can combine them with the goal to incorporate all relevant failure modes which turn up in a composite structure. This helps to design

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

robust structures and to work efficiently. A discussion related to the best practices for using this tool is available in the Guide to Composite Failure Criteria topic in the *Ansys Composite PrepPost User's Guide*.

Result Types

The **Composite Failure Tool** supports the following result objects:

Safety Factor (SF)

This result type is an indication of the margin to failure. That is, the applied load multiplied by the safety factor determines the failure load. Failure is experienced when you have a Safety Factor less than 1.

$SF \times F_{applied} = F_f$

Safety Factor values that are greater than one indicate a positive margin to failure and values less than one indicate a negative margin. Safety Factor values are always greater than zero.

Inverse Reserve Factor (IRF)

This result type is an inverse margin to the Safety Factor. The failure load can be defined as the load value divided by IRF. Failure is experienced when you have an Inverse Reserve Factor greater than 1.

$IRF = 1/SF$

The critical values of reserve factors lie between zero and one, whereas the non-critical values range from one to infinity. Whether the results are shown in numeric form or as contour plots, the non-critical values tend to be emphasized in comparison to critical values.

Safety Margin

The Safety Margin result is obtained from the safety factor. Failure is experienced when you have a Safety Margin less than 0.

$SM = SF - 1$

A positive safety margin indicates the relative amount that the applied load can be increased before reaching failure load. Correspondingly, a negative safety margin indicates how much the applied load should be decreased. Safety margins are typically expressed as percentages.

User-Defined Results

See the User-Defined Results (p. 2192) section of the Mechanical Help.

Also see the Composite Failure Tool (Group) object reference page for additional information about these result types.

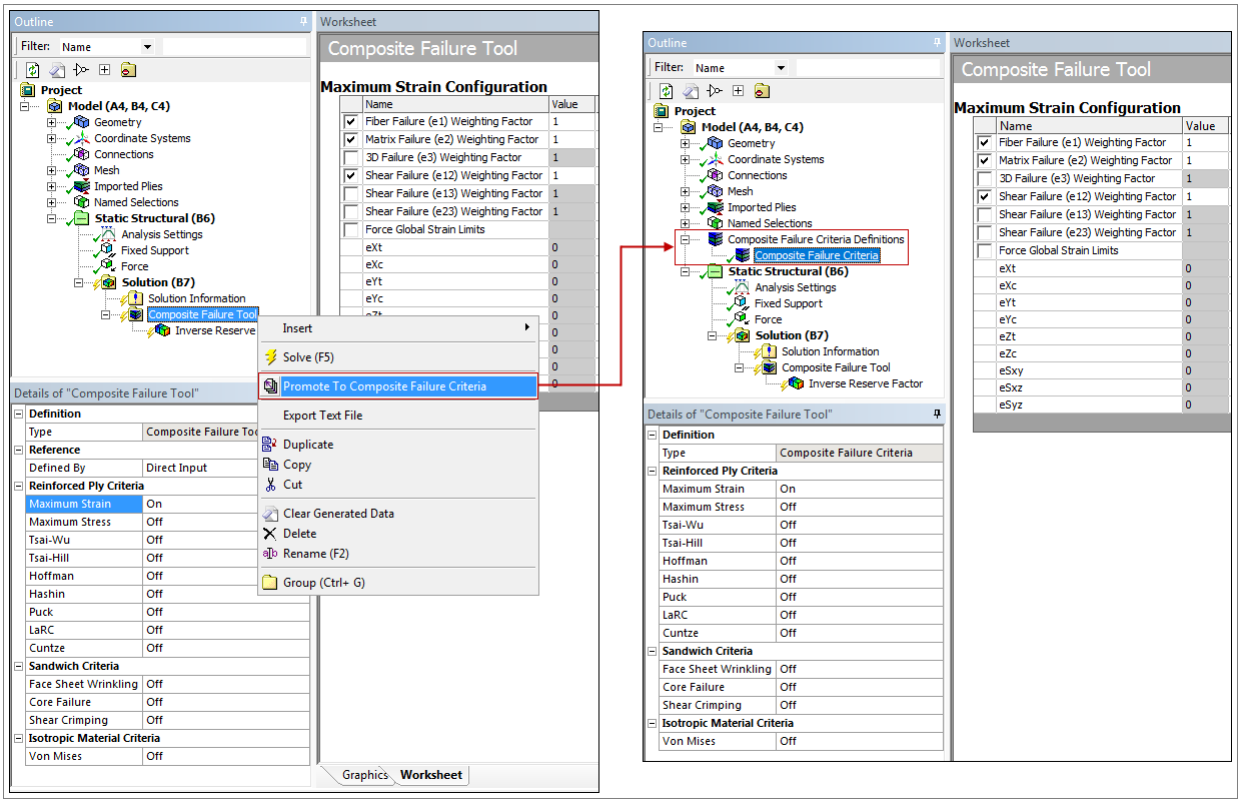
Composite Failure Tool Promotion

The purpose of this feature is to enable you to create and share one or more combinations of failure criteria by using them in various tools or across analyses. Through the promotion feature, as illustrated

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

below, you can promote a defined **Composite Failure Tool** to the Composite Failure Criteria Definitions object. By default, the promoted object maintains the name of the original object, such as the default tool name, "Composite Failure Tool."



After promotion, the properties of the corresponding **Composite Failure Tool** object become read only, except for the **Defined By** and the **Composite Failure Criteria** properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "Composite Failure Tool"	
Definition	
Type	Composite Failure Tool
Reference	
Defined By	Composite Failure Criteria Definitions
Composite Failure Criteria	Composite Failure Criteria
Reinforced Ply Criteria	
Maximum Strain	Off
Maximum Stress	Off
Tsai-Wu	Off
Tsai-Hill	Off
Hoffman	Off
Hashin	On
Puck	Off
LaRC	Off
Cuntze	Off
Sandwich Criteria	
Face Sheet Wrinkling	Off
Core Failure	Off
Shear Crimping	Off
Isotropic Material Criteria	
Von Mises	Off

Application

To define results using the **Composite Failure Tool**:

1. Make sure that your composite analysis is properly defined.

2. Highlight the **Solution** object and select **Composite Failure Tool** from the **Toolbox** drop-down menu or highlight the **Solution** object, right-click, and then select **Insert** > **Composites** > **Composite Failure Tool**. The **Composite Failure Tool** becomes the active object in the tree, it contains an **Inverse Reserve Factor** result object by default, and the Worksheet displays.

3. Specify the **Defined By** property as **Direct Input** (default) or **Composite Failure Criteria Definitions**. In order to select **Composite Failure Criteria Definitions**, you must first specify failure criteria in the **Reinforced Ply Criteria** category and promote the **Composite Failure Tool**. This creates the required criteria object and you must specify in the **Composite Failure Criteria** property that becomes available when you select the **Composite Failure Criteria Definitions** option. Review the above *Composite Failure Tool Promotion* topic as needed.

4. Specify failure criteria by turning the properties of the **Reinforced Ply Criteria** category of the Details On or Off. The Worksheet aromatically displays and populates with the criteria that you have specified. You can further customize the criteria using the check boxes available in the Worksheet. In addition, further failure criteria changes can be made in the Worksheet, specifically, **Weighting** values.

5. Insert results objects as desired (**Inverse Reserve Factor**, **Safety Factor**, and **Safety Margin**) from the **Composite Failure Tool** Context tab or using the right-click context menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

6. For individual result objects, specify elemental max or ply-wise results using the options of the **Sub Scope By** property, **Entire Section** (default) or **Ply**. Based on your selection, either **Ply Maximum** or **Elemental Maximum** displays in the **Display Option** property.

7. The properties **Show Critical Failure Mode**, **Show Critical Layer**, and **Threshold for Text Visualization** are used to configure the overlaid text plot as shown below. The acronyms indicate the failure mode and the indices the critical layer where failure occurs. The text labels are not shown if they are below the defined threshold which is, by default, **`0.25`**. If working with very large models, it is suggested that you set a meaningful threshold so that thousands of text labels do not display. This makes it easier to examine the structure and improves the performance.

8. **Composite Failure Tool** promotion may occur as desired.

Composite Sampling Point Tool

Go to a section topic:

- Overview (p. 2077)
- Application (p. 2077)
- Worksheet Display (p. 2078)

Overview

The **Composite Sampling Point Tool** is a post-processing option that you can use for composite ply structures that were modeled using Ansys Composite PrepPost (ACP). See the Sampling Points section in the *Ansys Composite PrepPost User's Guide* for more information.

This tool works in tandem with the **Composite Failure Tool**. That is, to generate and display result data, you must scope the **Composite Sampling Point Tool** to existing Composite Failure Criteria. This criteria is created by promoting a defined **Composite Failure Tool** object. Be sure to review the Composite Failure Tool (p. 2072) section as it is a prerequisite for using this feature.

Note:

This feature does not support the Linux platform.

Also see the Composite Sampling Point object reference section.

Application

To define results using the **Composite Sampling Point Tool**:

1. Make sure that your composite analysis is properly defined.

2. Define and promote a **Composite Failure Tool**.

3. With the **Composite Failure Tool** selected, right-click and select **Insert** > **Composites** > **Composite Sampling Point Tool**. The application inserts a **Composite Sampling Point Tool** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is automatically the active object and by default, includes the child object **Composite Sampling Point**.

Note:

You can also insert a **Composite Sampling Point Tool** by right-clicking the **Solution** object or in the geometry windows and selecting **Insert** > **Composites** > **Composite Sampling Point Tool**.

4. Specify the **Composite Failure Criteria** property. The **Worksheet** displays automatically and contains the criteria as specified in the **Composite Failure Criteria** object.
5. Select the **Composite Sampling Point** child object.
6. Using the **Geometry** property, select the desired faces on your geometry. The sampling point is created at the location of hit point on the geometry face and the direction of the sampling point is aligned with the face normal.
7. Solve or evaluate the results of the analysis. Once complete, the **Worksheet** displays automatically. Select desired result data from the available options (Layup, Text, Strain,, etc.). The example worksheet show below illustrates result data. This example illustrates that you can use the **Text** group to display the material, thickness, and angle as text labels for every ply in the plot. The angle displayed for modeling and production plies always matches the design angle in the modeling ply and material definitions.

Note:

The worksheet does not display result content using the units specified in the **Units** option.

Worksheet Display

The **Worksheet** displays the thickness distributions using the options of the **Layup** group as well as the post-processing results (**Strain**, **Stress**, and **Failure** groups) as 2D plots. Stresses and strains shown in the 2D plot display the values at the interpolated element center at the top and bottom of the layer. The 2D failure plot shows the worst IRF, RF or MoS factor of all failure criteria, failure modes evaluated, and integration point level. In addition, the display can show failure modes as text labels for every ply. And you can enable the polar properties of the laminate using the options of the **Polar Properties** group. In addition, there are several worksheet user interactions available, including:

- **Left Mouse Button**: Select and hold to move the content of all columns vertically. Select and hold on an individual column to move content horizontally.
- **Mouse Wheel**: This key zooms in and out on the vertical axis of all columns. Combine this action with the [ctrl] key to zoom in and out along the horizontal axis on individual columns.
- **Right Mouse Button**: Place the cursor over a graph location and then right-click and hold to display a tooltip that includes comma separated values for a stress, strain, failure, (left tooltip value) and a thickness (right tooltip value) for that associated worksheet value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

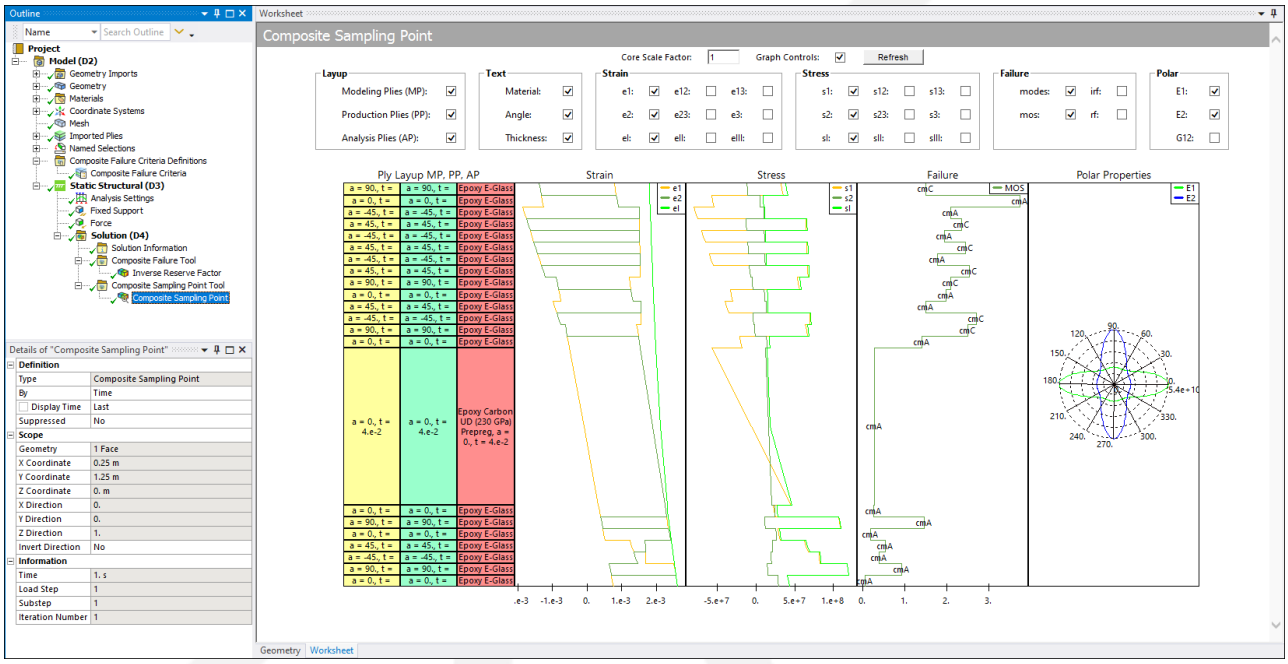
Published: 2023-10-23

- **Core Scale Factor**: This entry visually scales the core layer's thickness in order to allow more space within the columns to draw the other layers. This is especially useful when the core layer is much larger than other layers. The default value is 1. You can enter smaller values incrementally to reduce the size of the displayed core layer.

- **Escape Key**: Reset all charts to original view.

- **Note:**

  The **Polar Properties** chart does not support user interactions.



Contact Tool

The **Contact Tool** enables you to examine contact conditions on an assembly both before loading, and as part of the final solution to verify the transfer of loads (forces and moments) across the various contact regions. The **Contact Tool** is an object you can insert under a Connections object for examining initial contact conditions, or under a Solution or Solution Combination branch object for examining the effects of contact as part of the solution. The **Contact Tool** enables you to conveniently scope contact results (p. 2043) to a common selection of geometry or contact regions. In this way, all applicable contact results can be investigated at once for a given scoping.

A **Contact Tool** is scoped to a given topology, and there exist two methods for achieving this: the **Worksheet** method and the **Geometry Selection** method. Under the **Worksheet** method, the **Contact Tool** is scoped to one or more contact regions. Under the **Geometry Selection** method, the **Contact Tool** can be scoped to any geometry on the model. Regardless of the method, the scoping on the tool is applied to all results grouped under it.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

To use a **Contact Tool**, prepare a structural analysis for an assembly with contacts. You then use either the **Geometry Selection** or **Worksheet** scoping method for results.

Important:

If you are importing PCB traces from External Data, the trace materials must include the necessary structural properties to generate initial contact results, or the application generates an error. See the ECAD Analysis using Trace Mapping (p. 686) section for more information.

Evaluating Initial Contact Conditions

Note:

To calculate initial contact results, the **Contact Tool** assumes small deflection. This assumption affects the resulting pinball radius of the scoped contacts if their **Pinball Region** property is set to **Program Controlled**.

To evaluate initial contact conditions using the Worksheet method:

1. Insert a **Contact Tool** in the **Connections** folder (**Contact Tool** from the **Connections** Context tab or right-click the **Connections** object and then select **Insert** > **Contact Tool**). The tool is inserted and includes a default **Initial Information** child object.
2. In the Details of the **Contact Tool**, ensure that **Worksheet** (the default) is selected in the **Scoping Method** field. The **Worksheet** appears. Scoped contact regions are those that are checked in the table.
3. You can modify your selection of contact regions in the **Worksheet** using the following procedures:
   - To add or remove pre-selected groups of contact regions (**All Contacts**, **Nonlinear Contacts**, or **Linear Contacts**), use the drop-down menu and the corresponding buttons.
   - To add any number of contact regions, you can also drag-drop or copy-paste any number of contact regions from the **Connections** folder into the **Contact Tool** in the Tree View. Also, one or more contact regions can be deleted from the **Contact Tool** worksheet by selecting them in the table and pressing the **Delete** key.
   - To change the **Contact Side** of all contact regions, choose the option in the drop-down menu (**Both**, **Contact**, or **Target** from the drop-down menu and click the **Apply** button).
   - To change an individual **Contact Side**, click in the particular cell and choose **Both**, **Contact**, or **Target** from the drop-down menu.
4. Add contact result objects of interest under the **Contact Tool** folder using the option of the **Contact Tool** Context tab. Or, you can right-click the tool and select **Insert** > [desired result]. The specific contact result objects are inserted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. Obtain the initial contact results using a right mouse button click the **Contact** object, or **Contact Tool** object, or any object under the **Contact Tool** object, then choosing **Generate Initial Contact Results** from the context menu. Results are displayed as follows:

   - When you highlight the **Initial Information** object, a table appears in the **Worksheet** that includes initial contact information (p. 2084) for the contact regions that you specified in step 2 above. You can display or hide the various columns (p. 193) in the table. The table rows display in various colors that indicate the detected contact conditions. A brief explanation of each color is provided in the legend that is displayed beneath the table. Copies of the legend explanations are presented below in quotes, followed by more detailed explanations.

     – Red: "The contact status is open but the type of contact is meant to be closed. This applies to bonded and no separation contact types."

       Workbench has detected an open contact **Status** (p. 2043) condition, which is invalid based on the definitions of **Bonded** and **No Separation** contact types (p. 1234). It is very likely that the model will not be held together as expected. The geometry of the contact may be too far apart for the closed condition to be satisfied. Review of the Contact Region definition is strongly recommended.

     – Yellow: "The contact status is open. This may be acceptable."

       Workbench has detected an open contact **Status** (p. 2043) condition on a nonlinear contact type, **Frictionless**, **Rough**, or **Frictional**, which is probably acceptable under certain conditions as stated in their descriptions (p. 1234).

       If the **Status** is **Far Open**, the **Penetration** and the **Gap** will be set to zero even though the **Resulting Pinball** (p. 1254) is non-zero.

       Note:

       Currently, contact results are not saved to results (.rst) file for all contact elements that are outside the pinball region to optimize the file size. Results for Far-field contact elements were reported as zero in prior releases.

     – Orange: "The contact status is closed but has a large amount of gap or penetration. Check penetration and gap compared to pinball and depth."

       Workbench has detected that any of the following contact results are greater than 1/2 of the **Resulting Pinball**, or greater than 1/2 of the **Contact Depth**: **Gap**, **Penetration**, maximum closed **Gap**, maximum closed **Penetration**. This could lead to poor results in terms of stiffness of the contacting interface. Ansys recommends that you alter the geometry to reduce the gap or penetration.

     – Gray: "Contact is inactive. This can occur for MPC and Normal Lagrange formulations. It can also occur for auto asymmetric behavior."

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Refer to the individual descriptions for the **MPC** and **Normal Lagrange** formulations (p. 1240), and the description for **Auto Asymmetric** behavior (p. 1236).

Note:

The "not applicable" designation, **N/A** appears in the following locations and situations:

- All result columns when the contact pair is inactive (row is gray, or **Inactive** appears under the **Status** column).
- The **Geometric Gap** column for **Frictionless** (p. 1235), **Rough** (p. 1235), or **Frictional** (p. 1235) contact **Types** (p. 1234) and an **Interface Treatment** (p. 1257) set to **Add Offset** (p. 1259), **Ramped Effects**.

- When you highlight any of the contact result objects, the **Geometry** tab appears and displays the graphical result for the contact regions that you specified in step 2 above.

To evaluate initial contact conditions using the Geometry Selection method:

1. Select one or more bodies that are in contact.
2. Insert a **Contact Tool** in the **Connections** folder (**Contact Tool** from the **Connections** Context tab or right-click the **Connections** folder and then select **Insert** > **Contact Tool**). The tool is inserted and includes a default **Initial Information** child object.

   Note:

   The scoping of the **Initial Information** object is only available using the Worksheet method. Selecting bodies as in Step 1 above has no effect on **Initial Information** results.

   ---

3. In the Details of the **Contact Tool**, select **Geometry Selection** in the **Scoping Method** field. The bodies that you selected in step 1 are highlighted in the **Geometry** tab.
4. Add contact result objects of interest under the **Contact Tool** folder using the option of the **Contact Tool** Context tab. Or, you can right-click the tool and select **Insert** > [desired result]. The specific contact result objects are inserted.
5. Obtain the initial contact results using a right mouse button click the **Contact** object, or **Contact Tool** object, or any object under the **Contact Tool** object, then choosing **Generate Initial Contact Results** from the context menu. When you highlight any of the contact result objects, the **Geometry** tab appears and displays the graphical result for the bodies that you selected in step 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Evaluating Contact Conditions After Solution

Note:

The default method will be the last one that you manually chose in the **Scoping Method** drop-down menu. If you have already selected geometry, the **Scoping Method** field automatically changes to **Geometry Selection**. The default however will not change until you *manually* change the **Scoping Method** entry.

To evaluate contact conditions after solution using the Worksheet method:

1. Insert a **Contact Tool** from the **Toolbox** drop-down menu on the **Solution** Context tab or right-click the **Solution** folder and then select **Insert** > **Contact Tool** > **Contact Tool**). The tool is inserted and includes a default **Status** child object.

2. In the Details, select **Worksheet** in the **Scoping Method** field. The **Worksheet** appears. Scoped contact regions are those that are checked in the table.

3. You can modify your selection of contact regions in the **Worksheet** using the following procedures:

   - To add or remove pre-selected groups of contact regions (**All Contacts**, **Nonlinear Contacts**, or **Linear Contacts**), use the drop-down menu and the corresponding buttons.

   - To add any number of contact regions, you can also drag-drop or copy-paste any number of contact regions from the **Contact** folder into the **Contact Tool** in the Tree View. Also, one or more contact regions can be deleted from the **Contact Tool** worksheet by selecting them in the table and pressing the **Delete** key.

   - To change the **Contact Side** of all contact regions, choose the option in the drop-down menu (**Both**, **Contact**, or **Target** from the drop-down menu and click the **Apply** button).

   - To change an individual **Contact Side**, click in the particular cell and choose **Both**, **Contact**, or **Target** from the drop-down menu.

4. Add more contact results as needed in the **Contact Tool** folder (**Contact> [Contact Result, for example, Pressure]** from the **Contact Tool** Context tab, or right mouse button click **Contact Tool**, then **Insert> [Contact Result, for example, Pressure]**).

5. Solve database. Upon completion, you will see contact results with the common scoping of the **Contact Tool**.

To evaluate contact conditions after solution using the Geometry Selection method:

1. Select one or more bodies that are in contact.

2. Insert a **Contact Tool** from the **Toolbox** drop-down menu on the **Solution** Context tab or right-click the **Solution** folder and then select **Insert** > **Contact Tool** > **Contact Tool**). The tool is inserted and includes a default **Status** child object. Because you have already selected one or more bodies, **Geometry Selection** is automatically set in the **Scoping Method** field within the Details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

3. Add more contact results as needed to the **Contact Tool** folder either by right-clicking on the object and selecting **Insert** > [desired result] or using the options of the **Contact Tool** Context tab.

4. Solve database. Upon completion, you will see contact results with the common scoping of the **Contact Tool**.

The configuration of the **Contact Tool**, in particular the location (**Solution** vs **Solution Combination**) and the scoping method, affects the availability of results. A **Contact Tool** in the **Solution Combination** folder has the limitation that it supports only pressure, frictional stress, penetration and distance.

Limitation

Note that when the **Contact Tool**, whether inserted under the **Connection** folder or the **Solution** object, has a limitation for coincident contact elements. If you have 1) two contact conditions, and each condition shares elements from one of the bodies, and 2) in the **Worksheet** you have selected only one row for the contact conditions, the application uses the element data for both contact conditions to produce the Contact Tools Results (p. 2043) content. The numerical values at the shared node are contributed by all the contact elements that touch it.

To make sure that you have correct contact results, 1) obtain User Defined (p. 2194) results from the **Available Solution Quantities** of the Solution Summary Worksheet (p. 1961) and 2) scope the **User Defined** results using the **Result File Item** and **Element Type IDs** options. This enables you to specify only the desired contact elements. You can obtain **Element Type IDs** from the **Material and Element Type Information** option of the Solution Summary Worksheet.

Contact Tool Initial Information

When a Contact Tool is inserted under the Connections object, it includes a default object, **Initial Information**. This object provides the following information from the Worksheet.

- **Name**: Contact Region name.
- **Contact Side**: Selected contact side, either **Contact** or **Target**.
- **Type**: contact type, **Bonded**, **No Separation**, **Frictionless**, **Rough**, **Frictional**.
- **Status**: the status of the contact, **Open**, **Closed**, **Far Open**.
- **Number Contacting**: the number of contact or target elements in contact.
- **Penetration**: the resulting penetration.
- **Gap**: the resulting gap.
- **Geometric Penetration**: the penetration that initially exists between the **Contact** and **Target** surfaces.
- **Geometric Gap**: the gap that initially exists between the **Contact** and **Target** surfaces. For **Frictional** or **Frictionless** contact, this is the minimum gap. For **Bonded** or **No Separation** contact, this is the maximum closed gap detected.
- **Resulting Pinball**: user specified or the Mechanical APDL application calculated pinball radius.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Contact Depth**: average contact depth of elements.
- **Normal Stiffness**: the calculated maximum normal stiffness value.
- **Tangential Stiffness**: the calculated maximum tangential stiffness value.
- **Real Constant**: the contact Real Constant number.
- **Possible Overconstraint**: This field displays if there is a possible over constraint between multiple bonded contacts using **MPC** formulation. You can set the **MPC** option manually or it can be used by the **Program Controlled** option. The field displays **Yes** for over-constrained contact, otherwise it displays **No**. The field can also display **N/A** if there is an inactive side of the contact or for non-bonded contact regions.

The following table outlines how to interpret the Gap and Penetration columns in the Initial Contact Information when there is a true **initial geometric gap** at the contact interface.

Contact Type	Interface Treatment	Offset	Status	Penetration	Gap	Geometric Penetration	Geometric Gap
Bonded or No Separation	NA	NA	Closed	0	0	0	True Geometric Gap
Bonded or No Separation	NA	NA	Far Open	0	0	0	0
Frictionless, Rough, or Frictional	Add Offset, No Ramping	0	Far Open	0	0	0	0
Frictionless, Rough, or Frictional	Add Offset, Ramped Effects	0	Far Open	0	0	0	NA
Frictionless, Rough, or Frictional	Add Offset, No Ramping	< True Geometric Gap	Near Open	0	True Geometric Gap - Offset	0	True Geometric Gap
Frictionless, Rough, or Frictional	Add Offset, Ramped Effects	< True Geometric Gap	Near Open	0	True Geometric Gap - Offset	0	NA
Frictionless, Rough, or Frictional	Add Offset, No Ramping	> True Geometric Gap	Closed	Offset - True Geometric Gap	0	0	True Geometric Gap
Frictionless, Rough, or Frictional	Add Offset, Ramped Effects	> True Geometric Gap	Closed	Offset - True Geometric Gap	0	0	NA

The following table outlines how to interpret the Gap and Penetration columns in the Initial Contact Information when there is a true **initial geometric penetration** at the contact interface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Contact Type	Interface Treatment	Offset	Status	Penetration	Gap	Geometric Penetration	Geometric Gap
Bonded or No Separation	NA	NA	Closed	0	0	True Geometric Penetration	0
Bonded or No Separation	NA	NA	Far Open	0	0	0	0
Frictionless, Rough, or Frictional	Add Offset, No Ramping	| Offset | < Geometric Penetration	Closed	Offset + True Geometric Penetration	0	True Geometric Penetration	0
Frictionless, Rough, or Frictional	Add Offset, Ramped Effects	| Offset | < Geometric Penetration	Closed	Offset + True Geometric Penetration	0	True Geometric Penetration	NA
Frictionless, Rough, or Frictional	Add Offset, No Ramping	| – Offset | > Geometric Penetration	Near Open	0	| – Offset | - True Geometric Penetration	True Geometric Penetration	0
Frictionless, Rough, or Frictional	Add Offset, Ramped Effects	| – Offset | > Geometric Penetration	Near Open	0	| – Offset | - True Geometric Penetration	True Geometric Penetration	NA

Bolt Tool

When examining numerous bolt loads, use the **Bolt Tool** to obtain bolt-specific results.

Note:

If you are doing bolt analysis, you may wish to use the Bolt Tools Add-on (p. 1031) to quickly create bolt geometries.

Scoping

You scope the **Bolt Tool** to defined **Bolt Pretension** (p. 1566) boundary conditions that you activate using the **Worksheet**. **Bolt Pretension** boundary condition and **Worksheet** usage are the only supported methods for specifying the data of a **Bolt Tool**. Therefore, all of the support and scoping requirements of the **Bolt Pretension** boundary condition must be considered.

Result Types

The **Bolt Tool** provides the following results:

- **Adjustment**: This represents the displacement that occurs from the pretension. In Mechanical APDL terms, this is the displacement reported from the pretension node. This result is also available for reporting regardless of how the bolt is defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Working Load**: This represents a constrained force reaction from the pretension load. In Mechanical APDL terms, this is the constrained reaction reported from the pretension node. This is essentially the sum of all the forces acting through the pretension cut. This result is applicable for load steps when the load is defined by either **Locked** or **Adjustment** or **Increment**.

Application

To define results using the **Bolt Tool**:

1. Make sure that all Bolt Pretension (p. 1566) loads are properly defined.
2. Highlight the **Solution** object and select **Bolt Tool** from **Toolbox** drop-down menu on the **Solution** Context Tab (p. 62). You can also right-click the **Solution** object and select **Insert** > **Bolt Tool** > **Bolt Tool**. The **Bolt Tool** becomes the active object in the tree, it contains an **Adjustment** result object by default, and the **Worksheet** displays.
3. All existing **Bolt Pretension** loads populate the **Worksheet**. By default, all loads are applied, as shown by the activated check boxes for each table row. You can modify the table entries as needed.

   Add **Adjustment** and **Working Load** results to the tool as required.

   In addition, the following right-click menu options are available in the **Worksheet**:

   - **Scope To All Bolts**
   - **Clear Scoping**
   - **Activate all Selections**
   - **Deactivate all Selections**

Beam Tool

You can apply a **Beam Tool** to any assembly in order to view the linearized stresses on beam (line) bodies. It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately. Therefore, the stress outputs (which are linearized stresses) associated with beam bodies have been focused toward that design goal.

Note:

Review the following limitations for the **Beam Tool**:

- The **Beam Tool** is intended for beam models that feature structural results.
- The **Beam Tool** does not support bending or combined stress results when scoped to a geometry that:
  - Includes a User Integrated cross-section (or):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

  - Includes a user-defined cross-section when the **Cross Section (For Solver)** property (p. 875) is set to the **Pre-Integrated** option. Specify the **Mesh** setting for this property in order to obtain the stress results.
  - Originated from the SpaceClaim Eurocode Library.

- The **Beam Tool** is not supported for Modal, Harmonic Response, Eigenvalue Buckling, Random Vibration, Response Spectrum, Rigid Dynamics, or Structural Optimization analyses.
- Certain User Defined Results, such as BEAMDIRECT, could be available for an analysis (above) that does not support the **Beam Tool**, however, these results are not One-Sigma results.
- The **Beam Tool** is not available to the ABAQUS AUTODYN, LSDYNA, or Samcef solvers.

The **Beam Tool** is similar to the **Contact Tool** (p. 2079) in that the tool, not the results themselves control the scoping. By default, the scoping is to all beam bodies. You can change the scoping in the Details, if desired.

To insert a **Beam Tool**, highlight the **Solution** object then select **Beam Tool** from the **Toolbox** drop-down menu. Three beam stress results are included under the **Beam Tool** object: **Direct Stress**, **Minimum Combined Stress**, and **Maximum Combined Stress**. You can add additional beam stress results or deformation results (p. 2018) by highlighting the **Beam Tool** object and choosing the particular result from the **Beam Tool** Context tab. As an alternative, you can right-click the **Beam Tool** and select **Insert** > **Beam Tool** > [desired result].

Presented below are definitions of the beam stress results that are available:

- **Direct Stress**: The stress component due to the axial load encountered in a beam element.
- **Minimum Bending Stress**: From any bending loads a bending moment in both the local Y and Z directions will arise. This leads to the following four bending stresses: Y bending stress on top/bottom and Z bending stress on top/bottom. **Minimum Bending Stress** is the minimum of these four bending stresses.
- **Maximum Bending Stress**: The maximum of the four bending stresses described under **Minimum Bending Stress**.
- **Minimum Combined Stress**: The linear combination of the **Direct Stress** and the **Minimum Bending Stress**.
- **Maximum Combined Stress**: The linear combination of the **Direct Stress** and the **Maximum Bending Stress**.

Caution:

- Be cautious when adding **Beam Tool** results to the **Solutions Combination** (p. 2288) object. As stated above, the **Beam Tool** minimum and maximum results can originate from one of four different physical locations. As a result, the application could add

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

solution results from different physical locations. For this reason, carefully review stress results used with the **Solutions Combination** feature.

- **Beam Tool** results characterize the elastic behavior of a beam body *only*. You must be keenly aware of this when interpreting **Beam Tool** results on beam bodies whose material behaviors may become nonlinear. Large discrepancy can exist between the linearized Beam Tool results and the actual material stresses obtained from nonlinear constitutive relationships. Review the BEAM188, BEAM189, PIPE288, and PIPE289 element descriptions for more information.
- For the **Maximum Bending Stress** and **Minimum Bending Stress** result types, the application compares the Y & Z bending stresses on the top and bottom of the beam. These four basic bending stresses are contained in the local element coordinate system. They are **not** rotated to any other coordinate system prior to calculation.

Beta Feature: Currently, when you activate the **Beta Option** from the Workbench Project page (**Tools** > **Options** > **Appearance**), two new expressions (p. 2197) are generated in the Worksheet: BEAMMIX_TOTAL_COMBINED and BEAMMAX_TOTAL_COMBINED. These expressions enable you to create a **User Defined Result** for the linear combination of the **Direct Stress** and the sum of the minimum/maximum Y bending stress and minimum/maximum Z bending stress (**Total Minimum Combined Stress** and **Total Maximum Combined Stress**).

Beam Results

Beam results can be applied only to line body edges and are defined as follows in reference to the solution coordinate system of each beam or pipe element:

- **Axial Force**: the force along a beam element axis (X component).
- **Bending Moment**: the moment in the plane perpendicular to the beam element axis (Y and Z components).
- **Torsional Moment**: the moment about the beam element axis (X component).
- **Shear Force**: the force perpendicular to the beam element axis (Y and Z components).
- **Shear-Moment Diagram** (p. 2090): simultaneously illustrates the distribution of shear forces, bending moments and displacements, as a function of arc length along a path consisting of line bodies.

To apply a beam result, define a path by using edges, on the line body edges as described in "Defining a Path using an Edge" in Path (p. 883). For Shear-Moment Diagrams, the defined line body edges must be contiguous.

Beam results are not available to the Samcef or ABAQUS solver.

Note:

- User Defined Result (p. 2192) equivalents of the above results are BEAM_AXIAL_F, BEAM_BENDING_M, BEAM_TORSION_M, and BEAM_SHEAR_F.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- An **Axial Force** display will not include an arrow (that is, a vector). The force consists of only the X component. A positive force denotes tension: a negative force denotes compression.
- If a path is coincident with an edge, beam results from scoping to the path may not match beam results from scoping to the edge. The path for beams only allows contributions from beam elements with both endpoints in the path. An edge can allow contributions from elements that have only one node on the edge.

Shear-Moment Diagram

A shear-moment diagram is a beam result (p. 2089) that you can apply only to paths, which simultaneously illustrates the distribution of shear forces, bending moments and displacements, as a function of arc length along the path consisting of line bodies.

These three quantities are included in a shear-moment diagram because they are so closely related. For example, the derivative of the moment is the shear:

$dM/dx = V(x)$

You can pre-define the path by selecting a contiguous set of line body edges, then inserting a **Shear-Moment Diagram** object in the tree. Insert from the **Beam Results** drop-down menu on the **Solution** Context tab, or by a right-click the **Solution** folder and choosing **Insert> Beam Results** from the context menu.

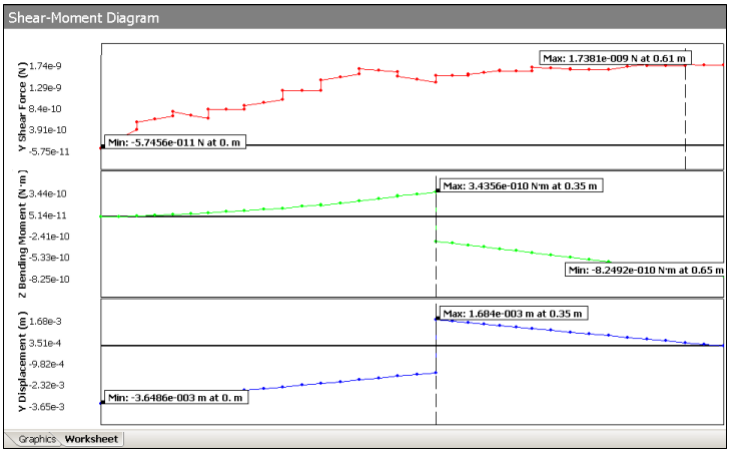
With the **Shear-Moment Diagram** object highlighted, the **Path**, **Type** and **Graphics Display** settings in the Details control the curves you can display in the **Worksheet** or the **Graph** window. Descriptions are presented below. When the X, Y, or Z component is indicated, they are in the local coordinate system whose X axis is directed instantaneously along the beam. The Y and Z axes can be inspected using an Element Triad (p. 1936) result. All **Type** and **Graphics Display** directions are referenced to this axis.

- **Path**: The specific path to which the shear-moment diagram is to apply. For ease of use, before inserting the **Shear-Moment Diagram** object, you can define the path by selecting a contiguous set of line body edges. You can choose to use this path or any other pre-defined paths (p. 883) that you have created for other path results (p. 1940).
- **Type**: The shear-moment diagram to display. Choices are:
  - **Total Shear-Moment Diagram**
  - **Directional Shear-Moment Diagram (VY-MZ-UY)**
  - **Directional Shear-Moment Diagram (VZ-MZ-UZ)**
- **Graphics Display**: Controls which quantity is plotted in the **Graph** window and reported as **Minimum** and **Maximum** values in the Details.

Example in Worksheet:

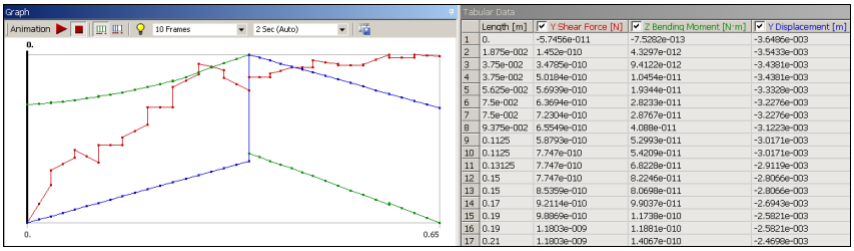
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



You can toggle the display of all the **Max** and **Min** annotation labels by right-clicking anywhere in the top diagram and choosing **Hide/Show Annotation Labels**.

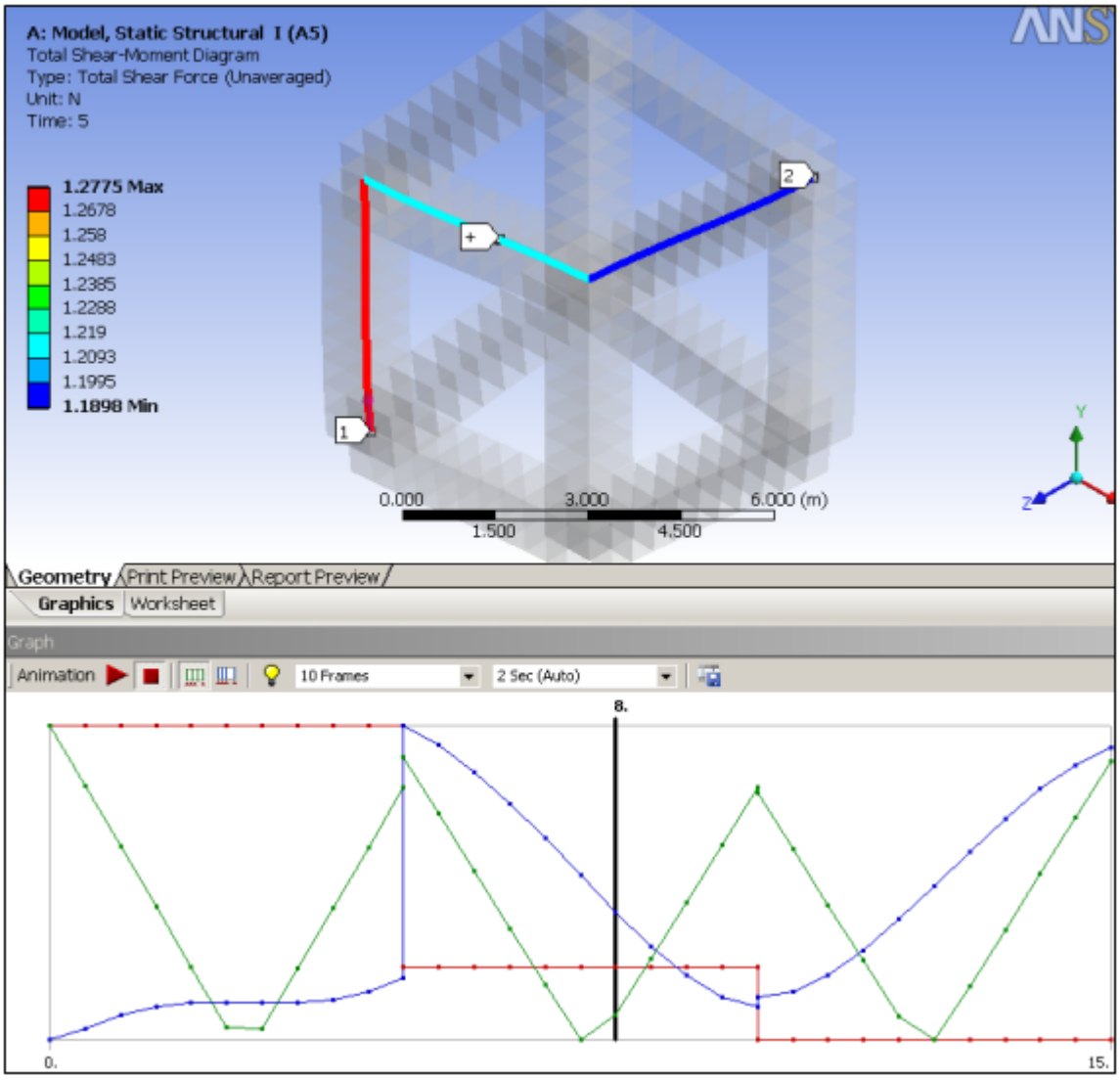
Example in Graph and Tabular Data Windows:



Example of Tracking Graph with Path Position:

When you click anywhere along the **Length** axis, the vertical bar and length that display corresponds to the position of the **+** annotation on the path as shown below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Structural Probes

This section provides:

- A general description of structural probe types (p. 2093).
- An explanation of the differences for rigid body probes (p. 2102) during Explicit Dynamics analyses.

Additional Probe Information

See the Probes (p. 1946) section for further information. In addition, see the following sections for details on these probe types:

Position

Energy

Reactions: Forces and Moments

Joint Probes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Response PSD Probe
Spring Probes
Bearing Probes
Beam Probes
Bolt Pretension Probes
Generalized Plain Strain Probes
Contact Distance Probes
Fracture Probes (Fracture Tool)

Structural Probe Types

The following structural probe types are available.

Probe Type	Applicable Analysis Types	Output	Characteristics
Deformation	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Rigid Dynamics, Explicit Dynamics, LS-DYNA	Deformation: X axis, Y axis, Z axis, Total	***Scope to:*** Flexible bodies, a single rigid body, or Remote Point. ***Scope by:*** Bodies (single body only if rigid), location (Hit Point Coordinate (p. 97)), vertex, edge, face, user-defined Coordinate System, or user-defined Remote Point. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Strain	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Explicit Dynamics	Strain: Components, Principals, Normal X, Normal Y, Normal Z, XY Shear, YZ Shear, XZ Shear, Minimum Principal, Middle Principal, Maximum Principal, Intensity, Equivalent (von-Mises)	***Scope to:*** Flexible body only. ***Scope by:*** Bodies, location (Hit Point Coordinate (p. 97)), vertex, edge, face. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
Stress	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Explicit Dynamics,LS-DYNA	Stress: Components, Principals, Normal X, Normal Y, Normal Z, XY Shear, YZ Shear, XZ Shear, Minimum Principal, Middle Principal, Maximum Principal, Intensity, Equivalent (von-Mises)	***Scope to:*** flexible body only. ***Scope by:*** Bodies, location (Hit Point Coordinate (p. 97)), vertex, edge, face. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Position (p. 2102)	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Rigid Dynamics, Explicit Dynamics	Result Selection: X axis, Y axis, Z axis	***Scope to:*** Rigid body only. ***Scope by:*** Bodies, coordinate system. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Flexible Rotation Probe	Coupled Field Static, Coupled Field Transient, Static Structural, and Transient Structural	Rotation of X, Y, and Z axes (in the Global Coordinate System only)	***Scope to:*** User-defined Remote Point or a Body. Note that this probe requires rotational degrees of freedom data (ROTX/ROTY/ROTZ), which is commonly associated with shell and beam bodies, but not solid bodies. If unavailable, the application displays result values of **0**.[a]
Velocity	Coupled Field Transient, Transient Structural, Rigid Dynamics,	Velocity: X axis, Y axis, Z axis	***Scope to:*** Flexible or rigid body. ***Scope by:*** Bodies (single body only if rigid), coordinate system (rigid bodies only), location (Hit Point Coordinate (p. 97)), vertex, edge, face.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
	Explicit Dynamics, LS-DYNA		***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Angular Velocity	Coupled Field Transient, Transient Structural, Rigid Dynamics	Angular Velocity: X axis, Y axis, Z axis	***Scope to:*** Rigid body only. ***Scope by:*** Bodies. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Acceleration	Coupled Field Transient, Transient Structural, Rigid Dynamics, Explicit Dynamics, LS-DYNA	Acceleration: X axis, Y axis, Z axis	***Scope to:*** Flexible or rigid body. ***Scope by:*** Bodies (single body only if rigid), coordinate system (rigid bodies only), location (Hit Point Coordinate (p. 97)), vertex, edge, face. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Angular Acceleration	Coupled Field Transient, Transient Structural, Rigid Dynamics	Angular Acceleration: X axis, Y axis, Z axis	***Scope to:*** Rigid body only. ***Scope by:*** Bodies. ***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Energy (p. 2105)	Coupled Field Harmonic, Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Rigid Dynamics, Explicit Dynamics	For Static Structural and Transient Structural: Kinetic, Strain, Damping, Artificial, and Nonlinear Stabilization. For Rigid Dynamics: Kinetic, Potential, External, and Total	**Scope to** your model using the **Geometry** property: **System Energy** is the default setting. Otherwise, only body scoping is supported for either flexible or rigid bodies. ***Scope by*** using the **Result Selection** property: • For Transient Structural and Static Structural: **All** (default) energy types or per body for the **Kinetic**, **Strain**, **Damping**, **Artificial**, or **Nonlinear Stabilization** options. • For Transient Rigid: **All** (default) energy types or per part for the **Kinetic**,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
		For Coupled Field Harmonic: Kinetic, Potential, and Damping For Coupled Field Static and Coupled Field Transient: Kinetic, Strain, Damping, Artificial, Nonlinear Stabilization, and Total. For Explicit Dynamics: Internal, Kinetic, Plastic Work, Hourglass, Contact, and Total	**Potential**, **External**, and **Total** options. **Note:** For the **External** and **Total** options, you must use the **System Energy** setting for the **Geometry** property setting. • For Coupled Field Harmonic: **All** (default) energy types or per body for the **Kinetic**, **Potential**, and **Damping** options. • For Coupled Field Static and Coupled Field Transient: **All**, **Kinetic**, **Strain**, **Damping**, **Artificial**, **Nonlinear Stabilization**, and **Total**. • For Explicit Dynamics: **All** (default) energy types or per body for the **Internal**, **Kinetic**, **Plastic Work**, **Hourglass**, **Contact**, and **Total** options.
Volume (p. 2152)	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Steady-State	Volume	***Scope to:*** Body. ***Scope by:*** Body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
	Thermal, Transient Thermal, Electric, and Thermal-Electric.		
Force Reaction (p. 2107)[b]	Coupled Field analyses, Static Structural, Transient Structural, Modal, Harmonic Response, Random Vibration, Response Spectrum, Explicit Dynamics[c]	Force Reaction: X axis, Y axis, Z axis	***Scope to:*** Flexible body, vertex, edge, or face[d]. In Explicit Dynamics analyses probes can also be scoped to faces, edges, and vertices of a rigid body. You can also scope to a section plane on a body by specifying **Surface** as the **Location Method**. ***Scope by:*** Boundary Condition[e], Contact Region (p. 2117), Remote Points[f], Beams[f], Springs[f], Mesh Connection (p. 2117), and Surface[g]. ***Orientation coordinate system:*** Any Cartesian: defaults to Global Cartesian. **Solution Coordinate System** (p. 1975) is the only valid option for Random Vibration and Response Spectrum.
Moment Reaction (p. 2107) [b]	Coupled Field analyses, Static Structural, Transient Structural, Modal, Harmonic, Random Vibration, Response Spectrum, Explicit Dynamics[c]	Moment Reaction: X axis, Y axis, Z axis	***Scope to:*** Flexible body, vertex, edge, or face[d]. In Explicit Dynamics analyses probes can also be scoped to faces, edges, and vertices of a rigid body. You can also scope to a section plane on a body by specifying **Surface** as the **Location Method**. ***Scope by:*** Boundary Condition[e], Contact Region (p. 2117), Remote Points[f], Beams[f], Mesh Connection (p. 2117), and Surface[g]. ***Orientation coordinate system:*** Any Cartesian: defaults to Global Cartesian. **Solution Coordinate System** (p. 1975) is the only valid option for Random Vibration and Response Spectrum.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
			Summation point: Centroid or orientation coordinate system.
Joint (p. 2118)	Static Acoustics, Static Structural, Transient Structural, Coupled Field Static, Coupled Field Transient, Explicit Dynamics, LS-DYNA, LS-DYNA Restart Analysis, Rigid Dynamics Analysis, and Ansys Motion[h]	See Joint Probes (p. 2118)	***Scope to:*** Joint only. ***Orientation coordinate system:*** The **Orientation Method** property for a Joint Probe defaults to the read-only setting, **Joint Reference System**. This option is the same for all outputs. Only Cartesian coordinate systems are valid for a Joint Probe. ***Summation point:*** Always at joint for Moment.
Spring (p. 2123)	All analysis types including Rigid Dynamics and LS-DYNA[h]	Elastic Force[i], Damping Force[j], Elongation, Velocity[k]	**Scope to:** Spring only. ***Orientation coordinate system:*** Spring axis only.
Bearing (p. 1413)	Coupled Field analyses, Static Structural, Transient Structural, Modal, Harmonic Response, Random Vibration, Response Spectrum	Elastic Force 1, Elastic Force 2, Damping Force 1, Damping Force 2, Elongation 1, Elongation 2, Velocity 1, Velocity 2	***Scope to:*** Bearing only. ***Orientation coordinate system:*** Bearing axes only.
Beam (p. 2125)	Coupled Field Static,	Axial Force, Torque,	***Boundary Condition:*** Select beam.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
	Coupled Field Transient, Static Structural, Transient Structural, LS-DYNA[h]	Shear Force at I, Shear Force at J, Moment at I, and Moment at J	
Bolt Pretension (p. 2125)	Coupled Field Static, Coupled Field Transient, Static Structural, Transient Structural, Random Vibration, Response Spectrum	Adjustment (Static and Transient Structural), Tensile Force	***Scope by:*** Boundary condition (Y pretension bolt condition). ***Orientation coordinate system:*** Along pretension direction only.
Generalized Plane Strain (p. 2125)	2D: Static Structural, Transient Structural	Rotation: X, Y: Moment: X, Y: Fiber Length Change: Force	***Orientation coordinate system:*** Any: defaults to Global Cartesian.
Response PSD (p. 2121) [l]	Random Vibration	X axis, Y axis, and Z axis. Displacement, Stress, Strain, Acceleration, Velocity	***Scope to:*** Flexible body only. ***Scope by:*** Location (Hit Point Coordinate (p. 97)) and vertex. ***Orientation Coordinate System:*** **Solution Coordinate System** (p. 1975) is the only valid option for Random Vibration.
Contact Distance (p. 2126)	Rigid Dynamics Only	The varied distance	***Scope to:*** Contact Region.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
		between the **Contact** and **Target** sides of the specified **Contact Region** for each time point in the analysis.	***Scope by:*** Contact Region.

[a] Solid bodies, which are composed of solid elements, have no rotational degrees of freedom, and therefore calculate maximum x, y, and z values as zero (Axis specified under the **Results** category equals **0**). However, if a solid body shares nodes with shell bodies or line bodies and if the scoping includes the solid body only, then this probe can report non-zero values (resulting from the shared nodes). But these rotation values may not represent the rotation of the body as a whole. Furthermore, if the body shares nodes with an element whose nodes include rotational degrees of freedom, such as contact or spring elements, or some condition that has inherent motion, the probe may also produce rotation values that do not represent the rotation of the body.

[b] **Force Reaction**and **Moment Reaction** probes:

- Will not solve if scoped to a **Contact Region** that includes a rigid body.
- Do not support **Mesh Connections** for Modal and Harmonic Response analyses.
- Do not support the **Location Method** option **Contact Region** when the corresponding **Contact Region** is scoped to element faces.
- A limitation exists when the scoping of a probe is applied to a geometric entity (**Location Method** = **Geometry Selection**) that shares more than one body. The (unscoped) elements that are adjacent to the scoped body contribute to the probe's results.

[c] For Explicit Dynamics, the only valid options for the **Location Method** property are **Geometry** or **Boundary Condition**.

[d] When you scope this result to geometry, including contact surfaces and cut surfaces, the application calculates the probe using element-nodal data. These calculations are equivalent to those of the **FSUM** command. That is, the sums for each component direction for the total selected node set and the nodal force and moment contributions of the selected elements attached to the node set.

Note:

You should examine your results carefully for geometry scoping. If all elements are selected, the sums of the result are usually zero except where constraints or loads are applied. Element-nodal results for geometry-based scoping should be the same as the node-based results when you specify **Boundary Condition** as the **Location Method** if the geometry-based scoping is the same as the boundary condition scoping. However, because of certain limitations associated with how Mechanical calculates scoping, and perhaps based on a model's geometry, the application may produce unexpected element-nodal results.

[e] When you specify **Boundary Condition** as the **Location Method**, the application calculates **Force Reaction** or **Moment Reaction** probes using nodal data. These calculations are

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

equivalent to those of the **PRRSOL** command, which provides the total reaction solution for the selected nodes.

[f] Remote Points must be constrained and Beams and Springs must be grounded.

[g] Setting the **Location Method** to **Surface** requires that you specify the following additional properties:

- **Surface**: Select a construction **Surface** (p. 890) created using the Construction Geometry (p. 883) feature.
- **Geometry**: Select the body or bodies that the construction **Surface** intersects (slices through).
- **Extraction**: Option include **Mesh From Positive Side** (default) and **Mesh From Negative Side**. The probe only examines the elements cut by the plane and only the nodes of those elements which are on specified side of the plane, positive or negative. The positive side is in the positive local Z direction from the construction surface and negative side is in the negative local Z direction.

Note:

Surfaces used for reaction probes do not currently intersect all geometries. For example, the feature does not intersect line bodies or bodies that include a joint, spring, or MPC contact.

Note:

This option has certain inherent limitations. The probe's results can be affected by:

- An application employed tolerance as you cut through the mesh. This calculation creates a small thickness value. This could allow nodes not included in the construction surface to be used in the solution.
- Facets shared by elements included in the construction surface.
- Loads and constraints that produce a solution in the body that is essentially zero.

If you experience doubts about your result values, compare reactions scoped to a surface against the reaction solutions scoped to boundary conditions and scoped to contact elements.

[h] For LS-DYNA, if your project was solved in a version earlier than 2020 R1, you will not be able to evaluate the following probes: **Spring**, **Beam Connection**, and **Joint**.

[i] Random Vibration and Response Spectrum analyses support the **Elastic Force** result only.

[j] The Damping Force result is calculated for Transient Structural analysis only when damping is defined.

[k] Velocity result is calculated only for Static Structural, Transient Structural, Rigid Dynamics, and LS-DYNA analyses.

[l] The Response PSD Probe (p. 2121) provides an excitation response plot across the frequency domain of an input PSD load. It also evaluates the root mean square (RMS) and expected frequency of a response PSD. It is assumed that the excitations are stationary random processes from the input PSD values.

Note:

- Refer to the Probe Details View (p. 1951) section for additional information about the above scoping options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- For more information about Explicit Dynamics, see Force Reaction and Moment Reaction Result Trackers for Explicit Dynamics in the *Explicit Dynamics Analysis Guide*.
- For a linked Mode-Superposition Harmonic Response analysis, the **Expand Results From** property (see Output Controls (p. 1473)) in the Harmonic Response analysis must be set to **Harmonic Solution** in order to support the Elongation result.
- For a linked Mode-Superposition Transient Structural analysis, the **Expand Results From** property (see Output Controls (p. 1473)) in the Transient Structural analysis must be set to **Transient Solution** in order to support the Elongation result and, if damping is defined, the Damping Force result.

Differences in Probes Applied to Rigid Bodies

The following table describes the differences between probes applied to rigid bodies in an Explicit Dynamics analysis, compared to probes applied to rigid bodies in a Static Structural or Transient Structural analysis.

Characteristic	**Explicit Dynamics Analysis**	**Static Structural or Transient Structural Analysis**
How rigid part is meshed	Meshed with solid element containing multiple nodes.	Meshed as a single element containing a single node.
Centroid of the rigid part	Need not be represented by any node in the mesh. The Mechanical application computes the part centroid by averaging the element centroids. Each element centroid is the average of the element's nodes.	Results at the single node represent the displacement, velocity, etc. at the centroid of the part.
Display of minimum and maximum results	Probe applied to rigid body displays both the minimum and maximum results at a given time because there are multiple elements and nodes reporting results. The position probe represents the sum of the minimum (or maximum) displacement with the average nodal coordinate.	Probe applied to rigid body does *not* display both the minimum and maximum results at a given time because there is only one element and one node reporting results.

Position

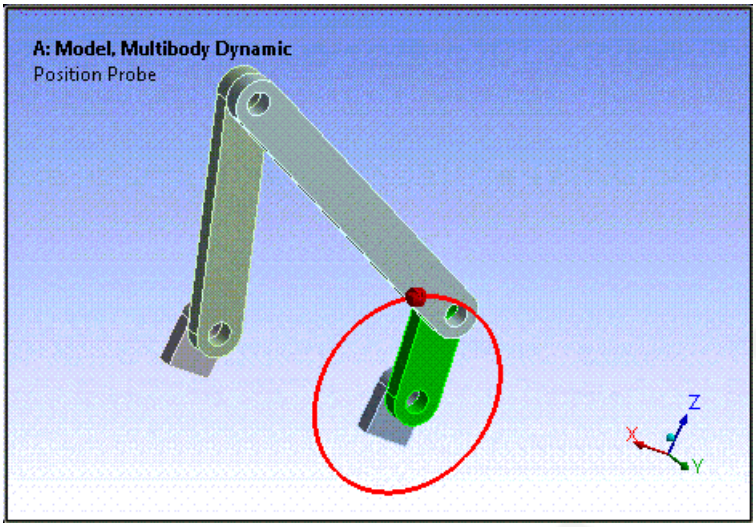
For this probe, result "positions" are created for time points along selected coordinate system axis or axes. These result values are plotted in the Graph window. This probe must be scoped to a rigid body. It is supported for Static Structural, Transient Structural, Rigid Dynamics, and Explicit Dynamics analyses.

Using the result values in **Tabular Data**, you can animate the scoped body in motion. In addition, if you set the **Result Selection** property to **All**, the **Position** probe provide a unique graphical

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

display feature. It displays a “trace” or red line that follows the movement of the scoped body. The trace is based on the result values of the scoped body for all three axes over time (as contained in **Tabular Data**). The example below shows a rotation. Linear movements are also supported. The small red ball shows the current position along the curve.



Rigid Dynamics Solver

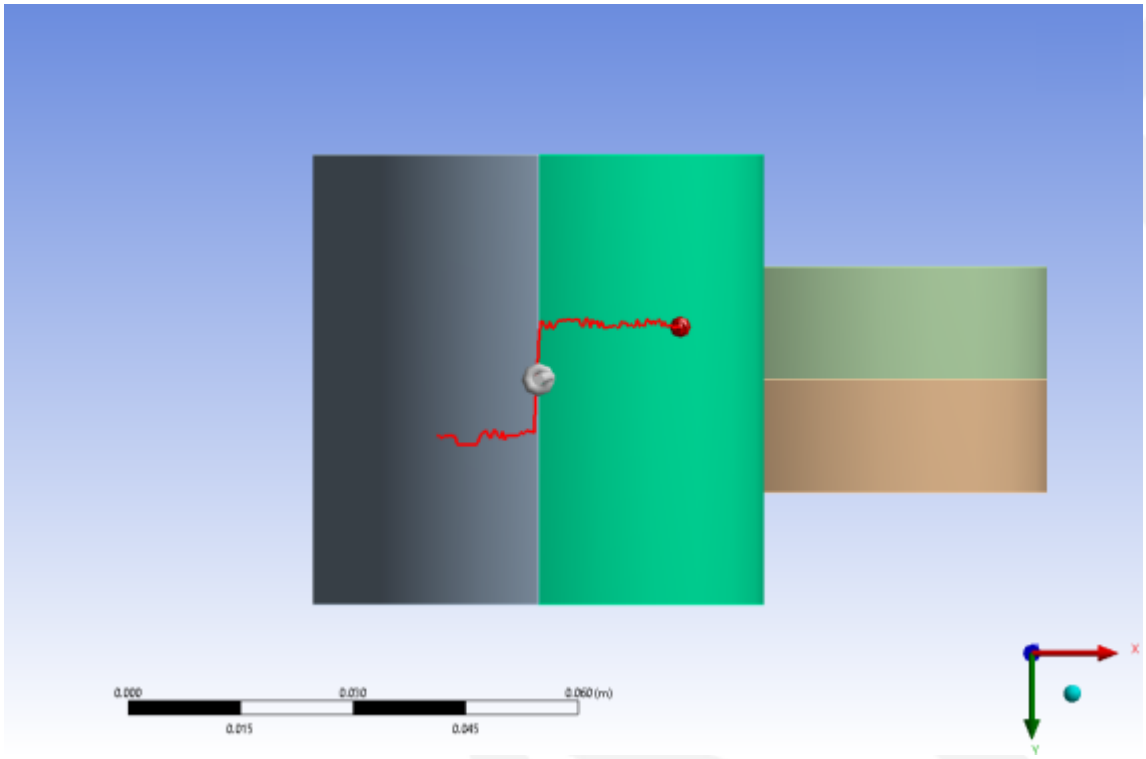
With the Rigid Dynamics Solver, the position probes can also be used to report the location of the resulting contact force. In order to get the contact force location, the **Location Method** field must be set to Contact Region. Then the **Contact Region** drop-down allows you to select the contact region for which the position will be reported.

Note:

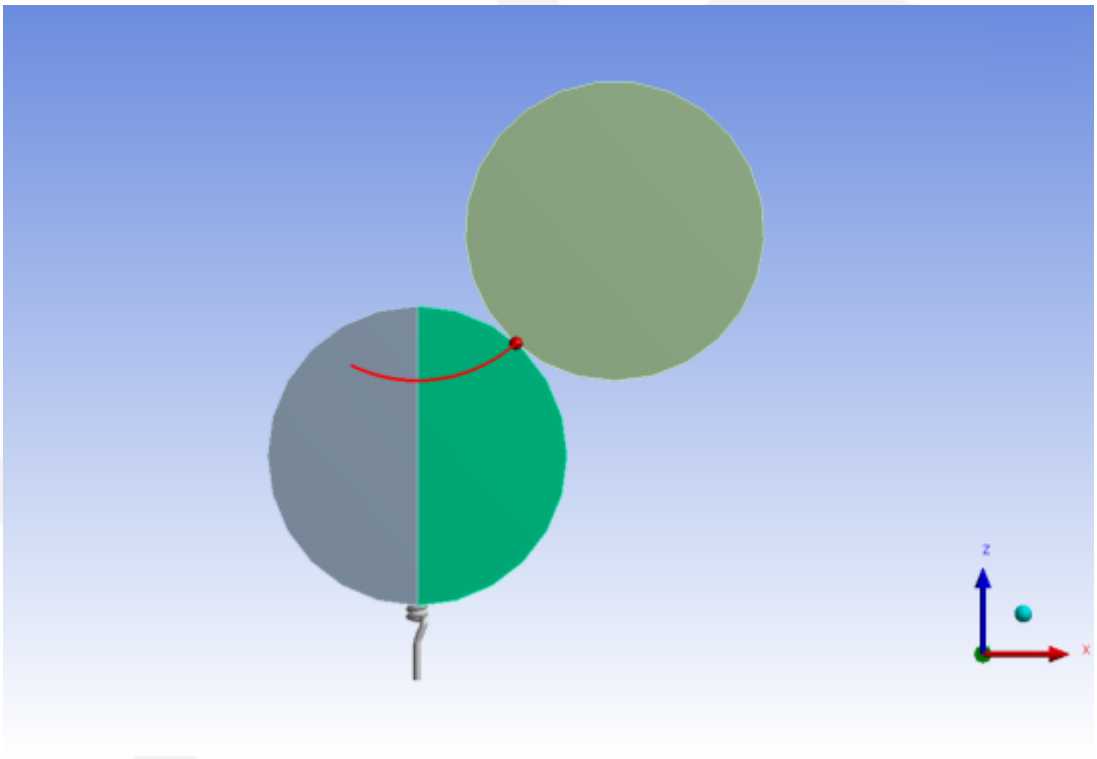
Contact regions between the same pair of parts are merged into a single contact region. Consequently, the probes will report the same values for the entire contact region.

If the contact is open, the position probe will report zero for all components, leading to abrupt jumps to the global origin in the trajectory plot. In some situations, the position probe seems to report erratic location information. Typically, in a cylinder/plane frictionless contact, the contact forces/torque can be equally reported anywhere along the contact line as in (a) below. The position in the plane perpendicular to the contact line is perfectly consistent in that case as in (b) below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



(a) contact probe viewed in X-Y



(b) contact probe viewed in X-Z

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Energy

This probe and the available energy outputs for the supported analysis types are listed below.

Piezoelectric and thermal coupled **Coupled Field Structural** and **Coupled Field Transient** analyses support the following energy outputs:

Kinetic: Kinetic energy due to the motion of parts in the analysis.

Strain: Energy stored in bodies due to deformation. This value is computed from stress and strain results. It includes plastic strain energy as a result of material plasticity.

Damping: Represents the average elastic and electric losses.

Artificial: Total artificial energy of the selected body(s). The artificial energy is a sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). It also includes artificial energy due to contact stabilization.

Nonlinear Stabilization: Total energy dissipation due to numerical stabilization in the selected body(s).

Total: This is the sum of all energies in the analysis.

A Piezoelectric analysis (**Coupled Field Harmonic**) supports the following energy outputs:

Kinetic: Kinetic energy due to the motion of parts in the analysis.

Potential: Sum of elastic strain energy and dielectric energy.

Damping: Represents the average elastic and electric losses.

Artificial: Total artificial energy of the selected body(s). The artificial energy is a sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). It also includes artificial energy due to contact stabilization.

Nonlinear Stabilization: Total energy dissipation due to numerical stabilization in the selected body(s).

Total: This is the sum of all energies in the analysis.

The **Static Structural** and **Transient** structural analyses supports the following energy outputs:

Kinetic: Kinetic energy due to the motion of parts in the analysis.

Strain: Energy stored in bodies due to deformation. This value is computed from stress and strain results. It includes plastic strain energy as a result of material plasticity.

Damping: Total damping energy of the selected body(s).

Artificial: Total artificial energy of the selected body(s). The artificial energy is a sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). It also includes artificial energy due to contact stabilization.

Nonlinear Stabilization: Total energy dissipation due to numerical stabilization in the selected body(s).

Total: This is the sum of all energies in the analysis.

A **Rigid Dynamics** analysis supports the following energy outputs:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Kinetic: Kinetic energy due to the motion of parts in a transient analysis is calculated as ½ *mass* velocity^2 for translations and ½ *omegaT*Inertia*omega for rotations.

Potential: This energy is the sum of the potential energy due to gravity and the elastic energy stored in springs and deformable bodies. The potential energy due to gravity is proportional to the height of the body with respect to a reference ground. The reference used in a Rigid Dynamics analysis is the origin of the global coordinate system. Because of this, it is possible to have a negative potential energy (and negative total energy) depending on your model coordinates. The elastic energy includes energy due to deformation of spring(s) in a rigid body dynamic analysis and is calculated as ½ * Stiffness * elongation^2. The elastic energy of the deformable bodies is calculated as $\frac{1}{2}U^TKU$ where K is the stiffness matrix and U is the elastic displacement.

External: This is all the energy the loads and joints bring to a system.

Total: This is the sum of potential, kinetic and external energies in a Rigid Dynamics analysis.

Note:

Energy results are not available for Rigid Dynamics analysis on a body per-body basis. An energy probe scoped on a body will return the energy of the whole part to which body belongs.

Note:

For stop, contact, and imperfect joints (radial gap, spherical gap and in-plane radial gap), the energy loss due to non-perfectly elastic collision (restitution factor<1) and/or friction is not taken into account in the external energy. Therefore, the total energy balance is not maintained and is expected to decrease.

An **Explicit Dynamics** analysis supports the following energy outputs:

Internal:

This is the energy stored in bodies due to deformation. This includes both Plastic Work and Hourglass Energies.

Kinetic:

Kinetic energy due to the motion of parts in the analysis.

Plastic:

This energy is the plastic strain energy as a result of material plasticity.

Hourglass:

The 2D and 3D Lagrangian volume and shell elements use hourglass control to remove zero energy modes of deformation. The forces of the hourglass control algorithms dissipate energy in resisting the zero energy modes of deformation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

For this analysis, this energy is calculated and accumulated in the element internal energy. The hourglass energy is also accumulated independently to the internal energy to allow users to interrogate and assess the relevance of hourglass energy in a simulation.

Contact:

Contact energy is accumulated on a per body basis as:

$$E^{n+1}_{nbody} = E^{n}_{nbody} + \Sigma_{Nodesonbody} F^{i}_{contact} \times \Delta x^{i}_{node}$$

For the purpose of energy accountancy, the contact energy is added to the work done on the system.

Total:

This is the sum of internal and kinetic energies in the analysis.

Reactions: Forces and Moments

You can obtain reaction forces and moments using **Force Reaction** probes or **Moment Reaction** probes. At the solver level, the output of reaction forces and moments is controlled via the Mechanical APDL **OUTRES** command. Support types marked RSOL are governed by the RSOL option, which refers to nodal constraint reactions. Those marked NLOAD and MISC are governed by the NLOAD and MISC options, which refer to the elemental nodal loads and elemental miscellaneous data, respectively. In addition, some analysis and support types require you turn them on in the **Output Controls**. If no setting is specified for a reaction type, the output occurs automatically.

When you request a **Force Reaction** or a **Moment Reaction** in a Cartesian coordinate system at a specific time point by setting **Display** to **Single Time Point** in the Details for Static Structural and Transient Structural Analysis, the **Force Reaction** or **Moment Reaction** is displayed by an arrow in the **Geometry** window. **Force Reaction** uses a single arrowhead and **Moment Reaction** uses double arrowhead. The arrows are drawn on the deformed mesh.

Similarly, when the force or moment reaction results are requested based on **Frequency** or **Set Number** and **Phase Angle** for Harmonic analysis or **Mode Number** for Modal analysis, the base of the arrow of the moment probe is placed at the **Summation Point** (or "centroid": the simple calculated average: unweighted by length, area, or volume). However, a **Moment Reaction** probe whose **Location Method** is a remote point will place the base of the arrow at the location of the remote point. In this case, there is no detail for Summation Point, and Mechanical does not employ a moment arm calculation. The moments are precisely the nodal moments for the remote point in the result file (as printed by the **PRRSOL** command in Mechanical APDL).

Large Scale Deformation

For large scale deformation analyses (**NLGEOM**,ON), both Mechanical APDL and Mechanical use the displaced mesh when calculating moment arms for moment probes:

```
moment_arm = (x+ux,y+uy,z+uz) - (sx,sy,sz)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Where **sx,sy,sz** is the setting of the **Summation** property, **x,y,z** is the nodal location, and **ux,uy,uz** is the displacement at the node location.

Important:

Currently for Mechanical, the specified **Summation** is not modified (or offset) to account for the displacements (or deformations).

When **NLGEOM ,OFF** is specified, the displacements are not used in the moment arm calculation:

```
moment_arm = (x,y,z) - (sx,sy,sz)
```

Rigid Dynamics Solver

For the Rigid Dynamics solver, when force or moment reaction probes use Contact Region for the **Location Method**, it allows you to display the resulting contact forces on a specific contact region. The contact region can be picked using the **Contact Region** drop-down menu. The **Contact Force** menu allows you to display either the Total force, the Normal, or the Tangent force. By definition, frictionless contact always reports a normal force and the tangent force reports zero for all components.

Note:

Contact regions between the same pair of parts are merged into a single contact region. Consequently, the probes will report the same values for the entire contact region.

Reaction Type Output Settings

The following topics discuss each type of reaction, the option that controls the output, and any required setting in the **Output Controls**. In addition, at the end of the section there is a listing of Support Requirements and Limitations (p. 2115). Make sure you review this content.

Fixed Boundary Conditions

For fixed boundary conditions, including:

- Face, Edge, and Vertex Rotations (do not include Force reactions)
- Displacements for Faces, Edges, and Vertices
- Cylindrical Support
- Frictionless face
- Simply Supported Edge and Vertex
- Finite Element (FE) Connection Boundary Conditions (Nodal Displacement and Nodal Rotation)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Reaction Type	Output Controlled By
Static Transient - Full	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Modal Harmonic Response - Full	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. To enable the output, set **Calculate Reactions** = **Yes** in the **Output Controls**.
Harmonic Response - Mode-Superposition Transient - Mode-Superposition	If results are expanded from a modal solution, then the output of these options are controlled by both the RSOL and NLOAD options of the **OUTRES** command. You must set both **Calculate Reactions** and **Nodal Forces** to either **Yes** or **Constrained Nodes** in the **Output Controls**. **Note:** **Constrained Nodes** is the preferred option, as the results file size will be smaller and the process time shorter. Otherwise, the output of these options are controlled by the **RSOL** option of the **OUTRES** command. Set **Calculate Reactions** = **Yes** in the **Output Controls**. **Important:** Force and Moment Reaction probes cannot be evaluated when you specify damping in combination with setting the **Expand Results From** property to either **Harmonic Solution** or **Transient Solution**. Review the **MXPAND** command in the *Mechanical APDL Command Reference* for additional information about this limitation.
Response Spectrum Random Vibration	In the upstream Modal analysis, under the Analysis Settings **Output Controls** (p. 1473), you must set the **Nodal Forces** property to either **Yes** or **Constrained Nodes** and set the **Store Modal Results** property to **Program Controlled** or **For Future Analysis**. **Note:** The **Constrained Nodes** setting is the preferred setting because the results file size will be smaller and the processing time shorter.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Remote Displacement

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient - Full	
Modal	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. To enable the output, set **Calculate Reactions** = **Yes** in the **Output Controls**.
Harmonic Response - Full	
Response Spectrum	
Random Vibration PSD	
Harmonic Response - Mode-Superposition	If results are expanded from a modal solution, then the output of these options are controlled by both the **RSOL** and **NLOAD** options of the **OUTRES** command. You must set both **Calculate Reactions** and **Nodal Forces** to either **Yes** or **Constrained Nodes** in the **Output Controls**. **Note:** **Constrained Nodes** is the preferred option, as the results file size will be smaller and the process time shorter. Otherwise, the output of these options are controlled by the **RSOL** option of the **OUTRES** command. Set **Calculate Reactions** = **Yes** in the **Output Controls**. **Important:** Force and Moment Reaction probes cannot be evaluated when you specify damping in combination with setting the **Expand Results From** property to either **Harmonic Solution** or **Transient Solution**. Review the **MXPAND** command in the *Mechanical APDL Command Reference* for additional information about this limitation.
Transient - Mode-Superposition	

Compression Only Support

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. Reaction probes scoped to a Compression Only boundary condition cannot display results if the solver did not converge.
Transient - Full	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Elastic Support

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **NLOAD** option of the **OUTRES** command. To enable the output, set **Nodal Forces** = **Yes** in the **Output Controls**.
Transient - Full	

Imported Displacement

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient - Full	

Weak Springs

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient - Full	

Grounded Beam

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient Full	
Modal	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. To enable the output, set **Calculate Reactions** = **Yes** in the **Output Controls**.
Harmonic Response - Full	
Harmonic Response - Mode-Superposition	If results are expanded from a modal solution, then the output of these options are controlled by both the **RSOL** and **NLOAD** options of the **OUTRES** command. You must set both **Calculate Reactions** and **Nodal Forces** to either **Yes** or **Constrained Nodes** in the **Output Controls**. **Note:** **Constrained Nodes** is the preferred option, as the results file size will be smaller and the process time shorter.
Transient - Mode-Superposition	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Reaction Type	Output Controlled By
	Otherwise, the output of these options are controlled by the **RSOL** option of the **OUTRES** command. Set **Calculate Reactions** = **Yes** in the **Output Controls**. **Important:** Force and Moment Reaction probes cannot be evaluated when you specify damping in combination with setting the **Expand Results From** property to either **Harmonic Solution** or **Transient Solution**. Review the **MXPAND** command in the *Mechanical APDL Command Reference* for additional information about this limitation.

Contact

Reaction Type	Output Controlled By
Static	The underlying element options are controlled by the **NLOAD** option of the **OUTRES** command. To enable the output, set **Nodal Forces** = **Yes** in the **Output Controls**. The contact element options are governed by the **MISC** option of the **OUTRES** command. To enable the output, set either the **Contact Miscellaneous** or **General Miscellaneous** property to **Yes** in the **Output Controls**.
Transient - Full	
Modal	These analysis types do not support contact reactions using the contact element option. They only support contact reactions using the underlying element option. You control the underlying element options using the **NLOAD** option of the **OUTRES** command. To enable the output, set the **Nodal Forces** property to **Yes** in the Output Controls category.
Standalone Harmonic Response - Full	
Standalone Harmonic Response - Mode-Superposition	
Harmonic Response Analysis Linked to Modal Analysis	
Transient - Linked to Modal Analysis	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Remote Point

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient - Full	
Modal	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. To enable the output, set **Calculate Reactions** = **Yes** in the **Output Controls**.
Harmonic Response - Full	
Harmonic Response - Mode-Superposition	If results are expanded from a modal solution, then the output of these options are controlled by both the **RSOL** and **NLOAD** options of the **OUTRES** command. You must set both **Calculate Reactions** and **Nodal Forces** to either **Yes** or **Constrained Nodes** in the **Output Controls**. **Note:** **Constrained Nodes** is the preferred option, as the results file size will be smaller and the process time shorter. Otherwise, the output of these options are controlled by the **RSOL** option of the **OUTRES** command. Set **Calculate Reactions** = **Yes** in the **Output Controls**. **Important:** Force and Moment Reaction probes cannot be evaluated when you specify damping in combination with setting the **Expand Results From** property to either **Harmonic Solution** or **Transient Solution**. Review the **MXPAND** command in the *Mechanical APDL Command Reference* for additional information about this limitation.
Transient - Mode-Superposition	

Grounded Spring

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **RSOL** option of the **OUTRES** command.
Transient - Full	
Modal	The output of these options are controlled by the **RSOL** option of the **OUTRES** command. To enable the output, set **Calculate Reactions** = **Yes** in the **Output Controls**.
Harmonic Response - Full	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

<table>
<tr><th>Reaction Type</th><th>Output Controlled By</th></tr>
<tr><td>Harmonic Response - Mode-Superposition</td><td rowspan="2">If results are expanded from a modal solution, then the output of these options are controlled by both the RSOL and NLOAD options of the OUTRES command. You must set both Calculate Reactions and Nodal Forces to either Yes or Constrained Nodes in the Output Controls.

Note:

Constrained Nodes is the preferred option, as the results file size will be smaller and the process time shorter.

Otherwise, the output of these options are controlled by the RSOL option of the OUTRES command. Set Calculate Reactions = Yes in the Output Controls.

Important:

Force and Moment Reaction probes cannot be evaluated when you specify damping in combination with setting the Expand Results From property set to either Harmonic Solution or Transient Solution. Review the MXPAND command in the Mechanical APDL Command Reference for additional information about this limitation.</td></tr>
<tr><td>Transient - Mode-Superposition</td></tr>
<tr><td>Random Vibration</td><td rowspan="2">To enable the output, set Calculate Reactions = Yes in the Output Controls of the upstream Modal analysis.</td></tr>
<tr><td>Response Spectrum</td></tr>
</table>

Mesh Connection

<table>
<tr><th>Reaction Type</th><th>Output Controlled By</th></tr>
<tr><td>Static</td><td rowspan="4">The output of these options are controlled by the NLOAD option of the OUTRES command.

To enable the output, set Nodal Forces = Yes in the Output Controls.</td></tr>
<tr><td>Transient - Full</td></tr>
<tr><td>Modal</td></tr>
<tr><td>Transient - Mode-Superposition</td></tr>
</table>

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Geometry

Reaction Type	Output Controlled By
Static	The underlying element options are controlled by the **NLOAD** option of the **OUTRES** command. To enable the output, set **Nodal Forces** = **Yes** in the **Output Controls**. **Note:** Mechanical supports geometry scoping for vertices, edges, and faces *only*.
Transient - Full	

Note:

A limitation exists when the scoping of a Force Reaction or Moment Reaction probe is applied to a geometry entity (**Location Method** = **Geometry Selection**) that shares more than one body. The (unscoped) elements that are adjacent to the scoped body contribute to the probe's results.

Surface

Reaction Type	Output Controlled By
Static	The output of these options are controlled by the **NLOAD** option of the **OUTRES** command. To enable the output, set **Nodal Forces** = **Yes** in the **Output Controls**.
Transient - Full	
Transient - Mode-Superposition	

Support Requirements and Limitations

- **Force Reaction** probes support Cartesian or cylindrical coordinate systems. **Moment Reaction** probes support Cartesian coordinate systems only.
- For solved **Force Reaction** and **Moment Reaction** probes, the arrow that displays for the probe represents a resultant vector, regardless of the axial direction that you specify in the **Result Selection** property of the probe. Vector displays in Mechanical always use the Global Coordinate System.
- A **Moment Reaction** probe cannot be scoped to a Grounded Spring.
- For the **Moment Reaction** probe, the **Summation** property is available for most **Location Method** property selections and enables you to select **Centroid** or **Orientation System** for your scoped entity. If your selected **Location Method** does not display this property, the application automatically uses the centroid of the scoped entity for the moment calculation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- A reported reaction may be inappropriate if that support shares a face, edge, or vertex with another support, contact pair, or load. This is because the underlying finite element model will have both loads and supports applied to the same nodes.

  If a model contains two or more supports that share an edge or vertex, use caution in evaluating the listed reaction forces at those supports. Calculation of reaction forces includes the force acting along bounding edges and vertices. When supports share edges or vertices the global summation of forces may not appear to balance. Reaction forces may be incorrect if they share an edge or face with a contact region.

- For a **Moment Reaction** scoped to a contact region, the location of the summation point may not be exactly on the contact region itself.

- If you set **Extraction** equal to **Contact (Underlying Element)** or **Target (Underlying Element)** in the Details of either a **Force Reaction** or **Moment Reaction** probe, the reaction calculations work by summing the internal forces on the underlying elements under a contact region. These probes can also extract reaction data from surface effect elements. The application creates surface effect elements during the solution process to simulate loads, such as pressures. However, the application does not currently display surface effect elements from the **Mesh** object or the **Connections** object.

  Therefore, a reported reaction may be inappropriate on a contact face if that face shares topology with another contact face/edge or external load (such as a force or fixed support), which would contribute to the underlying elements' internal force balance. In addition, during a Transient analysis, inertial and damping forces are also included. Another possible scenario could arise for MPC contact of solid surfaces. In this case, if a gap is detected, the solver may build constraints on an additional layer into the solid mesh from the TARGET elements. This produces a more accurate response but will invalidate any reactions from the underlying solid elements of the TARGET elements. If symmetric contact is chosen, be careful to verify which side becomes active for the TARGET elements so that the correct reaction can be determined.

- When scoping **Force Reaction** or **Moment Reaction** probes to geometry, it is possible that there may be elements (and as a result, element-based reactions) that are currently unavailable for summing purposes. For example, you scope a pressure to a face on your geometry. The solution process also applies surface effect elements to the same face to simulate the pressure loading. The probes, scoped to geometry, currently cannot extract reaction data from the surface effect elements and therefore, in the case of this example, only the underlying solid/shell/line elements of the original mesh contribute reaction data to the probe results.

- For Modal analysis, reaction results in damped modal analysis provide a **By** field option in the result definition to compute results based on **Mode Number**, **Phase of Maximum**, and **Maximum Over Phase**.

- For a Harmonic Response analysis, reaction results support all options of the result definition available for other harmonic results, and are reported based on the nearest frequency results available: no interpolation is done.

- In order to evaluate reactions successfully for a standalone Mode Superposition Harmonic Response analysis that has the **Clustering** property set to **On**, you need to also make sure that either the **Stress**, **Strain**, **Nodal**, or the **General Miscellaneous** properties in the **Analysis Settings**>**Output Controls** are also set to **Yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Reaction results sweep through a phase period of 0^o and 360^o at a specified increment. In previous releases of Mechanical (14.5 and earlier), the default value for this increment was 1^o in order to determine the **Phase of Maximum** and the **Maximum Over Phase** values. For Harmonic Response analyses only, the phase increment can be controlled using the **Phase Increment** option. A **Phase Increment** entry can be between 1^o and 10^o. The default Phase Increment value is 10^o but for legacy database results it is 1^o.

- For Random Vibration and Response Spectrum analysis, reaction results can only be scoped to a **Remote Displacement** boundary condition. Animation of reaction results is not supported for modal and harmonic analysis.

- Since **Beam Connections** are, by definition, three dimensional in nature, the reactions object scoped to grounded beams may produce reactions in all three directions/axes for two-dimensional analysis. The Tabular Data view will reflect the reactions in all three axes, while the Results view will only reflect values in two axes. The total reactions will be calculated taking into account the reaction components in all three axes.

- For a force reaction scoped to a contact region, if you set **Extraction = Contact (Contact Element)**, the reaction calculations come directly from the contact elements themselves. This results in accurate force reactions even when the contact region overlaps with other boundary conditions, such as other contact regions, supports, etc. Characteristics of the **Contact (Contact Element)** setting are that MPC contact is not supported, nor are reactions from the **Target (Underlying Element)** side. This feature should only be used with Asymmetric contact and requires that either the **Contact Miscellaneous** or the **General Miscellaneous** property be set to **Yes** in the **Output Controls**. A limitation of the **Contact (Contact Element)** setting is when you use linear contact (that is, either **Bonded** or **No Separation** contact types) with loads that are unrealistically very high or very low in magnitude. These situations can produce inaccurate force reactions.

  Furthermore, in certain rare cases that involve large or concentrated initial interference and/or thermal expansion applications, the reactions calculated by the contact element option may differ from those calculated by the underlying elements. If you experience such as scenario, the underlying element approach is more accurate. Try to tighten the tolerances of the **Newton-Raphson Option** property under the **Nonlinear Controls** (p. 1468) category of the **Analysis Settings** (also see the **CNVTOL** command) to improve the contact element reaction calculations.

- When a probe is scoped to a **Mesh Connection**, the Mechanical application reports the following reactions:

  – Forces and Moments summed from the element nodal forces and moments in the result file.

  – The **Extraction** detail determines which elements (Primary or Secondary) contribute to the force or moment sum.

- The **Surface** option of the **Location Method** property enables you to study reactions on cutting planes. You can extract generated member forces and reactions through a model by using a reaction probe scoped to a Surface (p. 890) object. For this probe type, you must explicitly select the body or bodies (via the **Geometry** property) to be sliced. You then specify for the **Extraction** property based upon whether you want to study the nodes in front or behind the plane. The **Force Reaction** probe operates the on elements cut by the plane (and only the nodes on those elements that are on the selected side of the plane). The application uses a slice algorithm for the **Surface** option. Because of very slight numerical accuracy issues, such as the number of significant digits used, the algorithm may not capture all expected nodes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The application can also fail to include a node based on how you slice an element. For example, if a slice plane produces a degenerate polygon, which can happen when a planar shell has only one node in the slice plane, the reaction calculation does not use the node from the element.

Important:

Do not apply reaction probes to cutting planes that intersect a boundary condition because the reported reactions become unreliable.

Note:

- Based on the **Extraction** property option you select, the probe uses the nodes of the cut elements, either from the positive or the negative side, of the specified coordinate system. This requires the application to create certain tolerances along the plane as well as calculated roundoffs of the points. Based on these calculations, the application could produce unexpected results. If the reactions of each side, positive and negative, differ significantly, you should change the location of the Surface plane slightly.
- Surfaces used in reaction probes do not currently intersect all geometries, such as line bodies, joints, springs, and MPC contact.
- Currently, surface probes cannot intersect a plane strain or an axisymmetric model and consequently no results display for this scoping.

Joint Probes

The joint type you select determines which results are available in the **Result Types** property of the **Joint Probe**. See the Joint Types (p. 1298) section for a description of the available joint types as well as free degrees of freedom.

Go to a section topic:

- Supported Analysis Types (p. 2118)
- Joint Probe Types (p. 2119)
- Characteristics and Requirements (p. 2119)

Supported Analysis Types

The following analysis types support the use of joint probes:

- Static Acoustics
- Static Structural (p. 652)
- Transient Structural (p. 665)
- Coupled Field Static (p. 391)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Coupled Field Transient (p. 395)
- Explicit Dynamics
- LS-DYNA
- LS-DYNA Restart Analysis
- Rigid Dynamics Analysis (p. 550)
- Motion

Joint Probe Types

The following table presents each of the joint probe results available through the **Result Type** drop-down menu in the Details.

Result Type Property Option	Applicable Joint Type(s)
Total Force	All
Total Moment	All except Slot and Spherical
Relative Displacement	All except Revolute, Universal, and Spherical
Relative Velocity	All except Revolute, Universal, and Spherical
Relative Acceleration	All except Revolute, Universal, and Spherical
Relative Rotation	All except Translational
Relative Angular Velocity	All except Translational
Relative Angular Acceleration	All except Translational
Damping Force	Bushing
Damping Moment	Revolute, Cylindrical, Bushing, and Spherical
Constraint Force	Revolute, Cylindrical, and Bushing
Constraint Moment	Revolute, Cylindrical, and Bushing
Elastic Moment	Revolute, Cylindrical, Bushing, and Spherical
Elastic Force	Bushing
Friction Force	Translational, Cylindrical
Friction Moment	Revolute, Cylindrical, and Spherical
Status	Spherical

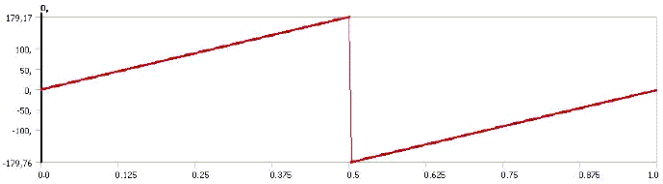
Characteristics and Requirements

Review the following characteristics and requirements to ensure that you properly configure your result.

- A joint defines the interface between two bodies. One of the bodies is referred to as a Reference body and the other as the Mobile body. The results from the joint measure the *relative motion* of the mobile body with respect to the reference body.
- A joint definition also includes specification of a local "reference" coordinate system for that joint. All results from the joint are output in this reference coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The reference coordinate system moves with the reference body. Depending on the motion of the reference body it might be difficult to interpret the joint results.
- All of these results have X, Y, and Z components in the reference coordinate system.
- Joint probes produce angular results in terms of intrinsic angles. The application calculates these results in an order, first X, then Y, and then Z, where Y is in the new position once rotated about the X axis, and Z is the new position once rotated about the Y axis.
- Relative rotation is expressed in Euler angles. When all three rotations are free, the general joint cannot report an angle that accounts for the number of turns. A typical behavior will be to switch from $+\pi$ radians to $-\pi$ radians for increasing angles passing the π limit, as illustrated below.



- For spherical and general joints the output of relative rotations is characterized by the Cardan (or Bryant) angles: the rotation around the joint Y axis is limited to between -90 degrees to +90 degrees. When this rotation magnitude value reaches 90 degrees, the output may "jump" to the opposite sign.
- The convention for the deformations differs for joints in a Rigid Dynamics analysis vs. those in a Transient Structural or Explicit Dynamics analysis. For the Rigid Dynamics type, the reference of zero deformation is taken after the model has been assembled, and the initial conditions have been applied. For the Transient Structural and Explicit Dynamics analysis types, the initial location of bodies is used as reference, before applying initial conditions.
- You can request a force or moment at a specific time point using the **Display Time** property. The result is displayed by an arrow in the **Geometry** window. Force will use a single arrowhead and moment will use double arrowhead.
- Joints compute no reactions forces or moments for the free degrees of freedom of the joint. However, Displacement, Velocity, Acceleration, Rotation, Rotational Velocity and Rotational Acceleration conditions - generate forces and moments, that are reported in the constraint force and moment.
- For analyses using the Mechanical APDL solver, the application calculates **Total Force** or **Total Moment** by adding the following values from miscellaneous output of general joint element:
  - Constraint Force/Moment
  - Stop Force/Moment
  - Lock Force/Moment
  - Elastic Force/Moment
  - Damping Force/Moment
  - Friction Force/Moment

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The inverse of the total is then used to convert the applied loads to the reaction loads.

You may wish to review NMISC and SMISC values of a joint element in the MPC184 General Joint Element Description section of the *Mechanical APDL Element Reference*.

- Joint forces and moment conditions are not reported in the joint force and moment probe.
- Joint force and moment are by definition the action of the reference body on the moving body. For the Mechanical APDL and Explicit Dynamics solvers, the joint constraint forces and moments are reported in the joint reference coordinate system. The elastic forces/moments and damping forces/moments in the joints are reported in the reference and mobile axes of the joint which follow the displacements and rotations of the underlying nodes of the joint element. When using the Ansys Rigid Dynamics solver, the joint forces and moments components are always reported in the joint reference coordinate system.
- For analyses using the Mechanical APDL solver, joint force and moment probes are not supported for **Body-Body** fixed joints when rigid bodies are present in the model. If these outputs are important, consider forcing the use of Joint elements via the **Solver Element Type** property for the fixed joint.
- For LS-DYNA analyses, only the **Total Force** and **Total Moment** results are available.

Response PSD Probe

The **Response PSD** probe provides a spectrum response of a structural component subjected to a random excitation. Response PSD is plotted as square of spectrum response over excitation frequency range. The plot provides an information as to where the average power is distributed as a function of frequency. The square root of the area under the response PSD is the so-called root-mean-square (RMS) value. It is a one-sigma, or one-standard-deviation, value in a statistical term. The centroid of the area under the response PSD probe, with respect to frequency, is referred to as the "expected frequency."

You can use the **Response PSD Tool** (p. 2130) to control the sampling points of the **Response PSD** probe.

The **Details View** properties and selections for the **Response PSD** object are described below.

Property	Control	Description
Definition	**Type**	Read-only property: **Response PSD**.
	Location Method	The response PSD probe is a point-based result. You use this property to specify how the location is defined. Options include **Geometry Selection** (default), **Coordinate System**, and **Remote Point**. For the **Geometry Selection** option, the application only supports vertex selection. For the **Coordinate System** option, you need to employ a user-defined coordinate system that specifies a specific location for evaluation. You can also specify a user-defined **Remote Point** defined on geometry. If you specify this probe on a surface body using vertex or coordinate system scoping, review the notes (p. 2123) at the end of the section for calculation differences.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Property	Control	Description
	Geometry	Appears if **Scoping Method** is set to **Geometry Selection**.
	Orientation	Read-only control: Only **Solution Coordinate System** is allowed for this result.
	Location	Appears if **Location Method** is set to **Coordinate System**.
	X Coordinate	Read-only field that displays coordinate that is based on the **Location** property of the coordinate system.
	Y Coordinate	Read-only field that displays coordinate that is based on the **Location** property of the coordinate system.
	Z Coordinate	Read-only field that displays coordinate that is based on the **Location** property of the coordinate system.
	Reference	Two options are available for the response PSD result evaluation: **Relative to base motion** (or relative motion) and **Absolute (including base motion)**. For the **Relative to base motion**, the response of any location in a structural component is calculated in term of a relative motion between the base and the structural component, and vice versa.
	Remote Points	Appears if **Location Method** is set to **Remote Points**.
	Suppressed	Include (**No**) or exclude (**Yes**) the result in the analysis.
Options	**Result Type**	Result Type: The result types include three basic motion characteristics (**Displacement**, **Velocity** and **Acceleration**), **Stress** (including normal and shear) and **Strain** (including normal and shear).
	Result Selection	Defines the direction, in **Solution Coordinate System**, in which response specified in the result type is calculated.
	Selected Frequency Range	Defines the frequency range for RMS calculation. Options include **Full** (default) and **Manual**. The **Full** setting uses the entire excitation frequency range.
	Range Minimum	Appears if **Selected Frequency Range** property is set to **Manual**. This property specifies the minimum frequency of the range.
	Range Maximum	Appears if **Selected Frequency Range** property is set to **Manual**. This property specifies the maximum frequency of the range.
Results	**Node ID**	A read-only property that displays the mesh Node ID used for the **Response PSD** probe result. The application selects the node nearest to the specified scoping of the probe.
	RMS Value	Read-only field that displays value calculated during solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Property	Control	Description
	RMS Percentage	Read-only field that displays the percentage of the root-mean-square of the selected frequency range over the entire available frequency range.
	Expected Frequency	Read-only field that displays the value calculated during the solution.

Note:

- For surface bodies solving for stress results, when you scope the Response PSD probe to a coordinate system, the application uses a specific algorithm, called "hit detect," that is based on the Newton–Raphson method. This algorithm is different than the one used for vertex scoping. Because the shell element has a thickness as well as both top and bottom stresses, the hit detect algorithm chooses to define the associated element as a hexagonal solid element. For the element that contains the origin of the coordinate system, the application calculates a stress tensor at this location through interpolation and includes contributions from all bottom and top nodal stresses. The application interpolates result values based on the element's shape functions. As a result, the mode shape input for the probe's calculation can be different for coordinate system scoping versus vertex scoping, and therefore the application may produce results that are different for each scoping method. This may be the case even if you scope the vertex or the coordinate system to the same location.
- The values for this response probe may differ compared to the results generated by the **RPSD** command (see Response PSDs in POST26) in Mechanical APDL. This is due to a difference in the number of sampling points as well as frequencies taken by the probe versus the Mechanical APDL command. To manually control the sampling points of the Response PSD probe, use the Response PSD Tool (p. 2130). This enables you to improve the accuracy of the result.
- The **Response PSD** probe uses the value 9.81456 m/s2 for acceleration due to gravity. This value is consistent with the default value of acceleration due to gravity used in **PSDUNIT** command.
- The square root of the area under a PSD curve represents the root mean square (rms) value of the excitation. The application computes this value using the linear trapezoidal rule and it can be different than the contour result.

Spring Probes

You can use a probe (p. 1946) to display the following longitudinal result items from a spring.

Elastic Force: The force is calculated as (Spring Stiffness * Elongation). The force acts along the length of the spring.

Damping Force: Damping force is calculated as (Damping Factor * Velocity) and acts to resist motion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Elongation: The elongation is the relative displacement between the two ends of the springs. The elongation could be positive (stretching the spring) or negative (compressing the spring).

Velocity: Velocity is the rate of stretch (or compression) of the spring. This quantity is only calculated in a Static Structural, Transient Structural, LS-DYNA, and Rigid Dynamics analysis.

Note:

During a Static Structural analysis, velocity is calculated as the difference of the relative displacement of two nodes (defining the spring) divided by the duration of the time step.

Bearing Probes

A Bearing is essentially a two-spring-damper system that is aligned in any two coordinate axes of a coordinate system: primarily a rotating plane.

For rotations in the X-Y plane, the result items for the first axis are in X direction and the results for the second axis are in Y direction. The application adds a suffix (number 1 and 2) to each result item. The X-Z and Y-Z rotation planes also use this convention.

You can use a Bearing probe to display the following result items.

Elastic Force 1

The force is calculated as (Spring Stiffness * Elongation). The force acts along the length of the spring along the first axis.

Elastic Force 2

The force is calculated as (Spring Stiffness * Elongation). The force acts along the length of the spring along the second axis.

Damping Force 1

Damping force is calculated as (Damping Factor * Velocity) and acts to resist motion along the first axis.

Damping Force 2

Damping force is calculated as (Damping Factor * Velocity) and acts to resist motion along the second axis.

Elongation 1

The elongation is the relative displacement between the two ends of the spring in the first axis. The elongation could be positive (stretching the spring) or negative (compressing the spring).

Elongation 2

The elongation is the relative displacement between the two ends of the spring in the second axis. The elongation could be positive (stretching the spring) or negative (compressing the spring).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Velocity 1

Velocity is the rate of stretch (or compression) of the spring in the first axis. This quantity is only calculated in a Transient Structural analysis.

Velocity 2

Velocity is the rate of stretch (or compression) of the spring in the second axis. This quantity is only calculated in a Transient Structural analysis.

Beam Probes

The Beam Probe results provide you the forces and moments in the beam from your analysis. Using the Beam Probe you can determine the Axial Force, Torque, Shear Force at I, Shear Force at J, Moment at I, Moment at J. You can also add the Force reaction and Moment Reaction probes to view reaction force moment for the beam. To add beam probes:

1. Select the **Solution** object.
2. On the **Solution** Context tab, select the **Probe** drop-down menu and select **Beam**.
3. In the Details, specify the desired **Boundary Condition** from the drop-down list.
4. Under **Options** category, select a desired result from the **Result Selection** property.

Bolt Pretension Probes

When a **Bolt Pretension** (p. 1566) load is applied, the Mechanical application reports the following reactions:

Adjustment: This represents the displacement that occurs from the applied pretension measured at the point where the bolt is sliced. In Mechanical APDL terms, this is the displacement reported from the pretension node. This result is also available for reporting regardless of how the bolt is defined.

Working Load/Preload Reaction: This represents a constrained force reaction from the pretension load. It is the reaction from the applied constraint when a bolt is either specified as **Locked**, **Adjustment**, or **Increment**, and reports a zero value during a step in which you have applied the preload (since there is no reaction at the bolt slice during preload step). In Mechanical APDL terms, this is the constrained reaction reported from the pretension node. This is essentially the sum of all the forces acting through the pretension cut. This result is only applicable for load steps when the load is defined by either **Locked** or **Adjustment** or **Increment**.

Generalized Plain Strain Probes

When a **Generalized Plane Strain** (p. 1582) load is applied (2D application), the Mechanical application reports the following reactions:

- **Fiber Length Change**: Fiber length change at ending point.
- **Rotation X Component**: Rotation angle of end plane about x-axis.
- **Rotation Y Component**: Rotation angle of end plane about y-axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

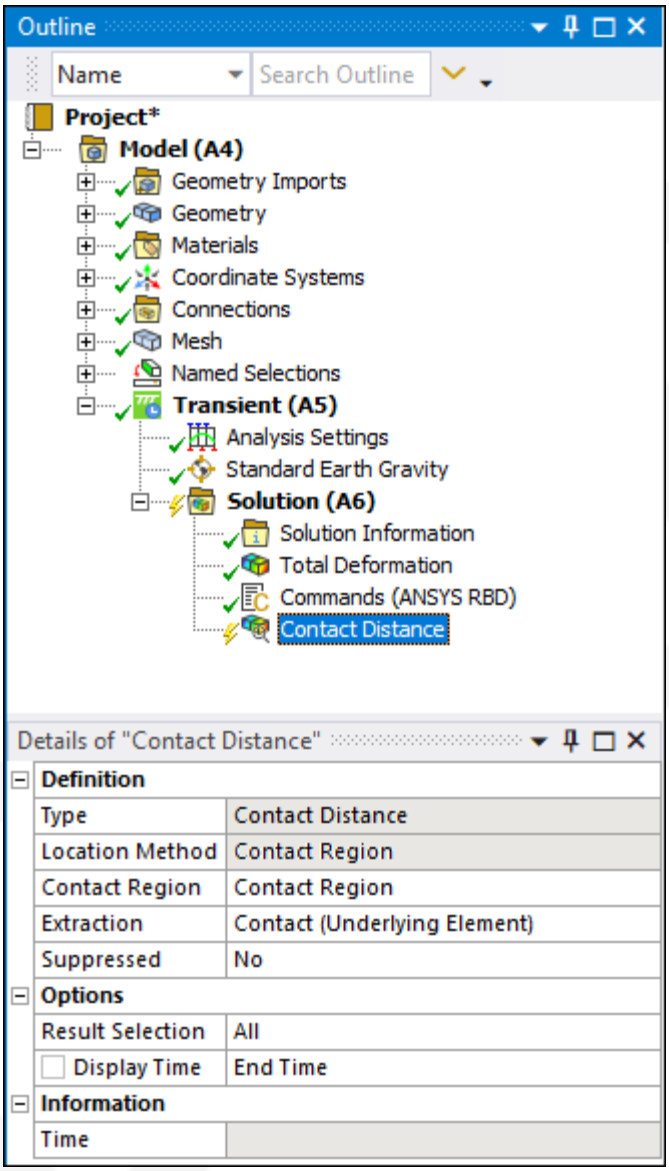
- **Force**: Reaction force at end point.
- **Moment X Component**: Reaction moment on end plane about x-axis.
- **Moment Y Component**: Reaction moment on end plane about y-axis.

Contact Distance Probes

For Rigid Dynamics systems, you can specify a **Contact Distance** probe. This probes produces a distance result, per time point, between the **Contact** and **Target** sides of a specified (or system generated) **Contact Region**.

Scoping

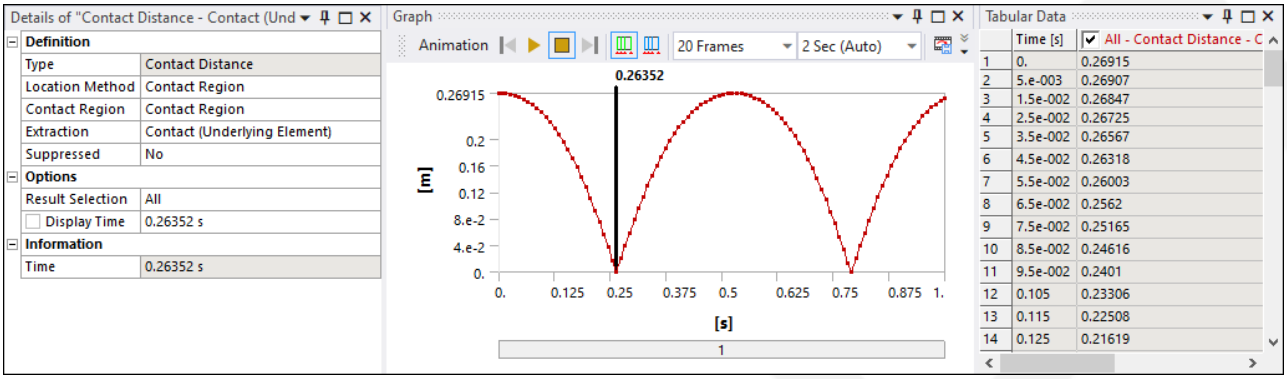
You scope this probe directly to an existing **Contact Region**.



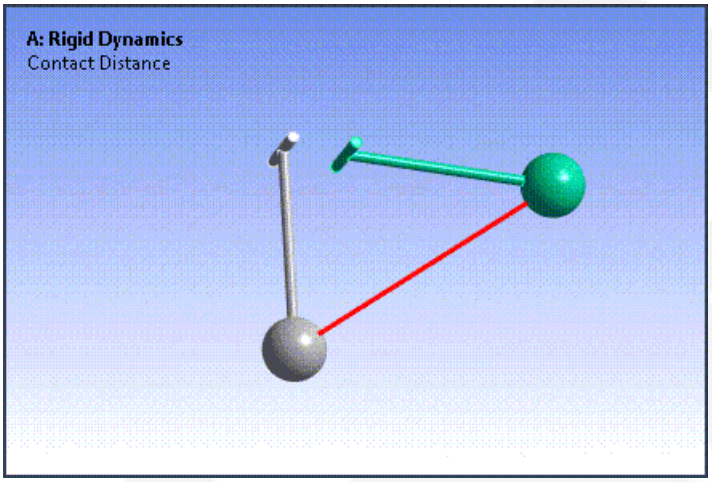
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Output

The application presents the **Contact Distance** probe results in the **Graph** and **Tabular Data** windows.



In addition, the application automatically inserts a tether (red line) between the contact sides. You can then animate the result to visually observe the change in contact distance over time, as illustrated below.



Fracture Probes (Fracture Tool)

When the crack selected in the **Crack Selection** property of your Fracture Tool is associated with a **SMART Crack Growth** (p. 468) object, a **Probe** group of options is available in the **Fracture Tool** Context tab and includes the fracture parameter probes listed below. Fracture probes enable you to view the time history of a fracture parameter (that is, SIFS, Crack Extension, etc.) for a specific crack front node along the crack front (tip) only.

- **J-Integral (JINT)**
- **SIFS (K1)/(K2)/(K2)**
- **Crack Extension**
- **Total Number of Cycles***
- **Equivalent SIFS Range***

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

***** Available when the **Crack Growth Option** property, of **SMART Crack Growth** object, is set to **Fatigue**.

Also see the Fracture Probes object reference page.

Go to a section topic:

- Extract Fracture Probe Results from a Result File (p. 2128)
- Display Features (p. 2128)
- Charting Fracture Probes (p. 2130)

Extract Fracture Probe Results from a Result File

When you have saved a results file (`file.rst`) from another fracture analysis, you can read this file into a different analysis and extract the result data with a **Fracture Tool**. Note the following requirements:

- For a Pre-Meshed Crack (p. 465), you can evaluate the fracture probes without defining the crack in the new analysis, but with similar boundary conditions.
- For all other crack types (p. 424), your analysis must include a defined **Fracture** folder, a matching mesh (including crack mesh), as well as defined boundary conditions.

Use the following steps to specify a **Fracture Tool** and to extract fracture results and probes from the result file.

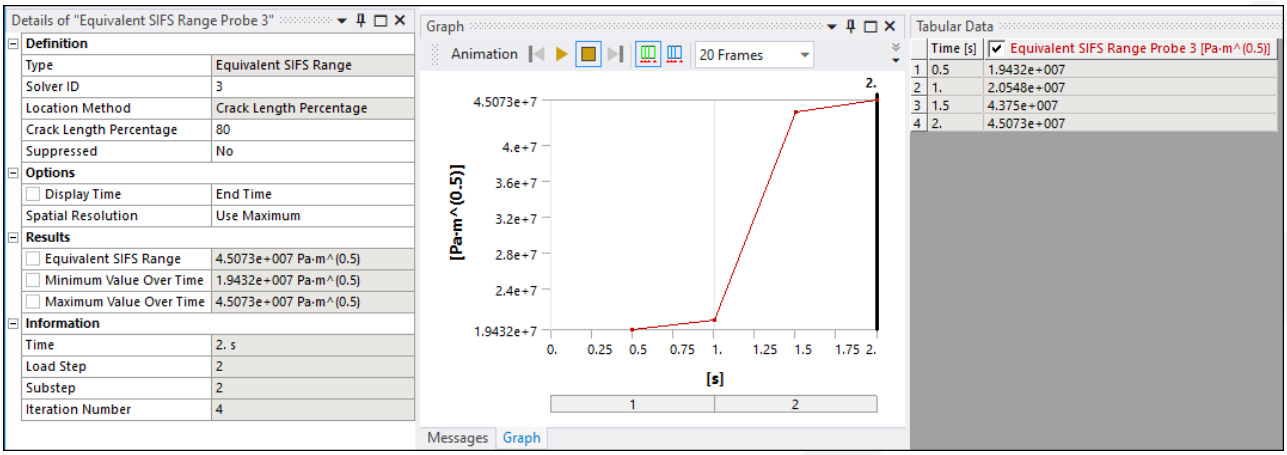
1. Read the results file into your analysis. As needed, review the Writing and Reading the Mechanical APDL Application Files (p. 1861) section.
2. Select the **Solution** object, open the **Toolbox** drop-down menu from the **Solution** Context tab, and select the **Fracture Tool** option. Based on the settings of the Fracture Controls (p. 1445) of the **Analysis Settings** object, a result object, such as **SIFS (K1)**, is automatically included under the **Fracture Tool**.
3. Set the **Scoping Method** property to **Result File Item**.
4. From the **Fracture Tool** Context tab, select and insert desired probe results.
5. From the drop-down list of the **Solver ID** property, select the appropriate solver ID value for each result. You can now evaluate the specific results.

Display Features

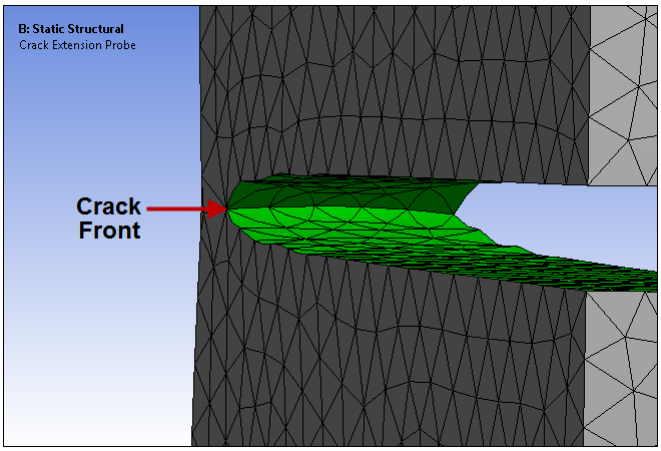
Once evaluated, probe results are plotted against time in the **Graph** window as illustrated below for a SIFS probe.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

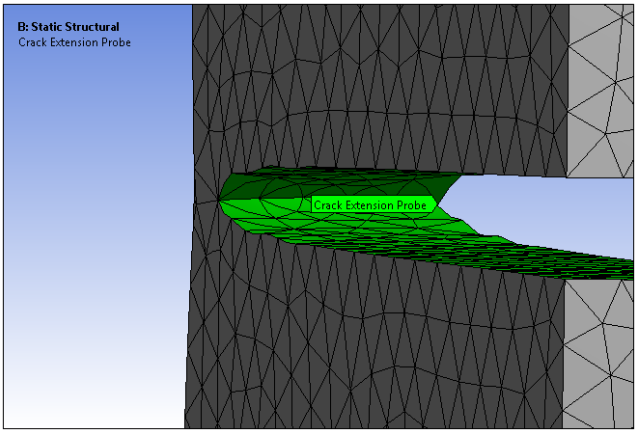


Another graphical display feature is that, when you have a fracture probe object selected, crack surfaces display in green in the **Geometry** window, as illustrated below. Furthermore, you can also see the tip of the crack.

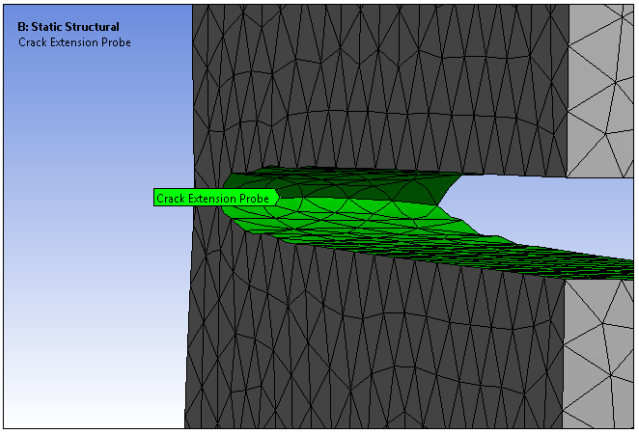


Using the **Crack Length Percentage** property, you can move the probe along the crack front. Because the results vary along the crack front, you can use this percentage entry to select a point along the tip you wish to examine. By changing the **Crack Length Percentage** you should receive different time history plots. An example is illustrated below.

Crack Length Percentage 0%



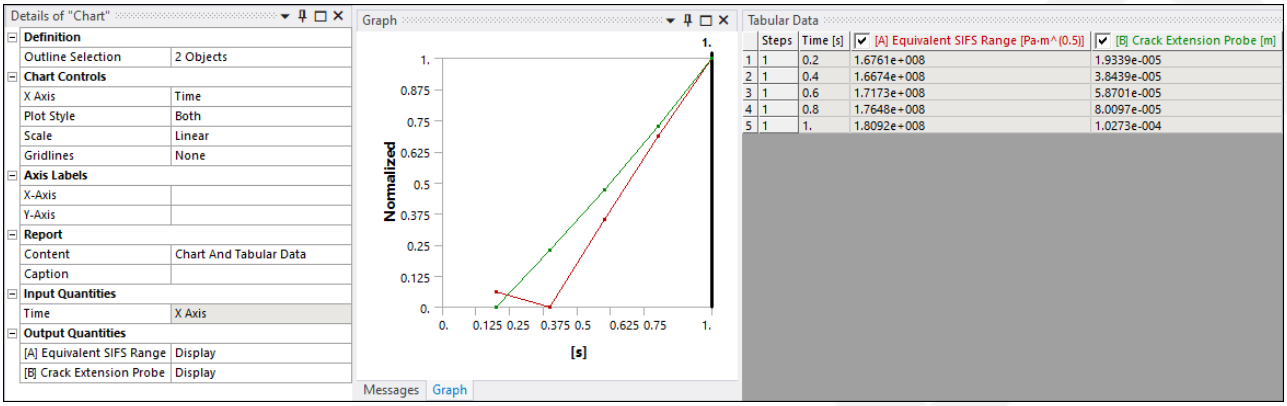
Crack Length Percentage 70%



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Charting Fracture Probes

Using the data generated by fracture probes, you can employ the New Chart and Table (p. 1929) option to plot your probes results against time or against one another. For example, the result of a SIFS probe is plotted versus the result of a Crack Extension probe is illustrated below.



Response PSD Tool

You can apply a **Response PSD Tool** to any Random Vibration (p. 524) analysis (only) in order to control the frequency sampling of Response PSD probes (p. 2121).

Scoping and Result Types

When inserted, the **Response PSD Tool** automatically generates a **Response PSD** probe (p. 2121) as its child object. You then properly scope both objects as desired.

Application

To define results using the **Response PSD Tool**:

1. Display the **Toolbox** drop-down menu on the **Solution** Context tab (p. 62) and select **Response PSD Tool** or right-click the Solution object or in the **Geometry** window and select **Insert**>**Response PSD Tool**>**Response PSD Tool**.

2. A **Response PSD Tool** object is placed under **Solution** object and includes a **Response PSD** probe. As needed, modify the following properties:

   - **Include All Excitation Frequencies**: Options include **Yes** (default) and **No**. Selecting **Yes** includes all excitation frequencies in the sampling. The **No** option includes minimum and maximum excitation frequencies in addition to the frequency samplings for natural frequencies.

   - **Include User Defined Frequencies**: Options include **Yes** and **No** (default). Selecting **Yes** enables you to manually enter frequencies in the **Tabular Data** table (**User Defined Frequency Steps**).

   - **Clustering Frequency Points**: The minimum value for this property, and default, is **20**. This property enables you to define the number of frequencies generated for both sides of the natural frequencies for response PSD result evaluations. If your response

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

PSD curve contains a number of spikes, you can add more frequency points to obtain a more accurate RMS result, however: this increases evaluation time.

3. Properly scope the properties of the Response PSD probe (p. 2121).

Note:

When a **Response PSD** probe is scoped to surface body, the application always reports the node result value using the smaller of the two values reported for the top and bottom layers.

For additional information, see the object reference page for the **Response PSD Tool**.

Gasket Results

Gasket results are structural results associated with Ansys interface elements. When used with Ansys structural elements, interface elements simulate an interface between two materials. The behavior at these interfaces is highly nonlinear.

To mesh a body using interface elements, use one of two ways:

1. Highlight the **Body** object in the tree and set **Stiffness Behavior** (p. 343) to **Gasket**. In this case, a Gasket Mesh Control will be added as a child of the gasket body in the model tree. You need to define the source face of the gasket in the Gasket Mesh Control to define the gasket material orientation.

2. Highlight the **Body** object in the tree and set the **Stiffness Behavior** (p. 343) to **Flexible**. In this case, you need to define a Gasket Mesh Control in the mesh folder.

The following gasket results are available in the Mechanical Application:

- **Normal Gasket Pressure** - corresponding to Mechanical APDL command **PLNSOL**,GKS,X
- **Shear Gasket Pressure** - corresponding to Mechanical APDL commands **PLNSOL**,GKS,XY and **PLNSOL**,GKS,XZ
- **Normal Gasket Total Closure** - corresponding to Mechanical APDL command **PLNSOL**,GKD,X
- **Shear Gasket Total Closure** - corresponding to Mechanical APDL commands **PLNSOL**,GKD,XY and **PLNSOL**,GKD,XZ

These results are only available in the solution coordinate system (p. 1975).

Warning:

Mechanical only supports the above gasket results. Note that there is a Mechanical APDL key option (KEYOPT(8)=1) that enables you to produce gasket results for standard stresses and strains (S, EPEL, and EPTH) on the interface elements. Mechanical gasket results do not support this key option and produce results with values of only zero. However, if you have the keyopt enabled, you can use a Commands (APDL) object to independently produce these stress results using the /POST1 command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

In addition, when using KEYOPT(8)=1, all Elemental Coordinate Systems (p. 1936) results will incorrectly display in the Global Coordinate System.

Review the individual element references for the Ansys interface elements for KEYOPT(8) element output quantities.

Campbell Diagram Chart Results

A Campbell diagram chart result is only valid in Modal analyses. The Campbell diagram chart result is mainly used in rotor dynamics for rotating structural component design. When a structural component is rotating, an inertial force is introduced into the system. The dynamic characteristics of the structural component change as a result of the inertia effect, namely, gyroscopic effect. To study changes in dynamic characteristics of a rotating structure, more than one solve point in Rotational Velocity (p. 1511) is required.

Prerequisites

In addition to being applicable to only Modal analyses, you must ensure that the following Analysis Settings are activated in order to properly apply a **Campbell Diagram**. Select the **Analysis Settings** object in your Modal Analysis and perform the following settings:

- Under **Solver Controls**: **Damped** = **Yes**.
- Under **Rotordynamics Controls**:
  - **Coriolis Effect** = **On, Stationary Frame of Reference**
  - **Campbell Diagram** = **On**

In addition, a Rotational Velocity (p. 1511) boundary condition must be created in order to properly scope the Campbell Diagram.

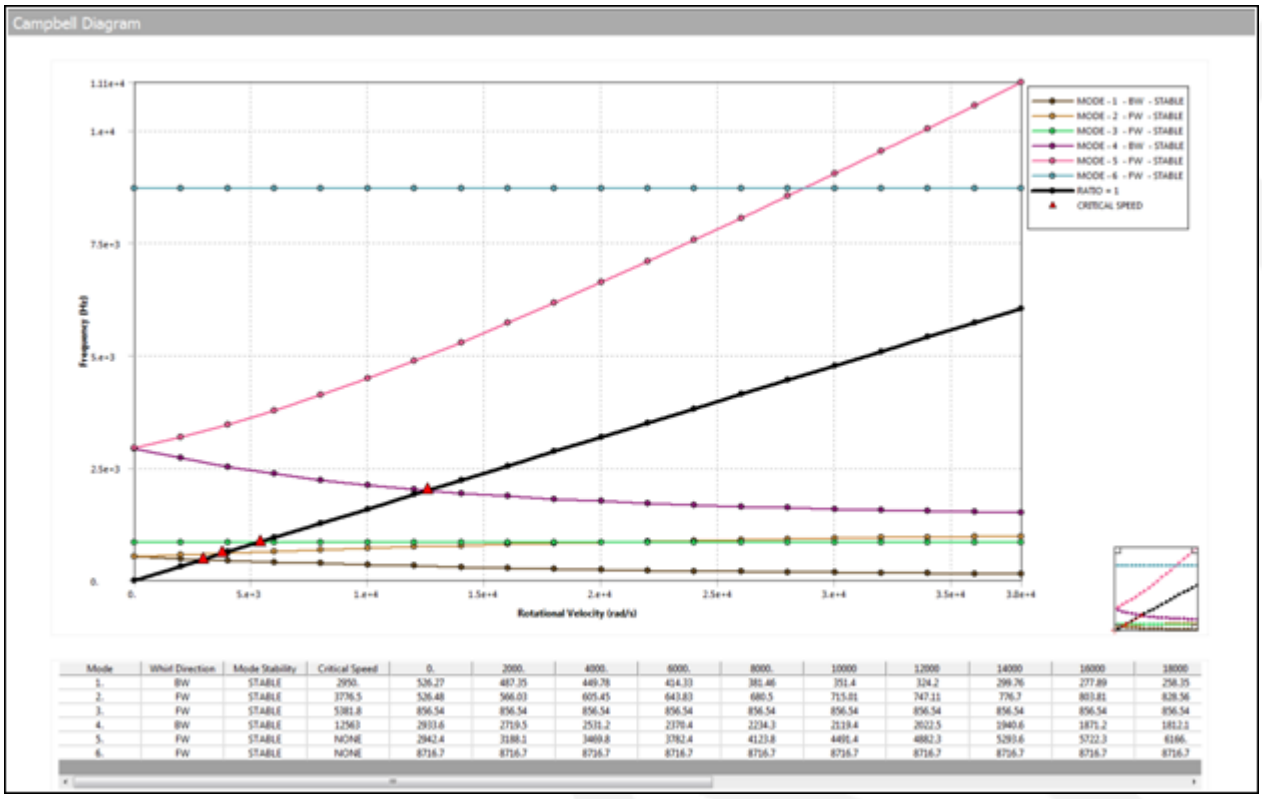
Applying a Campbell Diagram

To insert a Campbell diagram chart result, highlight the **Solution** object in the tree, then select Campbell Diagram from the Solution Context Tab (p. 62), or right-click the object and choose **Insert**>**Campbell Diagram**.

The following is an example of a Campbell diagram result chart:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Mode	Whirl Direction	Mode Stability	Critical Speed	0.	2000.	4000.	6000.	8000.	10000	12000	14000	16000	18000
1.	BW	STABLE	2950.	526.27	487.35	449.78	414.33	381.46	351.4	324.2	299.76	277.89	258.35
2.	FW	STABLE	3776.5	526.48	566.03	605.45	643.83	680.5	715.01	747.11	776.7	803.81	828.56
3.	FW	STABLE	5381.8	856.54	856.54	856.54	856.54	856.54	856.54	856.54	856.54	856.54	856.54
4.	BW	STABLE	12563	2933.6	2719.5	2531.2	2370.4	2234.3	2119.4	2022.5	1940.6	1871.2	1812.1
5.	FW	STABLE	NONE	2942.4	3188.1	3469.8	3782.4	4123.8	4491.4	4882.3	5293.6	5722.3	6166.
6.	FW	STABLE	NONE	8716.7	8716.7	8716.7	8716.7	8716.7	8716.7	8716.7	8716.7	8716.7	8716.7

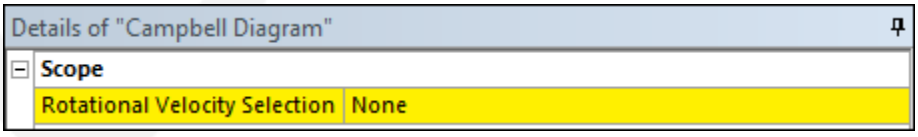
In this chart, each line represents a frequency evolution of a whirl mode with respect to increased rotational velocities. The whirl frequency value of an eigenmode at each rotational velocity is also listed in the table. For each whirl mode, it is either labeled as forward (FW) or backward (BW) whirl direction. In some cases, when there is no evident whirl direction, the whirl frequency is labeled as UNDETERMINED. If a whirl mode is identified as FW, the rotating structural component whirls the same direction as the rotation direction, and vice versa. If a whirl mode is evaluated to be unstable (marked as UNSTABLE), the whirl orbit will evolve into a divergent trajectory, instead of an elliptical trajectory.

In addition to whirl modes, a line (black color) of any ratio between whirl frequency and rotational velocity is plotted. The intersection between this line and each whirl mode is indicated with a red triangular marker. The rotational velocity corresponding to this intersection is called critical speed. At critical speed, the rotating structural component will experience a peak as the rotating frequency resonates with the natural whirl frequency.

The Campbell diagram chart result can be customized in Details of Campbell Diagram as follows:

Scope

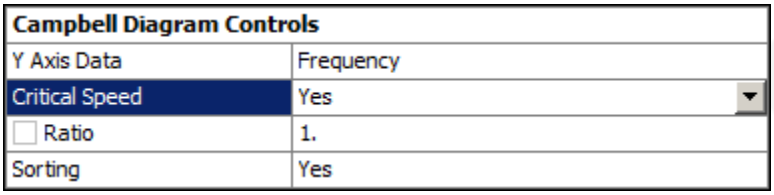
- **Rotational Velocity Selection**: This field displays the user-defined Rotational Velocity of the analysis for which the Campbell diagram chart result is evaluated. If one is not defined, the field is highlighted in yellow and displays the value **None**.



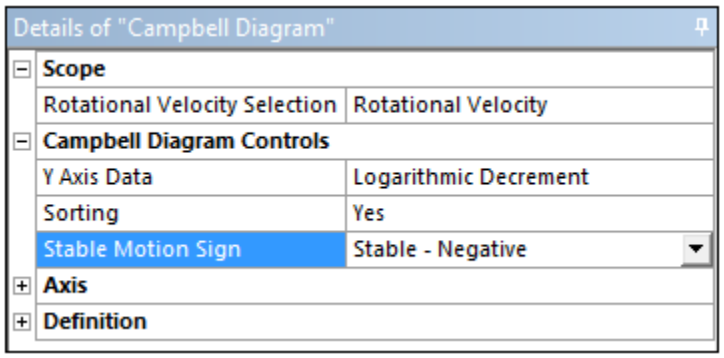
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Campbell Diagram Controls

- **Y Axis Data**: The data type options include: **Frequency** (default), **Stability**, **Logarithmic Decrement**, and **Modal Damping Ratio**.
- **Critical Speed**: Option for users to display critical speeds. The default is **Yes**. Requires you to provide a value in the Ratio field. The option is only valid for frequency.
- **Ratio**: Value used to evaluate critical speeds. The default value is **1.0**.
- **Sorting**: Option to display data in a sorted mode manner when some modes are crossing/intercepting each other. The default is **Yes**.



- **Stable Motion Sign**: This option displays when you specify **Logarithmic Decrement** for the **Y Axis Data** property. This option controls the sign of Logarithmic Decrement value for stable motion. The options include **Stable - Negative** (default) and **Stable - Positive**.



Note:

Any change made in these fields requires a result re-evaluation.

Axis

Note:

Two different unit types, rad/s and RPM, are available to define rotational velocity in the chart. The selection can be made in **Units** option on the **Home** tab.

- **X Axis Label**: Allows users to provide a customized label for rotational velocity.
- **X Axis Range**: There are two options to display the rotational velocity data range: **Program Controlled** and **User Defined**. The default setting is **Program Controlled**, which uses minimum and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

maximum determined by the system. The option of **User Defined** allows users to provide a customized range to be used in the chart. The minimum and maximum values are displayed in the **X Axis Minimum** and **X Axis Maximum** fields following the result evaluation.

- **X Axis Minimum**: Minimum rotational velocity value is displayed according to the selection made in **X Axis Range**.
- **X Axis Maximum**: Maximum rotational velocity value is displayed according to the selection made in **X Axis Range**.
- **Y Axis Label**: Allows users to provide a customized label for **Frequency**, **Stability**, **Logarithmic Decrement**, or **Modal Damping Ratio** depending on the selection made in **Y Axis Data**.
- **Y Axis Range**: The options include **Program Controlled** and **User Defined**, to display the frequency, stability, logarithmic, or modal damping ratio value range depending on the selection made in **Y Axis Data** property. The default setting, **Program Controlled**, uses the minimum and maximum determined by the system. The option of **User Defined** enables you to provide a customized range. The minimum and maximum values are displayed in the **Y Axis Minimum** and **Y Axis Maximum** fields following the result evaluation.
- **Y Axis Minimum**: Minimum frequency, stability, logarithmic decrement, or modal damping ratio value is displayed according to the selection made in **Y Axis Range** property.
- **Y Axis Maximum**: Maximum frequency, stability, logarithmic decrement, or modal damping ratio value is displayed according to the selection made in **Y Axis Range** property.

Axis	
X Axis Label	Rotational Velocity
X Axis Range	Program Controlled
X Axis Minimum	1. rad/s
X Axis Maximum	2. rad/s
Y Axis Label	Frequency
Y Axis Range	Program Controlled
Y Axis Minimum	0. Hz
Y Axis Maximum	0.33142 Hz

Waterfall Diagrams

For a multi-step Harmonic Response (p. 496) analysis that specifies RPMs as the step type, the application supports the following waterfall result options:

- **Velocity Waterfall Diagram**
- **Acceleration Waterfall**
- **Modal Coefficient Factors (MCF) Waterfall Diagram** (for **Mode Superposition** only when the On Demand Expansion Option (p. 1447) is set to **Yes**)
- **ERP Waterfall Diagram**
- **ERP Level Waterfall Diagram**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Far-field Sound Power Level Waterfall Diagram** and **Far-field SPL Mic Waterfall Diagram** (described in the Acoustic Waterfall Diagrams section)

Note:

- Because waterfall diagrams plot large amounts of data, the application provides several features, such as zoom and the use of probes, to manipulate the display. See the Waterfall Diagram Display Features (p. 2139) section for more information.
- You can specify a weighting filter (p. 1977) (A, B, and C) for the **ERP Level Waterfall Diagram** result.

See the following subsections for additional information about how to present data using a Waterfall Diagram:

Waterfall Diagram Data Filtering Features (RPM Only)

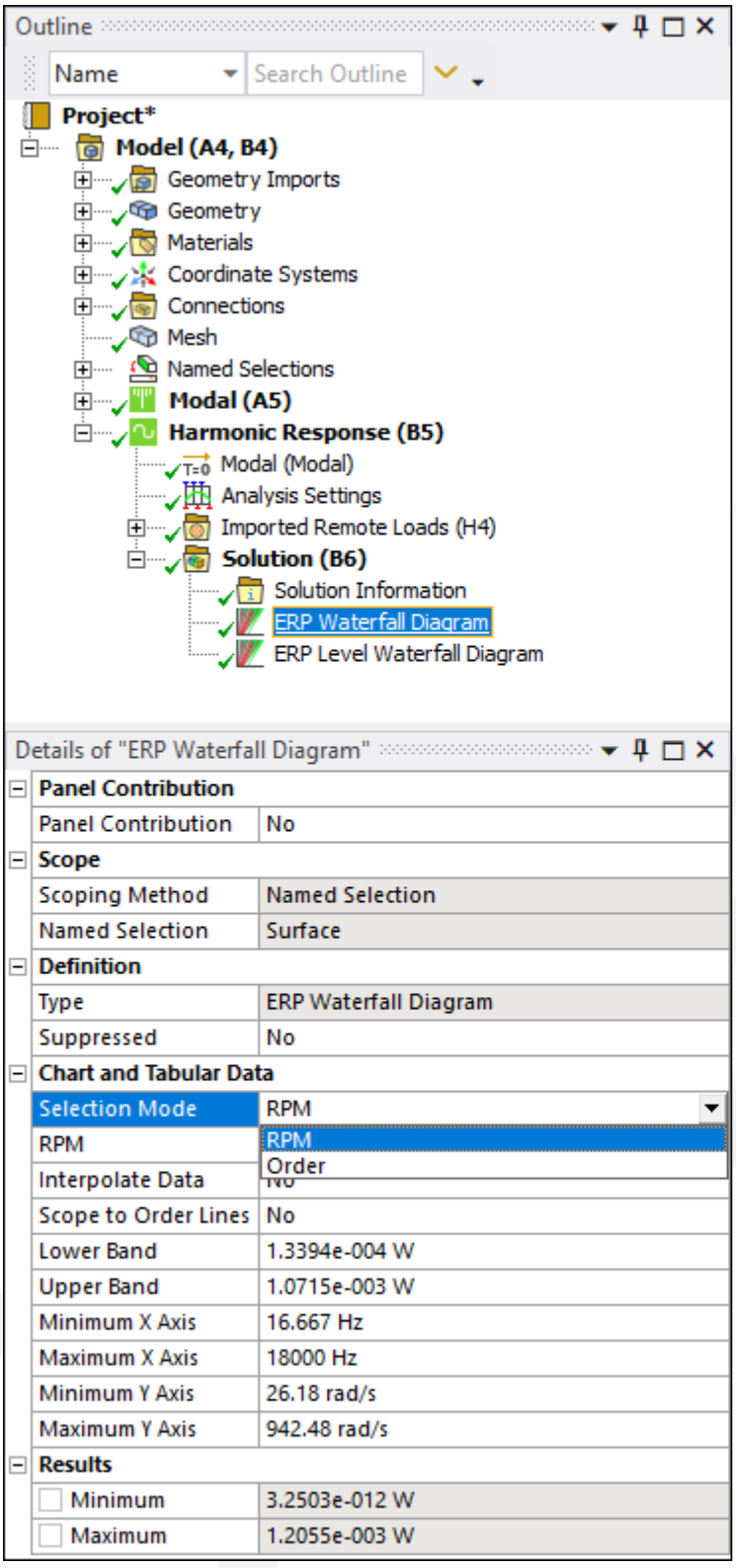
Waterfall Diagram Display Features

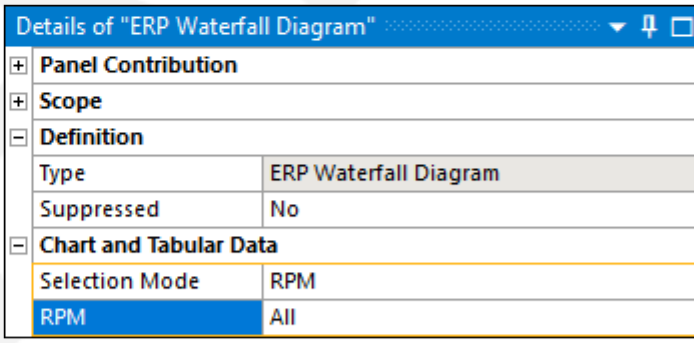
Waterfall Diagram Data Filtering Features (RPM Only)

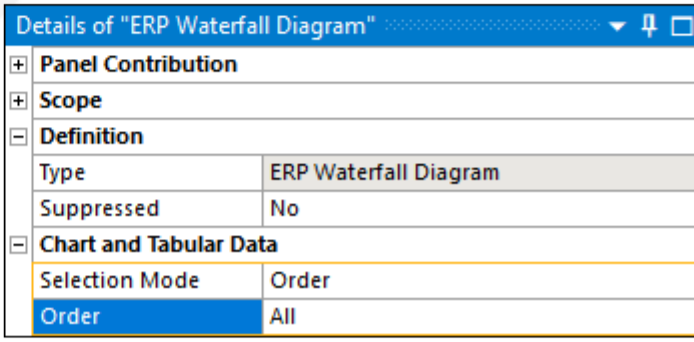
Waterfall results display large amounts of data. Use the properties of the **Chart and Tabular Data** category to filter data (**RPM**/**Order**) as well as to plot the data displayed in the **Worksheet** and the **Graph** and **Tabular Data** windows. As illustrated below, the Waterfall result types include the **Selection Mode** property. It is only visible for RPM-based waterfall diagrams. The options include **RPM** and **Order**. For each option, an additional entry property of the same name (**RPM** or **Order**) is also displayed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





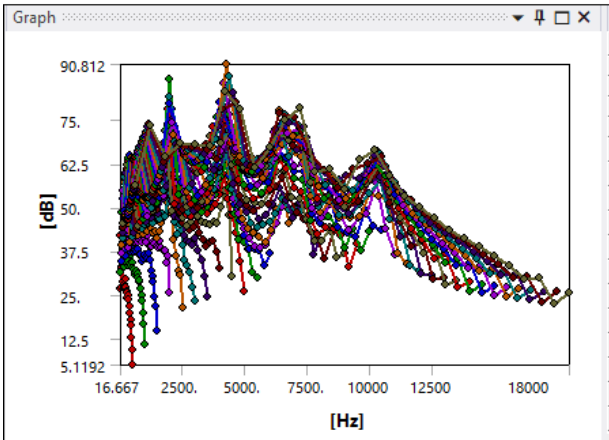


The default value for the **RPM** and **Order** entry properties is **All**, as shown in the following example. For this example, the **Graph** and **Tabular Data** windows include over 800 data points. Note that the **Tabular Data** window also includes a listing of the **Order** values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Details of "ERP Waterfall Diagram"

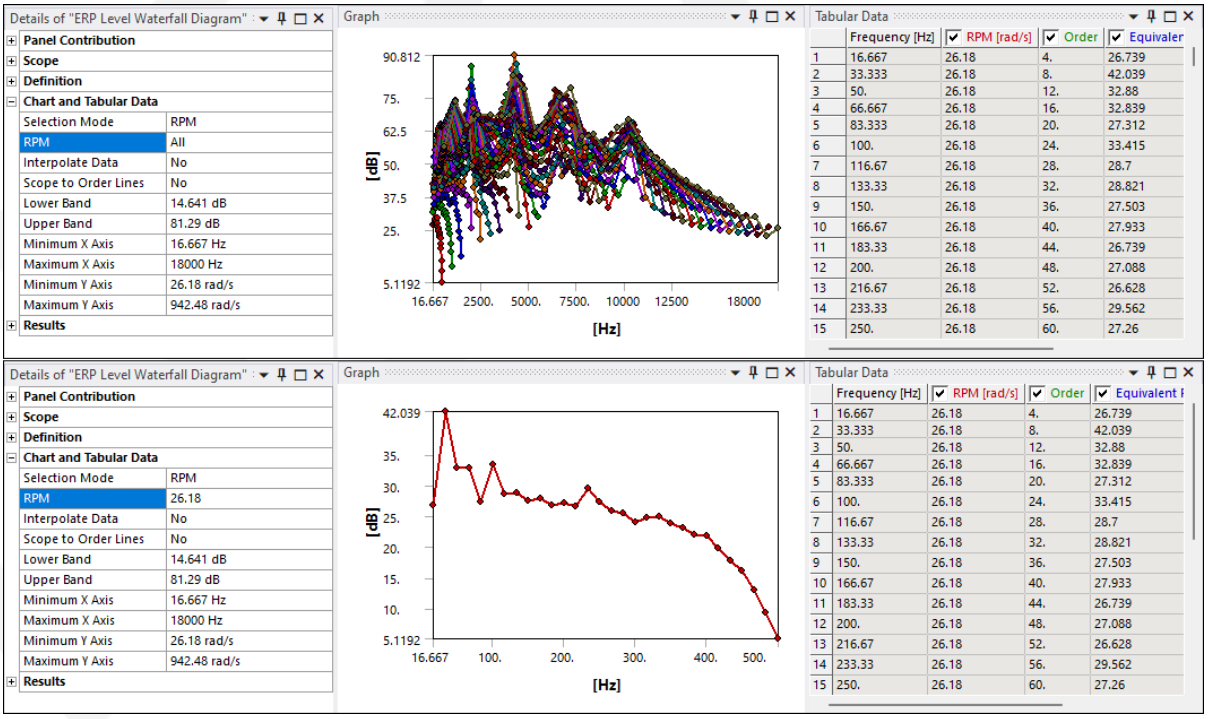
Panel Contribution	
Scope	
Definition	
Type	ERP Waterfall Diagram
Suppressed	No
Chart and Tabular Data	
Selection Mode	RPM
RPM	All
Interpolate Data	No
Scope to Order Lines	No
Lower Band	1.3394e-004 W
Upper Band	1.0715e-003 W
Minimum X Axis	16.667 Hz
Maximum X Axis	18000 Hz
Minimum Y Axis	26.18 rad/s
Maximum Y Axis	942.48 rad/s
Results	



Tabular Data

	Frequency [Hz]	RPM [rad/s]	Order	Equivalent Radiated Power
1	16.667	26.18	4.	26.739
2	33.333	26.18	8.	42.039
3	50.	26.18	12.	32.88
4	66.667	26.18	16.	32.839
5	83.333	26.18	20.	27.312
6	100.	26.18	24.	33.415
7	116.67	26.18	28.	28.7
8	133.33	26.18	32.	28.821
9	150.	26.18	36.	27.503
10	166.67	26.18	40.	27.933
11	183.33	26.18	44.	26.739
12	200.	26.18	48.	27.088
13	216.67	26.18	52.	26.628
14	233.33	26.18	56.	29.562
15	250.	26.18	60.	27.26
16	266.67	26.18	64.	25.88

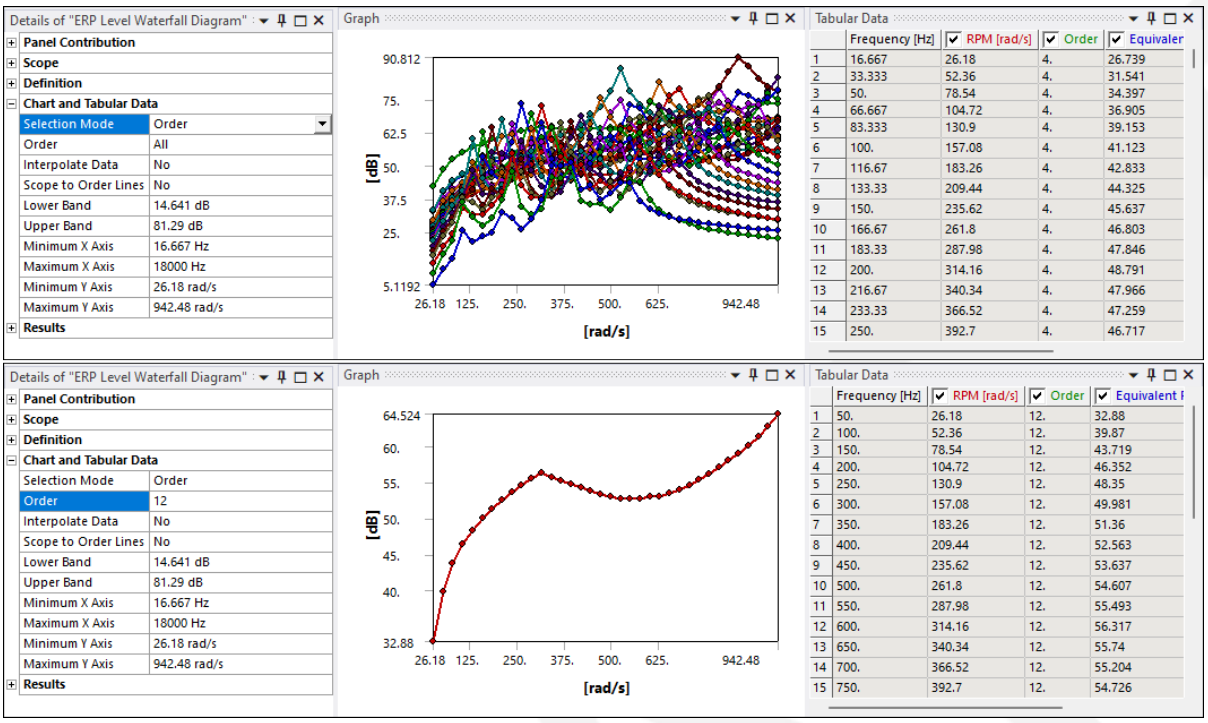
To refine this display, specify an **RPM** value in the **RPM** entry property. Here is an example for an **RPM** value of 26.18. The graph plots **Equivalent Radiated Power Levels** against **Frequency** for the specified RPM value.



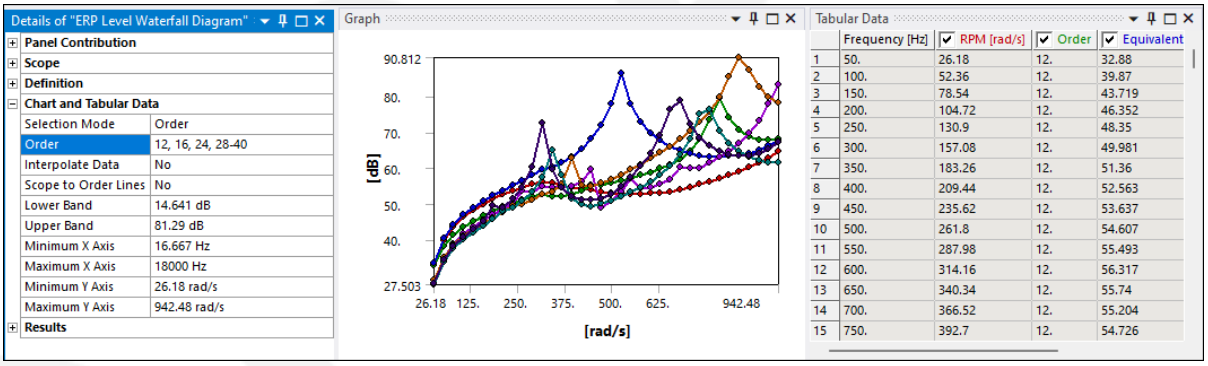
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The same is true for **Order** entries as shown here for **Order** value 12.



In addition to singular **Order** property entries, you can enter multiple values, separated by a comma (12, 16, 24) as well as range values separated by a hyphen (28-40), or a combination of entries, such as 12, 16, 24, 28-40.



Note:

For each order value you enter, the application includes all values within a decimal range of ±0.5 in the **Tabular Data** and **Graph** windows. For example, for the entry, 5,10, the selected order range is 4.5-5.5 and 9.5-10.5.

Waterfall Diagram Display Features

Because Waterfall Diagrams can plot large amounts of data, several display features exist to examine and present the data to suit your particular needs. Click the titles below to see how each option/property affects the display.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Go to a section topic for descriptions of how to use the feature or option:

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

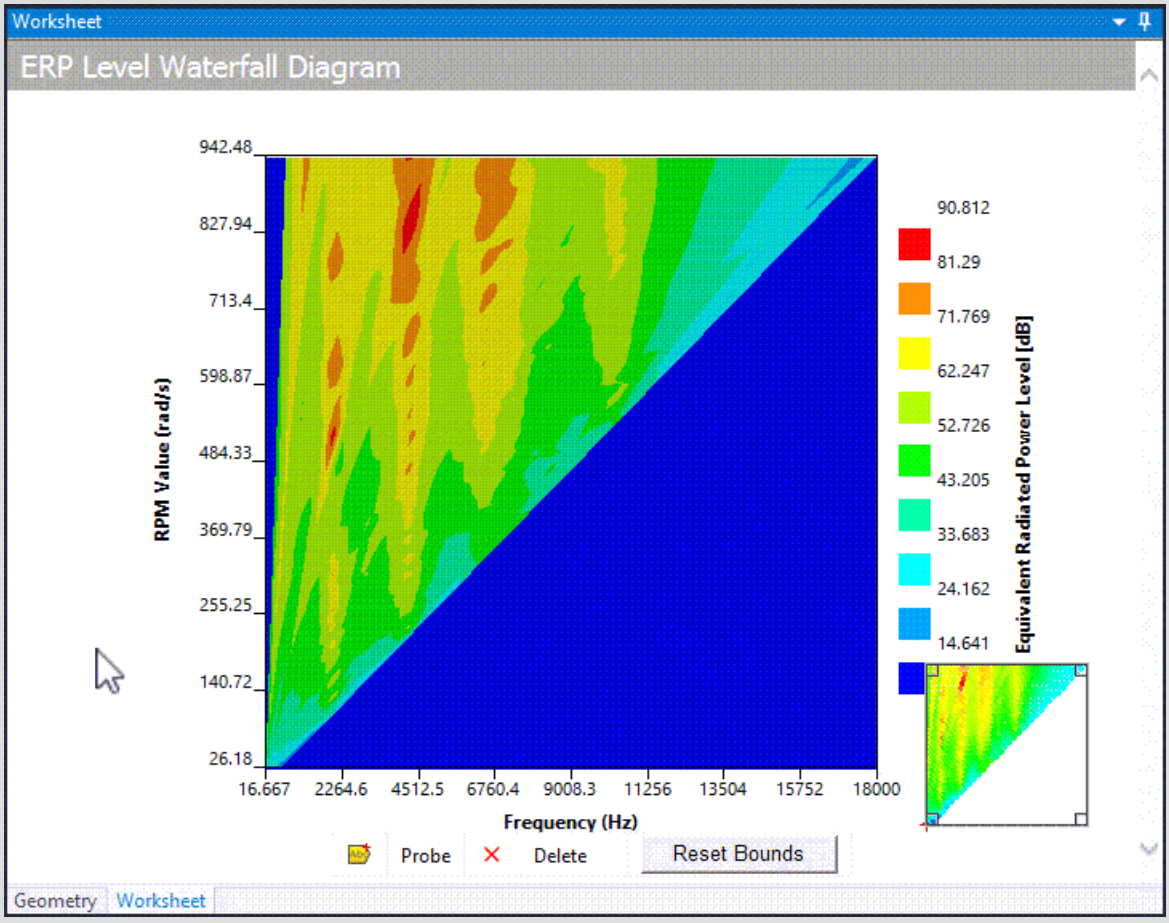
Published: 2023-10-23

Zooming

The worksheet for waterfall results includes a small box in the lower left-hand corner, as highlighted below, you use to zoom in on the diagram. Selecting a corner of this option and dragging the cursor enlarges that area of the diagram contained within the zoom option. A smaller control is placed inside of the zoom option once you drag a corner. Use this control to pan across the diagram to change the selected zoom area. Double-clicking the control returns the view to the default.

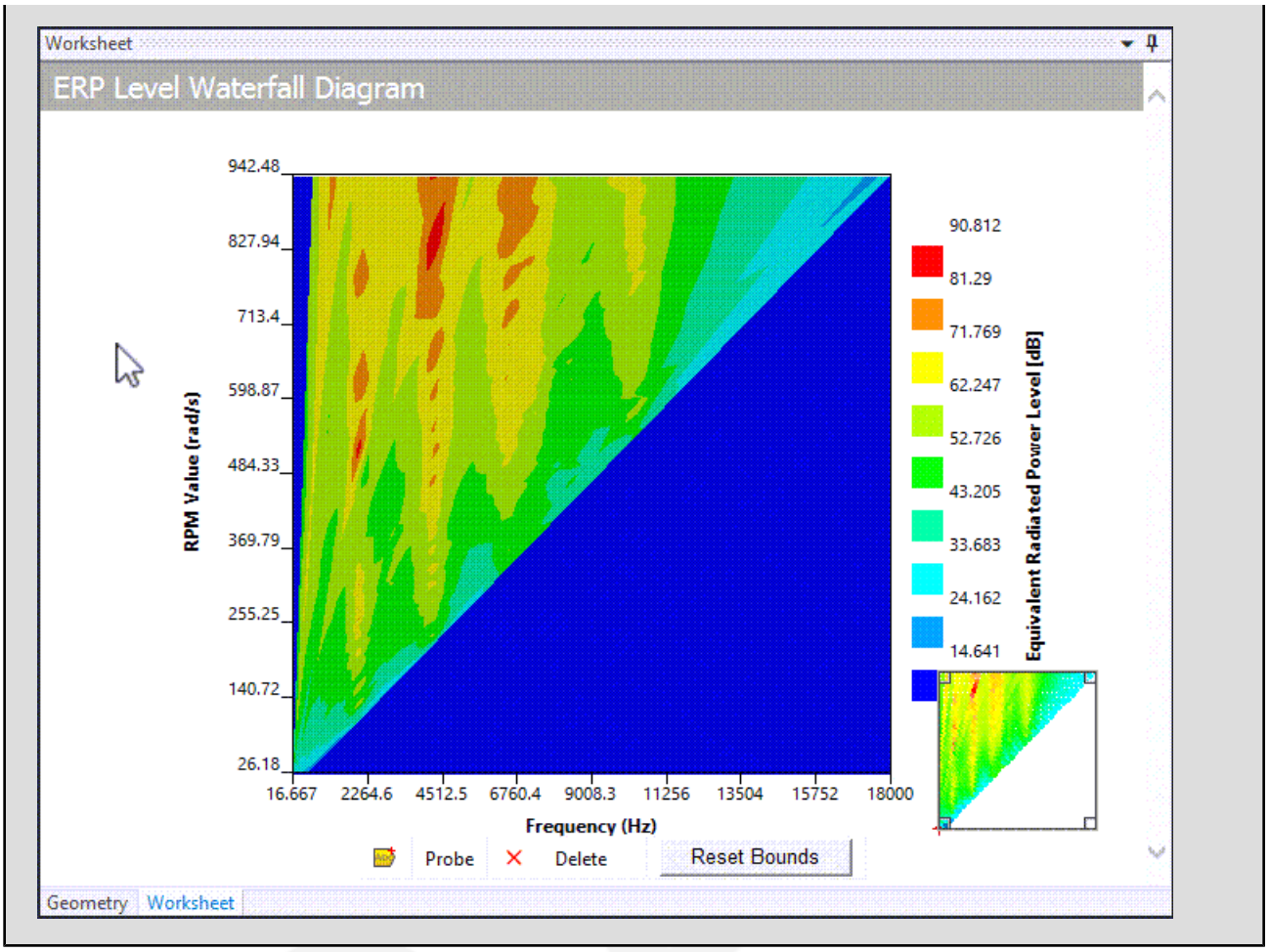
Note:

- Hide the display of the zoom feature in the worksheet by setting the **Show Zoom Pan Control** property to **No**.
- When zooming in on an area of the diagram, the values of the **Minimum**/**Maximum X Axis** and **Minimum**/**Maximum Y Axis** properties automatically update based on your selection.



In addition, use the you can and as shown below, you can select the **[Alt]** key and then select directly in the diagram to zoom on an area. Once you **[Alt]** + **click** in the diagram, dragging your cursor left-to-right enlarges a desired area. **[Alt]** + **click** and dragging your cursor from right-to-left returns the view to the default.

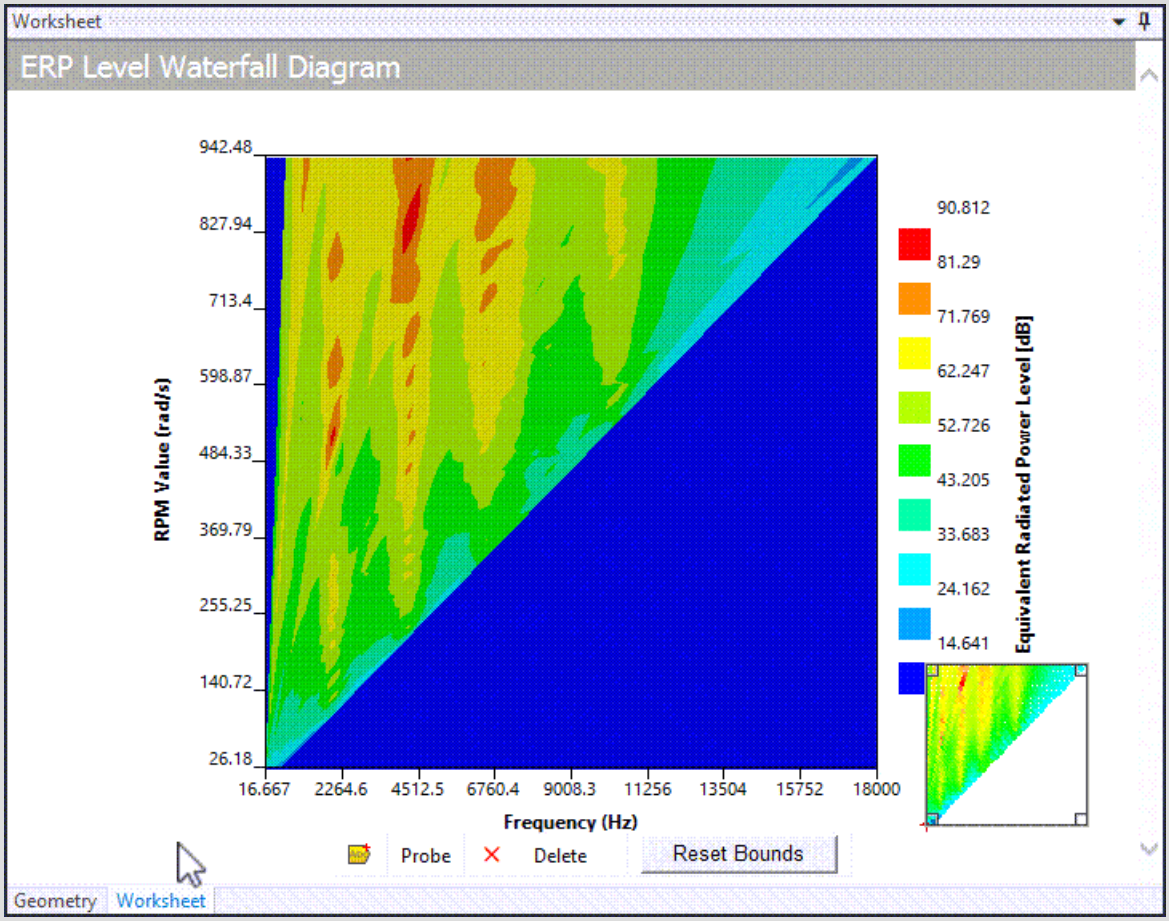
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifying Probes

Use the **Probe** option to place probes that display result values anywhere on the diagram. As you run the cursor over the result values, the probe displays a result value as well as a coordinate (X and Y) location. If you select a location that does not include data, the selection is ignored. Use the **Delete** option to delete probes by clicking on a probe's anchor.



Note:

Probes are automatically removed from the diagram when you select a different object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Panel Contribution

For Harmonic Response and Harmonic Acoustic analyses, the **Panel Contribution** option, and its associated properties, enables you to visually compare results for different geometry scopings via mosaic and line chart display tools. This feature supports face scoping only for a single **RPM** value. Supported waterfall diagram results include Velocity, Acceleration, ERP, and/or ERP Level Waterfall Diagram.

Procedure

1. Specify a supported waterfall diagram result.
2. Set the **Panel Contribution** property (**Panel Contribution** category) to **Yes**.
3. Specify the scoping of the supported waterfall diagram result. **Scoping Method** property options include:
   - **Geometry Selection**: Displays an associated **Geometry** property. Use this property to select a desired face or faces on the model.
   - **Named Selection**: Displays an associated **Named Selection** property. Use this property to select a desired face-based Named Selection from the drop-down menu.
   - **Named Selections**: Displays an associated **Named Selections** property. Use this property to display the drop-down menu, press and hold the [Ctrl] key, and select multiple face-based Named Selections from the list.

   Note:

   - For the **Geometry Selection** and **Named Selection** options, the application automatically creates a panel for each body contained in the scoped geometry.
   - For **Named Selections**, each named selection defines a panel. And the face selections of each named selection must be contained in a single body.

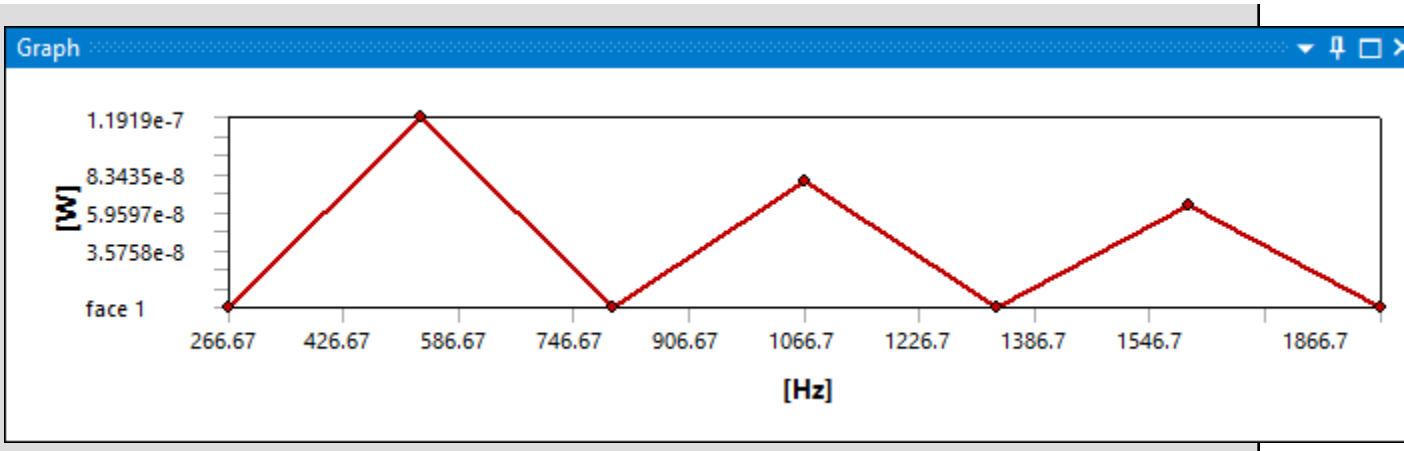
4. Set the **Selection Mode** property to **RPM** and then specify a desired value in the **RPM** property.
5. Evaluate the result.

Display Options

Once your result is evaluated, use the **Display Panel** property to display data. When set to **No** for the **ERP Waterfall Diagram** result, the **Graph** and **Tabular Data** windows display an accumulated result. That is, a summation of the result, for each frequency, for the specified RPM value. For all other supported waterfall results, these windows display the results of the accumulated results on the combined faces from all the panels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

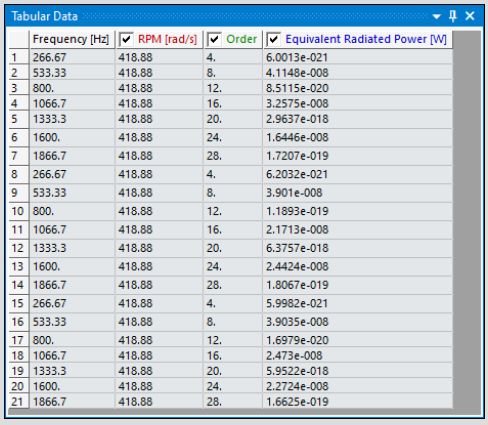
Published: 2023-10-23



Tabular Data

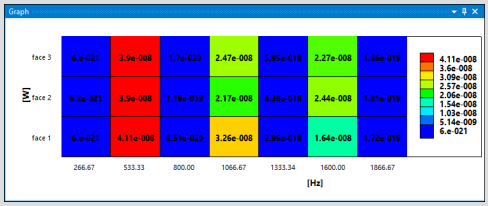
	Frequency [Hz]	RPM [rad/s]	Order	Equivalent Radiated Power [W]
1	266.67	418.88	4.	1.8203e-020
2	533.33	418.88	8.	1.1919e-007
3	800.	418.88	12.	2.2102e-019
4	1066.7	418.88	16.	7.9018e-008
5	1333.3	418.88	20.	1.5292e-017
6	1600.	418.88	24.	6.3594e-008
7	1866.7	418.88	28.	5.1899e-019

When set to **Yes**, you see result values for each panel for the specified **RPM** value in the **Tabular Data** window and the **Graph** window displays a mosaic chart of the frequency values versus the result values, such as power (W).



Tabular Data

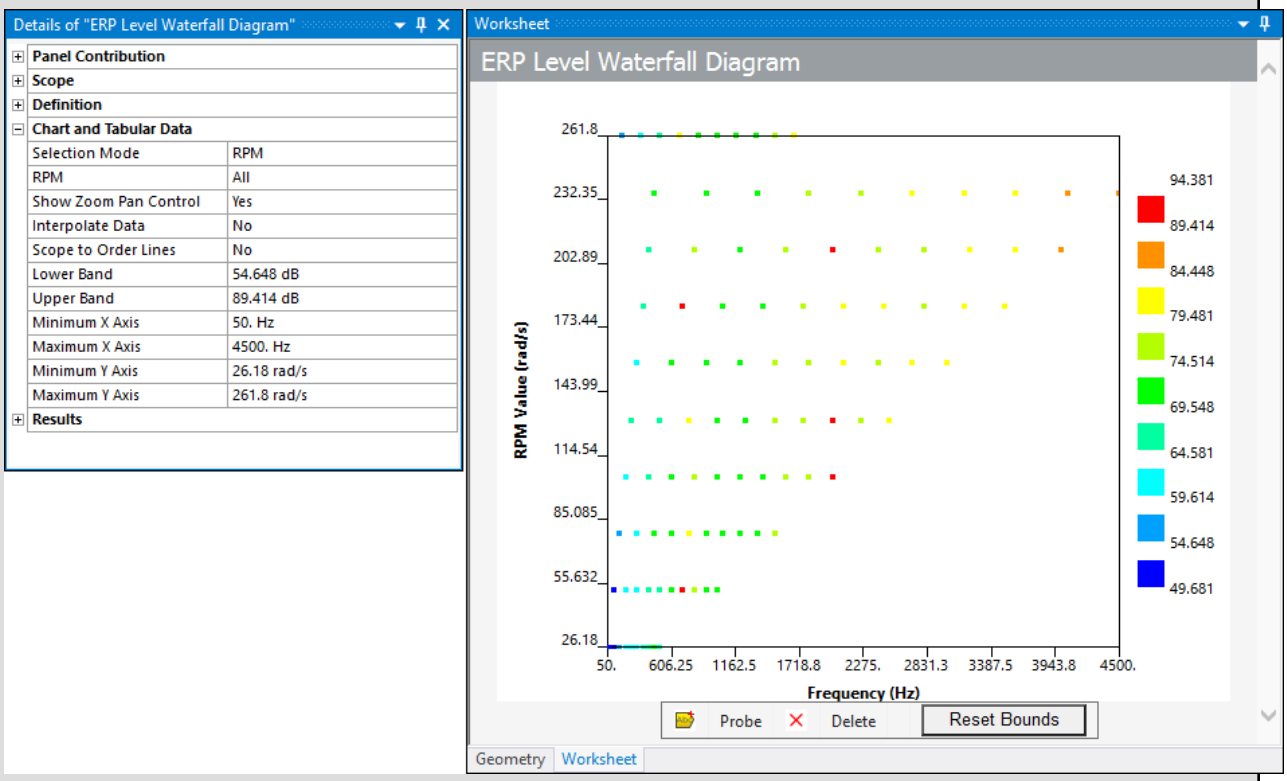
	Frequency [Hz]	RPM [rad/s]	Order	Equivalent Radiated Power [W]
1	266.67	418.88	4.	6.0013e-021
2	533.33	418.88	8.	4.1148e-008
3	800.	418.88	12.	8.5115e-020
4	1066.7	418.88	16.	3.2575e-008
5	1333.3	418.88	20.	2.9637e-018
6	1600.	418.88	24.	1.6446e-008
7	1866.7	418.88	28.	1.7207e-019
8	266.67	418.88	4.	6.2032e-021
9	533.33	418.88	8.	3.901e-008
10	800.	418.88	12.	1.1893e-019
11	1066.7	418.88	16.	2.1713e-008
12	1333.3	418.88	20.	6.3757e-018
13	1600.	418.88	24.	2.4424e-008
14	1866.7	418.88	28.	1.8067e-019
15	266.67	418.88	4.	5.9982e-021
16	533.33	418.88	8.	3.9035e-008
17	800.	418.88	12.	1.6979e-020
18	1066.7	418.88	16.	2.473e-008
19	1333.3	418.88	20.	5.9522e-018
20	1600.	418.88	24.	2.2724e-008
21	1866.7	418.88	28.	1.6625e-019



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Interpolate Data

By default, waterfall diagrams display interpolated **and** actual values. To display only the actual values, without interpolation, set the **Interpolate Data** property to **Yes** (the default is **No**).

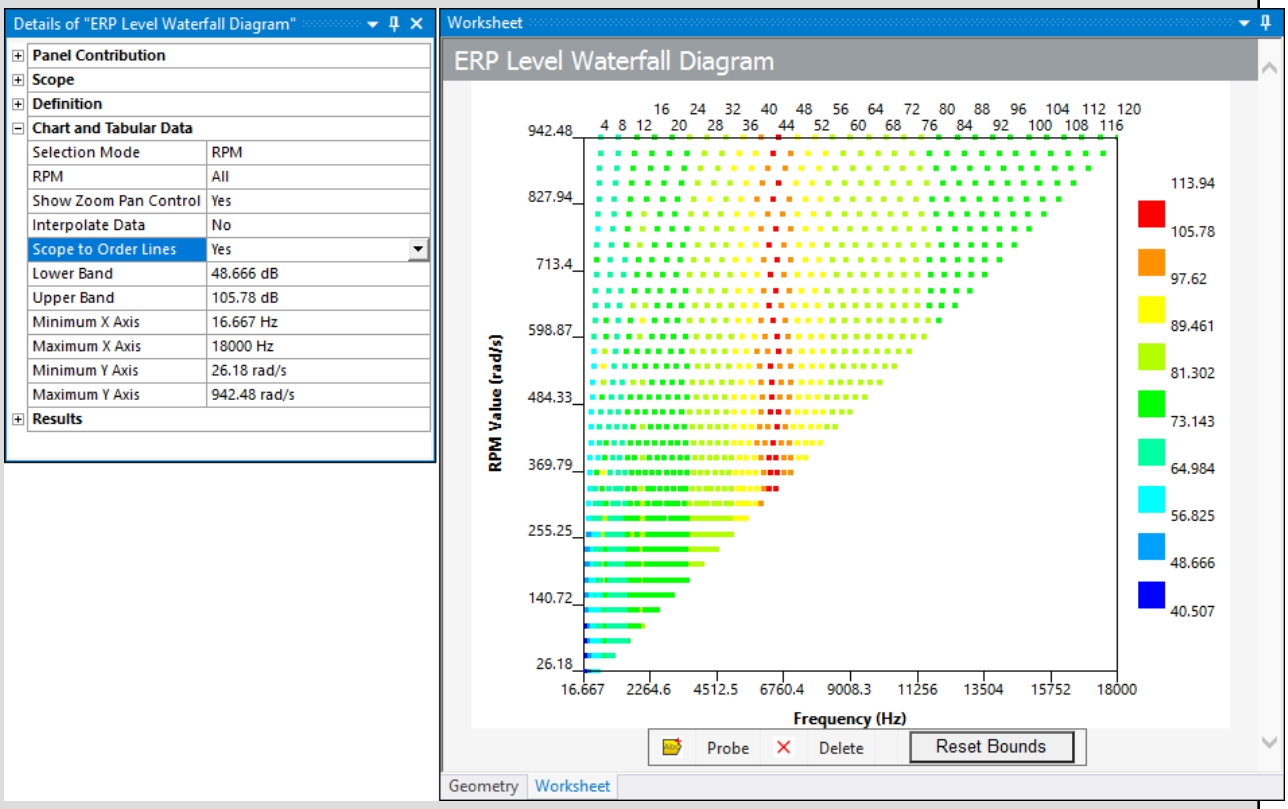


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Displaying Order Lines

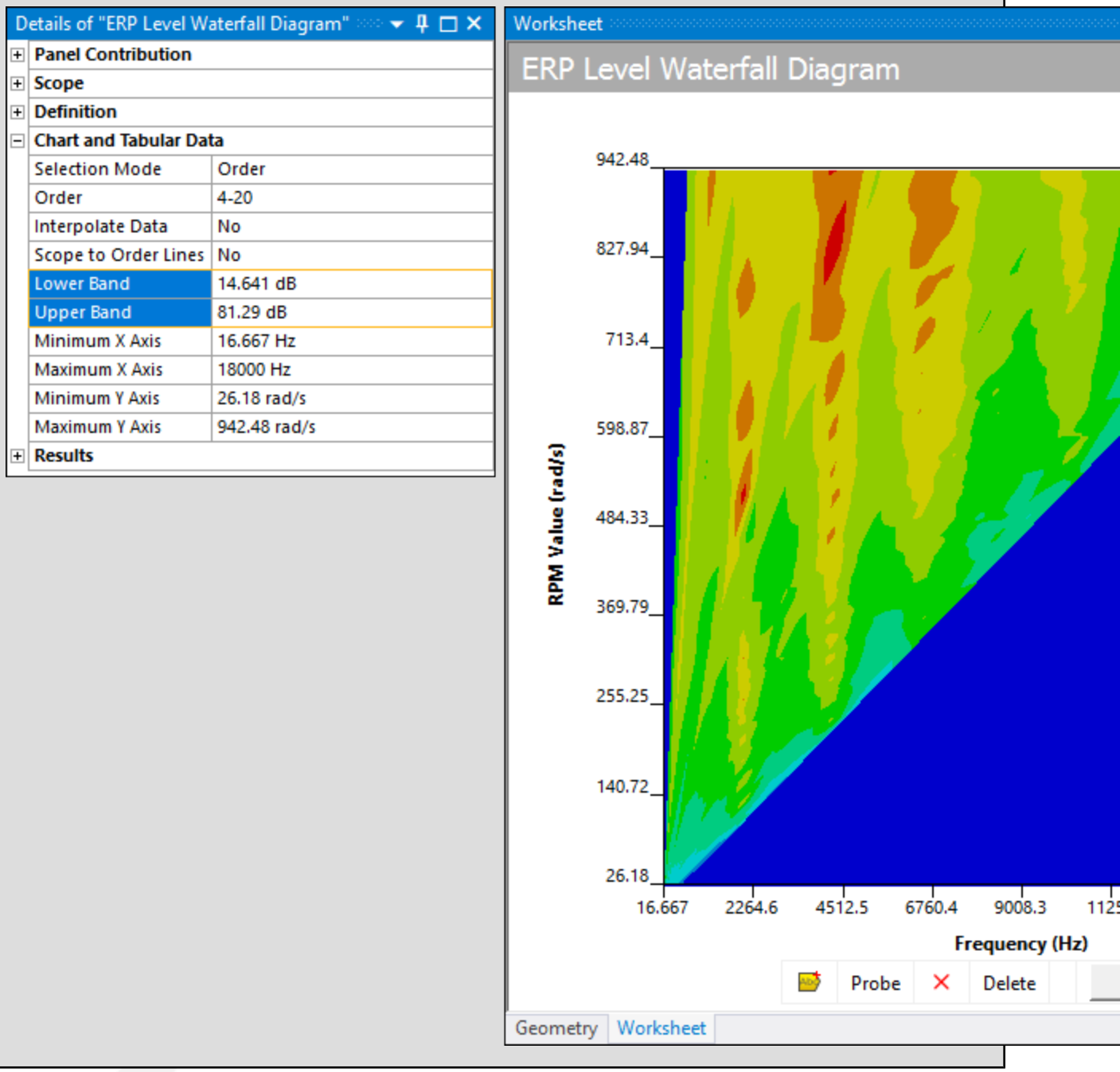
To display the values included in the order lines only, set the Use the **Scope to Order Lines** property to **Yes** (the default is **No**).



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifying Upper/Lower Bands

The application populates the **Lower Band** and **Upper Band** property values following the solution. These values represent a range of available data points. Not all regions of the diagram include data. Once available, you can manually specify a value for the lower or upper contour band to redistribute (in equal increments) the display. This can be beneficial when you have groupings of result data points at one end of the color assignment. Use the **Reset Bounds** button on the diagram to reset changes.

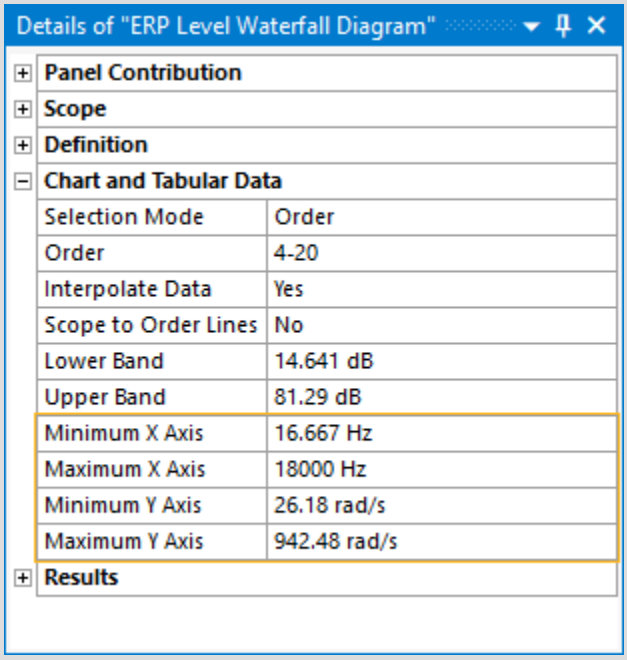


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Plotting Min/Max Data

The application calculates the values of the **Minimum/Maximum X Axis** and **Minimum/Maximum Y Axis** properties during the solution and automatically plots the values on the axes of the diagram. The diagram only displays values within this range. You can modify the range of these properties by manually entering frequency or RPM values presented on the diagram or in the associated result column of the **Tabular Data** window.



Note:

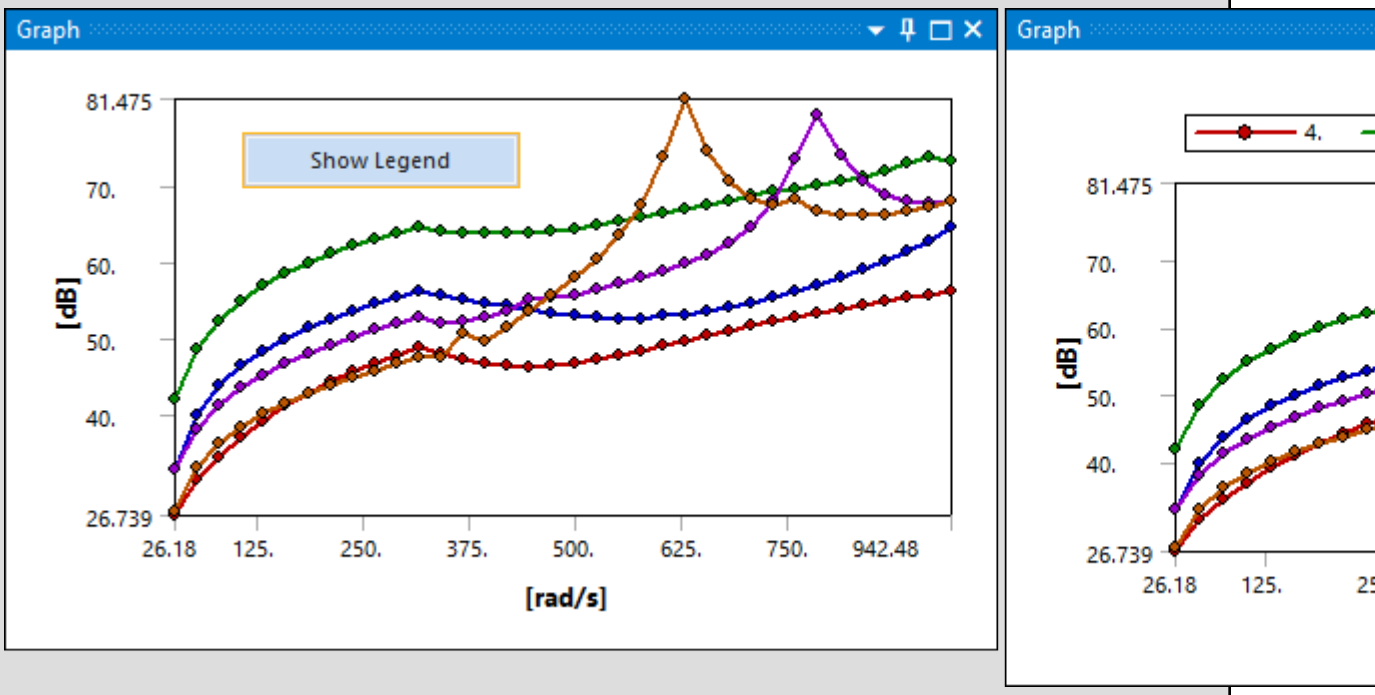
These values automatically update when you zoom in or out on an area of the diagram.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

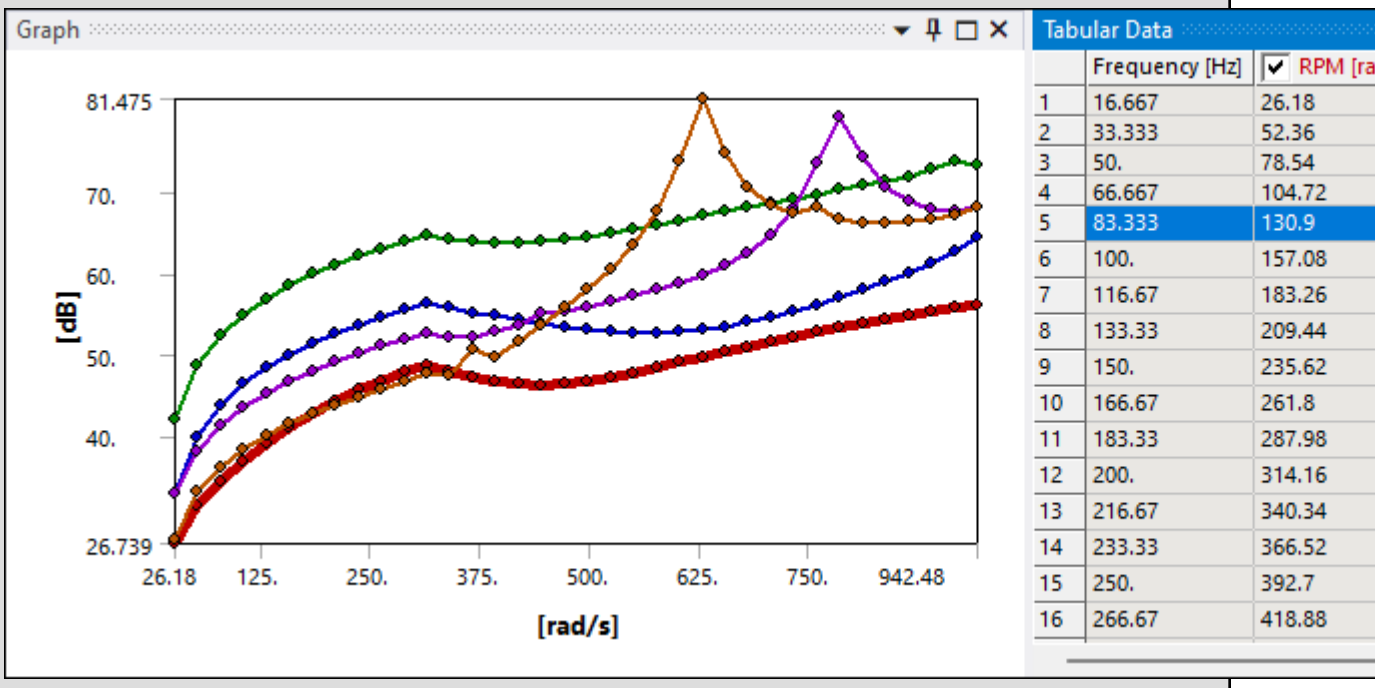
Displaying Graph Window Legend

Once you have displayed result data, you can:

- Right-click in the **Graph** window and select **Show Legend** (or **Hide Legend**) to display graph annotations. This example shows a specific set or **Order** numbers to simplify the display.



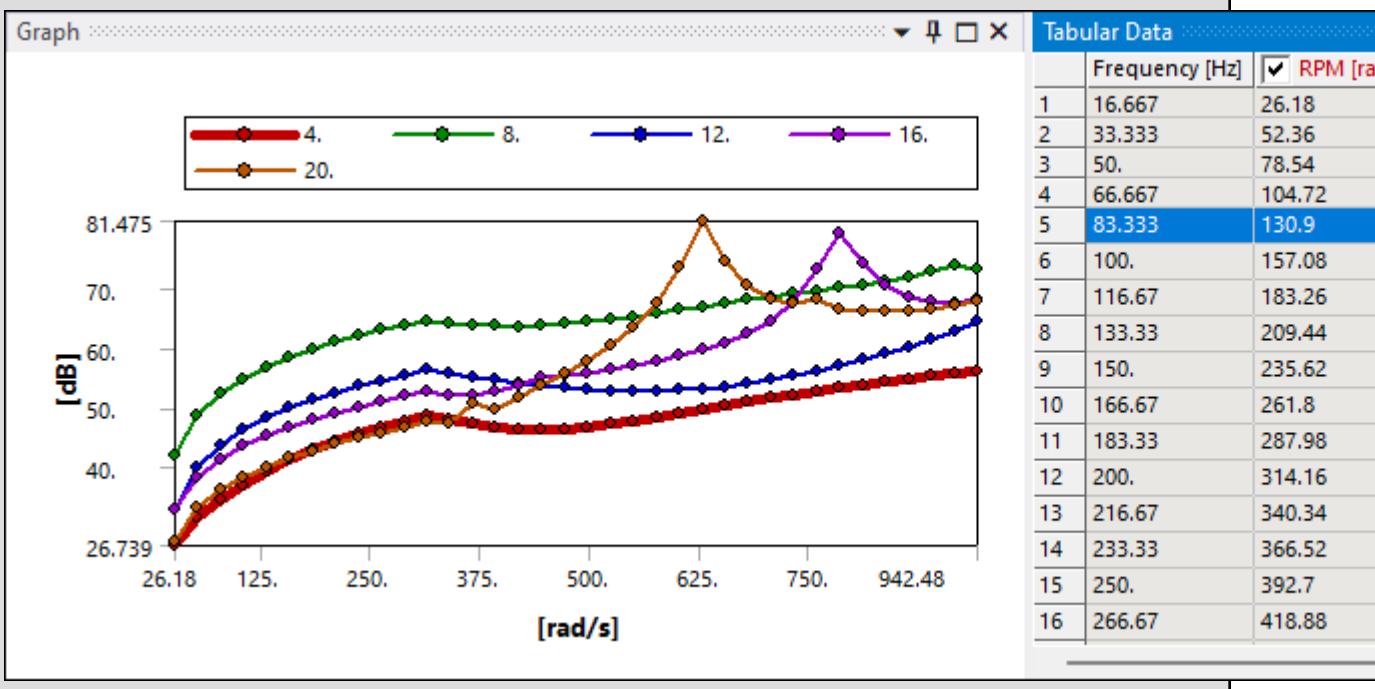
- Select one or more of the numeric rows in the **Tabular Data** window to highlight the value(s) in the graph.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The highlight applys to the legend value as well.



Line Pressure Result

You can scope this result type to a Construction Geometry **Path** (p. 883).

The **Line Pressure** result is supported for the following analysis types:

- Static Structural
- Transient Structural
- Modal
- Harmonic Response
- Eigenvalue Buckling
- Steady-State Thermal
- Transient Thermal
- Electric
- Thermal-Electric
- Response Spectrum

Volume Result

You can scope this result type to bodies and elements, either through direct geometry or element selection or using direct geometry-based or element-based Named Selections. You can also scope the result using the **Result File Item** (p. 2002) option. The default setting for the **Scoping Method** property

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

is **Geometry Selection** and the **Geometry** property is set to **All Bodies**. The **Volume** result displays the **Total** volume of your scoping as well as the **Minimum** and **Maximum** volume of all elements that result from the loading. This result is useful for visualizing the volume of elements when structural loading is applied.

Supported Analysis Types

The **Volume** result is supported for the following analysis types:

- Coupled Field Analyses
- Static Structural
- Transient Structural
- Modal
- Harmonic Response
- Eigenvalue Buckling
- Steady-State Thermal
- Transient Thermal
- Electric
- Thermal-Electric
- Response Spectrum

Volume Probe

The **Location Method** options for the **Volume** probe includes **Geometry Selection**. Geometry selections include bodies and mesh elements. Once evaluated, the probe displays the time history of the total volume computed over the geometry selection. See the Probe Details View (p. 1951) section for a description of all other **Details** view properties.

Supported Analysis Types

The **Volume** probe is supported for the following analysis types:

- Coupled Field Static
- Coupled Field Transient
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal
- Electric

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Thermal-Electric

Thermal Results

The following thermal result topics are addressed in this section:

Temperature

Heat Flux

Heat Reaction

Error (Thermal)

Thermal Flow Results

Thermal Probes

Thermal Contact Results

Thermal Probes (p. 2155) can be used to determine the following results:

- Temperature
- Heat Flux
- Heat Reaction

Note:

Currently, thermal analyses do not support the Contact Tool (p. 2079).

Temperature

In a steady-state or transient thermal analysis, temperature distribution throughout the structure is calculated. This is a scalar quantity.

Scoping allows you to limit the temperature display to particular geometric entities. Similarly scoping allows you to get reactions at specific boundary condition objects. Temperature results can be displayed as a contour plot. You can also capture the variation of these results with time by using a probe.

Heat Flux

The Mechanical application calculates the heat flux (q/A, energy per unit time per unit area) throughout the body. Heat flux can be output as individual vector components X, Y or Z.

You can display the X, Y, and Z components of heat flux in different coordinate systems.

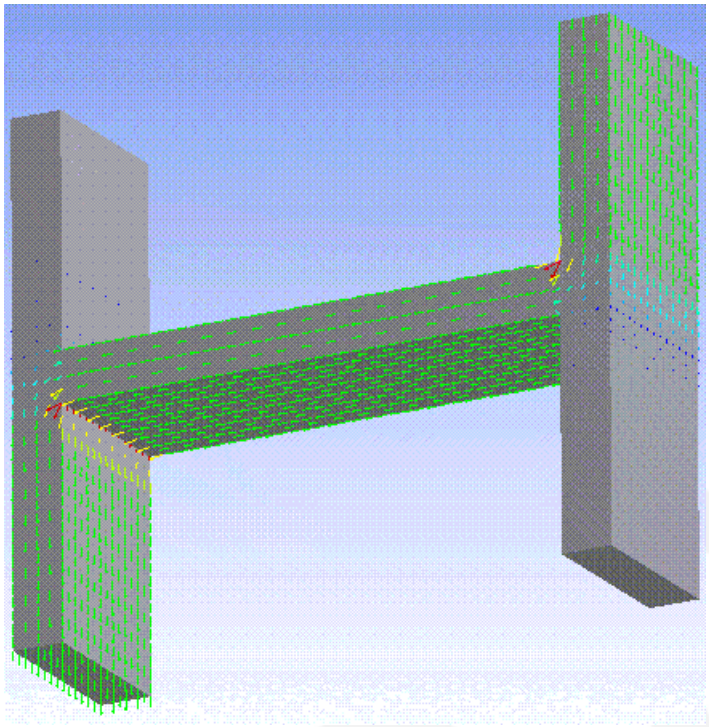
Scoping allows you to limit the heat flux display to particular geometric entities. Similarly scoping allows you to get reactions at specific boundary condition objects. Heat flux results can be displayed as a contour plot. You can also capture the variation of these results with time by using a probe.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Plots of Vector Heat Flux

A Vector Heat Flux plot provides the direction of heat flux (relative magnitude and direction of flow) at each point in the body.

The following graphic illustrates an example showing a high temperature area at the top and a low temperature area at the bottom. Note the direction of the heat flow as indicated by the arrows.



Request Vector Heat Flux plots in the same way that you would request any other result. After inserting the result object in the tree and solving, click the **Graphics** button (p. 74) in the **Result** Context tab.

Heat Reaction

You can obtain heat reaction (q, energy per unit time) at locations where a temperature, imported temperature, convection, or radiation boundary condition is specified. Heat reaction is a scalar.

To obtain a heat reaction result, insert a **Reaction** probe (p. 1927) and specify an existing **Boundary Condition**, **Geometry**, **Contact Region**, or **Surface** construction geometry. See Thermal Probes (p. 2155) for more information.

Error (Thermal)

The values for this result have no unit of measure. The description of this result is the same as Error (Structural) (p. 2030) except that heat flux is the basis for the errors instead of stresses. However, the structural error result differs in that it has an energy-based unit of measure (Joule, etc.).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Thermal Flow Results

You use the following flow-based result types for 2D and 3D thermal fluid flow analyses. Your model must include a line body (p. 872) that you specify as **Thermal Fluid** for these result types to be available for selection.

- **Fluid Flow Rate**: rate of fluid flow through the specified Line Body.
- **Fluid Heat Conduction Rate**: heat flow rate due to conduction within the fluid.

See FLUID116 in the *Mechanical APDL Element Reference*. The **Output Data** topic examines the solution output for these results.

Thermal Probes

The following thermal probe types are available.

Probe Type	Applicable Analysis Types	Output
Temperature	Coupled Field Static, Coupled Field Transient, Steady-State Thermal and Transient Thermal	Temperature: overall
Heat Flux	Coupled Field Static, Coupled Field Transient, Steady-State Thermal and Transient Thermal	Heat Flux: X axis, Y axis, Z axis
Heat Reaction	Coupled Field Static, Coupled Field Transient, Steady-State Thermal and Transient Thermal	Heat: overall

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output
Radiation (see note below)	Coupled Field Static, Coupled Field Transient, Steady-State Thermal and Transient Thermal	Outgoing Net Radiation, Emitted Radiation, Reflected Radiation, Incident Radiation

Note:

- Refer to the Probe Details View (p. 1951) section for additional information about the above scoping options.
- If you receive a negative value for the **Outgoing Net Radiation** result, then the application is detecting that heat is entering the scoped entity.
- For 2D plane stress models the Radiosity Solver method assumes an infinite third dimension so the **Radiation Probe** results will be proportional to the Workbench model thickness.
- For plane and cyclic symmetries the **Radiation Probe** results are calculated for the full symmetry model.

See the Probes (p. 1946) section for further information.

Thermal Contact Results

If your model contains **Contact Regions**, you can define the contact results as listed below under the **Connections** object or the **Solution** object by inserting a **Contact Tool**. See the Reviewing the Results section of the *Mechanical APDLContact Technology Guide* for additional information.

Thermal Flux: available only for evaluating contact conditions after solution. This data represents the total heat flux at the contact surface.

Status: status codes include:

- **-3**: MPC bonded contact.
- **-2**: MPC no-separation contact.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **0**: open and not near contact.
- **1**: open but near contact.
- **2**: closed and sliding.
- **3**: closed and sticking.

The labels **Far**, **Near**, **Sliding**, and **Sticking** are included in the legend for **Status**.

Note:

- MPC-based contact definitions use negative values. They indicate the intentional removal of one or more contact constraints to prevent over-constraint.
- Contact that has been deactivated via **Auto Asymmetric** behavior will be displayed with a status of **Far-Open**. Results for deactivated pairs can be suppressed in the **Contact Tool** by changing **Both** to either **Contact** or **Target** as necessary.

Please review the descriptions, restrictions, and caveats for contact results and displays that are included in the Contact Results (p. 2043) section under Structural Results (p. 2017).

Magnetostatic Results

A magnetostatic analysis offers several results items for viewing. Results may be scoped to bodies and, by default, all bodies will compute results for display. You can use the Details view to view vector results in several ways. Magnetic Flux Density, Magnetic Field Intensity, and Force represent the magnitude of the results vector and can be viewed as a contour or as a directional vector. Any directional solution represents direction vector components (X, Y, Z) of the vector. They may be displayed as a contour.

The following electromagnetic result topics are addressed in this section:

Electric Potential
Total Magnetic Flux Density
Directional Magnetic Flux Density
Total Magnetic Field Intensity
Directional Magnetic Field Intensity
Total Force
Directional Force
Current Density
Inductance
Flux Linkage
Error (Magnetic)
Magnetostatic Probes

Magnetostatic Probes (p. 2161) can be used to determine the following results:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Flux Density
- Field Intensity
- Force Summation
- Torque
- Energy
- Magnetic Flux

Known Limitation

In some instances, during the solution process for magnetic analyses, the application may automatically generate new elements that are not included in the original mesh or within a geometry body. Furthermore, these elements may contain magnetic force (FMAG) results.

These elements are considered to be supplementary elements. If one of these supplementary elements share a node with an element from the original mesh, the forces from the supplementary elements can contribute to the force value at the shared node.

Regardless of whether you scope results to a body, the supplementary elements may still contribute to the force results at nodes of the scoped body.

Using the **/POST1** command, you can counteract this effect by unselecting the element type associated with the supplementary elements with the command string: **ESEL**,**U**,**TYPE**.

Electric Potential

Electric potential represents contours of constant electric potential (voltage) in conductor bodies. This is a scalar quantity.

Total Magnetic Flux Density

Magnetic Flux Density is computed throughout the simulation domain and is a vector quantity. Selecting this option allows you to view the magnitude of the vector as a contour or as a directional vector.

Directional Magnetic Flux Density

Magnetic Flux Density vector components are computed throughout the simulation domain. Selecting this option allows you to view individual vector components (X, Y, Z) as a contour.

Total Magnetic Field Intensity

Magnetic Field Intensity is computed throughout the simulation domain and is a vector quantity. Selecting this option allows you to view the magnitude of the vector as a contour or as a directional vector.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Directional Magnetic Field Intensity

Magnetic Field Intensity vector components are computed throughout the simulation domain. Selecting this option allows you to view individual vector components (X, Y, Z) as a contour.

Total Force

Total Force results represent electromagnetic forces on bodies. This is a vector quantity. Selecting this option allows you to view the magnitude of the vector as a contour or as a directional vector.

Directional Force

Vector components of force and torque are computed throughout the simulation domain. They are meaningful only on non-air bodies. Selecting this option allows you to view individual vector force components (X, Y, Z) as a contour. The total summed forces and torque are available in the Details view. For example, requesting the z component of directional force/torque will report the net force acting in the z direction and the net torque acting about the z axis of the specified coordinate system.

Current Density

Current density can be computed for any solid conductor body. It is displayed as a vector and is best viewed in wireframe mode. You can use the **Vector** group to adjust the vector arrow viewing options. You can use the element-aligned option in the **Vector** group for current density vectors, but not the grid-aligned option.

Inductance

Inductance can be computed for conductor bodies. It is defined as a measure of the differential change in flux linkage to the differential change in current. This is represented by the equation below, where $d\psi$ is the differential change in flux linking conductor j produced by a differential change in current for conductor i. Note that this is valid for linear and nonlinear systems, the inductance will be a function of current.

$$L_{ij} = \frac{d\psi_{ij}}{di_i}$$

Inductance is often used as a parameter in electric machine design and in circuit simulators.

A conductor body must have a current load to be considered in inductance calculations. Inductance results are presented in the Worksheet View. The results are presented in table form. The example below shows inductance results for a two-conductor system. The diagonal terms represent self-inductance, while the off-diagonal terms represent mutual inductance. In this case, L_{11} = 1*e* - 4, L_{22} = 8*e* - 4, L_{12} = L_{21} = 4*e* - 4 Henries.

	Cond1 (H)	Cond2 (H)
Cond1	$1e^{-4}$	$4e^{-4}$
Cond2	$4e^{-4}$	$8e^{-4}$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The Details view for inductance allows you to define a Symmetry Multiplier. Use this if your simulation model represents only a fraction of the full geometry. The multiplier should be set to compensate for the symmetry model. For example, if you create a half-symmetry model of the geometry for simulation, set the Multiplier to '2.' Changing the multiplier will update the Worksheet results.

Note:

- Computing inductance can be time-consuming and should only be used if needed.
- Loads (Voltage, and Current) must be constant when Inductance is specified. Tabular and function loads are not supported.
- Inductance can only be used with a single step, single substep solution. User settings to the contrary will be overridden.
- Inductance requires the **Direct** solver setting (default) for the **Solver Type** property of **Analysis Settings**. User settings to the contrary will be overridden.

Flux Linkage

Flux linkage can be computed for any system incorporating a conductor. Solving for flux linkage calculates the flux, ψ, linking a conductor. This is commonly referred to as the "flux linkage." For nonlinear systems, the flux linkage will be a function of current. Flux linkage is also a function of stroke (for example, displacement of an armature).

Flux linkage is often used to compute the emf (electromotive force) in a conductor, defined using the equation below, where *V* is the electromotive force, typically expressed in volts.

$$v = -\frac{d\psi}{dt}$$

Conductor bodies must have defined current loads to be considered in flux linkage calculations. Flux linkage results are presented in the Worksheet View. The results are presented in table form. The example below shows flux linkage results for a two-conductor system.

	Flux Linkages (Wb)
Cond1	$5e^{-4}$
Cond2	$10e^{-4}$

The Details view for flux linkage allows you to define a Symmetry Multiplier. Use this if your simulation model represents only a fraction of the full geometry. The multiplier should be set to compensate for the symmetry model. For example, if you create a half-symmetry model of the geometry for simulation, set the Multiplier to '2.' Changing the multiplier will update the Worksheet results.

Note:

- Computing flux linkage can be time-consuming and should only be used if needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Loads (Voltage, and Current) must be constant when flux linkage is specified. Tabular and function loads are not supported.
- Flux linkage can only be used with a single step, single substep solution. User settings to the contrary will be overridden.
- Flux linkage requires the **Direct** solver setting (default) for the **Solver Type** property of **Analysis Settings**. User settings to the contrary will be overridden.

Error (Magnetic)

The description of this result is similar to Error (Structural) (p. 2030) except that flux density is the basis for the errors instead of stresses.

When all materials are linear, Workbench uses relative permeability (MURX, MURY, MURZ) values which are available in the material properties.

When nonlinear materials are present, Workbench does *not* extract relative permeability from the material properties. Instead, for a given element, Workbench first sums the flux density vectors of the result nodes to form a vector called **B**. Workbench next sums the field intensity vectors of the result nodes to form a vector called **H**.

MURX, MURY, and MURZ are all assigned the value (|B|/|H|) / MUZERO, where:

- |B| is the length of the **B** vector,
- |H| is the length of the **H** vector,
- MUZERO is free space permeability.

If the **H** vector has a zero length, the contribution of this element to the energy error will be set to 0.

Magnetostatic Probes

The following magnetostatic probe types are available.

Probe Type	Applicable Analysis Types	Output	Characteristics
Flux Density	Magnetostatic	Flux Density: X axis, Y axis, Z axis	**Scope to:** body. **Scope by:** bodies, **Orientation coord**
Field Intensity	Magnetostatic	Flux Intensity: X axis, Y axis, Z axis	**Scope to:** body. **Scope by:** bodies, **Orientation coord**
Force Summation[a]	Magnetostatic	Force Sum: X axis, Y axis, or Z axis; Symmetry Multiplier	**Scope to:** body.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
			Scope by: bodies **Orientation coord**
Torque[b][a]	Magnetostatic	Torque: X axis, Y axis, or Z axis; Symmetry Multiplier	**Scope to:** body. **Scope by:** bodies **Orientation coord** **Summation:** Orie
Energy	Magnetostatic	Magnetic Co-energy	**Scope to:** body. **Scope by:** System
Magnetic Flux[c]	Magnetostatic	Magnetic Flux	**Scope to:** body. **Scope by:** edge.

[a] A limitation exists when the scoping of a **Force Summation** or a **Torque** probe is applied to a geometry entity (**Location Method** = **Geometry Selection**) that shares more than one body. The (unscoped) elements that are adjacent to the scoped body contribute to the probe's results.

[b] Torque results represent the torque on a body due to electromagnetic forces. Torque is specified about the origin of a coordinate system. By default, the global coordinate system is used. To change the specification point, create a local coordinate system and specify the results about the new origin. The torque result is listed in the Details pane.

[c] Magnetic Flux is computed along the scoped edge. The scoping should produce a single continuous path along the edge. Flux is reported as magnitude only.

See the Probes (p. 1946) section for further information.

Electric Results

The application supports the electric results described below. In addition, see the Electric Probes (p. 2163) and Frequency Response for Electric Results (p. 2164) sections for additional result options.

Result	Description
Electric Voltage	Represents contours of constant electric potential (voltage) in conductor bodies. This is a scalar quantity.
Total Electric Field Intensity	Is computed throughout the simulation domain and is a vector sum quantity. Selecting this result enables you to view the total magnitude of the vectors as a contour.
Directional Electric Field Intensity	Its vector components are computed throughout the simulation domain. This result enables you to view individual vector components (X, Y, Z) as contours.
Total Electric Flux Density	Is computed throughout the simulation domain and is a vector sum quantity. Selecting this result enables you to view the total magnitude of the vectors as a contour.
Directional Electric Flux Density	Its vector components are computed throughout the simulation domain. This result enables you to view individual vector components (X, Y, Z) as contours.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Result	Description
Total Electrostatic Force	Available for **Coupled Field Harmonic**, **Coupled Field Static**, and **Coupled Field Transient** analyses. This result to computed throughout the simulation domain and is a vector sum quantity. Specifying this result enables you to view the total magnitude of the vectors as a contour.
Directional Electrostatic Force	Available for **Coupled Field Harmonic**, **Coupled Field Static**, and **Coupled Field Transient** analyses. The vector components for this result are computed throughout the simulation domain. This result enables you to view individual vector components (X, Y, Z) as contours.
Total Current Density	Can be computed for any solid conductor body. It is displayed as a vector and is best viewed in wireframe mode. You can use the **Vector Display** group to adjust the vector arrow viewing options. You can use the element-aligned option in the **Vector Display** group for current density vectors, but not the grid-aligned option.
Directional Current Density	Its vector components are computed throughout the simulation domain. This result enables you to view individual current density vector components (X, Y, Z) as contours.
Joule Heat	Occurs in a conductor carrying an electric current. Joule heat is proportional to the square of the current, and is independent of the current direction. **Note:** • This result is not supported when generated by non-zero contact resistance. • During a Coupled Field Harmonic (p. 380) (piezoelectric) analysis, this result only contributes damping losses from coupled elements. Any structural elements do not contribute to the result.

Electric Probes

The following electric probe types are available. Refer to the Probe Details View (p. 1951) section for additional information about the above scoping options. Also see the Probes (p. 1946) section for further information.

Probe Type	Applicable Analysis Types	Output	Characteristics
Electric Voltage	Coupled Field Static, Coupled Field Transient, and Electric	Voltage	**Scope to:** body. **Scope by:** bodies, location (Hit Point Coordinate (p. 97)), vertex, edge, face.
Field Intensity	Coupled Field Static, Coupled Field Transient, and Electric	X axis, Y axis, Z axis, Total	**Scope to:** body. **Scope by:** bodies, location (Hit Point Coordinate (p. 97)), vertex, edge, face.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Probe Type	Applicable Analysis Types	Output	Characteristics
			Orientation coordinate system: any; defaults to Global Cartesian.
Current Density	Coupled Field Static, Coupled Field Transient, and Electric	X axis, Y axis, Z axis, Total	**Scope to:** body. **Scope by:** bodies, location (Hit Point Coordinate (p. 97)), vertex, edge, face. **Orientation coordinate system:** any; defaults to Global Cartesian.
Charge Reaction	Coupled Field Harmonic, Coupled Field Static, Coupled Field Transient	Overall Charge Reaction in Coulomb (C)	**Scope to**: Boundary Condition
Impedance	Coupled Field Harmonic	Overall Impedance in Ohms	**Scope to**: Boundary Condition
Reaction	Coupled Field Static, Coupled Field Transient, and Electric	Current: overall	**Scope to:** body. **Scope by:** Boundary Condition.
Joule Heat	Coupled Field Harmonic, Electric Analysis, Thermal-Electric Analysis, and Coupled Field Transient	Power	**Scope to:** body only. **Scope by:** body.
Quality Factor	Coupled Field Harmonic and Coupled Field Transient	Quality Factor	**Scope to**: body only. **Scope by**: body only.
Electromechanical Coupling Coefficient	Coupled Field Harmonic, Coupled Field Static, and Coupled Field Transient	Electromechanical Coupling Coefficient	**Scope to**: body only. **Scope by**: body only.

Frequency Response for Electric Results

For Coupled Field Harmonic analyses, the application supports the following electric frequency response results. The application plots the amplitude of these result values versus the frequency range in a

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

graph displayed in the Worksheet. By default, the application also plots Amplitude versus Phase Angle. See the Frequency Response (p. 2046) and Phase Response (p. 2056) section for additional information.

- **Voltage**: This result supports geometry or Named Selection scoping.
- **Charge Reaction**: This result supports Boundary Condition scoping. It represents the reaction force for the degrees of freedom for the voltage.
- **Impedance**: This result supports Boundary Condition Scoping only. It represents the resistance of current flow. It is computed using charge as follows.

  For this result, the charge Q is retrieved at the terminal and then current is defined as: $I = \frac{dQ}{dt}$ and $Q = Q_o e^{j\omega t}$, then $I = \omega Q_o e^{j\omega t} j_\infty$

  Resulting Impedance = $Z = \frac{V}{I}$

Fatigue Results

Fatigue provides life, damage, and safety factor information and uses a stress-life or strain-life approach, with several options for handling mean stress and specifying loading conditions. An example of the strain-life approach is a part/body with notch where, although the nominal response is elastic, the local response may present as a plastic. Review the following sections with regards to fatigue material properties, the theory of the calculations associated with the material properties, loading conditions, and results.

Fatigue Material Properties

Fatigue Stress Life versus Strain Life

Frequency-Based Fatigue

Fatigue Analysis Application

Fatigue Results

Fatigue Combination

Mechanical Embedded DesignLife UI

Fatigue Material Properties

Engineering Data contains example materials which may include fatigue curves populated with data from engineering handbooks. You can also add your own fatigue curves. The **Fatigue Tool** will use the information from these curves for each material in the model when calculating life, damage, safety factors, etc. If Young's Modulus is temperature dependent, then the fatigue calculations are carried out using the Young's Modulus computed at the reference temperature of the body.

For the strain-life approach, the materials must have Strain-Life Parameters defined. For the Stress-Life approach, the materials must have S-N Curve defined. To add this data to a material follow the Add Material Properties procedure (see Adding Material Properties in the *Engineering Data User's Guide*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Strain-Life Parameters

The following four strain-life parameter properties and the two cyclic stress-strain parameters must have data defined:

- **Strength Coefficient**
- **Strength Exponent**
- **Ductility Coefficient**
- **Ductility Exponent**
- **Cyclic Strength Coefficient**
- **Cyclic Strain Hardening Exponent**

S-N Curve

The alternating stress, or stress-life (SN), mean curve data can be defined with respect to mean stress, r-ratio, or temperature. The **Interpolation** method (**Log-Log**, **Semi-Log**, or **Linear**) can be defined. The curve data must be defined to be greater than zero.

- **Mean Stress**

  Use this definition if experimental SN data was collected at constant mean stress for individual SN curves.

- **R-Ratio**

  Use this definition if multiple SN curves were collected at a constant r-ratio. The r-ratio is defined as the ratio of the second loading to the first: r = L2 / L1. Typical experimental r-ratios are -1 (fully reversed), 0 (zero-based), and .1 (to ensure that a tensile stress always exists in the part).

- **Temperature**

  Use this definition if multiple SN curves were collected at different temperatures.

  If you choose the **Temperature** Field Variable while defining an S-N Curve material property, and you enter multiple S-N Curves (**Cycles**/**Alternating Stress**) for different Temperatures, the **Fatigue Tool** (p. 2176) automatically uses the appropriate S-N curve for the interpolation based on the temperature at each node of the body.

  Note:

  – The Fatigue Tool supports temperature dependent fatigue for Static Structural and Transient Structural analyses only, and not for frequency-based fatigue.

  – For non proportional loading, we use the average temperature for the calculations.

It is possible to define multiple SN curves to account for different mean stress or r-ratio values. The values of mean stress/r-ratio values are only important if multiple curves are defined in the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Engineering Data workspace and the **Mean Stress Curves** option is specified for the **Mean Stress Theory** property in the **Fatigue Tool** (p. 2176).

Linear/Bilinear S-N Curve

For a Frequency-Based Fatigue analysis, the S-N Curve material properties enable you to examine frequency-based fatigue formulations using either a linear or bi-linear curve for Stress (S) versus the number of life cycles (N). Material property options include:

- **Linear**

  Use this definition for single slope SN curve data.

- **Bi-Linear**

  Use this definition for when you have two sloped of SN curve data.

Refer to the next section, Fatigue Stress Life Versus Strain Life (p. 2167), for additional information about strain-life parameters.

Note that in Engineering Data, in the **Display Curve Type** drop-down menu, you can plot either a **Strain-Life** or **Cyclic Stress-Strain** curve.

Fatigue Stress Life versus Strain Life

Within the Ansys fatigue module, the first decision that needs to be made in performing a fatigue analysis is which type of fatigue analysis to perform, Stress-life or Strain-life. Stress-life is based on empirical S-N curves which are modified by a variety of factors. Strain-life is based upon the Strain Life Relation Equation where the Strain-life Parameters are values for a particular material that best fit the equation to measured results. The Strain Life Relation Equation requires a total of 6 parameters to define the strain-life material properties: four strain-life parameter properties and the two cyclic stress-strain parameters. The Strain Life Relation Equation is shown below:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E}\left(2N_f\right)^b + \varepsilon'_f\left(2N_f\right)^c$$

The two cyclic stress-strain parameters are part of the equation below:

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}}$$

Where:

$\frac{\Delta\varepsilon}{2}$ = Total Strain Amplitude

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$\Delta\sigma$ = 2 x the Stress Amplitude

E = Modulus of Elasticity

N_f = Number of Cycles to Failure

$2N_f$ = Number of Reversals to Failure

And the parameters required for a Strain Life analysis include:

σ'_f = Fatigue Strength Coefficient

B = Fatigue Strength Exponent (Basquin's Exponent)

ε'_f = Fatigue Ductility Coefficient

C = Fatigue Ductility Exponent

K' = Cyclic Strength Coefficient

n' = Cyclic Strain Hardening Exponent

Note in the above equation, total strain (elastic + plastic) is the required input. However, running a finite element analysis to determine the total response can be very expensive and wasteful, especially if the nominal response of the structure is elastic. An accepted approach is to assume a nominally elastic response and then make use of Neuber's equation to relate local stress/strain to nominal stress/strain at a stress concentration location.

To relate strain to stress we use Neuber's Rule:

$\varepsilon\sigma = K_t^2 e S$

Where:

ε = Local (Total) Strain

σ = Local Stress

K_t = Elastic Stress Concentration Factor

e = Nominal Elastic Strain

S = Nominal Elastic Stress

The Fatigue Tool assumes nominal elastic behavior and therefore $e=S/E$, and by simultaneously solving Neuber's equation along with cyclic strain equation, we can calculate the local stress/strains (including plastic response) given only elastic stress input. Note that this calculation is nonlinear and is solved using iterative methods. In addition, Ansys fatigue calculations use a value of 1 for the Elastic Stress Concentration Factor (K_t), assuming that the mesh is refined enough to capture any

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

stress concentration effects. K_t in this case is not be confused with the Stress Reduction Factor option which is typically used in a Stress-life analysis to account for factors such as reliability and size effects.

Important:

The Strain-life approach is currently only valid for linear stress analyses (elastic material models only).

Frequency-Based Fatigue

Mechanical enables you to perform Frequency-Based Fatigue analyses using the Harmonic Response and Random Vibration analysis types or using a combination of the two.

Random Vibration Analysis

When analyzing the dynamic response of the structure, the application uses the frequency domain for calculations and therefore loading is a function of frequency or random vibrations. Examples of this include:

- Electronic vibration in PCB board
- Automobile engine vibrations
- Vibrations of a helicopter in flight
- Vibrations of a shaker table
- Natural vibrations of structures due to sea waves or winds

Refer to the Fatigue Material Properties for Random Vibration (Spectral) Fatigue (p. 2169) and Fatigue Result Methods for Random Vibration (Spectral) Fatigue (p. 2173) sections for additional technical details for Frequency-Based Fatigue calculations for Random Vibration analyses.

Harmonic Response Analysis

If the response of the system is harmonic in nature, not random, the application uses the maximum stress response for a either a single target frequency or sweeping multiple target frequencies, along with the duration of exposure to calculate the fatigue effects. Refer to the Fatigue Result Methods for Harmonic Fatigue (p. 2175) for additional information for Harmonic Fatigue analyses.

Combination Analysis

You can also choose to calculate fatigue damage from Random Vibration and Harmonic analyses and then use the Fatigue Combination (p. 2188) object to combine the effects in order to analyze the total damage to the system.

Fatigue Material Properties for Random Vibration (Spectral) Fatigue

All Frequency-Based Fatigue formulations are driven by the material's relationship between Stress (S) and number of life cycles (N). This S-N relation can be defined in the Engineering Data Workspace using any of the following:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Linear S-N Curve (p. 2170)
- Bi-linear S-N Curve (p. 2171)
- S-N Curve Table (p. 2172)

Important:

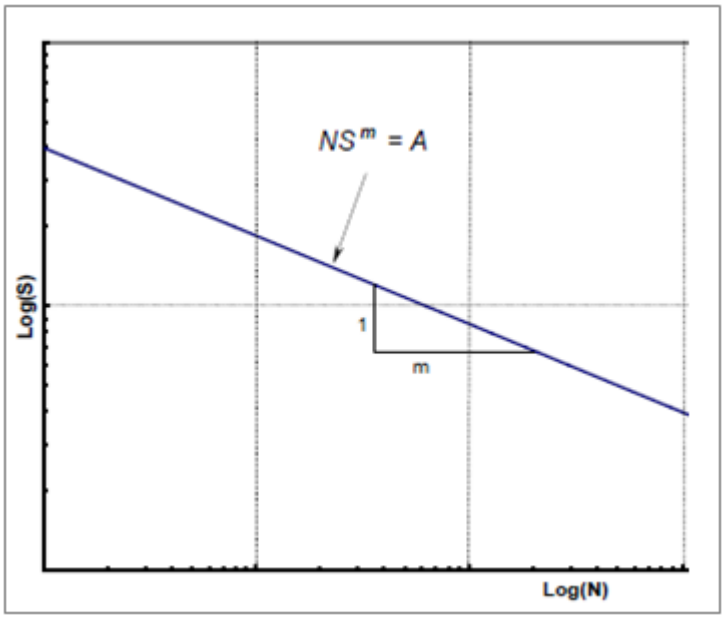
- For Frequency-Based Fatigue calculations, you need to employ at least one of the SN Curve formulations listed above in order to proceed with a solution.
- For both the Linear S-N Curve and the Bi-linear S-N Curve, the reference units for the parameters is [Pa]. No other unit or unit system is currently supported.
- In the various S-N Curve formulas covered here, the "S" value always refers to Stress Amplitude.

As long as you define material properties using one of the above formulas, you can perform Frequency-Based Fatigue calculations.

Linear S-N Curve Formula

This is a single segment S-N Curve formula of the form:

$NS^m = A$



Where:

A = Fatigue Strength Coefficient

m = Fatigue Strength Exponent

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

S = Stress Amplitude (in Pa)

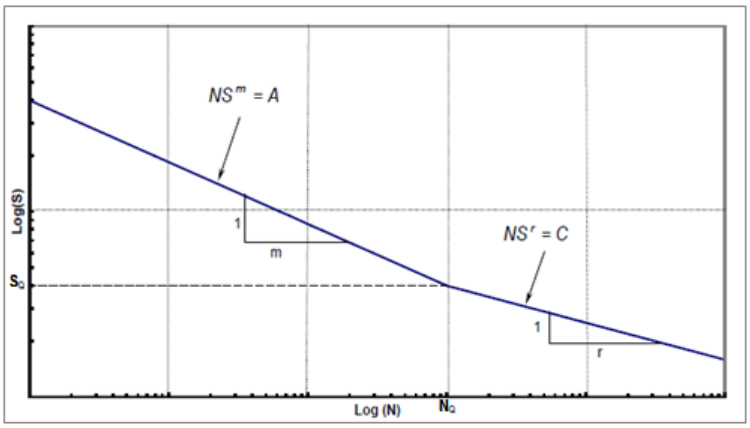
Note:

The value "m" is the inverse negative slope of the Linear S-N Curve.

Bi-Linear S-N Curve Formula

This is a two segment S-N Curve formula of the form:

$NS^m = A$ & $NS^r = C$



Where:

A = First Fatigue Strength Coefficient

m = First Fatigue Strength Exponent

C = Second Fatigue Strength Coefficient

r = Second Fatigue Strength Exponent

S_Q = Stress Amplitude at Transition Point (in Pa)

N_Q = Number of cycles at Transition Point

Using the **Derive from** property in Engineering Data Workspace, select one of two methods of definition:

- **Coefficients and Exponents**: Users define A, m, C, and r.
- **Transition Point**: define m, r, N_Q, and S_Q.

All other properties are automatically calculated.

Note:

The "m" and "r" values are the inverse negative slopes of the Bilinear S-N Curve.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

S-N Curve Table

As illustrated below the default data for the S-N Curve is contained in the Engineering Data Workspace tabular data of corresponding Alternating Stresses and Cycles of life.

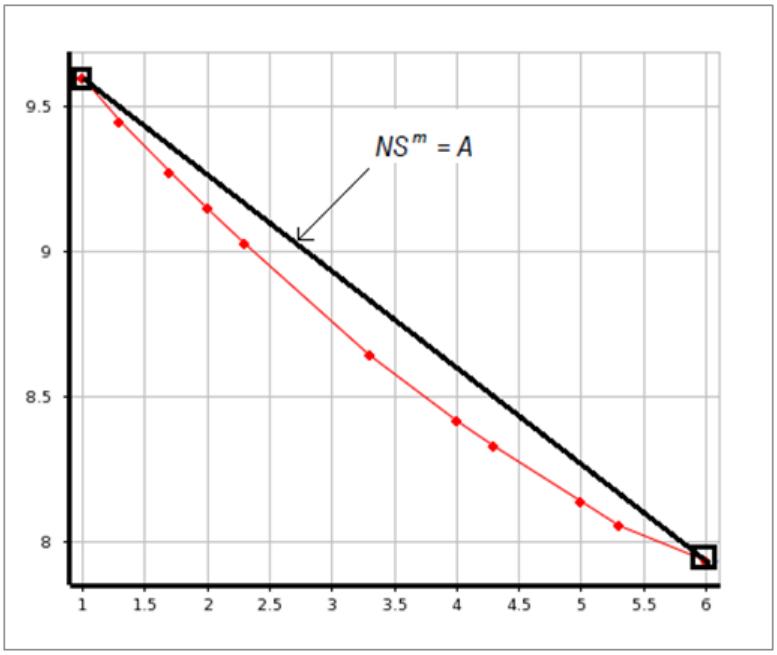
	B	C
1	Cycles	Alternating Stress (Pa)
2	10	3.999E+09
3	20	2.827E+09
4	50	1.896E+09
5	100	1.413E+09
6	200	1.069E+09
7	2000	4.41E+08
8	10000	2.62E+08
9	20000	2.14E+08
10	1E+05	1.38E+08
11	2E+05	1.14E+08
12	1E+06	8.62E+07
*		

Note:

If only the SN table is provided (and not the Linear/Bilinear parameters themselves), then the solver will use the first and last points of the table to perform a linear interpolation using equation $NS^m=A$ and arrive at the required parameters of A and m. Once the A and m values are obtained, we can proceed with the solution normally. If either of the parameters (Linear/Bilinear) are provided directly through the material definition, they will be used directly. (The table, if present, will be ignored in this case).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Fatigue Result Methods for Random Vibration (Spectral) Fatigue

There are three different formulations of Random Vibration (Spectral) Fatigue that are currently supported.

Steinberg Formulation

Steinberg Formulation utilizes all three stress occurrences (1σ, 2σ, 3σ) and their rate of occurrence along with the Miner's rule in order to compute the total fatigue damage of the system.

$$D = \frac{n_{1\sigma}}{N_{1\sigma}} + \frac{n_{2\sigma}}{N_{2\sigma}} + \frac{n_{3\sigma}}{N_{3\sigma}}$$

Where:

$n_{1\sigma}$ = actual number of cycles at or below the 1σ level (0.6831 $f_0 t$).

$n_{2\sigma}$ = actual number of cycles at or below the 2σ level (0.271 $f_0 t$).

$n_{3\sigma}$ = actual number of cycles at or below the 3σ level (0.0433 $f_0 t$).

$N_{1\sigma}$, $N_{2\sigma}$, $N_{3\sigma}$ = allowable number of cycles (from fatigue curve) at 1σ, 2σ, 3σ stress levels.

$N_{1\sigma}$, $N_{2\sigma}$, $N_{3\sigma}$ are obtained by using the S-N relation and the (1σ, 2σ, 3σ) stresses to find the corresponding number of cycles. If the Bilinear curve is used, the solver chooses the appropriate curve to interpolate on, based on the value of the stresses.

f_0 is defined as the statistical frequency which is obtained as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\omega_0 = \frac{\sigma_{\dot{x}}}{\sigma_x}, f_0 = \frac{\omega_0}{2\pi}$$

Where:

$\sigma_{\dot{x}}$ = RMS Stress Velocity Result (stress based on velocity result)

σ_x = RMS Stress Result

Note:

$\dot{x}/x$ equal a user-specified measure of stress.

Narrow Band Formulation

Narrow Band formulation is a generalized method where the stress ranges are assumed to have a Rayleigh distribution. The formula to calculate Fatigue damage is given by,

$$D_{NB} = \frac{f_o t}{A}\left(\sqrt{2}\,\sigma\right)^m \Gamma\left(\frac{m}{2}+1\right)$$

Where:

f_0 = Statistical frequency

t: Exposure Duration

σ: Equivalent Alternating Stress

Γ: Gamma function.

A, *m*: SN curve properties from the equation $NS^m = A$, where S = Stress Amplitude.

If we are using the Bilinear form of the SN curve, this formula changes to:

$$D_{NB} = \frac{f_0 t}{A}\left(\sqrt{2}\,\sigma\right)^m \Gamma(a,z) + \frac{f_0 t}{C}\left(\sqrt{2}\,\sigma\right)^r \Gamma_0(b,z)$$

where:

$a = \frac{m}{2} + 1$

$b = \frac{r}{2} + 1$

$z = \left(\frac{S_Q}{\sqrt{2}\,\sigma}\right)^2$

and:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

S_Q is the stress point of intersection of the two SN Curves.

Γ_0 = lower incomplete gamma function.

$$\Gamma_0(b,z) = \Gamma(b) - \Gamma(b,z)$$

Wirsching Formulation

The Wirsching Formulation can be described as a correction factor to the Narrow Band Formulation in order to account for Wideband scenarios. Instead of using a different, more complicated method for Wideband cases, we calculate Fatigue Damage using Narrow Band formulation and simply apply the Wirsching correction factor to it, as shown:

$$D = \lambda D_{NB}$$

Where λ is the *wide band correction factor*.

$$\lambda = a(m) + [1 - a(m)](1 - \varepsilon)^{b(m)}$$

Where:

$a(m)$ = 0.926-0.033m

$b(m)$ = 1.587m - 2.323

$\varepsilon = \sqrt{1-\gamma^2}$ = Bandwidth Factor

$\gamma = \sqrt{\frac{m_2}{m_0 \times m_4}}$ = Irregularity Factor

$m_0 m_2 m_4$ = Spectral Moments

m is the fatigue strength exponent obtained from the Linear SN Curve.

Since Wirsching does not have a specialized formula for the Bilinear SN Curve, the average of the two fatigue strength exponents (*m* and *r*) is used in the Bilinear case.

Fatigue Result Methods for Harmonic Fatigue

If the response of the system is harmonic in nature, not random, the application uses the maximum stress response for a either a single target frequency or sweeping multiple target frequencies, along with the duration of exposure to calculate the fatigue effects.

There are three different options available for Harmonic fatigue calculations:

- **Single Frequency**: The stress at the defined Frequency is considered for the fatigue analysis. This stress is used along with the defined SN Curve to interpolate for the fatigue life.
- **Multiple Frequencies**: You can manually specify frequency values and their corresponding **Exposure Durations**. Stresses at each of these distinct frequencies are considered for the fatigue analysis and fatigue damage is accumulated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Sine Sweep**: You can manually specify frequency values and their corresponding **Scale Factors**. A **Sine Sweep** analysis is performed from the lowest to highest frequency value in steps of 0.5 Hz using the Sweep Rate (to account for total exposure time) and the appropriate amplitude scaling at each step. The fatigue damage is accumulated.

Note:

In the Harmonic Response fatigue analysis, the S-N Curve Table will take precedence and it will be used if available. If the table is not present, then the solver will use either the Linear/Bilinear parameters, whichever is present, to construct a pseudo curve and that will be used for the interpolation.

Fatigue Analysis Application

After you have defined the stress-life or strain-life curves for all materials in your model, you can choose your fatigue options and run the fatigue analysis.

The **Fatigue Tool** option is available from the **Toolbox** drop-down menu on the **Solution Context Tab** (p. 62) or through the right-click context menu.

Worksheet View

The Worksheet displays by default when you insert the **Fatigue Tool** during a Static Structural or Harmonic Response analysis. It displays default data, in graph form, based on the property settings of the object. The display changes as you make changes to the associated properties. In addition, this graphical Worksheet display is available for the options of the **Graph Results** menu. See the Fatigue Tool (Group) object reference page for additional information.

Property Definition

Once inserted into the tree, based on your analysis type, define the Details View properties as described below:

- Fatigue Strength Factor (Kf) (p. 2177)
- Loading Type (p. 2177)
- Scale Factor (p. 2178)
- Analysis Type (p. 2178)
- Mean Stress Theory (p. 2178)
- Method Selection (p. 2178) (Random Vibration analysis only)
- Stress Component (p. 2178)
- Exposure Duration (p. 2179) (Random Vibration and Harmonic Response analyses only)
- Frequency Selection (p. 2179) (Harmonic Response analysis only)
- Frequency (p. 2179) (Harmonic Response analysis only)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Sweep Rate (p. 2179) (Harmonic Response analysis only)
- Units Name (p. 2179)
- 1 "Unit" is Equal To (p. 2180)
- Bin Size (p. 2180)
- Use Quick Rainflow Counting (p. 2180)
- Infinite Life (p. 2180)
- Maximum Data Points To Plot (p. 2181)

Fatigue Strength Factor (Kf)

This property defines the fatigue strength reduction factor. This setting enables you to adjust the stress-life or strain-life curve(s) by a specified factor while the fatigue analysis is running. The default value is **1**. You set the property by making an entry or you can use the slider feature associated with the property. The slider represents the entry range from minimum (**0.01**) to maximum (**1**). You use this property to account for a "real world" environment that may be harsher than a rigidly-controlled laboratory environment in which the data was collected. Common fatigue strength reduction factors to account for such things as surface finish can be found in design handbooks.

Loading Type

The options of the Type property are described below. Their availability is based upon your analysis type.

- **Zero-Based** (r=0)
- **Fully Reversed** (r=-1)
- **Ratio**
- **History Data**
- **Non-proportional Loading** (available only for stress-life applications)

The **Zero-Based** (r=0), **Fully Reversed** (r=-1), and **Ratio** options are all constant amplitude, proportional loading types and are graphically illustrated in the Worksheet.

The **History Data** option enables you to import a file containing the data points. This option is a non-constant amplitude proportional loading type. This data is depicted in a graph on the Worksheet. You can specify the number of data points this graph will display using the **Maximum Data Points To Plot** (p. 2181) property in the **Options** category.

The **Non-proportional Loading** option is a non-proportional constant amplitude loading type for models that alternate between two different stress states (for example, between bending and torsional loading). Problems such as an alternating stress imposed on a static stress can be modeled with this feature. Non-proportional loading is only supported for **Fatigue Tool** objects used with **Solution Combination** (p. 2288) where you select exactly two stress states (only two **Base Cases** present in the **Solution Combination** Worksheet).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Scale Factor

This setting scales the load magnitude. For example, if you set this to 3, the amplitude (and mean) of a zero-based loading will be 1.5 times the stress in the body. The graph in the **Worksheet** window will update to reflect this setting. This option is useful to see the effects of different finite element loading magnitudes without having to run the complete structural analysis repeatedly. Note that this scale factor is applied after the stresses have been collapsed from a tensor into a scalar. Thus any multiaxial stress collapse methods that are sensitive to the sign (Von-Mises, Maximum Shear, Maximum Principal) may not give the same answer had the scale factor been applied to the environment load itself.

Analysis Type

Define the fatigue analysis as either **Stress Life** or **Strain Life**.

Mean Stress Theory

This setting specifies how the mean stress effects should be handled.

- If **Analysis Type** is set to **Stress Life**, choose from **None** (default), **Goodman**, **Soderberg**, **Gerber**, **ASME Elliptical**, and **Mean Stress Curves**. The **Goodman**, **Soderberg**, **Gerber**, and **ASME Elliptical** options use static material properties along with S-N data to account for any mean stress while **Mean Stress Curves** use experimental fatigue data to account for mean stress. You can specify the default setting for this property using the Mechanical application **Fatigue** settings (p. 223) in the **Options** dialog box (p. 217).
- If **Analysis Type** is set to **Strain Life**, choose from **None**, **Morrow**, and **SWT** (Smith-Watson-Topper).

Note:

A sample plot of each of these theories is shown at the bottom of the **Worksheet** view. This plot does not use live data, but is rather a generic representation of each theory. For more information on these theories, see "Metal Fatigue In Engineering" by Ralph I. Stephens, et. al.

Method Selection

This property is only available for Random Vibration analyses. The options include: Narrow Band, Steinberg (default), and Wirsching. This setting specifies the method to calculate the Probability Density Function (PDF) from the available PSD stress response.

Stress Component

Because stresses are multiaxial but experimental fatigue data is usually uniaxial, the stress must be converted from a multiaxial stress state to a uniaxial one. A value of 2 times the maximum shear stress is used. You can choose from several types, including component stresses, von Mises, and a

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

signed von Mises, which takes the sign of the absolute maximum principal stress. The signed von Mises is useful for accounting for any compressive mean stresses.

Note:

For a fully reversed case (no mean stress), the Max Principal and Abs Max Principal options will give the same result because alternating stress is always calculated using Abs Max Principal stress.

Exposure Duration

This property is only available for Random Vibration and Harmonic Response analyses. It specifies the duration (in time) the loading is applied. The resulting damage is for this entire duration. The default setting is one second (1). This means that the calculated damage is based on damage per second.

In a Harmonic Response analysis, the number of cycles is internally calculated as the Exposure Duration multiplied by the chosen frequency.

Frequency Selection

This property is only available for Harmonic Response analyses. You use this property to select a analysis. Options include:

- **Single Frequency** (default): This option specifies that only the stress at the defined Frequency is considered for the fatigue analysis.
- **Multiple Frequencies**: This option enables you to manually specify frequency values and their corresponding **Exposure Durations** in the **Tabular Data** window. Stresses at each of these distinct frequencies are considered for the fatigue analysis and fatigue damage is accumulated.
- **Sine Sweep**: This option enables you to manually specify frequency values and their corresponding **Scale Factors** in the **Tabular Data** window. A **Sine Sweep** analysis is performed from the lowest to highest frequency value in steps of 0.5 Hz using the **Sweep Rate** and the appropriate amplitude scaling at each step. The fatigue damage is accumulated.

Frequency

This property is only available for Harmonic Response analysis. It specifies the frequency (in Hz) for which the stress response is calculated and used for the fatigue analysis. The phase angle for which the maximum stress occurs is automatically chosen for a chosen frequency.

Sweep Rate

The Sweep Rate property is only available for Harmonic Response analysis. It is only available when the Sine Sweep option is selected for the Frequency Selection property. This property specifies the rate of frequency sweep in Hz/s units.

Units Name

This property enables you to specify the name for the **Life Units**. The unit options include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **cycles**
- **blocks**
- **seconds**
- **minutes**
- **hours**
- **days**
- **months**
- **User Defined**

User Defined

Selecting the **User Defined** option displays the **Custom Units Name** field. Enter the name for your customized unit name in this field. The specified unit is reflected in the Details view for all applicable fatigue settings.

1 "Unit" is Equal To

Where "unit" is either cycle or block based on the **Units Name** selection. Modify the field's value based on the desired number of cycles or blocks for the units.

Bin Size

This option appears only if **Type** is set to **History Data** (non-constant amplitude loading). This setting defines how many divisions the cycle counting history should be organized into for the history data loading type. Strictly speaking, this is number specifies the dimensions of the rainflow matrix. A larger bin size has greater precision but will take longer to solve and use more memory.

Use Quick Rainflow Counting

This option appears only if **Type** is set to **History Data** (non-constant amplitude loading). Since rainflow counting is used, using a "quick counting" technique substantially reduces runtime and memory, especially for long time histories. In quick counting, alternating and mean stresses are sorted into bins before partial damage is calculated. This means that with quick counting active, calculations will be performed for maximum of binsize. Thus the accuracy will be dictated by the number of bins. Without quick counting, the data is not sorted into bins until after partial damages are found and thus the number of bins will not affect the results. The accuracy of quick counting is usually very good if a proper number of bins are used when counting. To see the effects of using quick counting, compare the results of constant amplitude loading to simulated constant amplitude loading from a load history file. With quick counting off, the result should match exactly but with quick counting on, there will be some error depending on the bin size and alternating stress value in relation to the midpoint of the bin the count is sorted into.

Infinite Life

Stress Life Analysis

This option appears only if **Type** is set to **History Data** (non-constant amplitude loading) and defines what life will be used if the stress amplitude is lower than the lowest stress on the SN curve. It may be important in how damaging small stress amplitudes from the rainflow matrix are.

Strain Life Analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Since the strain-life method is equation based it has no built-in limit, unlike stress-life for which the **Fatigue Tool** uses a maximum life equal to the last point on the SN curve. Thus to avoid skewed contour plots showing very high lives, you can specify **Infinite Life** in a strain-life analysis. For example, if you set a value of 1e9 cycles as the **Infinite Life**, the maximum life reported is 1e9.

Maximum Data Points To Plot

This option is only applicable for **History Data** loading and allows you to specify the number of data points to display in the corresponding graph that appears in the **Worksheet**. The default value is 5000 points. The graph displays the full range of points and all points are used in the analysis. However, depending on the value you set, every second or third point may not be displayed in the interest of avoiding clutter and making the graph more readable.

Fatigue Results

Once you insert a Fatigue Tool object into your analysis, you select from the result options listed below. By default, the results are scoped to **All Bodies**. However, you can modify scoping and apply it to individual parts or faces as desired. If you are working with a shell model, note that the application always specifies the top face value.

The result options are provided on the **Fatigue Tool** Context menu or by right-clicking on the object and selecting **Insert** > **[desired result]**.

Note:

The application does not support the Fatigue Tool when you have symmetry specified in the analysis.

Contour Results	Graph Results
• Life (p. 2181) • Damage (p. 2182) • Safety Factor (p. 2182) • Biaxiality Indication (p. 2182) • Equivalent Alternating Stress (p. 2183)	• Rainflow Matrix (History Data only) (p. 2183) • Damage Matrix (History Data only) (p. 2184) • Fatigue Sensitivity (p. 2185) • Hysteresis (p. 2186)

Life

This result contour plot shows the available life for the given fatigue analysis. If loading is of constant amplitude, this represents the number of cycles until the part will fail due to fatigue. If loading is non-constant, this represents the number of loading blocks until failure. Thus if the given load history represents one month of loading and the life was found to be 120, the expected model life would be 120 months.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

During a constant amplitude analysis, if the alternating stress is lower than the lowest alternating stress defined in the S-N curve table, the application uses the Life result for that point. In addition, for this case, the application imposes a limit of 1e06 cycles if the Linear S-N curve definition is used.

Damage

Fatigue damage is defined as the design life divided by the available life. The default design life may be set through the **Options** dialog box (p. 217). A damage of greater than 1 indicates the part will fail from fatigue before the design life is reached.

Note:

When you set the **Type** property to **History Data Loading**, and specify a **Damage** result, the damage per stress cycle is the reciprocal value of the **Infinite Life** property value if the stress cycle is below the Endurance limit. As a result, loading histories that have numerous stress cycles below the endurance limit may exhibit a greater accumulated damage value than expected. You can mitigate this effect by setting a high **Infinite Life** value.

Safety Factor

This result is a contour plot of the factor of safety (FS) with respect to a fatigue failure at a given design life. The maximum FS reported is 15.

These are the steps at each node to calculate **Safety Factor**:

1. Calculate the alternating and mean stress tensor.
2. Collapse alternating and mean stress from tensor to scalar using selected stress component.
3. Calculate **Safety Factor** from the mean stress equation using S_{eqv} as queried from the SN curve for the design life.

   $1/FS = S_{alt}/S_{eqv} + S_{Mean}/S_{Ultimate}$

Biaxiality Indication

This result is a stress biaxiality contour plot over the model that gives a qualitative measure of the stress state throughout the body. Note the following biaxiality values:

- **-1** = Pure shear
- **0** = Uniaxial stress
- **1** = Pure biaxial state

For non-proportional loading, specify the desired biaxiality using the **Method** property. Options include:

- **Average**: The application will sum the biaxiality for both single load cases and divide by 2.
- **Standard Deviation**: This calculation is the difference between single load case biaxiality and Average biaxiality. Therefore, a small Standard Deviation indicates behavior that is close to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

proportional loading whereas a large Standard Deviation indicates a significant change in the principal stress directions.

Equivalent Alternating Stress

The equivalent alternating stress is the stress we use to query the SN curve after accounting for multiaxial loading and mean stress effects. This result is not valid if the loading has non-constant amplitude, (that is, **Loading Type** = **History Data**). The result is useful for cases where the design criterion is based on the equivalent alternating stress that you specify.

These are the steps at each node to calculate **Equivalent Alternating Stress**:

1. Calculate the alternating and mean stress tensor.
2. Collapse alternating and mean stress from tensor to scalar using selected stress component.
3. Calculate the **Equivalent Alternating Stress** using the desired empirical stress theory, as specified by the **Mean Stress Theory** property of the **Fatigue Tool** object. For example, when you set the **Mean Stress Theory** property to **Goodman**, the **Equivalent Alternating Stress** calculation becomes:

   $EqvAltStress = S_{alt}/(1 - S_{Mean}/S_{Ultimate})$

Therefore, that is the value reported as Equivalent Alternating Stress and this is used to query Fatigue Life from the SN Curve.

Important:

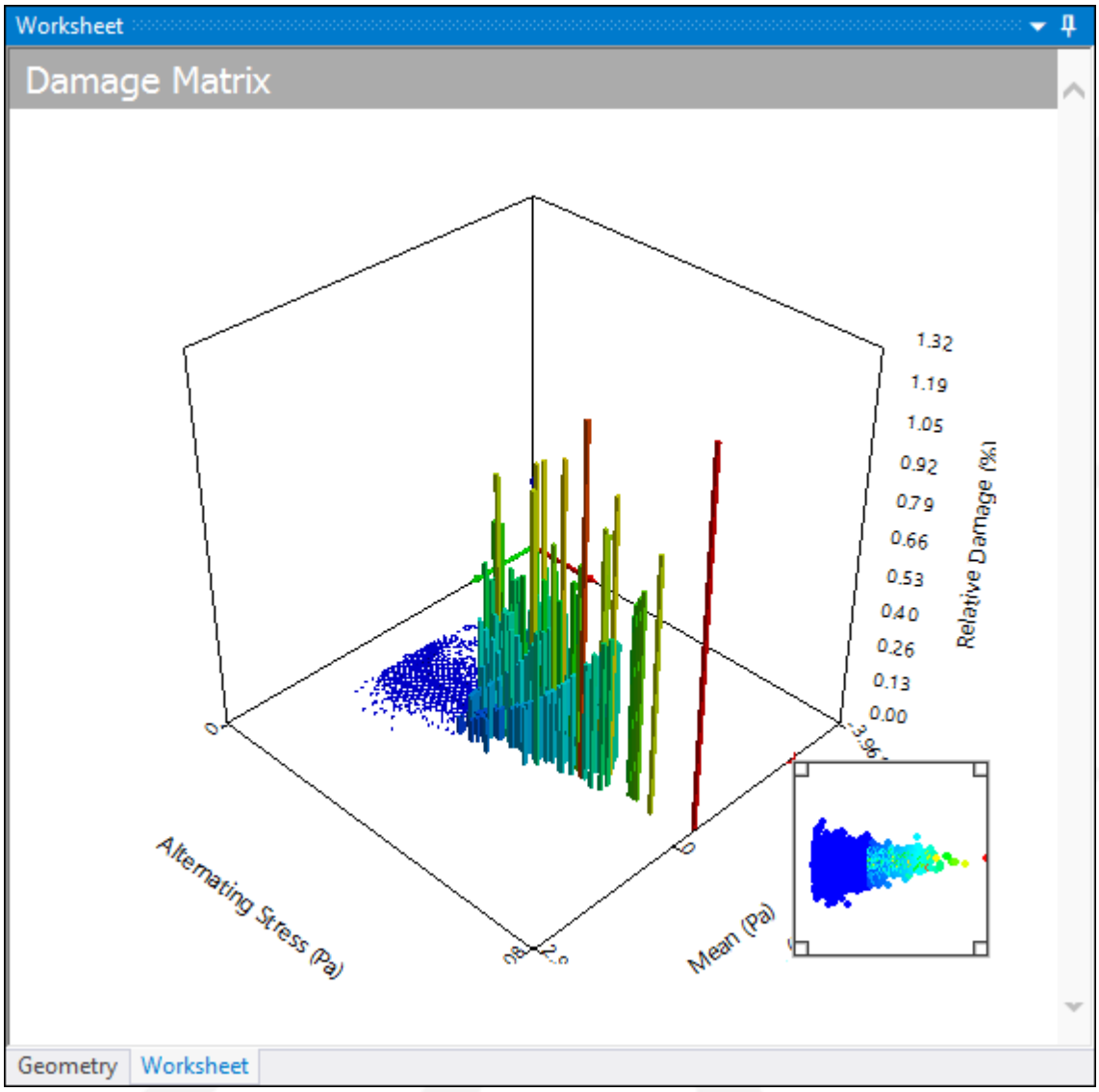
If you specify a **Mean Stress Theory** and static failure is predicted, the reported equivalent alternating stress is reported as 1e32 Pa (this value is always reported when there is static failure).

Rainflow Matrix (History Data only)

This graph depicts how many cycle counts each bin contains. This is reported at the point in the specified scope with the greatest damage.

The Navigational Control at the bottom right-hand corner of the graph can be used to zoom and pan the graph. You can use the double-sided arrow at any corner of the control to zoom in or out. When you place the mouse in the center of the Navigational Control, you can drag the four-sided arrow to move the chart points within the chart.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



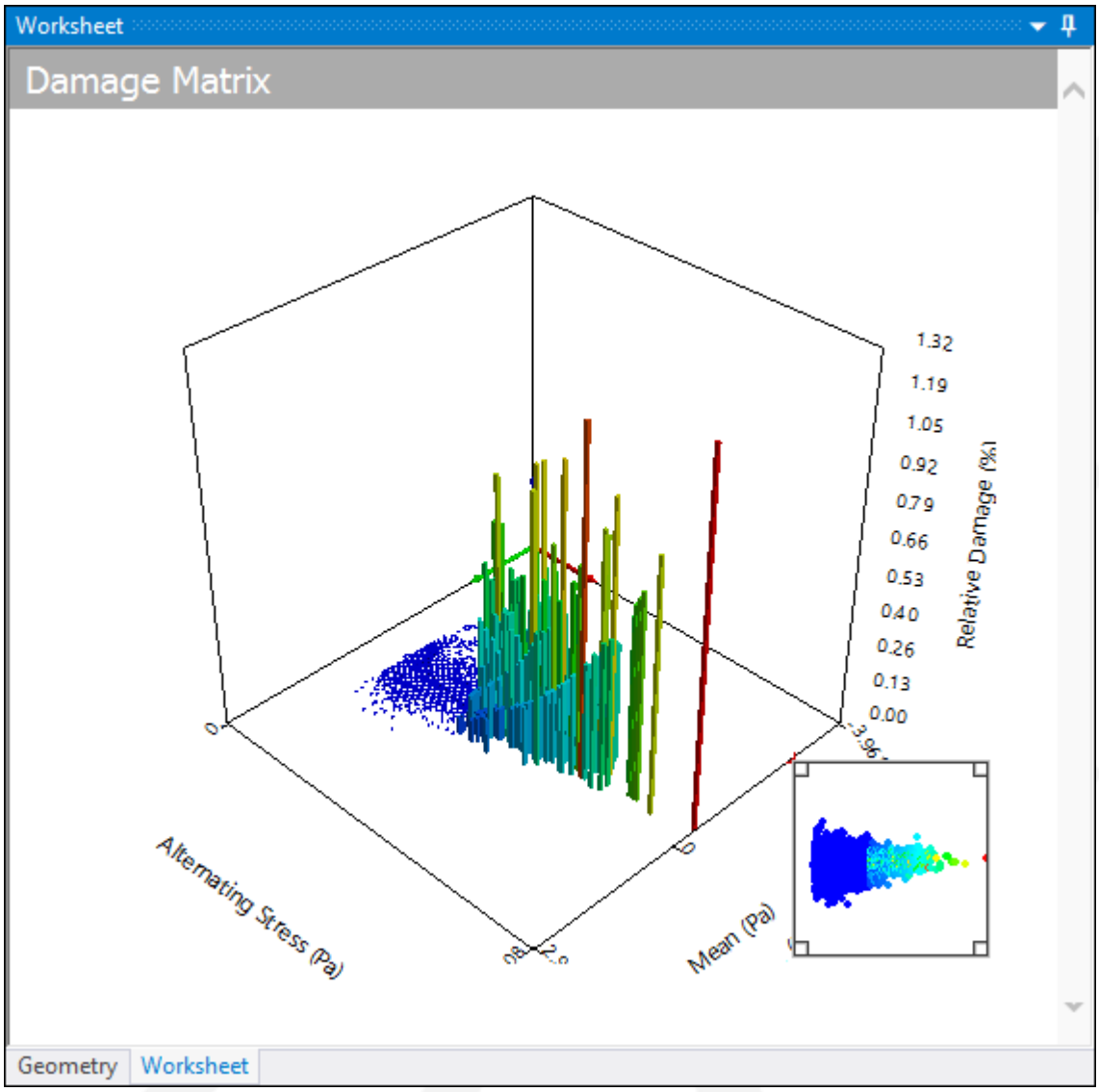
Damage Matrix (History Data only)

Similar to the rainflow matrix, this graph depicts how much relative damage each bin has caused. This result can give you information related to the accumulation of the total damage (such as if the damage occurred though many small stress reversals or several large ones).

The Navigational Control at the bottom right hand corner of the graph can be used to zoom and pan the graph. You can use the double-sided arrow at any corner of the control to zoom in or out. When you place the mouse in the center of the Navigational Control, you can drag the four-sided arrow to move the chart points within the chart.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



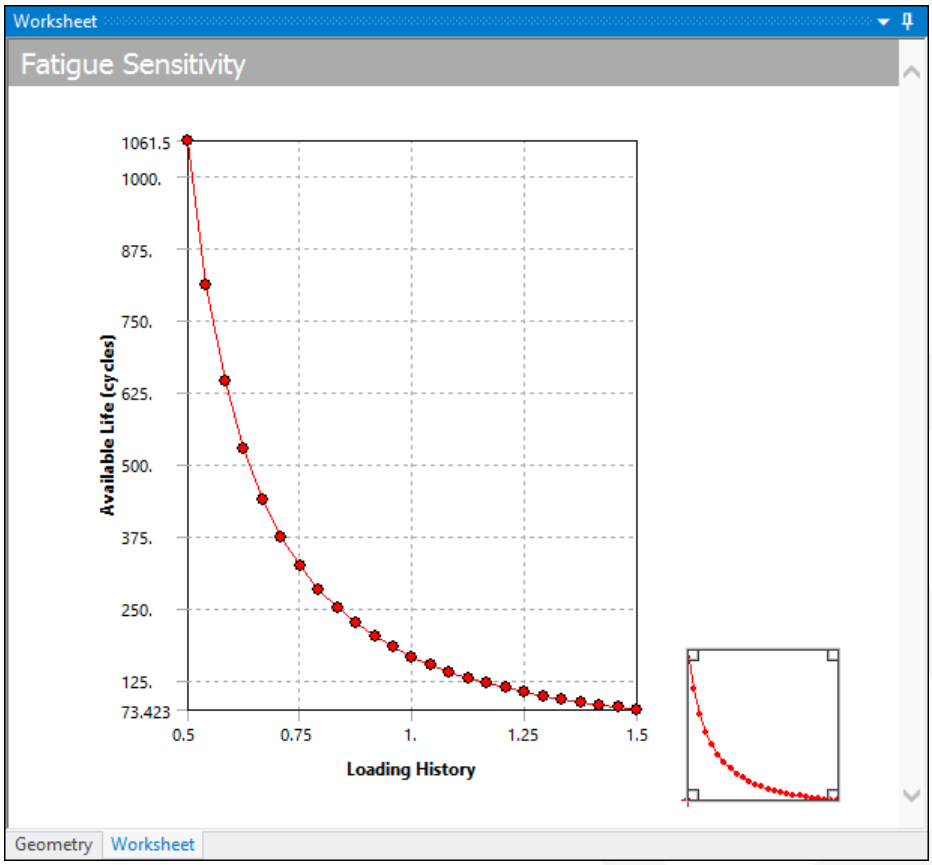
Fatigue Sensitivity

This plot shows how the fatigue results change as a function of the loading at the critical location on the scoped region. Sensitivity may be found for life, damage, or factory of safety. For instance, if you set the lower and upper fatigue sensitivity limits to 50% and 150% respectively, and your scale factor to 3, this result will plot the data points along a scale ranging from a 1.5 to a 4.5 scale factor. You can specify the number of fill points in the curve, as well as choose from several chart viewing options (such as linear or log-log).

The Navigational Control at the bottom right hand corner of the graph can be used to zoom and pan the graph. You can use the double-sided arrow at any corner of the control to zoom in or out. When you place the mouse in the center of the Navigational Control, you can drag the four-sided arrow to move the chart points within the chart.

To specify a result item, you must be under a **Solution** object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



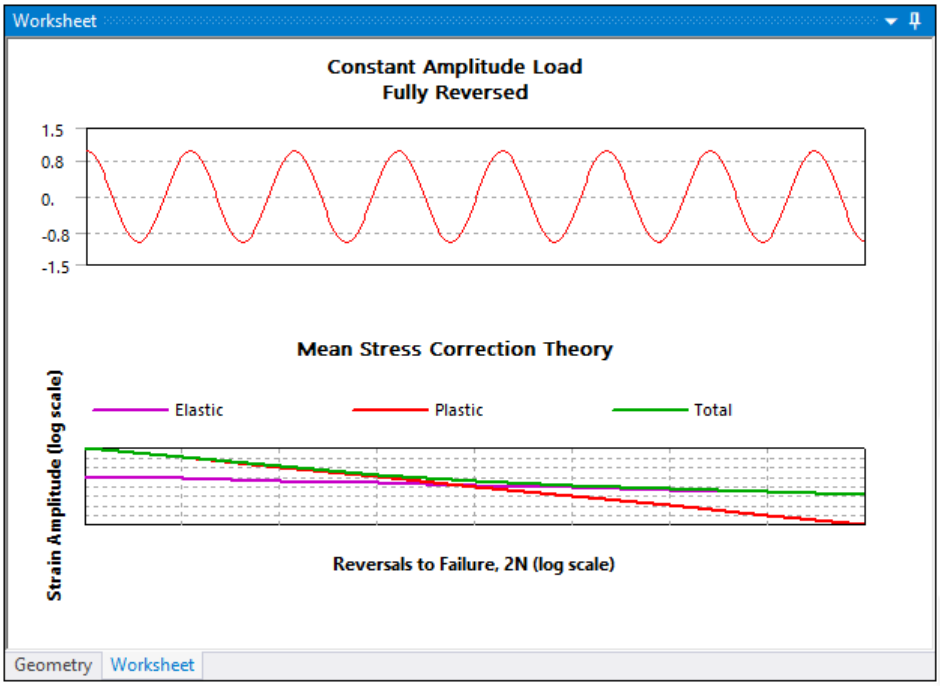
Hysteresis

In a strain-life fatigue analysis, although the finite element response may be linear, the local elastic/plastic response may not be linear. The Neuber correction is used to determine the local elastic/plastic response given a linear elastic input. Repeated loading will form close hysteresis loops as a result of this nonlinear local response. In a constant amplitude analysis a single hysteresis loop is created although numerous loops may be created via rainflow counting in a non-constant amplitude analysis. The **Hysteresis** result plots the local elastic-plastic response at the critical location of the scoped result (the **Hysteresis** result can be scoped, similar to all result items). **Hysteresis** is a good result to help you understand the true local response that may not be easy to infer. Notice in the example below, that although the loading/elastic result is tensile, the local response does venture into the compressive region.

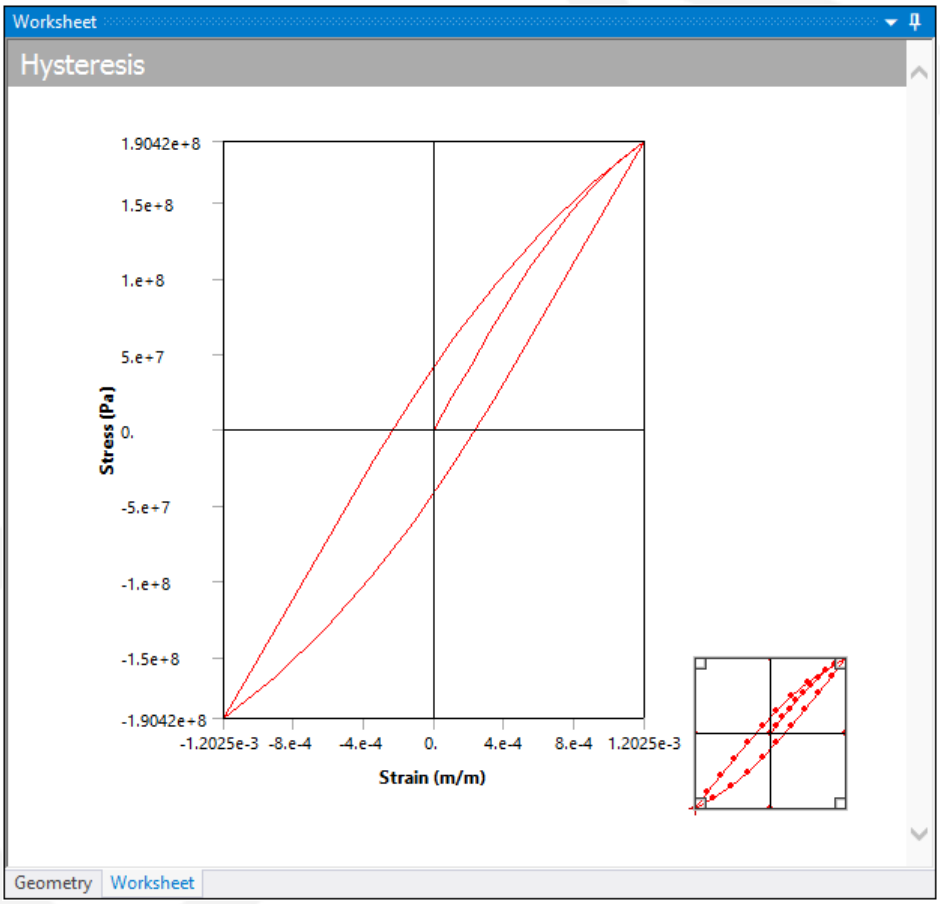
Loading/Elastic Response

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Corresponding Local Elastic Plastic Response at Critical Location



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Fatigue Combination

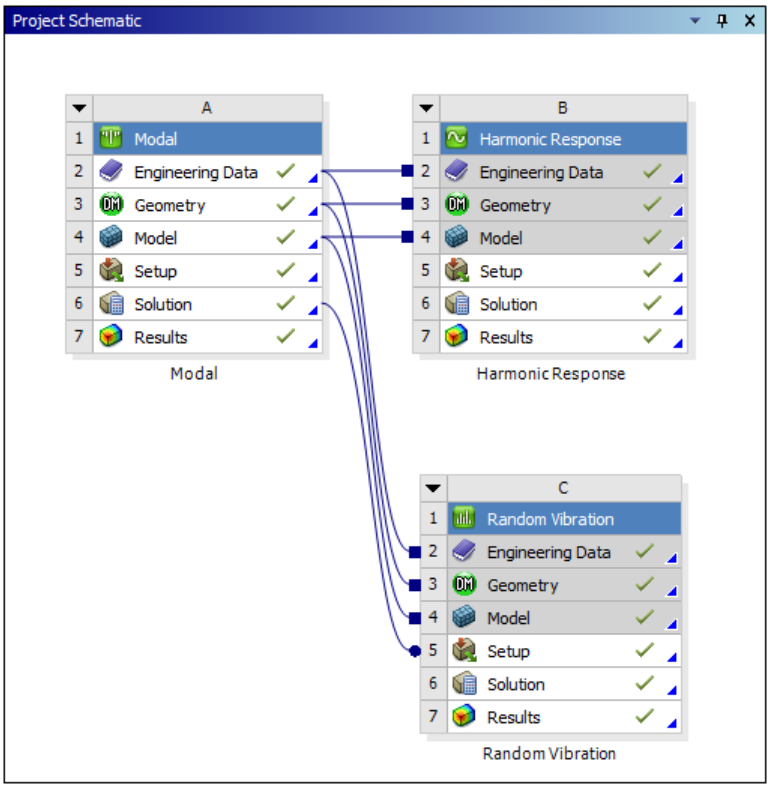
When you are running multiple analyses that are using the **Fatigue Tool** and **Damage** results, you can use the Fatigue Combination object to combine or sum-up the damage results, generated by individual Fatigue Tools , from each analysis. The feature enable you to create a sum total of the damage results from multiple systems.

Important:

If you have imported a legacy database (wbpj or wbpz), you must first clear the data of any existing Fatigue Tools, and re-evaluate the results, in order to then use the **Fatigue Combination** feature.

Workflow

From the Workbench **Project Schematic**, configure your analysis. The example illustrated here includes a combination of Modal, Harmonic Response, and Random Vibration analyses.



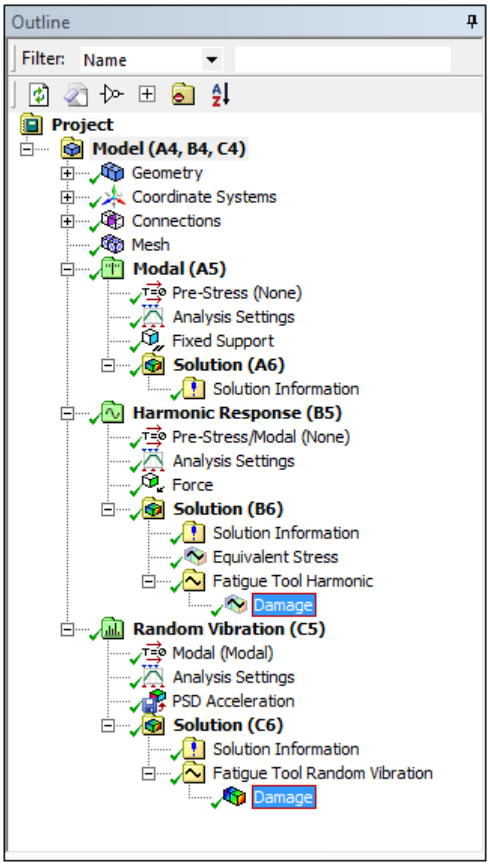
For your target analyses, insert **Fatigue Tools** and **Damage** results.

Important:

For the **Fatigue Combination** feature to work correctly, the first **Damage** result in each Environment must be scoped to **All Bodies**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

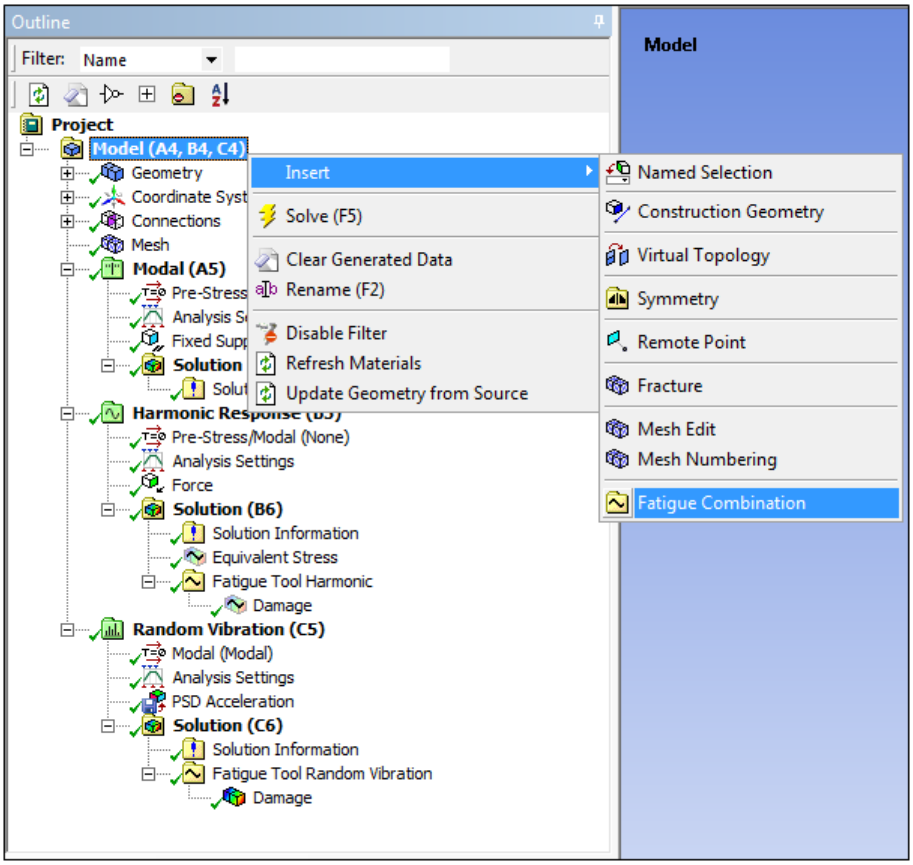
Published: 2023-10-23



Select the Model object, right-click the mouse and select **Insert** > **Fatigue Combination**.

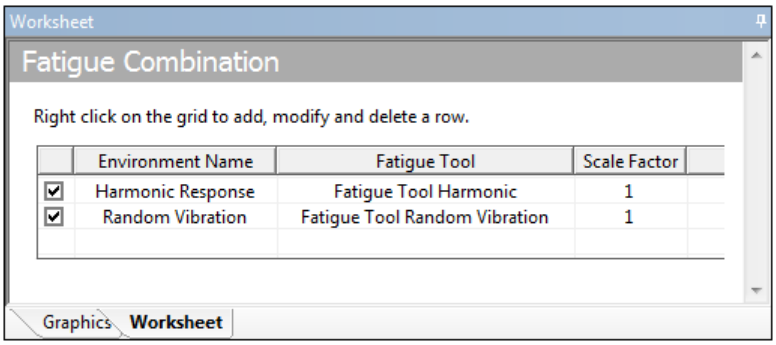
DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



As illustrated below, when you select the **Fatigue Combination** object, the **Worksheet** displays. The **Worksheet** enables you to add rows and specify, from a drop-down menu, the desired Environment, **Fatigue Tool**, and you can specify a **Scale Factor** of the Damage results that you wish to sum. The default **Scale Factor** is **1**.

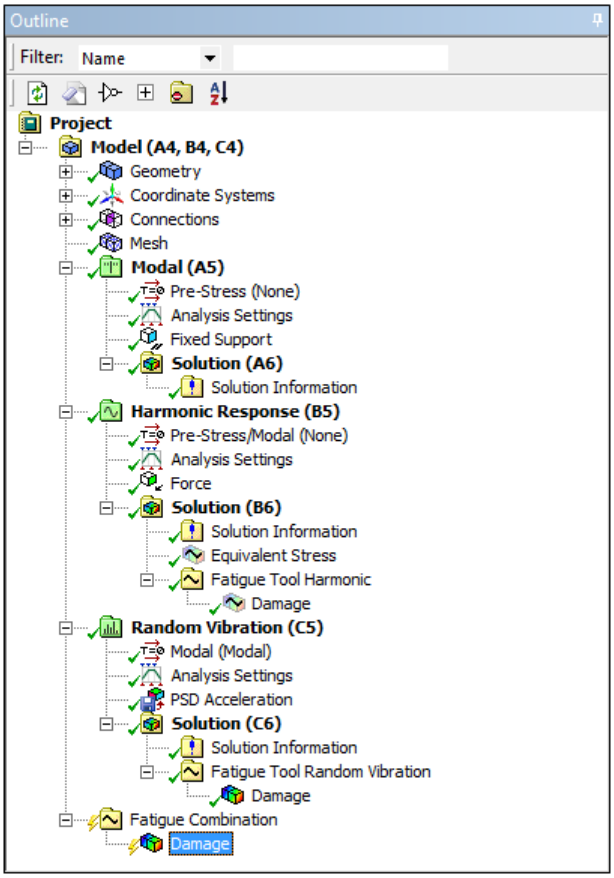
No **Details** view categories or properties exist for this object. See the Fatigue Tool object reference page for the Details associated with **Damage** results.



Once the **Worksheet** is complete, insert a **Damage** result to the **Fatigue Combination** object using the **Fatigue Combination** option or by right-clicking the mouse on the **Fatigue Combination** object or in the **Geometry** window and select **Insert** > **Damage**. As needed, specify available **Damage** object properties. Once complete, evaluate the result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Mechanical Embedded DesignLife UI

The **Mechanical Embedded DesignLife UI** is an add-on to the Ansys Mechanical User Interface that makes the fatigue analysis capabilities of the nCode DesignLife product available from within the Mechanical environment. The nCode DesignLife product is a partner product from HBM Prenscia.

For more information see DesignLife Add-on in the *Mechanical Add-ons Guide*.

Noise, Vibration, and Harshness

NVH is an add-on to the Ansys Mechanical User Interface that enables you to preprocess/postprocess Noise, Vibration, and Harshness (NVH) results:

- It streamlines Modal Assurance Criterion (MAC) calculations through FEA-Experimental model comparison.
- It is able to reconstruct elastic stress/strain fields from modal superposition/harmonic analyses or tests.
- It allows computation of the Frequency Response Function (FRF) in terms of Compliance, Mobility or Accelerance.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- It offers a pre-test environment that allows identification of optimum sensor and exciter locations for vibration tests.

Note:

The NVH Add-on is only available on the Windows operating system.

For more information about this add-on, see NVH Toolkit Add-on in the *Mechanical Add-ons Guide*.

Forced Response

The **Forced Response** application is an add-on to the Ansys Mechanical user interface that enables you to use cyclic modeling to investigate the aeromechanics and reliability of turbomachinery blade rows.

For more information about this add-on, see Forced Response Add-on in the *Mechanical Add-ons Guide*.

User Defined Results

This section examines the purpose, operation, and use of the **User Defined Result** feature of Mechanical.

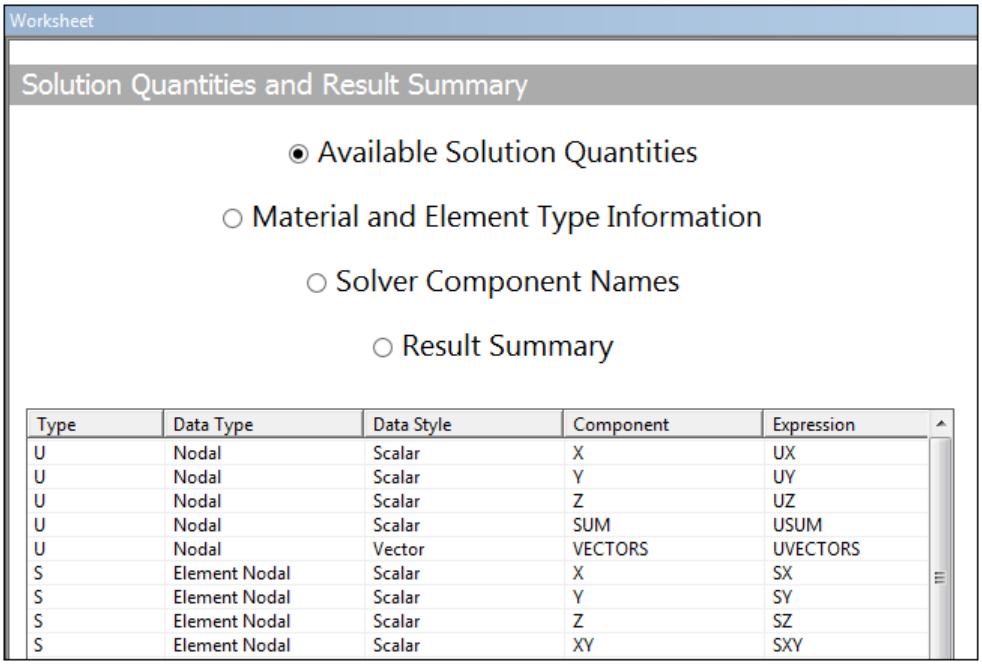
Overview
Characteristics
Application
Node-Based Scoping
User Defined Result Expressions
Unit Description
User Defined Results for the Mechanical APDL Solver
User Defined Results for the Ansys Rigid Dynamics Solver

Overview

Based on the settings of the **Output Controls** (p. 1473), the application writes result quantities, such as stresses and strains, to the result file during the solution process. These quantities are displayed in the Solution Quantities and Result Summary Worksheet (p. 1961) (illustrated below). The **User Defined Result** feature enables you to create result objects for these quantities or use these quantities to derive more complex Expressions (p. 2197) involving them, such as adding and summing result values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Type	Data Type	Data Style	Component	Expression
U	Nodal	Scalar	X	UX
U	Nodal	Scalar	Y	UY
U	Nodal	Scalar	Z	UZ
U	Nodal	Scalar	SUM	USUM
U	Nodal	Vector	VECTORS	UVECTORS
S	Element Nodal	Scalar	X	SX
S	Element Nodal	Scalar	Y	SY
S	Element Nodal	Scalar	Z	SZ
S	Element Nodal	Scalar	XY	SXY

Refer to the following sections for descriptions of user defined result entries in the worksheet:

- User Defined Results for the Mechanical APDL Solver (p. 2203)
- User Defined Results for Explicit Dynamics Analyses

Characteristics

General:

- All analysis types and solver targets can produce User Defined Results. A User Defined Result *may* be unique to a particular solver and analysis. After clicking on the **Solution** object, you must click the **Worksheet** to produce the complete listing of the results that are applicable to the analysis type and solver being used.
- All result types can be combined except for results which have different dimensions. For example, displacement vectors, which contain 3 items, cannot be added to stress tensors, which contain 6 items.
- User Defined Results which are elemental (such as stress or strain results) can be displayed as averaged or unaveraged results (p. 1981). It takes Mechanical longer to display a result which is not averaged.

Like most result types that display using contours, user defined results:

- Are scoped to a geometry (vertex, edge, face, body), named selection (geometry, node-based, or element-based), path, or surface. However, you cannot scope user defined results based on Contacts to a path or surface.
- Require a set, time, and frequency/phase, to be fully specified (depending on the analysis type).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Requires the selection of **Response Type** (Displacement/Velocity/Acceleration/Other Result) option for Random Vibration and Response Spectrum analysis types.
- In a Random Vibration Analysis, you need to select a **Scale Factor** to get a result corresponding to the **Probability** (read-only).
- Display minimum/maximum values and a **Graph**.
- Display nodal averaged data.
- Can be added to a Chart (p. 1929)
- Can be examined using probe annotations, slice planes, isosurface, etc.
- Can be cleared.
- Can be duplicated.

Unlike other contour results, user defined results:

- Can have a variable unit category assigned to its contour.
- Become obsolete if a user defined result is dependent upon another user defined result that has been modified, cleared, or deleted. In this instance, the graphic of the geometry displays without results.
- User defined results cannot employ **Probes**.

Application

User defined results enable you to define expressions using mathematical operators. For example, you can enter the mathematical combination **UX+UY** in the **Expression** (p. 2197) field and then retrieve a new result.

To apply a user defined result, first insert a **User Defined** result object (p. 2194) then specify its properties (p. 2195). Make sure that you review the limitations (p. 2196) topic as well as the file format requirements (p. 2196) topic if you plan to import a result text file.

Insert Object

Use one of the following methods to insert the object.

- Select the **User Defined Result** option on the Solution Context tab.
- Right-click the Solution object and the select the **Insert** > **User Defined Result** option.
- After the solution is complete and with the Solution object selected, select the Worksheet option from the Home (p. 40) tab to display the Solution Quantities and Result Summary Worksheet (p. 1961). Right-click on a desired result (row) in the worksheet and select **Create User Defined Result** (see figure below).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Type	Data Type	Data Style	Component	Expression
U	Nodal	Scalar	X	UX
U	Create User Defined Result	Scalar	Y	UY
U		Scalar	Z	UZ
U	Nodal	Scalar	SUM	USUM
U	Nodal	Vector	VECTORS	UVECTORS
S	Element Nodal	Scalar	X	SX

The application automatically names the object and populates an expression when you use the worksheet to create a user defined result, while the other two insertion methods above require you to manually define the result. Until you become familiar with this feature, it is recommended that you insert user defined results using the worksheet to ensure results are valid and applicable for the particular analysis type and solver used in your simulation.

Note:

NMISCxxx and SMISCxxx results are not displayed in the worksheet and can only be accessed by typing in the keyword directly. See User Defined Results for the Mechanical APDL Solver (p. 2203) for details.

Specify Properties

Once you insert a **User Defined Result** object, specify the properties.

Manually Define the Result	Automatic Result Generation via Worksheet
1. If you create a user defined result manually, you need to first specify the Source property. Options include **Expression** and **Imported File**, as illustrated below.	When you create the result using the Worksheet, the application automatically names the object and populates the **Expression** (p. 2197) property. Except for an **Identifier** (p. 1978) property, all remaining details are also automatically generated based on the information provided by the result type, such as **Input Unit System** (Metric) and **Output Unit** (Displacement) as shown in the example below.

Details of "User Defined Result"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Source	Expression
Expression	Expression / Imported File
Input Unit System	
Output Unit	
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Results	
Information	

<table>
<tr><th>Manually Define the Result</th><th>Automatic Result Generation via Worksheet</th></tr>
<tr><td>
2. Are you defining an expression or importing a result file?
<table>
<tr><th>Expression?</th><th>Imported File?</th></tr>
<tr><td>Enter your desired result expression.</td><td>Select the Imported File option. This option displays the Imported File Name property. Use this property to select and import the desired file.</td></tr>
</table>
Important:
If you're using an imported file, be sure to review the limitations (p. 2196) and file format requirements (p. 2196) below.
3. Select a desired Output Unit.
4. Evaluate your results.
Note:
See the User Defined Result Expressions (p. 2197) and Unit Description (p. 2202) sections for more information.
</td><td></td></tr>
</table>

Limitations

Note the following when specifying an imported file:

- The data must be based on the same mesh as the current mesh in Mechanical. This feature is currently only supported for solid meshes.
- The file must be specified using a supported format (see below).
- The result displays basic contours only. The result does not support animation, time history, deformations, etc.

File Format Requirements

It is your responsibility to make sure that your result file conforms to the following formats.

Format 1: Exported from Mechanical

```
<No header>
| Node/Elem Number | Result Value |
1, 42.5
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
2, 65.8
…
```

See the Exporting Results (p. 2276) section.

Format 2: Exported from AM Solve

```
AM Result File
ANSYS 2022 R1         BUILD 22.1BETA    UP20210811    WINDOWS x64
MM/DD/YYYY      00:00:00
Title: Example--Static Structural (A5)
| Node/Element Number | X Coordinate | Y Coordinate | Z Coordinate | Result Value |
1, 42.5
2, 65.8
…
```

Format 3: Exported from LS-DYNA PrepPost

```
*KEYWORD
$SOLID_ELEMENT_RESULTS
1, 42.5
2, 65.8
…
…
/END
```

Node-Based Scoping

In regard to usage, suppose two user defined results (with identifiers A and B, respectively) are scoped to ScopeA and ScopeB. The algorithm to draw the contours for C = A + B (scoped to ScopeC) proceeds as follows:

- The results A and B are combined on all common bodies (determined from ScopeA and ScopeB and referred to as CommonBodies).
- The scope (ScopeC) of the newly defined result C is then employed: the contours of C are drawn on the intersection of ScopeC and CommonBodies.

Note, each of ScopeA, ScopeB, and ScopeC can be any set of geometric entities: vertices, edges, faces, bodies, or named selections (consisting of geometric entities or even nodes in the mesh).

Example 9: Nodal Scoping

Assumptions: A is scoped to bodies 1 and 2 and B is scoped to two faces, one in body 2 and one in body 3. The combination C = A+B is scoped to two vertices, one in body 2, and the other in body 3.

Result: A+B will be computed on nodes common to the underlying bodies of A and B; these nodes will exist only in body 2. Then the combination C = A + B will be displayed only on the vertex belonging to body 2 (the one belonging to body 3 is not in the intersection of the two original scoping bodies).

User Defined Result Expressions

The term "expression" has more than one use when defining user defined results. An expression is:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Primarily, the combination of mathematical values, based on syntax rules (p. 2200) and the available math operations (p. 2200).
- A column displayed on the **Solution Worksheet** that indicates the result type.
- An entry field in the Details view of a user defined result where you enter mathematical values, such as **UX+UY+UZ**.

The example of the **Solution Worksheet** shown below highlights the **Expression** column.

Type	Data Type	Data Style	Component	Expression	Output Unit
U	Nodal	Scalar	X	UX	Displacement
U	Nodal	Scalar	Y	UY	Displacement
U	Nodal	Scalar	Z	UZ	Displacement
U	Nodal	Scalar	SUM	USUM	Displacement
U	Nodal	Vector	VECTORS	UVECTORS	Displacement
S	Element Nodal	Scalar	X	SX	Stress
S	Element Nodal	Scalar	Y	SY	Stress
S	Element Nodal	Scalar	Z	SZ	Stress
S	Element Nodal	Scalar	XY	SXY	Stress

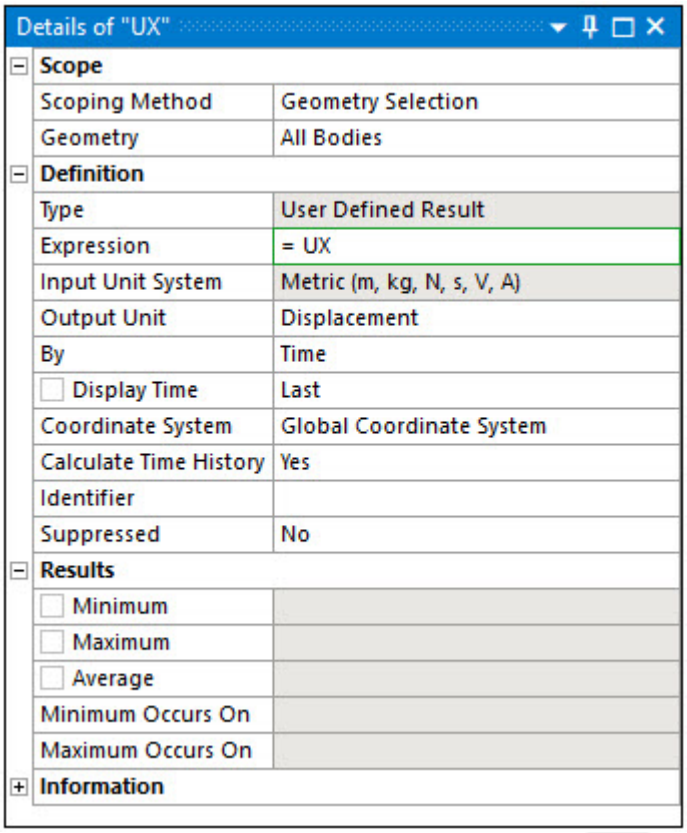
When a **User Defined Result** is applied (p. 2194), the content of the above column populates the **Expression** field of the user defined result's Detail View. In this example, **UX**.

Note:

You can manually enter an expression using the **Expression** property. An expression must include a variable and not only a combination of constants. For example, the expressions "3" or "3+5"are not supported. However, "3 + UX" or "6 + VOLUME" are accepted expressions. This expression result is evaluated for each node or element, depending upon the variable you use.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

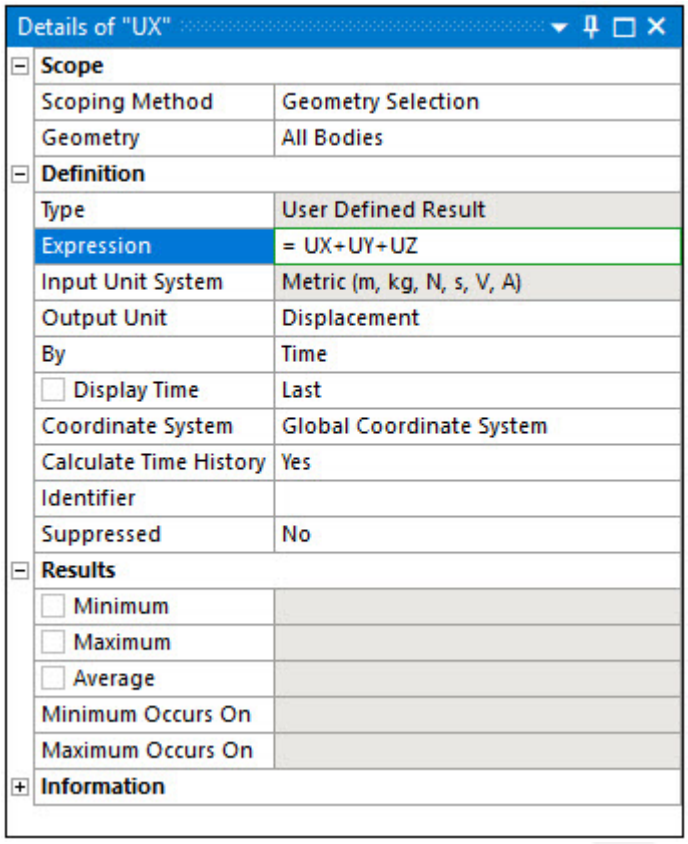
Published: 2023-10-23



Details of "UX"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	User Defined Result
Expression	= UX
Input Unit System	Metric (m, kg, N, s, V, A)
Output Unit	Displacement
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Minimum	
Maximum	
Average	
Minimum Occurs On	
Maximum Occurs On	
Information	

The content of the **Expression** field can be modified using mathematical operators to further define the expression. As shown below, you can combine the X, Y, and Z components and then retrieve a new customized result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Details of "UX"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	User Defined Result
Expression	= UX+UY+UZ
Input Unit System	Metric (m, kg, N, s, V, A)
Output Unit	Displacement
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Minimum	
Maximum	
Average	
Minimum Occurs On	
Maximum Occurs On	
Information	

Expression Syntax

Expressions support the following syntax:

- **Operands**: ('+', '-', '*', '/', '^', '**')
- **Functions**: (sqrt(), min()…) - *always use lower case*
- **Numbers**: (scalar quantities such as 1.0, 25, -314.23, or 2.5e12)
- **Identifiers** (p. 1978): unique user defined names

Supported Mathematical Operations

The following is a list of the mathematical operations currently supported for user defined results. The shorthand notation "s" defines a single-valued quantity (constant values such as 1.34) and "a" defines an array. An array is distinguished by its dimension which includes the length, based on the number of rows (that is, number of nodes or elements), and the width, consisting of 1, 3, or 6 columns depending on the type of result stored.

- Addition (+): s1+s2, a1+a2, a+s (s+a is not supported)
- Subtraction (-): s1-s2, a1-a2, a-s
- Multiplication (*): s1*s2, a1*a2, a*s, s*a
- Division (/): s1/s2, a1/a2, a/s

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Power of (^ or **): s1^s2, a1^a2, a^s, s^a, (undefined if s1 = 0 and s2 < 0 *or* if s1<0 and s2 is a non-integer). The characters "**" perform the same action as the caret symbol (^) and you can use them interchangeably.
- Log base ten (log10): log10(s), log10(a), (s and a > 0.0)
- Square root (sqrt): sqrt(s), sqrt(a), (s and a should be >= 0.0)
- Dot product (dot): dot(a1,a2) (results in a single-column array consisting of the inner products, one for each row of a1 and a2; therefore, a1, a2 should have the same dimensions)
- Cross product (cross): cross(a1,a2) (a1, a2 must have 3 columns)
- Add Comp (addcomp): If the argument, such as "uvectors," has 3 columns, they are added to produce a single-column array. If the argument is a single-column array, the result will be a scalar summing all the array entries. This operation returns a variable with a different dimension than its argument , such as addcomp (UVECTORS). This particular expression is evaluated as a scalar (one value) on a per node/element contour result. However, as another example, such as addcomp (VOLUME) evaluates to a singular number and this is not a valid value to be displayed as a contour result. To make this case valid, you can convert to an expression containing valid variables instead, for example: **`addcomp(VOLUME) + VOLUME - VOLUME`**.

  This expression evaluates to a constant result value for all of the elements of the body to which it is scoped.
- Maximum (max): s = max(s1,s2), a = max(a1,a2)
- Minimum (min): s = min(s1,s2), a = min(a1,a2)
- Absolute Value (abs): s = abs(s1), a=abs(a1)
- Trigonometric Functions (sin, cos, tan): sin(s), cos(s), tan(s), sin(a), cos(a), tan(a) (s and a are both in radians)
- Inverse Trigonometric Functions (asin, acos, atan): asin(s), acos(s), atan(s), asin(a), acos(a), atan(a) (return values are in radians; where -1 <= s <= 1 and -1 <= a <=1 for asin and acos)
- atan2: atan2(s1,s2), atan2(a1,a2) (return values are in radians; calculates the arctangent of s1/s2 or a1/a2 and uses the sign of the arguments to determine the quadrant of the returned angle)

Note:

- A comma "," is used as the separator, if a period "." is used the decimal sign.

  and...

  A semi-colon ";" is used as the separator, if a comma "," is used the decimal sign.
- The current expression list does not allow input parameters from the Parameter Workspace. Only output parameters are allowed for Min and Max values of a user defined result.
- All operations involving two vector arrays must have the same dimensionality.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Any result whose expression contains the addcomp function must be scoped to exactly one body.

Unit Description

The units of a user defined result are defined by the following Detail view settings:

- **Input Unit System**: A read-only field that displays the active Mechanical application unit system. To evaluate an expression, a user defined result's units must be converted to the Input Unit System. As a result, the expression is most easily verified when the intervening data is viewed in the Input Unit System.
- **Output Unit**: The physical dimension assigned to a user defined result. It determines which factors are used to convert the result from its **Input Unit System** to the current unit system selection. A large list of unit types is available. Because you can assign such a wide variety of unit types to a result, Ansys recommends that you choose carefully and be consistent.

Units are defined in a two step process.

1. Before you evaluate an expression, the units are converted to the **Input Unit System**.
2. Once evaluated, values are converted from the input system to the active Mechanical application unit system using the appropriate factor.

For example, given the following user defined result expressions with MKS (m, kg, N, °C, s, V, A) units:

- FORCE_MKS=**FSUM**
- STRESS_MKS=**SEQV**
- DISP_MKS=**USUM**

If you change the unit system to CGS (cm, g, dyne, °C, s, V, A) and create a new user defined result with **Expression=FSUM+SEQV+USUM** while assigning **Volume** as the **Output Unit**, you will produce the following user-defined results:

Custom Identifier	Expression	Input Unit System	Output Units
FORCE_MKS	FSUM	Metric (m, kg, N, s, V, A)	Force
STRESS_MKS	SEQV	Metric (m, kg, N, s, V, A)	Stress
DISPL_MKS	USUM	Metric (m, kg, N, s, V, A)	Displacement
VOLUME_CGS	FSUM+SEQV+USUM	Metric (cm, g, dyne, s, V, A)	Volume

The expression **VOLUME_CGS** is easy to verify for its Input Unit System, CGS. If FSUM=3 dyne, SEQV=17 dyne/cm² and USUM=2 cm, (as seen in when CGS is selected in the Mechanical application), **VOLUME_CGS** produces the value 22 cm³. Any subsequent changes to the unit system in the Mechanical application cause each of the user defined results to convert based on their required factors. In this manner, VOLUME_CGS will use a factor of 1000 to convert from Metric CGS to Metric mm, because it represents a Volume. FORCE_MKS, STRESS_MKS and DISPL_MKS will convert differently, based on the selected Output Units.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

User Defined Results for the Mechanical APDL Solver

This section examines supported expressions for the **User Defined Result** as well as certain requirements and limitations for their use with the Mechanical APDL solver.

Refer to the **PRNSOL** and **PRESOL** command pages in the Mechanical APDL application Commands Reference for descriptions of most **Component** and **Expression** entries in the table. Some other entries are self-explanatory (**SUM** for example). **VECTORS** refer to vector plot (p. 1968) results that include arrows in the display.

The following tables for node- and element-based results list additional User Defined Result names not included in the **PRESOL**/**PRNSOL** listings. The Solution object Worksheet lists these result options following a solution (see Application (p. 2194)). Using this data, you can explicitly define your User Defined Result, such as Total Deformation by using the component deformations across all of the nodes in the model, identified by **UX**, **UY**, and **UZ**. You can use these component values to mathematically produce a user defined result for Total Deformation: **SQRT(UX^2+UY^2+UZ^2)**.

Go to a section topic:

- Node-Based Results Expressions (p. 2204)
- Element-Based Results Expressions (p. 2205)
- Characteristics, Requirements, and Limitations (p. 2208)
  - Display Option (p. 2208)
  - Contact Results (p. 2208)
  - SPSD Results (p. 2209)
  - Nodal Average Results (p. 2209)
  - PNUM Command Results (p. 2209)
  - Nonlinear Analysis Components (p. 2210)
  - Solution Coordinate System Results (p. 2211)
- User Defined Results Not Displayed in Worksheet (p. 2211)
- Limitations of Vector Displays (p. 2211)

Important:

Review the Notes (p. 2208) below for specific requirements and characteristics regarding certain User Defined Results as well as the topics for User Defined Results Not Displayed in Worksheet (p. 2211) and Limitations of Vector Displays (p. 2211).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Node-Based Results

The following table lists the available expressions that you can use to define your **User Defined Result**. Node-based user defined results are most often associated with degree of freedom solutions (like nodal reactions).

Name	Description
AMPS	Nodal electric current (reaction)
CSG	Nodal magnetic current segments (reaction)
DOMG	Nodal rotational accelerations in a structural transient dynamic analysis (analogous to PRNS,DMG)
F	Nodal structural forces (reaction)[a]
HEAT	Nodal thermal heat flow (reaction)
LOC	Nodal locations (x,y,z)
LOC_DEF	Deformed nodal locations (x+ux,y+uy,z+uz)
M	Nodal structural moments (reaction)[a]
MVP_AZ	Nodal Z magnetic vector potential in an axisymmetric electromagnetic analysis (analogous to PRNS,A)
NDIR	Nodal THXY, THYZ, and THZX values. The NDIRVECTORS display consists of triads.
NAR_EPEL	Component (X, Y, Z, XY, YZ, XZ) strain and Principal (1, 2, 3, INT) elastic strain.[b]
NAR_EPCR	Component (X, Y, Z, XY, YZ, XZ) strain and Principal (1, 2, 3, INT) creep strain.[b]
NAR_EPPL	Component (X, Y, Z, XY, YZ, XZ) strain and Principal (1, 2, 3, INT) plastic strain.[b]
NAR_EPTH	Component (X, Y, Z, XY, YZ, XZ) strain and Principal (1, 2, 3, INT) thermal strain.[b]
NAR_EPTO	Component (X, Y, Z, XY, YZ, XZ) strain and Principal (1, 2, 3, INT) total strain.[b]
NAR_EPELEQV_RST	Equivalent elastic strain.[b]
NAR_EPCREQV_RST	Equivalent creep strain.[b]
NAR_EPPLEQV_RST	Equivalent plastic strain.[b]
NAR_EPTHEQV_RST	Equivalent thermal strain.[b]
NAR_EPTOEQV_RST	Equivalent total strain.[b]
NAR_S	Component (X, Y, Z, XY, YZ, XZ) stress and Principal (1, 2, 3, INT, EQV) stress.[b]
OMG	Nodal rotational velocities in a structural transient dynamic analysis (analogous to **PRNSOL,OMG**)
R	Nodal rotations in a structural analysis (analogous to **PRNSOL,ROT**)
REULER	Structural rotations displayed as Euler triads.
UACOUSTIC	Nodal displacement for an Acoustic Environment which equals the pressure value divided by the product of density and gravity.

[a] The application computes the result values of Forces and Moments (FX/MX, FY/MY, FZ/MZ, FSUM/MSUM, and FVECTORS/MVECTORS) only at constrained nodes. Consequently, if you create a **User Defined Result** from one of these values and scope it to a Construction Geometry **Path** (p. 883), it is possible that no contours will display. Since Mechanical cannot properly interpolate the nodal values at the path point if a path point touches an element with unconstrained nodes, it displays no contour color.

[b] See the Nodal Average Result (p. 2209) note below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element-Based Results

The following table lists the available expressions that you can use to define your element-based User Defined Result. Element-based user defined results can exist at the nodes (like stress and strain) or can exist at the centroid (like volume).

Name	Description
AI	Element nodal acoustic intensity (pressure times the complex conjugate of the acoustic velocity vector (PG)). Components include AIX, AIY, AIZ, AISUM, and AIVECTORS. **Note:** • This result is derived from the pressure and velocity results at the corner nodes. • This result is **not** based upon the sound intensity solution (SNDI).
SPSD	Element nodal equivalent stress as calculated by the solver. See SPSD Result (p. 2209) note below.
SFPRES	Surface pressure load applied to SOLID187 elements in models with cracks. The load is defined by the **SF** command in the solution. The SFPRES result is displayed as a vector normal to the element face.
ELEMENTAL_REAL	Element real data from the Mechanical APDL **R** command.
ELEMENTAL_STATUS	Element birth/death status associated with the **EKILL** command. If element is DEAD, the status is 1; if element is not DEAD, status is 0..
EPCREQV_RST	Element nodal equivalent creep strain as calculated by the solver.
EPELEQV_RST	Element nodal equivalent elastic strain as calculated by the solver.
EPPLEQV_RST	Element nodal equivalent plastic strain as calculated by the solver.
EPTTEQV_RST	Element nodal equivalent total strain (plus thermal strain) as calculated by the solver, that is, EPTTEQV_RST is total mechanical and thermal strain: EPTTEQV_RST = EPELEQV_RST + EPPLEQV_RST + EPCREQV_RST + EPTHEQV_RST.
EPTOEQV_RST	Element nodal equivalent total strain as calculated by the solver, that is, EPTOEQV_RST is total mechanical strain: EPTOEQV_RST = EPELEQV_RST + EPPLEQV_RST + EPCREQV_RST.
ETOP	Element nodal densities used for topological optimization (same as TOPO).
BEAM	Element nodal beam stresses: direct, minimum bending, maximum bending, minimum combined, maximum combined.
SVAR	Element nodal state variable data.
CONTJHEA	Element nodal Joule heat for CONTA174.
CONTFORC	Element nodal contact normal forces for CONTA175.
CONTFNODE	Component (X, Y, Z) contact element forces.[a]
CONTHNODE	Contact element heat.[a]
CONTAREA	Contact element area.[a]
BEAM_AXIAL_F	Element nodal axial force vectors for BEAM188/189.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Name	Description
BEAM_BENDING_M	Element nodal bending moment vectors for BEAM188/189.
BEAM_TORSION_M	Element nodal torsion moment vectors for BEAM188/189.
BEAM_SHEAR_F	Element nodal shear force vectors for BEAM188/189.
BEAMDIRECT	Direct Stress, the stress component due to the axial load encountered in a beam element.
BEAMMIN_BENDING	Minimum Bending Stress. From any bending loads a bending moment in both the local Y and Z directions will arise. This leads to the following four bending stresses: Y bending stress on top/bottom and Z bending stress on top/bottom. Minimum Bending Stress is the minimum of these four bending stresses.
BEAMMAX_BENDING	Maximum Bending Stress, the maximum of the four bending stresses described under Minimum Bending Stress.
BEAMMIN_COMBINED	Minimum Combined Stress, the linear combination of the Direct Stress and the Minimum Bending Stress.
BEAMMAX_COMBINED	Maximum Combined Stress, the linear combination of the Direct Stress and the Maximum Bending Stress.
PIPE_INTERNAL_PRESSURE	Internal pressure at integration point for PIPE288.
PIPE_EXTERNAL_PRESSURE	External pressure at integration point for PIPE288.
PIPE_EFFECTIVE_TENSION	Effective tension at integration point for PIPE288.
PIPE_HOOP_STRESS	Hoop Stress at integration point for PIPE288/PIPE289.
ENFO	Element nodal reaction forces for structural analyses.
ENMO	Element nodal reaction moments for structural analyses.
EHEAT	Element nodal heat values for thermal analyses.
CURRENTSEG	Element nodal magnetic current segments.
VOLUME	Element volumes.
ENERGY	Element potential and kinetic energies.
RIGID_ANG	Element Euler angles for MASS21 elements (rotation about x-axis, rotation about y-axis, rotation about z-axis).
CONTSMISC	Element summable miscellaneous data for contact elements. CONTSMISC is completely analogous in implementation to SMISC (see "User Defined Results Not Displayed in Worksheet" below), except that CONTSMISC, for display purposes, extrapolates the single elemental value to the corner nodes.
CONTNMISC	Element non-summable miscellaneous data for contact elements. CONTNMISC is completely analogous in implementation to NMISC (see "User Defined Results Not Displayed in Worksheet" below), except that CONTSMISC, for display purposes, extrapolates the single elemental value to the corner nodes.
EDIR	Elemental THXY, THYZ, and THZX values: (1) currently only angles of first node in solution record are employed; (2) the EDIRVECTORS display consists of triads.
ECENT	Element centroids (x,y,z)
PNUMTYPE	Element type reference numbers.[b]
PNUMREAL	Real constant set numbers.[b]
PNUMMAT	Material set numbers.[b]
PNUMSEC	Section numbers.[b]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Name	Description
PNUMESYS	Element coordinate system numbers (note: a 0 value corresponds to the global Cartesian system).[b]
PNUMELEM	Mechanical APDL element ID.[b]
PNUMENAM	Mechanical APDL element identifying number (such as 181 for SHELL181 elements).[b]
CONTPNUMTYPE	Element type reference numbers for contact elements.[b]
CONTPNUMREAL	Real constant set numbers for contact elements.[b]
CONTPNUMMAT	Material set numbers for contact elements.[b]
CONTPNUMSEC	Section numbers for contact elements.[b]
CONTPNUMESYS	Element coordinate system numbers for contact elements.[b]
CONTPNUMELEM	Mechanical APDL Element ID for contact elements.[b]
CONTPNUMENAM	Mechanical APDL element identifying number for contact elements.[b]
SMISC	Element summable miscellaneous data.
NMISC	Element non-summable miscellaneous data.
EFFNU_ZERO_EPTOEQV	Element nodal equivalent total strain (EPEL + EPPL + EPCR) as calculated by the post-processor. For average results, the solver averages the element nodal component strains at common nodes and performs a Von Mises calculation with effective Poisson's Ratio set to ZERO.
EFFNU_ZERO_EPTTEQV	Element nodal equivalent total strain plus thermal strain (EPEL + EPPL + EPCR + EPTH) as calculated by the post-processor. For average results, the solver averages the element nodal component strains at common nodes and performs a Von Mises calculation with effective Poisson's Ratio set to ZERO.
LAYNUMBER	Number of layers, as defined by the section database, for a shell element. If no section database exists, the number of layers is displayed as zero.
LAYTHICK	Thickness of a layer, as defined by the section database, for a shell element. The layer number is specified using the **Layer** property. If the **Layer** property is set to **Entire Section**, the thickness of the entire element is displayed.
LAYMATERIAL	Material number for an element, displayed in a manner similar to Mechanical APDL's graphic for **/PNUM**,**MAT,1**. If a shell element contains layers defined by the section database and if the **Layer** property is set to a layer number greater than **0**, then the material number for the layer is displayed.
LAYANGLE	Angle of a layer's coordinate system with respect to the element coordinate system, as defined by the section database, for a shell element. The layer number is specified by the **Layer** property. If the **Layer** property is set to **Entire Section**, a value of zero is displayed.[b]
LAYOFFY	Section offset in the Y direction, as defined by the section database, for a shell element. The Y offset is the same for all layers.
THERMAL_FLUID_HEAT_FLOW_RATE	Rate of heat flow through a specified Line Body (p. 872).
THERMAL_FLUID_HEAT_FLOW_RATE	Heat flow rate due to conduction within the fluid of a Line Body.
MESH_ELEMENT_QUALITY	Composite quality of meshed elements.

Name	Description
MESH_ASPECT_RATIO	Aspect ratio for triangles and quadrilaterals of meshed elements.
MESH_JACOBIAN_RATIO	Jacobian Ratio of meshed elements.
MESH_WARPING_FACTOR	Warping Factor of meshed elements.
MESH_PARALLEL_DEVIATION	Parallel Deviation of meshed elements.
MESH_MAXIMUM_CORNER_ANGLE	Maximum Corner Angle of meshed elements.
MESH_SKEWNESS	Skewness of meshed elements.
MESH_ORTHOGONAL_QUALITY	Orthogonal Quality of meshed elements.
MESH_CHARACTERISTIC_LENGTH	Characteristic Length of meshed elements.
COMBI250_F	Structural force (reaction) using element COMBI250. Components include X, Y, Z, Sum, and Vectors.
COMBI250_M	Structural moment (reaction) using element COMBI250. Components include X, Y, Z, Sum, and Vectors.
TARGFPRS	This expression displays fluid penetration pressure at the target side of a contact condition. The application stores this data in the result file for the target element.

[a] The solver does not calculate these element nodal results. The application calculates these results using the centroid value of each scoped element and divides that value by the number of nodes included in the element. The application then assigns this derived value to each node. This creates an elemental-nodal result. The application does not perform any interpolation for these calculations.

[b] See the PNUM (p. 2209) note below.

Note:

The section database is created by the Mechanical APDL **SECDATA** command.

Characteristics, Requirements, and Limitations

Review the following specific characteristics, requirements, and limitations associated with certain User Defined Results.

Display Option

If the **Display Option** is set to:

- **Averaged**: For the ENFO, EHEAT, and CURRENTSEG expressions, the result at each node represents the sum (or contributions) of all the elements that contain the node.
- **Unaveraged**: The ENFO expression is analogous to **PLESOL,FORCE**.

Contact Results

By default, Contact Results (p. 2043) (accessible through User Defined Results (p. 2192) via CONTSTAT or CONTFLUX) are not written to the result file in a thermal analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

SPSD Result

SPSD is a User Defined Result that is unique to the Mechanical APDL result file. For any element that supports stresses, the SPSD result represents the equivalent stress, for each corner node in the element, as stored on the result file. Hence, SPSD is the equivalent stress as calculated by the Mechanical APDL solver for the corner nodes. For this result, **SPSD** is the expression displayed in the **Type** column and **Stress** is displayed in the **Output Unit** column. Prior to release 13.0, SPSD represented the equivalent stress as calculated from component stresses during postprocessing, that is, it was not calculated by the Mechanical APDL solver.

Nodal Average Result

Note the following Nodal Average Result (NAR) requirements. NAR results are:

- Supported by Mechanical APDL solver only.
- Controlled by the **OUTRES,NAR** command.
- Results averaged across all bodies.
- Displayed in the Global Coordinate System only.
- Not supported the MAXSHEAR component.
- Not supported for Random Vibration Analyses and Response Spectrum analyses.
- Not supported for Equivalent Strain data (zero is reported) for the element LINK180.

Note:

If you use the **OUTRES** command in a **Commands (APDL)** object to 1) disable a tensor result and 2) enable the corresponding NAR, the application only supports the **Averaged** option for the **Display Option** and **Average Across Bodies** properties for the specified result. Furthermore, the label "Nodal Averaged Result" is displayed in the legend of the **Geometry** window for the result.

PNUM Command

Displays of **/PNUM** results are analogous to EPLOTs with the following commands in Mechanical APDL:

- /PNUM,TYPE,1
- /PNUM,REAL,1
- /PNUM,MAT,1
- /PNUM,SEC,1
- /PNUM,ESYS,1
- /PNUM,ELEM,1

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

For example, the range of the values of the **PNUMTYPE** result vary from the smallest element type to the largest element type, as created by Ansys **ET** commands.

Important:

Mechanical supports up to 200 SVAR results (SVAR1, ..., SVAR200). Do not exceed this value. If more than 200 SVAR results exist in the result file, Mechanical does not evaluate or display **any** SVAR results. All SVAR results in the result file are ignored.

Note:

- PNUM results are available for all analysis types.
- When you are analyzing shell elements, the PNUMMAT result displays a Material Number for each layer when the following conditions are met:
  - The shell element contains layers defined by the section database (via **SECDATA** command).

    And...

  - The **Layer** property of the User Defined Result is set to a number greater than **0**.

Nonlinear Analysis Components

During a nonlinear analysis (**Large Deflection** property set to **Yes**), the application generates the following components for the Nonlinear (NL) Item of the **PLESOL** command.

Component	Description
SEPL	Equivalent stress (from stress-strain curve).
SRAT	Stress state ratio.
HPRE	Hydrostatic pressure.
EPEQ	Accumulated equivalent plastic strain.
CREQ	Accumulated equivalent creep strain.
PLWK	Plastic work per volume/plastic strain energy density.
CRWK	Creep strain energy density.
ELWK	Elastic strain energy density.
SGYT	Yield stress (tensile).
PEQT	Equivalent plastic strain (tensile).

[a] This expression is the equivalent of the command **PLESOL,SEND,CREEP**.

[b] This expression is the equivalent of the command **PLESOL,SEND,ELASTIC**.

[c] No Mechanical APDL equivalent.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Solution Coordinate System Results

If you evaluate a **User Defined Result** with Element-Nodal result quantities (e.g., stress) that are 1) averaged, or 2) interpolated, the application could produce incorrect result values when using the **Solution Coordinate System**. Commonly, you see this situation when using shell bodies, where the individual element coordinate systems may not be aligned with each other. Some examples of averaging or interpolation operations include calculating the result value on a node shared between two elements, calculating the result value on a midside node from corner nodes, and calculating result values on paths/surfaces. Standard element-nodal results in Mechanical are always automatically evaluated in **Global Coordinate System**, and therefore do not face this problem.

User Defined Results Not Displayed in Worksheet

For the Mechanical APDL solver, there are User Defined Results associated with summable miscellaneous data (SMISC) and non-summable miscellaneous data (NMISC) on the result file. These results are *not* listed in the Solution Worksheet. Because this data can be voluminous, by *default*, Mechanical does not write it to the result file for all element types in the model (examples of MISC records always written to the result file include beam, joint, and spring element types). You activate miscellaneous output for all elements or just contact elements using the Output Controls (p. 1473) available in the Details of the Analysis Settings (p. 1417) object. Mechanical has adopted a convention that miscellaneous data for contact elements be called CONTSMISC and CONTNMISC. This means that SMISC and NMISC data will only display on non-contact elements and that CONTSMISC and CONTNMISC data will only display on contact elements. You can also request and store state variables such as USERMAT or USERCREEP if you wish to utilize user-defined materials. Like miscellaneous data, SMISC and NMISC, the state variables do not display in the Solution Worksheet. You access state variables using the expression field entry SVAR followed by the state variable number.

To display these results:

1. Select the **User Defined Result** option on the Solution Context tab.
2. In the Details view **Expression** field, type the string SMISC or NMISC followed by the sequence number which indicates the desired datum.

For example, to display the 2nd sequence number for SMISC, enter SMISC2 for the Expression. The graphics contour display will be similar to the Mechanical APDL display for the command **PLESOL,SMISC,2**. When you evaluate this result, the Details view will show no units and no coordinate system for this data. That is, no unit conversions and no coordinate transformations are performed. If you enter a data expression that does not exist on the result file, the result will not be evaluated. To display the 2nd sequence number for summable miscellaneous data on scoped contact elements, enter CONTSMISC2 for the Expression.

Limitations of Vector Displays

The following limitations are associated with vector display for user defined results:

- BEAM_SHEAR_FVECTORS (based upon section shear forces, SFy and SFz, in the BEAM188 SMISC record).
- BEAM_BENDING_MVECTORS (based upon the bending moments, My and Mz, in the BEAM188 SMISC record).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The beam is defined by nodes I(end 1) and J(end 2) and an optional orientation node K. Depending upon direction from node I to node J, the displayed vector for these results may be flipped 180 degrees.

User Defined Results for the Ansys Rigid Dynamics Solver

The following table lists the available **User Defined Result** expressions for the Ansys Rigid Dynamics Solver. These expressions are available in the Solution Summary Worksheet (p. 1961) following the solution.

Note:

Even though S and/or EPEL values may be available in the worksheet, these stress and strain values are not written to the rigid solver file. Their availability may be a result of flexible parts existing in the model.

Type	Description/Components
U	Displacements: X, Y, Z, SUM, VECTORS
A	Accelerations: X, Y, Z, SUM, VECTORS
V	Velocities: X, Y, Z, SUM, VECTORS
OMG	Rotational Velocities: X, Y, Z, SUM, VECTORS
DOMG	Rotational Accelerations: X, Y, Z, SUM, VECTORS
CFFTOT	Total Contact Force: X, Y, Z, SUM
CFT-TOT	Total Contact Torque: X, Y, Z, SUM
CFFNOR	Normal Contact Force: X, Y, Z, SUM
CFT-NOR	Normal Contact Torque generated by Normal Force: X, Y, Z, SUM
CFFTAN	Tangent Contact Force: X, Y, Z, SUM
CFT-TAN	Tangent Contact Torque (torque generated by tangent force): X, Y, Z, SUM
CF-LOC	Contact Force Location: X, Y, Z
CDIST	Contact Distance Value
CDPTC	Location of Contact Side for Contact Distance: X, Y, Z
CDPTT	Location of Target Side for Contact Distance: X, Y, Z

Python Result

The Python Result object enables you to evaluate output quantities by executing an Iron-python script based on the Data Processing Framework (DPF) post-processing toolbox.

Go to a desired topic:

Insert the Python Result Object

Understanding Data Processing Framework

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using DPF with the Python Result Feature
Limitations of the Python Result Object
Workflow Examples
Animation

Python Result Disclaimer

The **Python Result** feature advances capabilities for you to create and evaluate a variety of scripts. Since the core architecture of this object works like a generic Python interpreter, you are free to use all that Python has to offer.

However, if your goal is to transfer information back to Mechanical in the form of a standard output, such as contours, tables, charts, etc., there are limitations to what Mechanical can handle and display. There are also performance considerations for complicated calculations within the script.

The examples provided in this section illustrate the most commonly used scenarios with a variety of inputs (geometry and finite element scoping, time and frequency selection, data sources, etc.) and outputs (Contours, deformations, table/charts) that Mechanical currently supports for this feature.

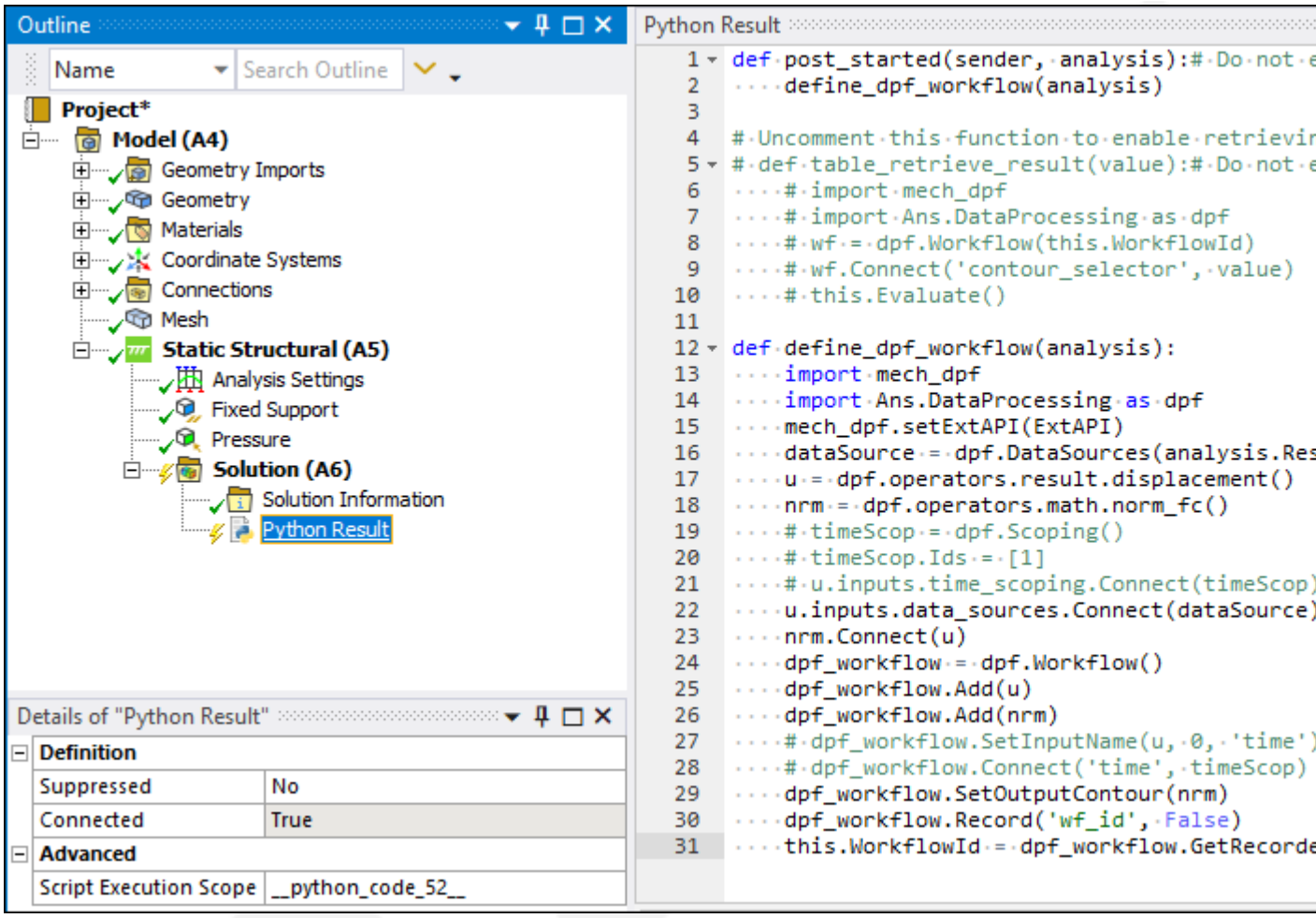
We encourage you to be cautious while using this feature. Python and DPF offer many features, however, there are limits to what Mechanical can do in terms of supported outputs.

Insert the Python Result Object

To insert a **Python Result** into the Outline:

1. Insert a **Python Result** object under the **Solution** object using either the contextual (right-click) menu or the **Python Result** option on the **Solution** Context ribbon tab. By default, the object includes a template script for a **Total Deformation** result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

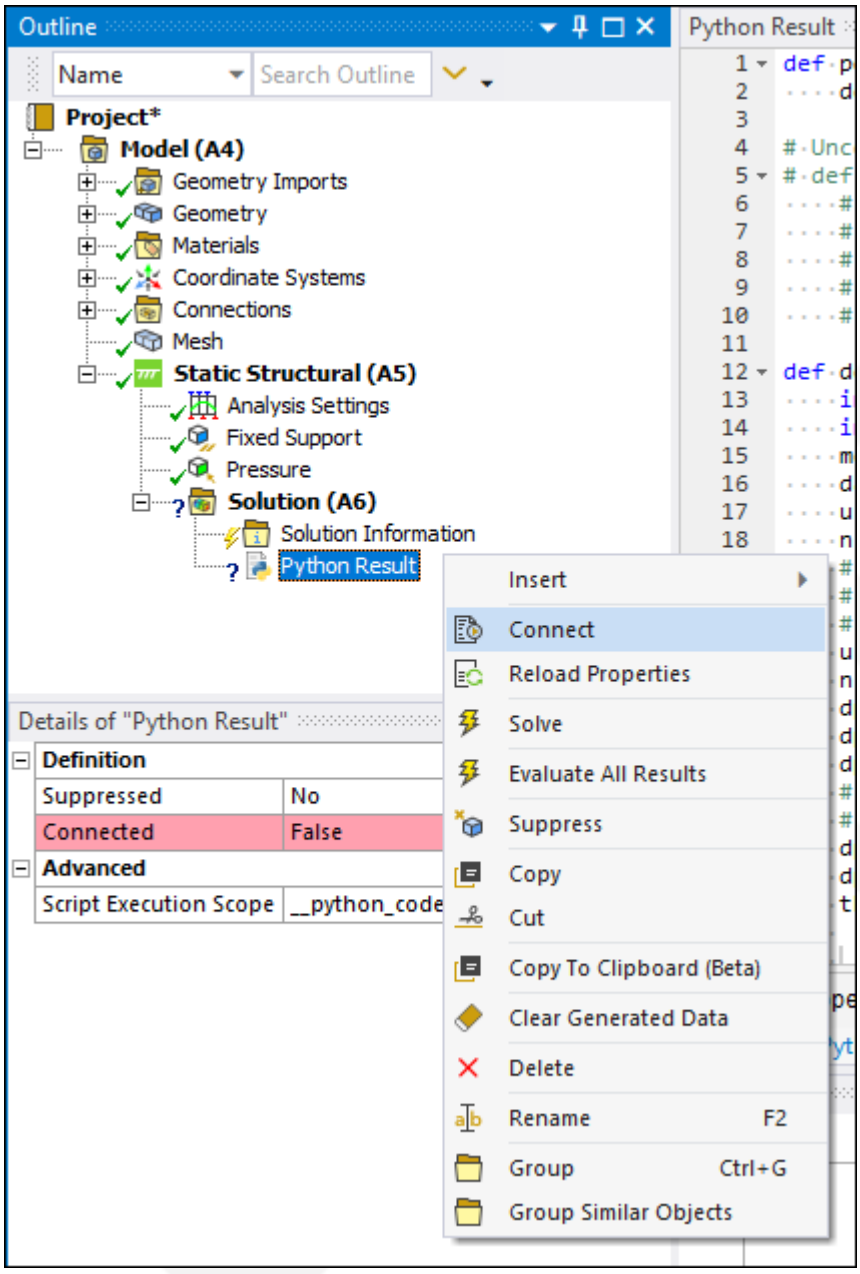


2. Write your desired scripts. When the text of the **Python Result** object is modified, the object becomes under-defined as the Python code is no longer connected to the callback. It is necessary to connect the code changes once completed or the code will not be executed. Note that as soon as you start editing the text in the editor, the **Python Result** object will become under-defined, and the **Details** pane will indicate that the **Connected** property of the **Python Result** object has become false. At this point, once you are finished making your code changes or use the right-click context menu on the object in the tree to **Connect** the **Python Result** object.

3. Right-click the object and select **Connect**. Make sure that the **Connected** property of the object shows the value **True**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

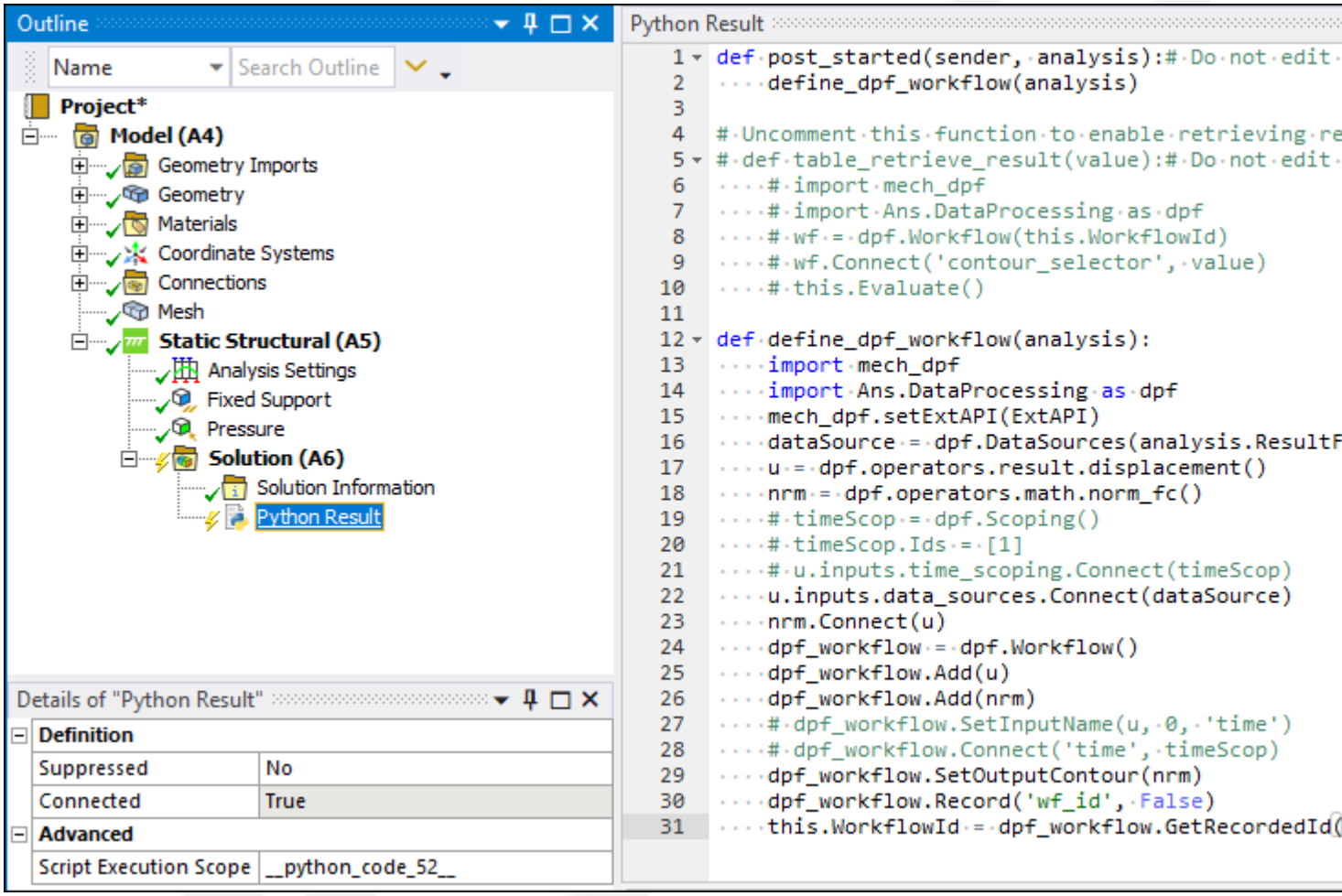
Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

4. **Note:**

   If you make a change to the script once it is "connected" causes the object to become “undefined” (question mark icon). Select the **Connect** option again to update the object.



Understanding Data Processing Framework

The Data Processing Framework (DPF) is designed to provide numerical simulation users/engineers with a toolbox for accessing and transforming simulation data. DPF can access data from solver result files as well as several neutral formats (csv, hdf5, vtk, etc.). Various operators are available enabling the manipulation and the transformation of this data. DPF is a workflow-based framework that enables simple as well as complex evaluations by chaining operators. The data in DPF is defined based on physics agnostic mathematical quantities described in a self-sufficient entity called field. This allows DPF to be a modular and easy to use tool with a large range of capabilities. It is a product designed to handle large amounts of data.

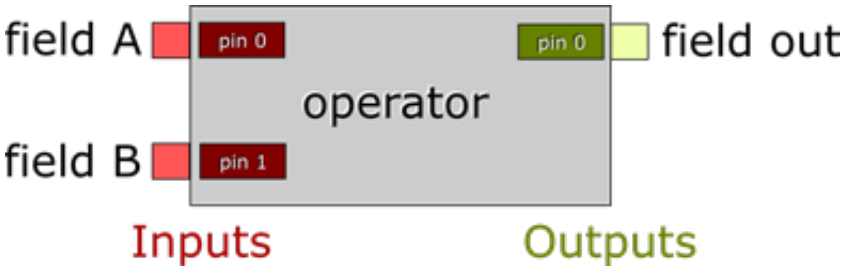
In summary, DPF can be explained through the following concepts:

- **Operator**: Is the only object used to create and transform the data. It is composed of a “core” (which will operate the calculation part), along with input and output “pins.” Those pins allow

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

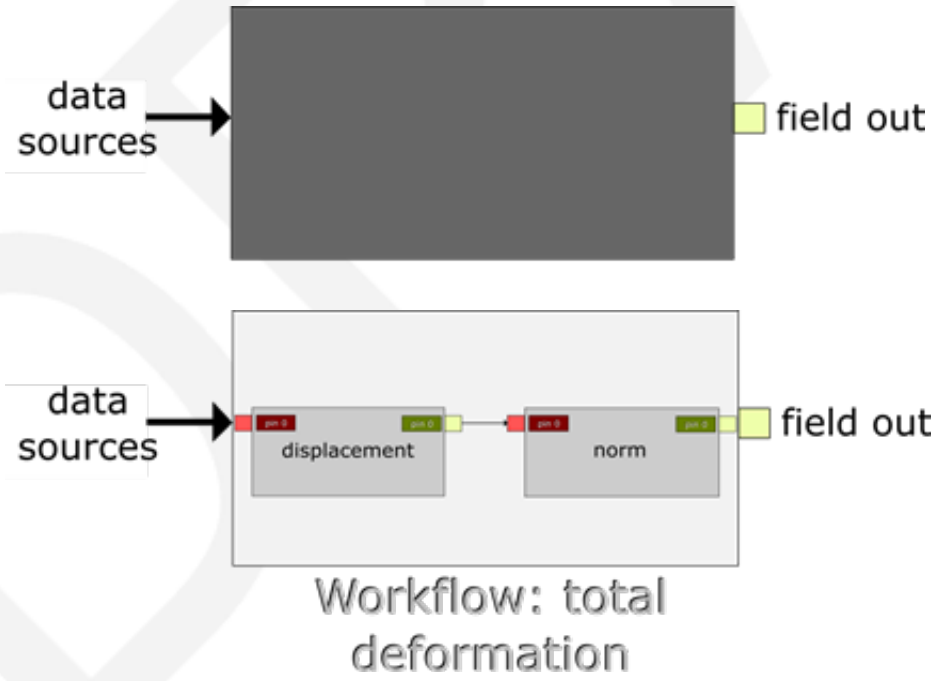
the user to provide input data to each operator. When the operator is evaluated, it will process the input information to compute its output ones. Operators can be chained together to create workflows.



- **Workflow**: Is the global entity that you will use. Built by chaining operators, it will evaluate the data processing defined by the used operators. It needs input information, and it will compute the requested output information. See workflows' examples below.
- **Data Sources**: Are one or more files in which analysis results can be found.
- **Field**: Is the main simulation data container. In numerical simulations, results data are defined by values associated with entities (scoping), and these entities are a subset of a model (support). In DPF, field data is always associated to its scoping and support, making the field a self-describing piece of data.

  For example: For a Field of a nodal displacement, the displacements are the simulation data, the associated nodes are the scoping. The field also contains homogeneity and the unit of the data to complete the data definition to have it self-described.

  Those are the four main data types. For any data processing, the workflow can be compared with a black box (illustrated below) in which some operators are chained, computing the information it is made for.



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Other Data Types

Additional data types include:

- **Fields container**: A fields container is a container of fields. It is used for example in transient/harmonic/modal or multi - steps static analysis,
- **Support**: Is the physical entity to which the field is associated. For example, it can be a mesh, a geometrical entity, time, or frequency values.
- **Scoping**: Is a subset of the model's support. Typically, scoping can represent nodes ids, element ids, time steps, frequencies, joints, etc. Therefore, the scoping can describe a spatial subset and / or a temporal one on which a Field is scoped. It is a part of the support.

Location

Location is the type of topology associated to the data container. DPF uses, for example, three different spatial locations, that are **Nodal**, **Elemental**, and ElementalNodal for finite element data. The Nodal type (resp. Elemental) describes data computed on the Nodes (resp. on the Element itself). These nodes and elements are identified by an Id – typically a node or element number. An ElementalNodal location describes a data defined on the Nodes of the elements, but you will need to use the Element Id to get it. Thus, you can define Elemental or Nodal scoping.

You can find the PDF-based documentation following the process presented below, using the **Scripting** option on the **Automation** tab.

The documentation will be generated at the requested location:

```
import mech_dpf
import Ans.DataProcessing as dpf
doc_op = dpf.operators.utility.html_doc()
doc_op.inputs.output_path.Connect(r'MY_FILE_PATH\MY_FILE_NAME.html')
doc_op.Run()
```

Using DPF with the Python Result Feature

When you add a **Python Result** object, the following script template is automatically included. The different parts of the script are described below:

Script	Description
`def post_started(sender, analysis):` `    define_dpf_workflow(analysis`	PART 1, call define_dpf_workflow (auto generated).

`# Uncomment this function to enable retrieving results from the table/chart`
 	
`# def table_retrieve_result(value):`
`    # import mech_dpf`
`    # import Ans.DataProcessing as dpf`
 	
`    # wf = dpf.Workflow(this.WorkflowId)`
`    # wf.Connect('contour_selector', value)`
 	
`    # this.Evaluate()`
	PART 2, optional callback function for retrieving results from the table/chart.
	
	Get the stored workflow by id and

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Script	Description
	set the input value to the workflow. Evaluate the workflow.
`def define_dpf_workflow(analysis):`	PART 3, start define_dpf_workflow

`import mech_dpf`
`import Ans.DataProcessing as dpf`
`mech_dpf.setExtAPI(ExtAPI)`
	PART 3.1, load dpf environment.

`dataSource = dpf .DataSources(analysis.ResultFileName)`	PART 3.2, take the file in the current analysis system that contains the source results (a dataSource).
`u = dpf.operators.result.displacement()` `nrm = dpf.operators.math.norm_fc()`	PART 3.3, create needed operators

`# timeScop = dpf.Scoping()`
`# timeScop.Ids = [1]`
`# u.inputs.time_scoping.Connect(timeScop)`
	PART 3.4, optional, set time scoping to the operator

`u.inputs.data_sources.Connect(dataSource)` `nrm.Connect(u)`	PART 3.5, make the connections between the elements
`dpf_workflow = dpf.Workflow()` `dpf_workflow.Add(nrm)`	PART 3.6, create the workflow and add operators.

`dpf_workflow.Add(u)`
`# dpf_workflow.SetInputName(u, 0, 'time')`
`# dpf_workflow.Connect('time', timeScop)`
	PART 3.7, optional, set time scoping to the workflow.

`dpf_workflow.SetOutputContour(nrm)`
`dpf_workflow.Record('wf_id', False)`
`this.WorkflowId = dpf_workflow.GetRecordedId()`
	PART 3.8, set the workflow's output and record it.

Note:

- Part 2 is optional and allows the user to retrieve results from the table/chart. More details and use case can be found in next section.
- Part 3.1 imports the DPF environment. This will load all the mandatory DPF tools allowing you to use workflows, operators, fields container, etc.
- Part 3.2 shows how you can access the result file from the current environment that the Python Result is operating under.
- Part 3.3 shows how you can access various operators available in the DPF libraries.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Part 3.4 is optional and is to get results of desired time steps by setting time scoping to the operator.
- Part 3.5 shows how to make connections between operators. All operators have certain inputs that they expect and certain outputs that they produce. Output from one operator can be connected as an input to another.
- Part 3.6 consists of creating the workflow to evaluate and adding operators into it. All participating operators must be connected to a workflow.
- Part 3.7 is optional and allows to set the desired input to the workflow (the given example is to set the time scoping to the input of the workflow, which works the same as Part 3.4.)
- Then, part 3.8 shows how to set the output result you want (here it will be the output gotten from the norm operator). The recording step returns the workflow id, allowing you to get back the stored workflow using this scripting line:

```
dpf_workflow = dpf.Workflow(WorkflowId)
```

Limitations of the Python Result Object

The following are known limitations related to the **Python Result** object. In addition to these limitations, you may encounter additional scenarios where the feature is not supported. Review the Python Result Disclaimer (p. 2213) for more information.

- It is recommended to clear and re-evaluate a Python Result after resuming Mechanical to get the deformation scale factor and retrieving results from the table/chart work.
- Not all analysis types are supported.
- Not all non-standard element types/mesh features are supported, such as Nonlinear Adaptive Region, Reinforcements, and General Axisymmetric.
- Some result object APIs are not supported, such as Legend manipulation, Plot Data API.
- Not all result types are supported, such as Complex Results, Heat Flux, beam strains/stresses.
- There is limited access to result types in LS-DYNA.
- Certain interface operations are not supported, such as Cut/Copy/Paste, Duplicate, and Evaluate from Workbench.

Workflow Examples

The examples content uses the script under the function:

```
define_dpf_workflow(analysis)
```

This is the script in Part 3, where the parameter analysis is referring to the analysis system which the current Python Result object belongs to.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Go to an example:

Display Results for Shared Shell Topologies

Total Deformation

Von Mises Stress

Retrieving Total Deformation from Table/Chart for Time Point

Strain Tensors - Named Selection Scoping

Total Deformation Scoped on Time Step using Property Provider

Total Deformation Scoped on Geometry using Property Provider

Migrating to New Python Result

Display Results for Shared Shell Topologies

This workflow is an example of how to display results on shells for multiple bodies that share topology. In this case, displacement results. The unique feature of this example is that the result field is created from scratch.

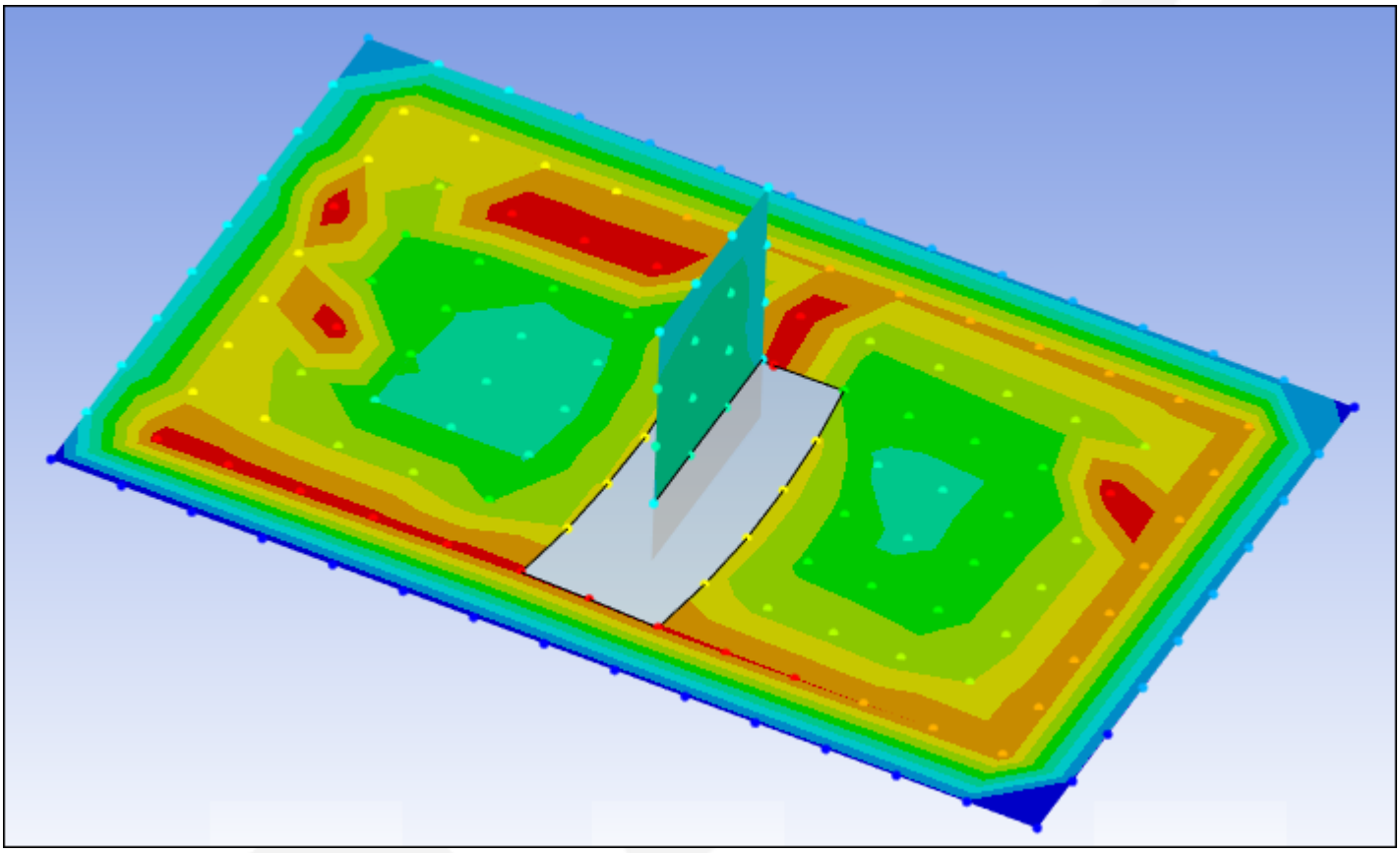
```
import mech_dpf
import Ans.DataProcessing as dpf
mech_dpf.setExtAPI(ExtAPI)
dataSource = dpf.DataSources(analysis.ResultFileName)
```

Script Example

```
#Read a mesh from a results file
  mesh_op = dpf.operators.mesh.mesh_provider()
  mesh_op.inputs.data_sources.Connect(dataSource)
  mesh = mesh_op.outputs.mesh.GetData()

#Create a field of results from scratch with these attributes
#Location : "Nodal"
#Mesh support : mesh
#Shell Layers definition: Layer Independent
#Geometries with shells can be defined by:
#NoneLayer
#LayerIndependent,
#TopBottom
#TopBottomMid

  my_field = dpf.FieldsFactory.CreateScalarField(numEntities=0, location='Nodal')
  my_field.MeshedRegionSupport = mesh
  my_field.eShellLayers = dpf.enums.shellLayers.LayerIndependent

#Adding the results at nodes to the field (here with no physical sense)
  for i in mesh.NodeIds:
  my_field.Add(i,[float(i)])

#Use the forward operator to plot the field
  forward = dpf.operators.utility.forward_field()
  forward.inputs.field.Connect(my_field)

  dpf_workflow = dpf.Workflow()
  dpf_workflow.Add(forward)
  dpf_workflow.SetOutputContour(forward,dpf.enums.GFXContourType.FENodalScoping)
  dpf_workflow.Record('wf_id', False)
  this.WorkflowId = dpf_workflow.GetRecordedId()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

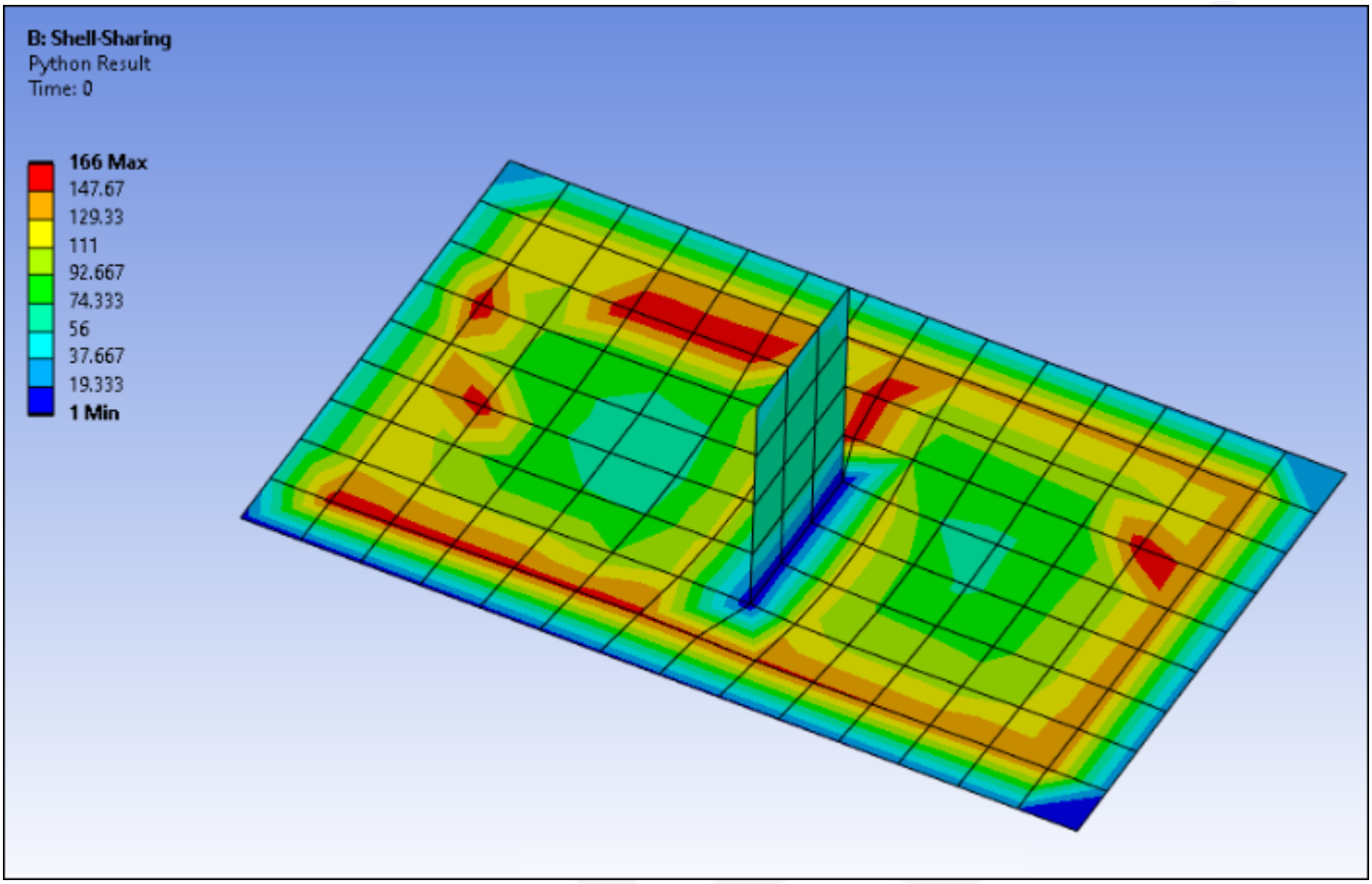
For this specific use-case of shared topology with shell elements, you can face issues concerning the result display. If the **`eShellLayers`** attribute is not set, the shared shell elements (here at the edge between the two topologies) will not appear.



To ensure that the contour result on these shared elements are displayed, you must ensure that the attribute **`eShellLayers`** is set when creating the field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

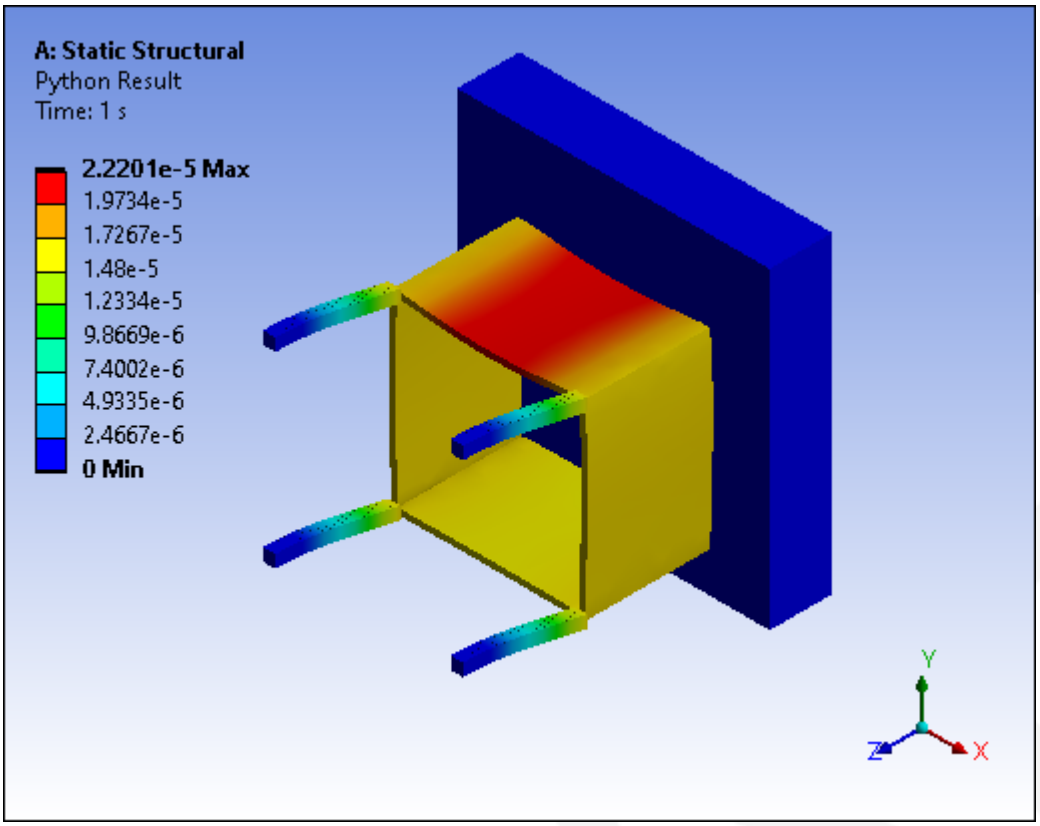


Total Deformation

This elementary workflow enables you to read a displacement from an analysis and compute its norm.

```
import mech_dpf
import Ans.DataProcessing as dpf

my_data_sources = dpf.DataSources()
my_data_sources.SetResultFilePath(analysis.ResultFileName)

u_op = dpf.operators.result.displacement()
u_op.inputs.data_sources.Connect(my_data_sources)

norm_op = dpf.operators.math.norm_fc()
norm_op.Connect(u_op)

wf = dpf.Workflow()
wf.Add(u_op)
wf.Add(norm_op)

wf.SetOutputContour(norm_op) #to get the contour result
wf.SetOutputWarpField(u_op) #to get the warping result (displacement field needed to be used)

wf.Record("wf_id", True)
this.WorkflowId = wf.GetRecordedId()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



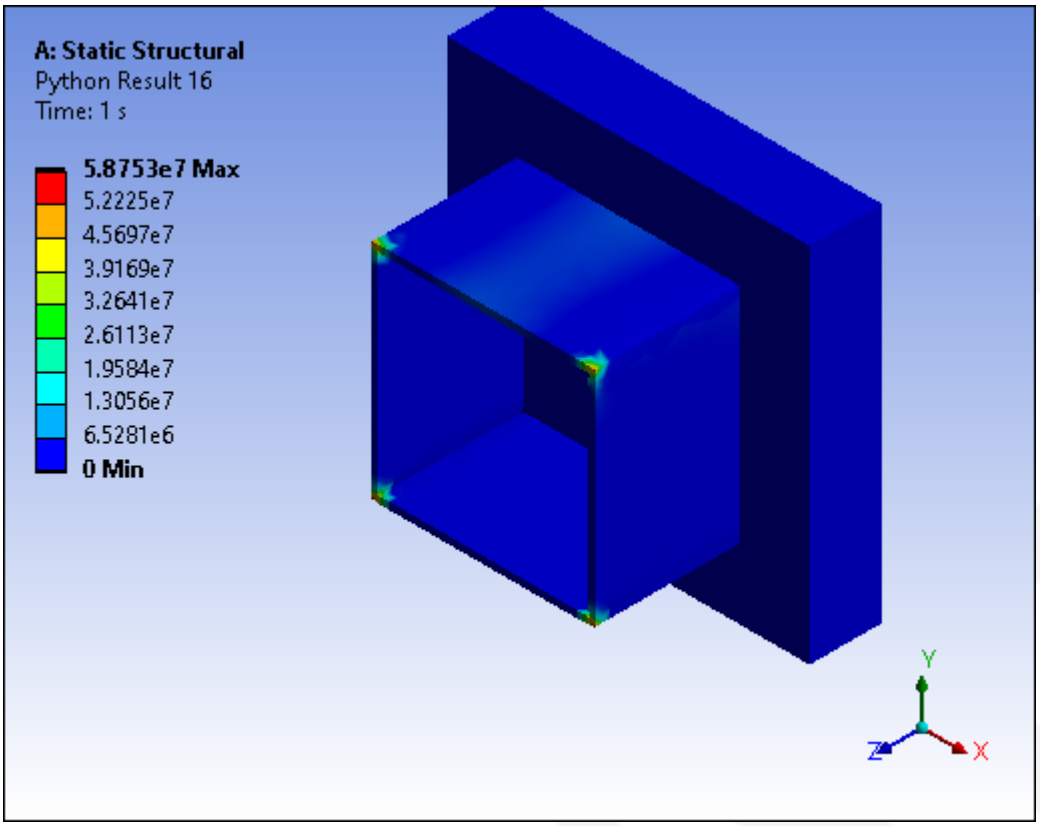
Von Mises Stress

This workflow enables you to read the stress tensors for a range of time steps from an analysis, to average those elemental nodal tensors to a nodal location, and to compute it's Von Mises equivalent. This workflow is an example of how to use the time scoping pin. This pin expects a scoping input and enables you to choose the time or frequency sets of the results.

```
import mech_dpf
import Ans.DataProcessing as dpf

my_data_sources = dpf.DataSources(analysis.ResultFileName)
my_time_scoping = dpf.Scoping()
my_time_scoping.Ids = [1] # the first set

s_eqv_op = dpf.operators.result.stress_von_mises()
s_eqv_op.inputs.requested_location.Connect('Nodal')
s_eqv_op.inputs.data_sources.Connect(my_data_sources)
s_eqv_op.inputs.time_scoping.Connect(my_time_scoping)

dpf_workflow = dpf.Workflow()
dpf_workflow.Add(s_eqv_op)
dpf_workflow.SetOutputContour(s_eqv_op)
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Beam Result Evaluation

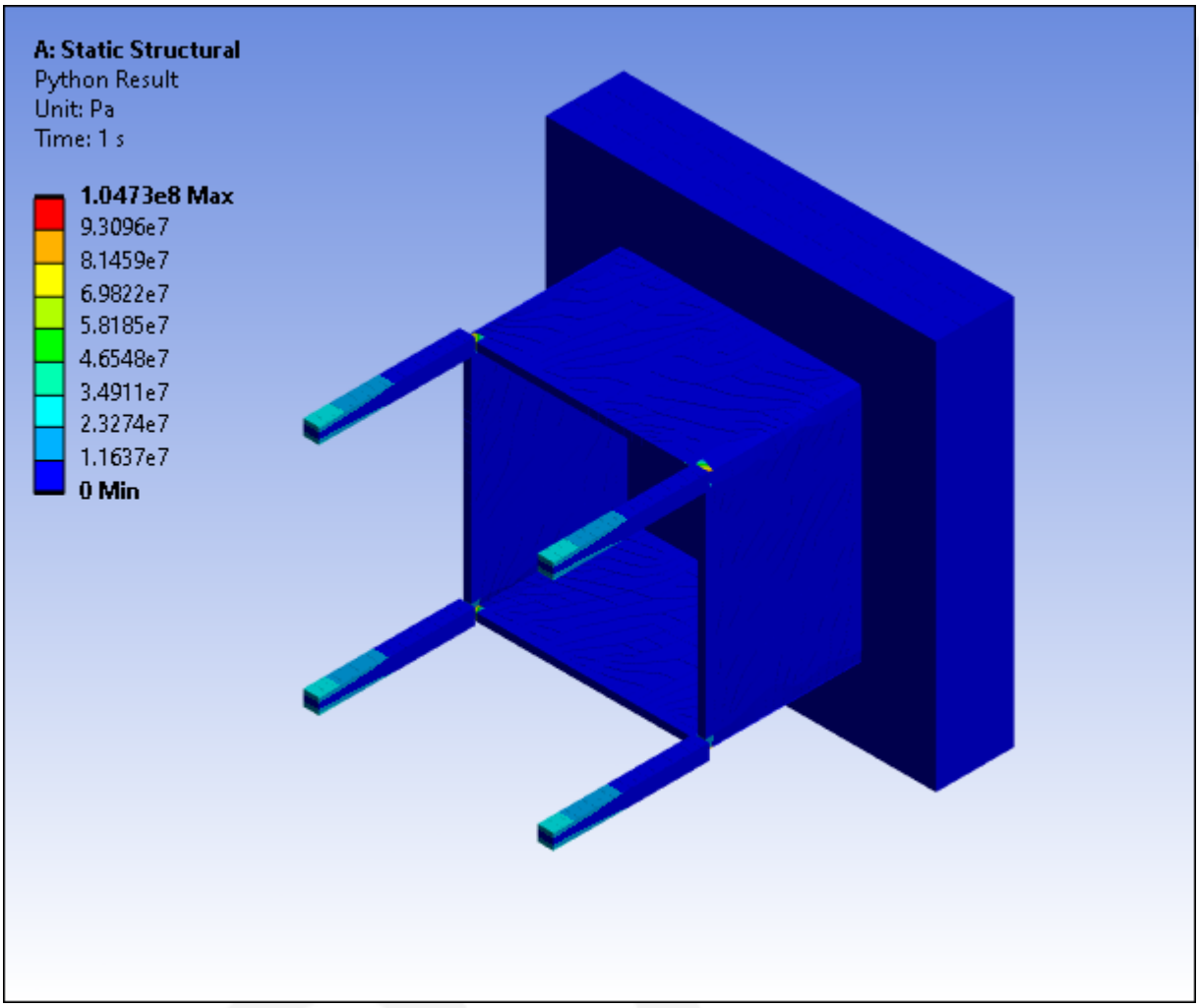
To enable result evaluation on beams, you need to add the following three lines to the script:

Note:

Only unaveraged results (p. 1981) are supported.

```
s_eqv_op.inputs.requested_location.Connect('ElementalNodal')
s_eqv_op.inputs.read_beams.Connect(True)
dpf_workflow.SetOutputName(my_data_sources, "my_data_source")
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Retrieving Total Deformation from Table/Chart for Time Point

This workflow is an example of retrieving results at a desired time point from the table/chart to review results in the **Geometry** window. The full script (not just Part 3) is shown below for a better view.

```
def post_started(sender, analysis):# Do not edit this line
    define_dpf_workflow(analysis)

# Uncomment this function to enable retrieving results from the table/chart
def table_retrieve_result(value):# Do not edit this line
    import mech_dpf
    import Ans.DataProcessing as dpf
    wf = dpf.Workflow(this.WorkflowId)
    wf.Connect('contour_selector', value)
    this.Evaluate()

def define_dpf_workflow(analysis):
    import mech_dpf
    import Ans.DataProcessing as dpf
    dataSource = dpf.DataSources(analysis.ResultFileName)
    u = dpf.operators.result.displacement()
    nrm = dpf.operators.math.norm_fc()
    timeScop = dpf.Scoping()
    timeScop.Ids = [1, 3, 4, 5, 6, 7, 9]
    u.inputs.time_scoping.Connect(timeScop)
    u.inputs.data_sources.Connect(dataSource)
```

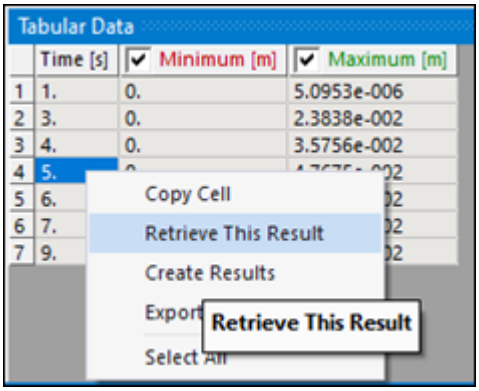
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
    nrm.Connect(u)
    dpf_workflow = dpf.Workflow()
    dpf_workflow.Add(u)
    dpf_workflow.Add(nrm)
    # dpf_workflow.SetInputName(u, 0, 'time')
    # dpf_workflow.Connect('time', timeScop)
    dpf_workflow.SetOutputContour(nrm)
    dpf_workflow.Record('wf_id', False)
    this.WorkflowId = dpf_workflow.GetRecordedId()
```

First, uncomment the callback function table_retrieve_result (value) in Part 2 from the default script.

Moreover, multiple time points scoping is suggested to attach to the workflow. Evaluate the python result. The tabular data and chart will reflect the time points chosen in the script. Select the time point in the Tabular Data window, then click the right mouse button and select Retrieve This Result.



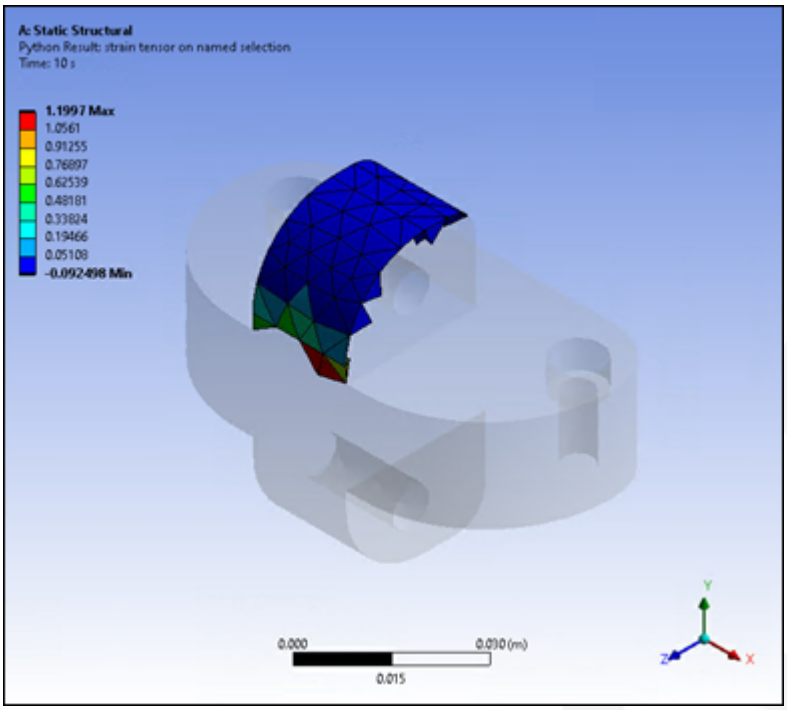
Strain Tensors - Named Selection Scoping

This workflow is an example of the use of mesh scoping for the input of a result reader. Here, you can extract a mesh scoping (a set of spatial entities: node or elements for example) from a named selection created in Mechanical or Mechanical APDL. This mesh scoping is then used as an input for a strain tensors reader. The output of this workflow is the elemental mean of normal elastic strain on X axis scoped only on the named selection.

```
  import mech_dpf
  import Ans.DataProcessing as dpf
  my_data_sources = dpf.DataSources(analysis.ResultFileName)

  ns_op = dpf.operators.scoping.on_named_selection()
  ns_op.inputs.data_sources.Connect(my_data_sources)

# first version with elemental scoping
  ns_op.inputs.requested_location.Connect('Elemental')
  ns_op.inputs.int_inclusive.Connect(1)
  ns_op.inputs.named_selection_name.Connect('SELECTION')

  strain_op = dpf.operators.result.elastic_strain_X()
  strain_op.inputs.data_sources.Connect(my_data_sources)
  strain_op.inputs.mesh_scoping.Connect(ns_op.outputs.mesh_scoping)

# make an elemental average on the elemental nodal field
  to_elemental_op = dpf.operators.averaging.to_elemental_fc()
  to_elemental_op.Connect(strain_op)

  dpf_workflow = dpf.Workflow()
  dpf_workflow.Add(ns_op)
  dpf_workflow.Add(strain_op)
   dpf_workflow.Add(to_elemental_op)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
dpf_workflow.SetOutputContour(to_elemental_op, dpf.enums.GFXContourType.FEElementalScoping)
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```



Note:

- The Name Selection name must match what was used in the input file. For example, if you are using the MAPDL solver, irrespective of the name used in Mechanical, the name of the selection is written as an upper-case string in the input file.
- Regarding the location of the strain operator (ElementalNodal location), and using the elemental averaging operator, it shows how to chain several operators.
- A better scoping representation can be utilized by setting a proper enum for different contour scoping types:

  ```
  dpf_workflow.SetOutputContour(to_elemental_op, dpf.enums.GFXContourType.FEElementalScoping)
  ```

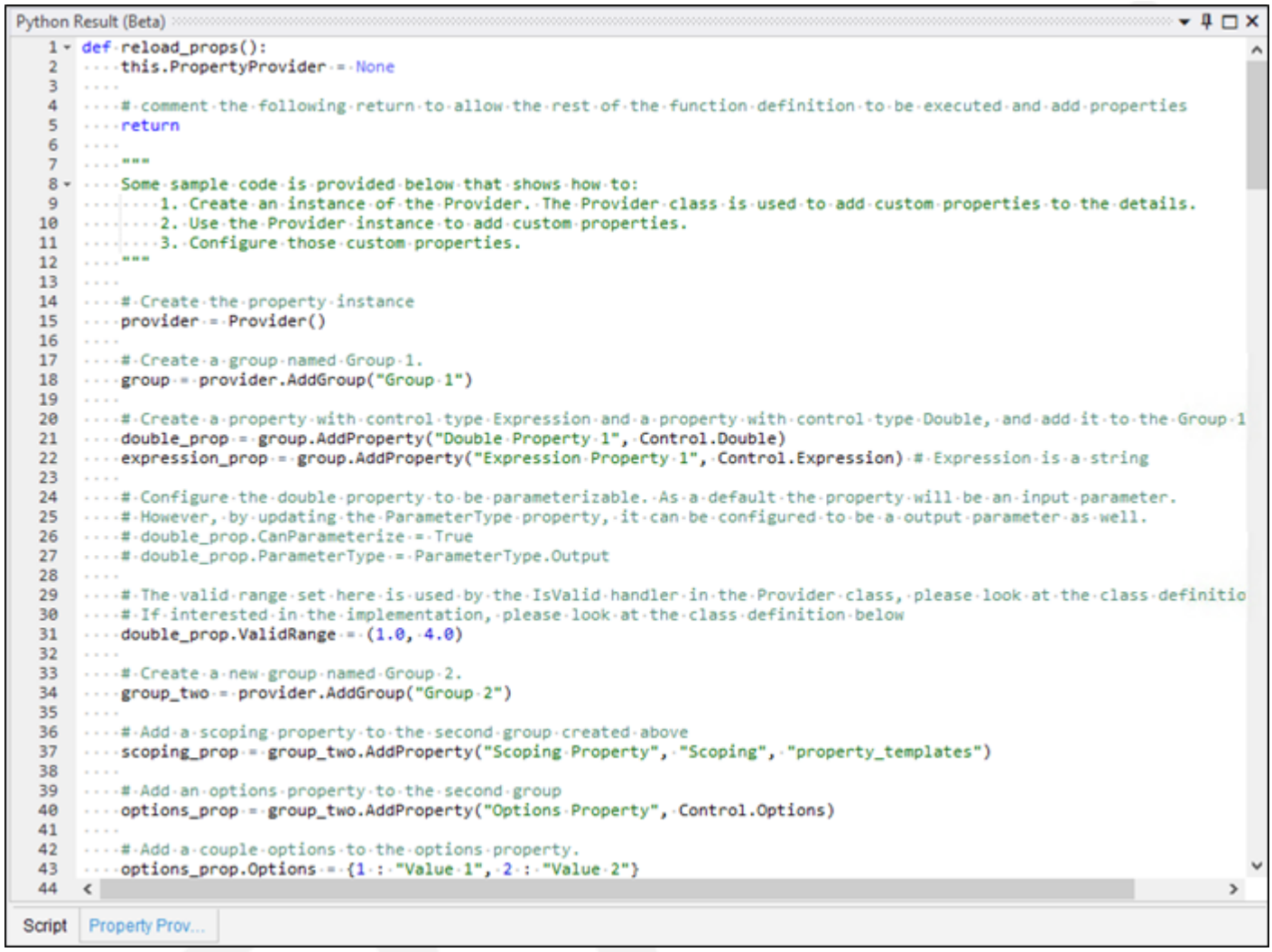
Total Deformation Scoped on Time Step using Property Provider

This workflow enables you to read a displacement from an analysis and compute its norm at a specified time step by using the **Property Provider** (p. 2330) pane. The **Property Provider** pane enables you to add custom properties to the Python Result object.

Below is the **Property Provider** tab in the **Python Result**.

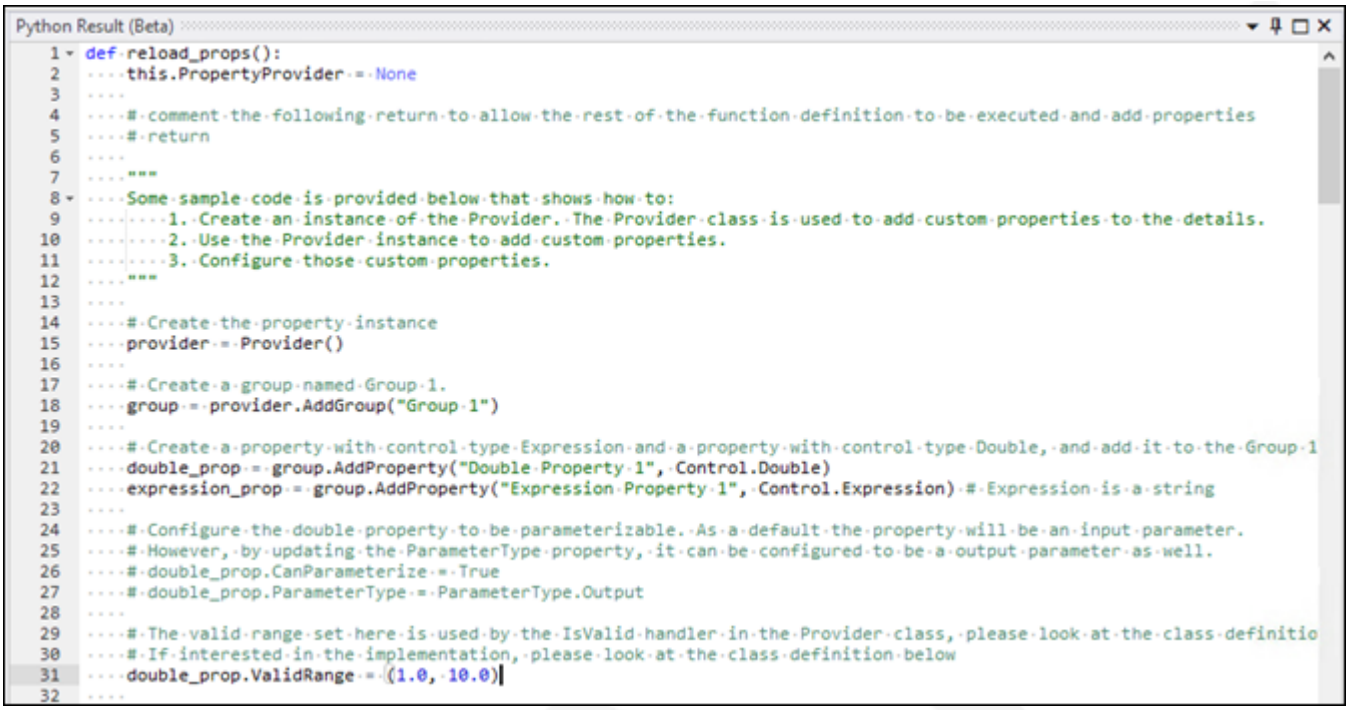
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



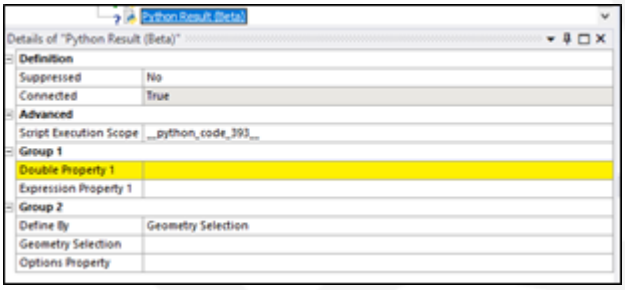
Comment Line 5 as shown below to add the properties for the python result. Assume there are 10 steps in the current analysis. Line 31 can also be modified for a better time set scoping.

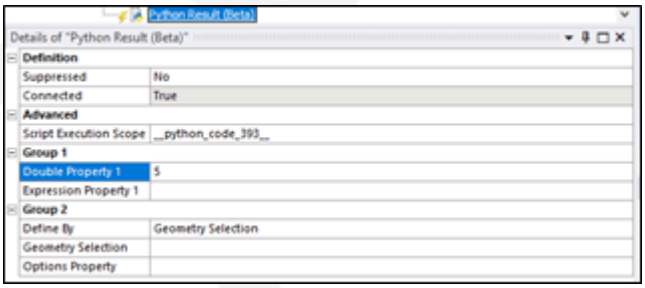
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Right-click on the **Python Result** object and select **Reload**.

The properties defined in the sample code will be added in the result Details view and requires an input for the Double Property 1 under Group 1. With a valid input, the status of the **Python Result** will change from Underdefined to Needs to be Updated.





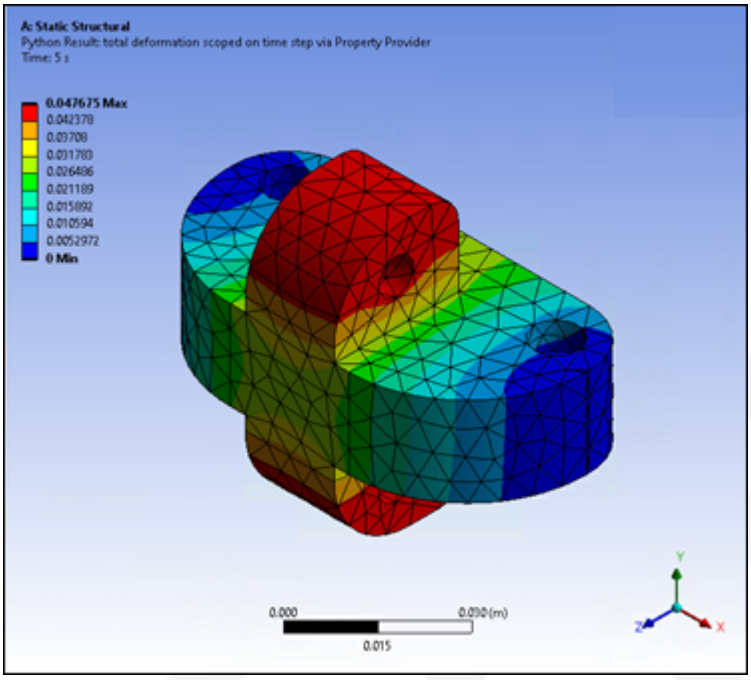
The workflow below is an example of use of the time scoping pin and connected to the value 5 read from Double Property 1.

Total Deformation Scoped on Time Step via Property Provider Pane

```
import mech_dpf
import Ans.DataProcessing as dpf
dataSource = dpf.DataSources(analysis.ResultFileName)
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
u = dpf.operators.result.displacement()
nrm = dpf.operators.math.norm_fc()
prop_val = this.GetCustomPropertyByPath("Group 1/Double Property 1").Value
timeScop = dpf.Scoping()
timeScop.Ids = [int(prop_val)]
u.inputs.time_scoping.Connect(timeScop)
u.inputs.data_sources.Connect(dataSource)
nrm.Connect(u)
dpf_workflow = dpf.Workflow()
dpf_workflow.Add(u)
dpf_workflow.Add(nrm)
dpf_workflow.SetOutputContour(nrm)
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```



Total Deformation Scoped on Geometry using Property Provider

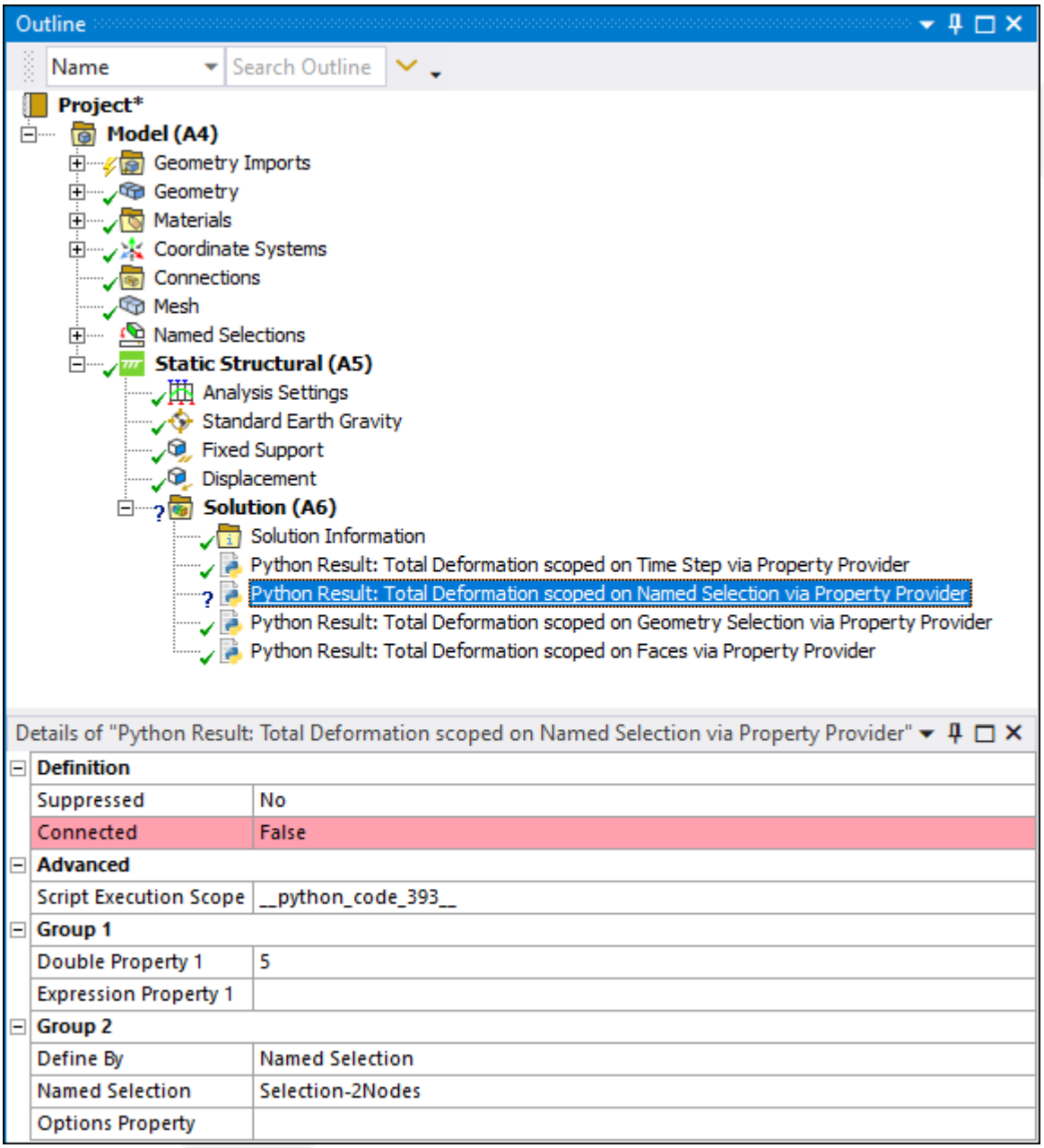
This section describes examples for:

- Scoping to FE Entities (Elements and Nodes) (p. 2231)
- Scoping to Geometry (Faces, Bodies, etc.) (p. 2234)

Scoping to FE Entities (Elements and Nodes)

Pre-define a nodal named selection "Selection-2Nodes," with two nodes selected. Reload the property and set it as below:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The workflow below is an example of:

- Use of the time_scoping pin and connected to the set number 5 read from Group 1.
- Use of the mesh_scoping pin and connected to the named selection read from Group 2.

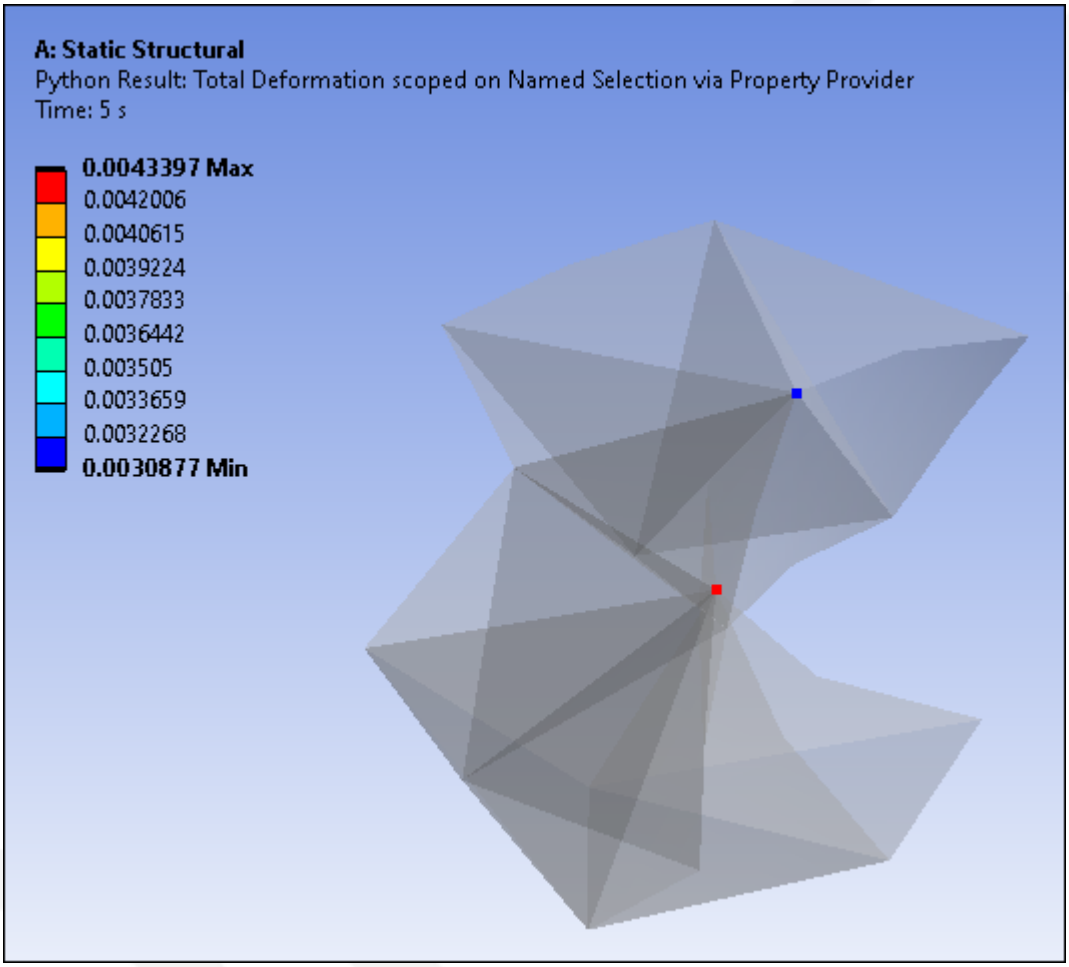
Total Deformation scoped on Time Step and Name Selection via Property Provider Pane

```
import mech_dpf
import Ans.DataProcessing as dpf

dataSource = dpf.DataSources(analysis.ResultFileName)
u = dpf.operators.result.displacement()
nrm = dpf.operators.math.norm_fc()

prop_val = this.GetCustomPropertyByPath("Group 1/Double Property 1").Value
timeScop = dpf.Scoping()
timeScop.Ids = [int(prop_val)]
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
u.inputs.time_scoping.Connect(timeScop)
u.inputs.data_sources.Connect(dataSource)

scoping_refs = this.GetCustomPropertyByPath("Group 2/Scoping Property/Named Selection").Value
mech_dpf.setExtAPI(ExtAPI)
nodes_loc = mech_dpf.GetNodeScopingByRefId(scoping_refs)
u.inputs.mesh_scoping.Connect(nodes_loc)

nrm.Connect(u)

dpf_workflow = dpf.Workflow()
dpf_workflow.Add(u)
dpf_workflow.Add(nrm)
dpf_workflow.SetOutputContour(nrm)
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```



Two helper methods, GetNodeScopingByRefId() and GetElementScopingByRefId() are provided in mech_dpf to create the corresponding nodal/elemental scoping with the connected nodes/elements from the selection.

Similarly, if the geometry scoping is defined by Geometry Selection, and several nodes are selected manually by using the node select tool in the Graphics Toolbar (p. 97), in the above script, just replace the line:

```
nodes_val = this.GetCustomPropertyByPath("Group 2/Scoping Property/Named Selection").Value
```

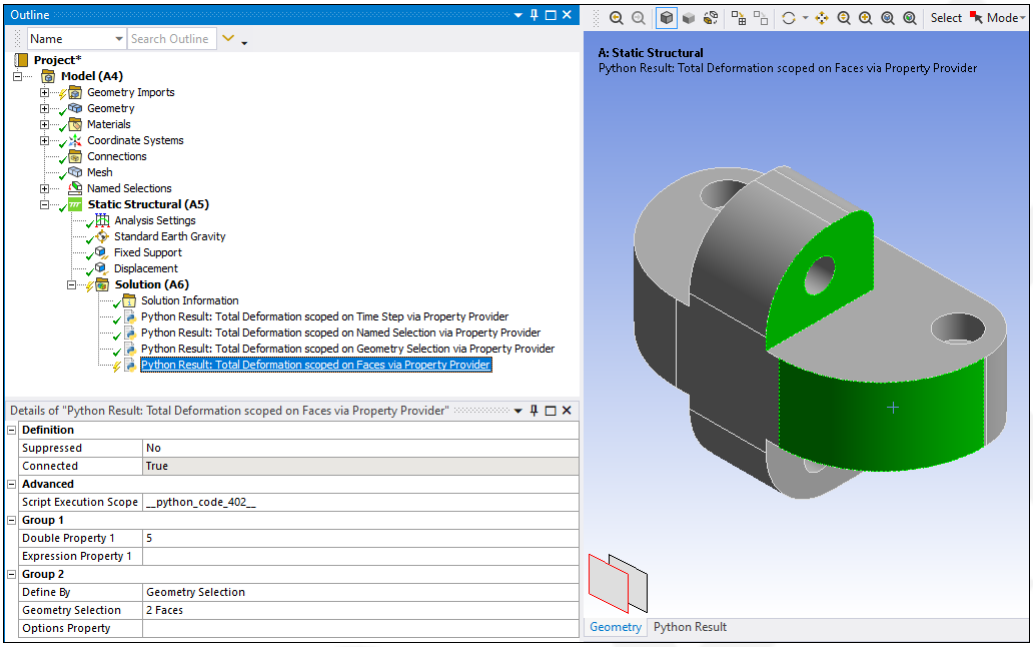
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

with:

```
nodes_val = this.GetCustomPropertyByPath("Group 2/Scoping Property/Geometry Selection").Value
```

Scoping to Geometry (Faces, Bodies, etc.)

The same script in the above section can be used if a geometry is selected (faces, bodies, vertices, etc.) with the property set as shown below.



The workflow below is an example of getting all nodes belong to the selected faces and display the total deformation on these nodes.

Total Deformation Scoped on Time Step and Two Faces via Property Provider Pane

```
import mech_dpf
import Ans.DataProcessing as dpf

dataSource = dpf.DataSources(analysis.ResultFileName)
u = dpf.operators.result.displacement()
nrm = dpf.operators.math.norm_fc()

prop_val = this.GetCustomPropertyByPath("Group 1/Double Property 1").Value
timeScop = dpf.Scoping()
timeScop.Ids = [int(prop_val)]
u.inputs.time_scoping.Connect(timeScop)

u.inputs.data_sources.Connect(dataSource)

scoping_refs = this.GetCustomPropertyByPath("Group 2/Scoping Property/Geometry Selection").Value

mech_dpf.setExtAPI(ExtAPI)
nodes_loc = mech_dpf.GetNodeScopingByRefId(scoping_refs)
u.inputs.mesh_scoping.Connect(nodes_loc)

nrm.Connect(u)
dpf_workflow = dpf.Workflow()
dpf_workflow.Add(u)
dpf_workflow.Add(nrm)
dpf_workflow.SetOutputContour(nrm, dpf.enums.GFXContourType.GeomFaceScoping)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```

Similarly, dpf.enums.GFXContourType.GeomFaceScoping is used to get a better contour display as discussed next.

Von Mises Stress with Different Scoping Representations

The below enum is introduced for a better contour result display in various scoping:

- dpf.enums.GFXContourType.FENodalScoping
- dpf.enums.GFXContourType.FEElementalScoping
- dpf.enums.GFXContourType.FEElementalFaceScoping
- dpf.enums.GFXContourType.GeomBodyScoping
- dpf.enums.GFXContourType.GeomFaceScoping
- dpf.enums.GFXContourType.GeomEdgeScoping
- dpf.enums.GFXContourType.GeomVertexScoping

It can be set as a second parameter when calling the below method in the script:

```
dpf_workflow.SetOutputContour(to_elemental_op, dpf.enums.GFXContourType.FENodalScoping)
```

while the default scoping type is:

```
dpf.enums.GFXContourType.FENodalScoping
```

The workflow below is an example of displaying the Von Mises Stress on named selection with different scoping representations via Property Provider.

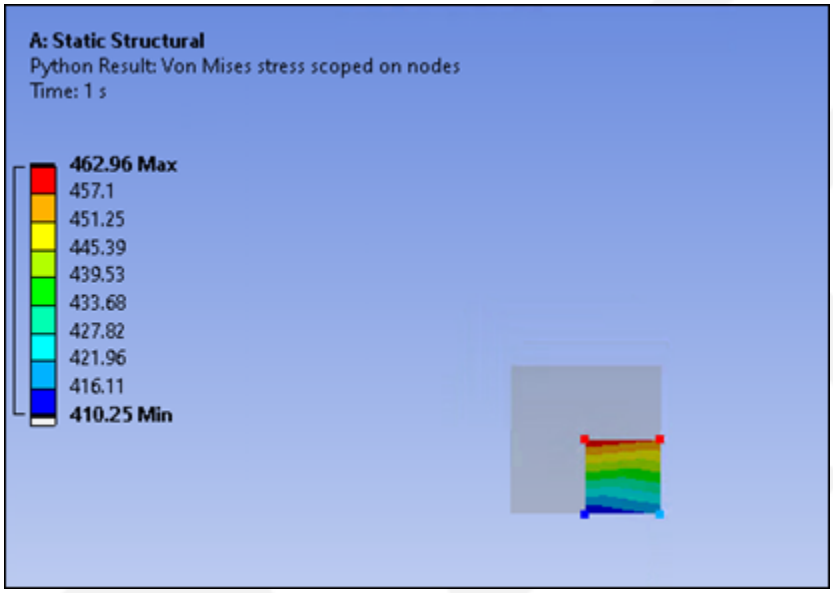
Von Mises Stress Scoped on Time Step and Named Selection via Property Provider Pane

```
import mech_dpf
import Ans.DataProcessing as dpf
dataSource = dpf.DataSources(analysis.ResultFileName)
s_eqv_op=dpf.operators.result.stress_von_mises()

prop_val = this.GetCustomPropertyByPath("Group 1/Double Property 1").Value
timeScop = dpf.Scoping()
timeScop.Ids = [int(prop_val)]
s_eqv_op.inputs.time_scoping.Connect(timeScop)

s_eqv_op.inputs.data_sources.Connect(dataSource)

scoping_refs = this.GetCustomPropertyByPath("Group 2/Scoping Property/Named Selection").Value
mech_dpf.setExtAPI(ExtAPI)
nodes_loc = mech_dpf.GetNodeScopingByRefId(scoping_refs)
s_eqv_op.inputs.mesh_scoping.Connect(nodes_loc)

dpf_workflow = dpf.Workflow()
dpf_workflow.Add(s_eqv_op)
dpf_workflow.SetOutputContour(s_eqv_op, dpf.enums.GFXContourType.FENodalScoping)
dpf_workflow.Record('wf_id', True)
this.WorkflowId = dpf_workflow.GetRecordedId()
```

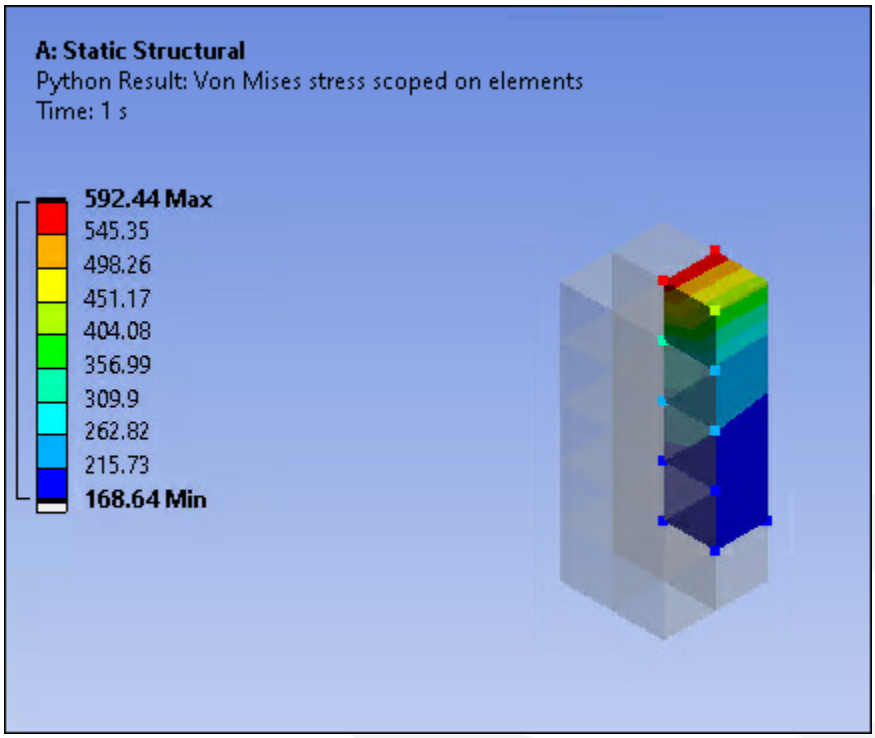
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Different scoping representations and the proper enum to select are summarized in the below table.

Scoping Representation and the Corresponding GFX Contour Type

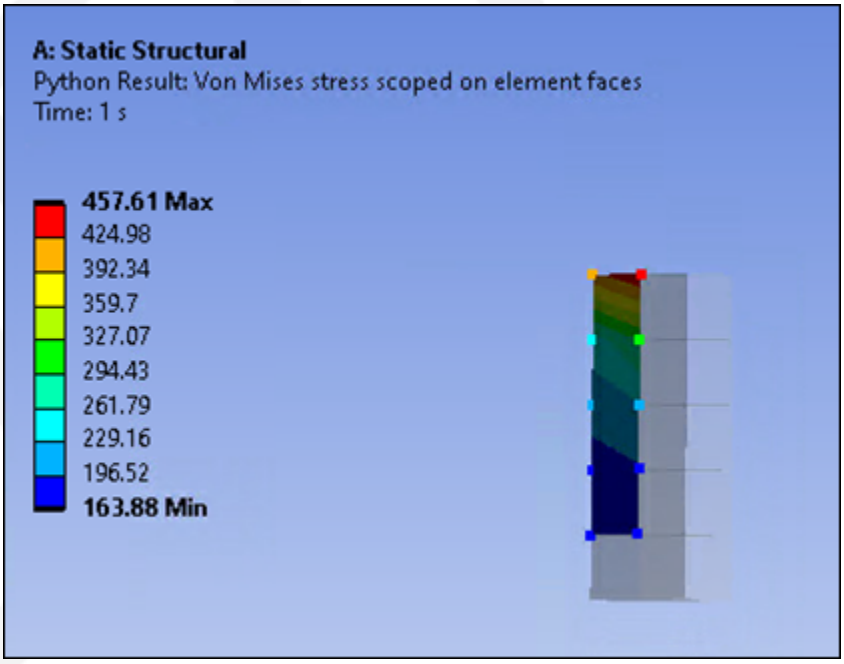
Named Selection	**Method in `mech_dpf`**	**`dpf.enums.GFXContour-Type`**
Four Nodes	GetNodeScopingByRefId()	dpf.enums.GFXContourType.. FENodalScoping
 Averaged Von Mises stress scoped on four nodes.		
Four Elements	**`GetElementScopingByRe-fId ()`**	**`dpf.enums.GFXContour-Type.`** **`FEElementalScoping`**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Averaged Von Mises stress scoped on four elements

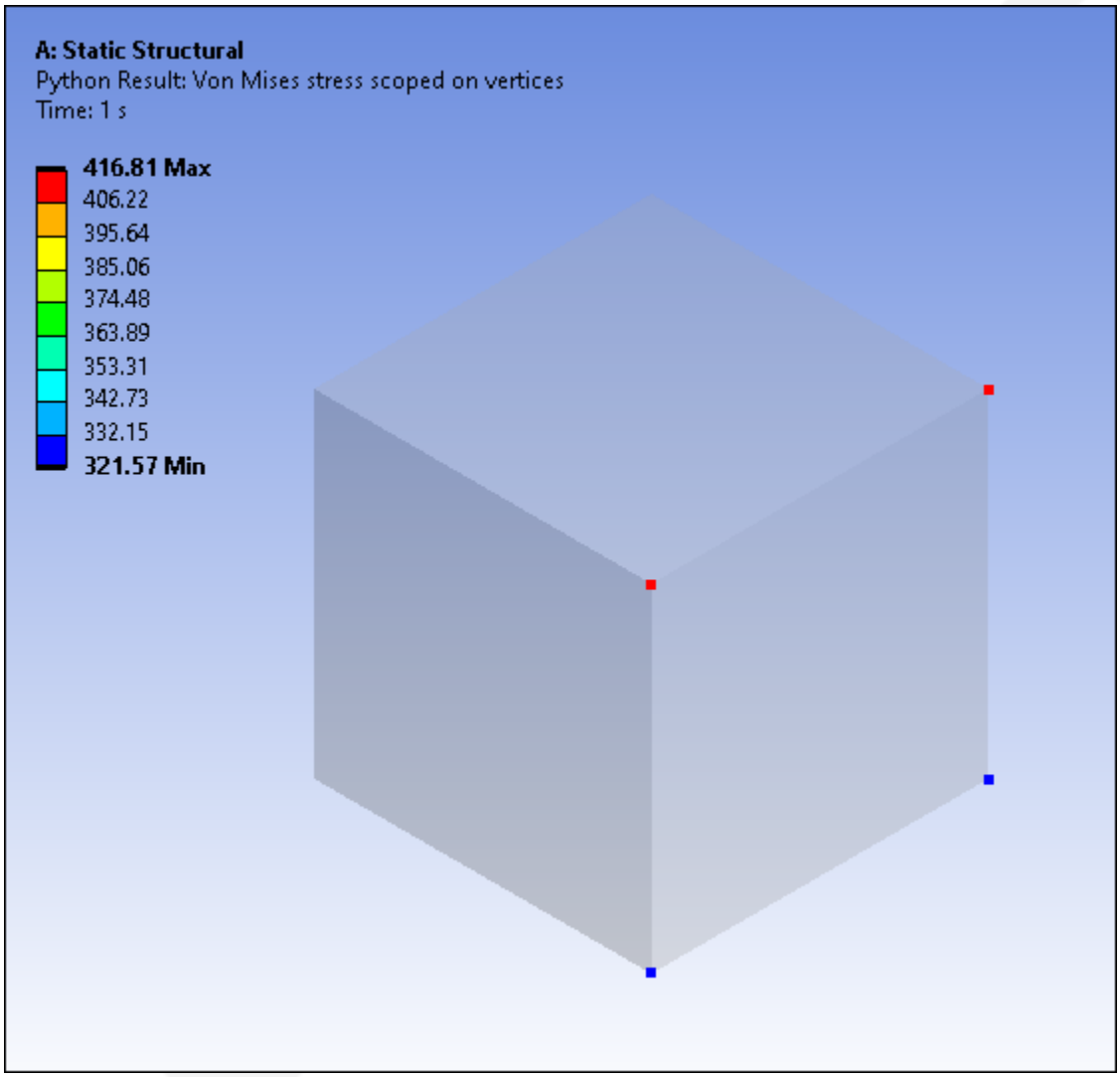
Four Element Faces	`GetNodeScopingByRefId()`	`dpf.enums.GFXContourType.FEElementalFaceScoping`



Averaged Von Mises stress scoped on four element faces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

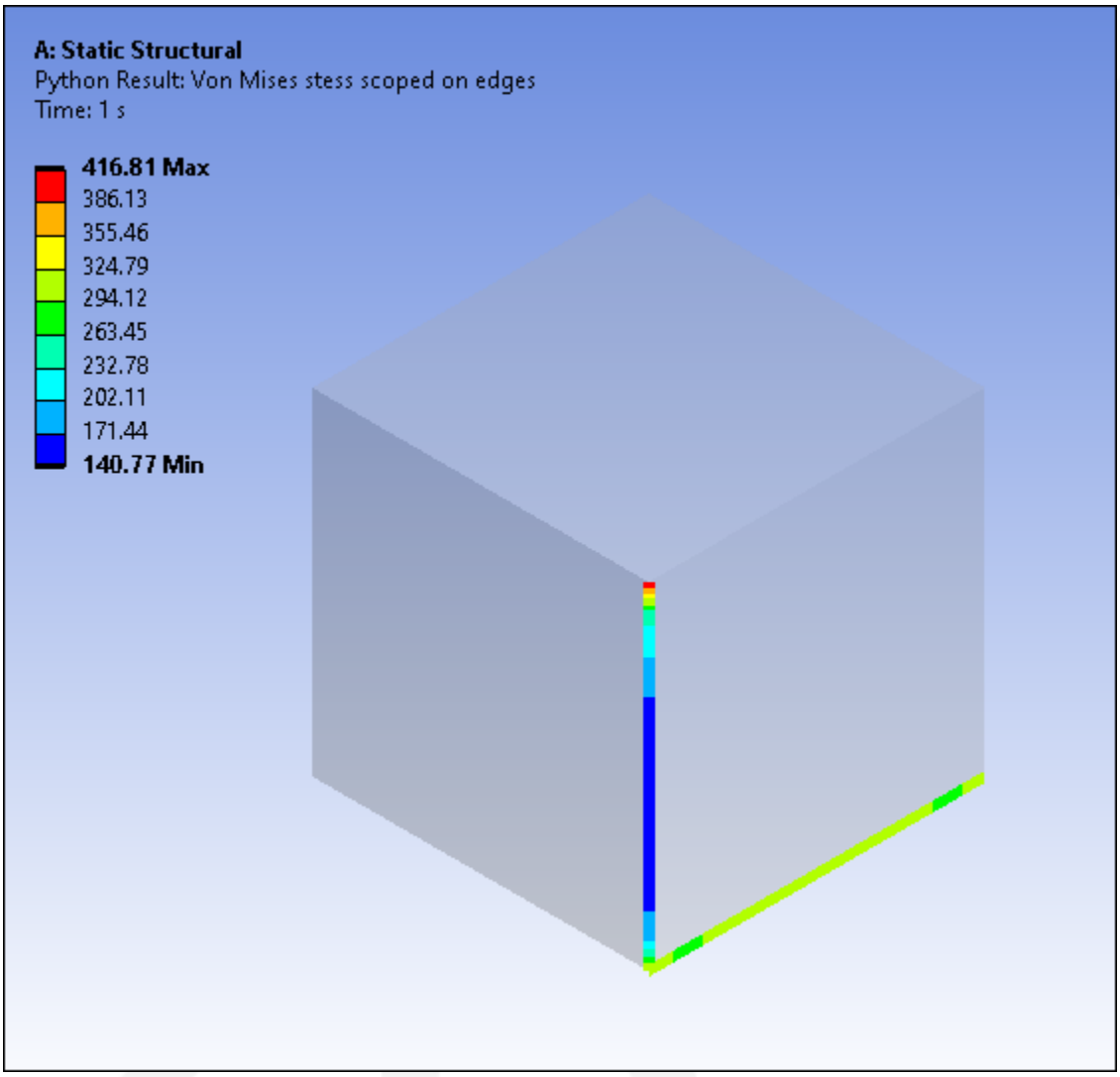
Four Vertices	`GetNodeScopingByRefId()`	`dpf.enums.GFXContour-Type.GeomVertexScoping`



Averaged Von Mises stress scoped on four vertices

Two Edges	`GetNodeScopingByRefId()`	`dpf.enums.GFXContour-Type.GeomEdgeScoping`

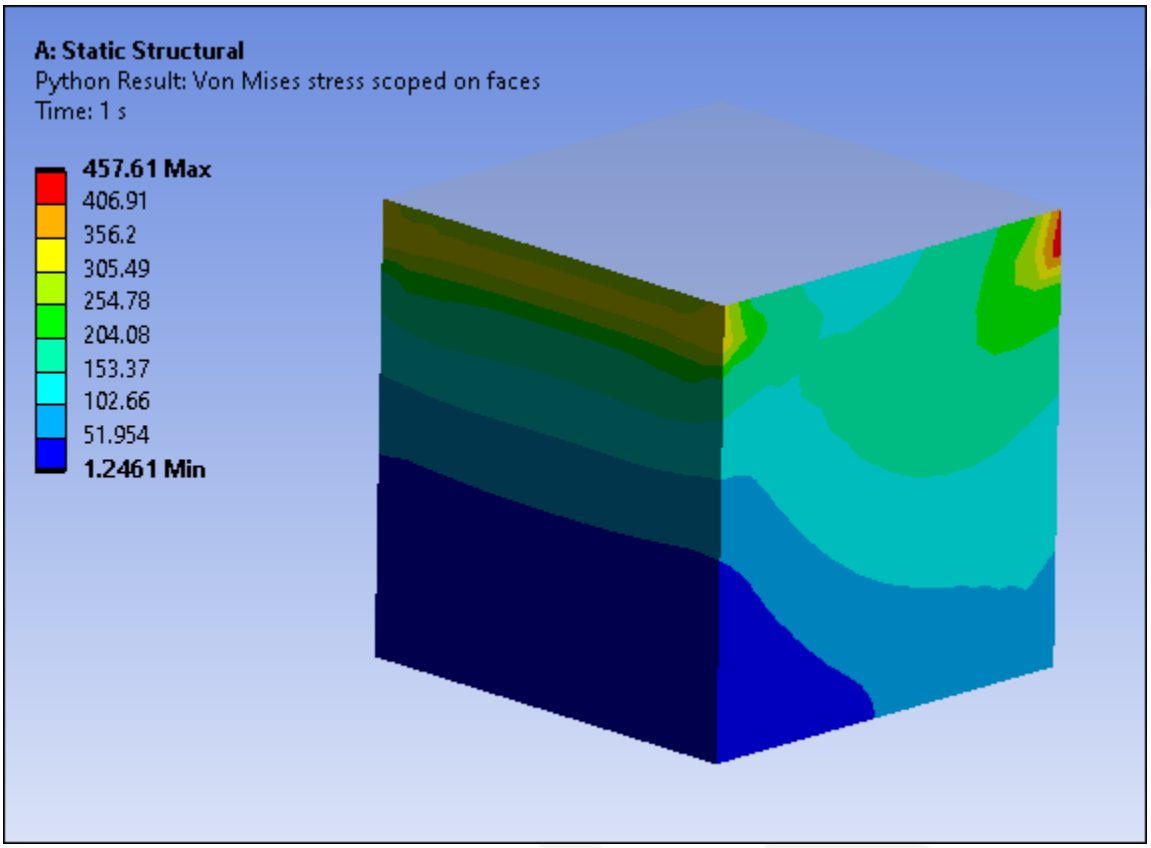
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Averaged Von Mises stress scoped on two edges.

Two Faces	`GetNodeScopingByRefId()`	`dpf.enums.GFXContour-Type.`
		
		`GeomFaceScoping`

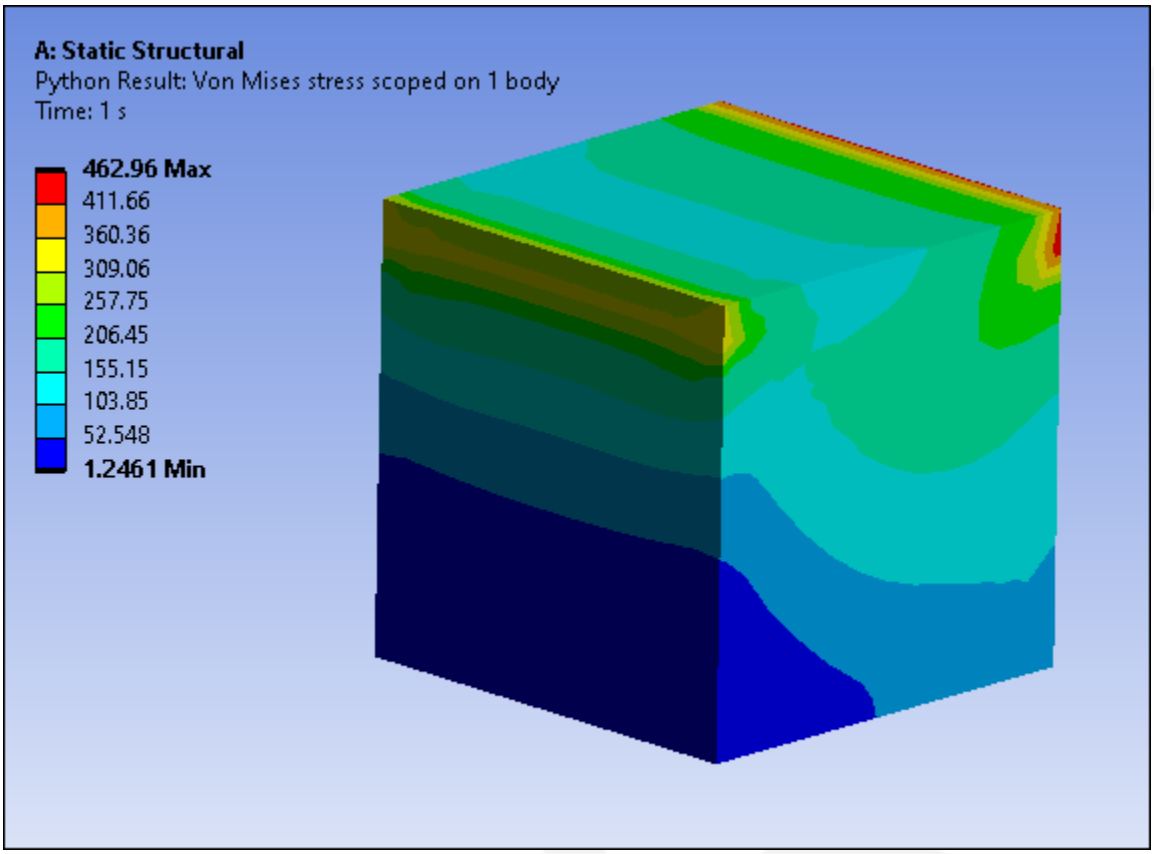
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Averaged Von Mises stress scoped on two faces.

One Body	`GetNodeScopingByRefId()`	`dpf.enums.GFXContour-Type.`
		
		`GeomBodyScoping`

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Averaged Von Mises stress scoped on 1 body

Notes

- Unaveraged elemental-nodal result such as Von Mises stress can be obtained by replacing the lines.

  Averaged Von Mises Stress

  ```
  scoping_refs = this.GetCustomPropertyByPath("Group 2/Scoping Property/Named Selection").Value
  mech_dpf.setExtAPI(ExtAPI)
  nodes_loc = mech_dpf.GetNodeScopingByRefId(scoping_refs)
  s_eqv_op.inputs.mesh_scoping.Connect(nodes_loc)
  ```

  With:

  Unaveraged Von Mises Stress

  ```
  scoping_refs = this.GetCustomPropertyByPath("Group 2/Scoping Property/Named Selection").Value
  mech_dpf.setExtAPI(ExtAPI)
  elems_loc = mech_dpf.GetElementScopingByRefId(scoping_refs)
  s_eqv_op.inputs.mesh_scoping.Connect(elems_loc)
  s_eqv_op.inputs.requested_location.Connect('ElementalNodal')
  ```

- However, contour display of unaveraged elemental-nodal result scoped on different entities besides Body may not be well supported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Migrating to New Python Result

Migrating from old Script Defined Result (2021R2 and older) to new Python Result

The older Script Defined Result object (from version 2020R2) will not work in the newer version of ANSYS Mechanical (2022R1 or beyond).

Old Script Defined Result from 2021R1 and older, you must manually insert new Python Result objects and recreate their scripts and workflows. You can either use the new templates to create a new script from scratch or simply copy the old script and paste it (with proper indentation) within the define_dpf_workflow(analysis) section (PART 3) of the new Python Result script. You also need to add this line at the end of your script within the section.

```
this.WorkflowId = wf.GetRecordedId()
```

Old Python Result from 2021R2, you must manually remove the numerical suffix from the auto generated method post_started(sender, analysis).

For example, if the first part of your script is:

```
def post_started_132(sender, analysis):# Do not edit this line
```

You need to change it to:

```
def post_started(sender, analysis):# Do not edit this line
```

To use the retrieving results from the table/chart feature, you also need to:

1. Include the callback function **`table_retrieve_result (value)`** provided in the default template, and:
2. Change the record workflow line from **`dpf_workflow.Record('wf_id', True)`** to **`dpf_workflow.Record('wf_id', False)`**.

Animation

To animate a **Python Result**, time scoping is needed in the script:

```
timeScop = dpf.Scoping()
timeScop.Ids = [1, 2, 3, 4, 5, 6, 7, 8, 9, 10]
u.inputs.time_scoping.Connect(timeScop)
```

In this example, **`timeScop.Ids`** represents all steps defined in the analysis. In this case, 10 steps are defined.

Note:

Ansys recommends that you clear and re-evaluate Python Results after resuming the project in Mechanical, otherwise the animation may fail.

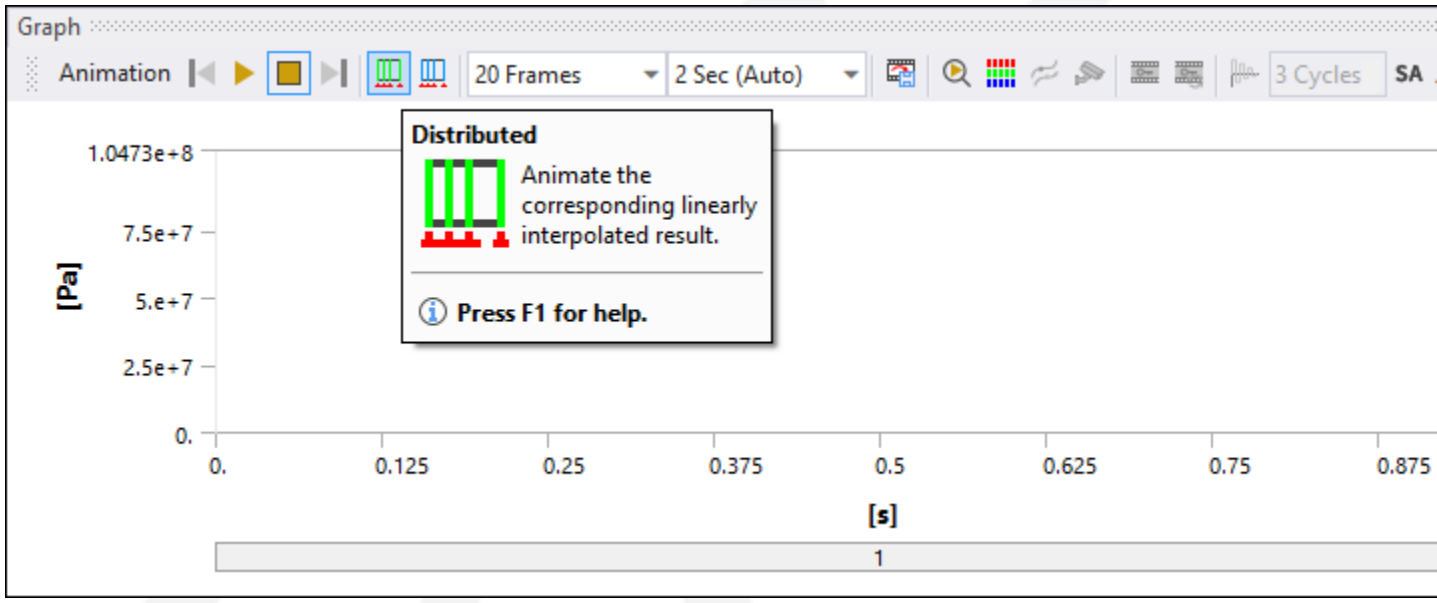
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Limitations

Note the following limitations for the animation capability:

- Animations of the **Python Result** is supported for Static Analysis, Transient Structural, Steady-State Thermal, and Thermal Transient analyses and Modal analyses when there is no complex result.
- Animation is only supported when all the available time steps are selected in the time scoping. Otherwise, the animation fails. For example, if you make the following entry: `timeScop.Ids` = [1, 2, 5, 6, 9, 10], the animation will fail.
- For the **Distributed** mode, the animation shows the last contour and deformation when t = 0.



User Defined Criteria

Overview

You can define User Defined Criteria using the **Primary Criterion** and **Composite Criterion** objects. These objects enable you to create scalar-based results on the mesh, or on the solution, of an upstream system. This permits you to quantify the performance of your current design. In addition, you can use the result data in a downstream Structural Optimization analysis as an Objective or as a Response Constraint. The User Defined Criterion options are available for the following:

- Measures Object (computes geometric quantities on the mesh)
- Harmonic Response Analysis (p. 496)
- Modal Analysis (p. 515)
- Static Structural Analysis (p. 652)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Depending upon your application (mesh or solution), the **Primary Criterion** object enables you to:

- Select a base result, such as mass, displacement, or reaction force.
- Set scoping options to generate the scalar value for an entity of interest (body, remote point, boundary condition, etc.).

Note:

Based on how you specify criterion, using the **Measures** object or from the solution of the upstream system, a prefix is added to the available options to indicate their source, either "Meas:" or the analysis designation, such as "A5," "B5," etc.

To create more complex criterion, you can combine multiple scalar criteria (**Primary Criterion** objects) into a single scalar value using the **Composite Criterion** object/option.

See the Understanding the Criterion Calculations (p. 2250) section for a description of the calculations used to obtain the criterion.

Application and Background

The following sections describe the steps to specify the **Primary Criterion**, how to combine your results (via **Composite Criterion** object) as well as the formulas used to calculate the criterion.

Define Primary Criterion for the Measures Object

Define Primary Criterion for a Harmonic Response Analysis

Define Primary Criterion for a Modal Analysis

Define Primary Criterion for a Static Structural Analysis

Combine Primary Criterion Objects

Understanding the Criterion Calculations

Define Primary Criterion for the Measures Object

To specify a Primary Criterion object under the Measures object:

1. To apply criteria to the mesh, first insert a Measures object from the Model Context tab (p. 48) or right-click the **Model** object and select **Insert** > **Measures**.

   Once the **Measures** object is inserted, the **Primary Criterion**/**Composite Criterion** options are available from the **Measures** Context tab, or by right-clicking the **Measures** object and selecting **Insert** > **Primary Criterion** or **Composite Criterion**.

2. Select the desired **Scoping Method**. Options include:

   - **Geometry** (default): Displays an associated Geometry selection property you use to select a geometric entity using the Vertex, Edge, Face, or Body selection options.
   - **Named Selection**: Displays an associated Named Selection property you use to select an available user–defined Named Selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

3. From the **Base Result** property, select the type of nodal vector field from which the extraction occurs. Property options for this analysis type include:

   - **Mass**
   - **Volume**
   - **Center of Gravity**
   - **Moment of Inertia**

4. As needed, change the **Coordinate System**. This property provides a drop-down list of available coordinate systems. The selected coordinate system must be Cartesian. The default setting is **Global Coordinate System**.

   Note:

   When the **Base Result** property is set to **Moment of Inertia**, this property also includes and is automatically set to the option **Coordinate System At Center Of Gravity**. Using this option means that the Moment of Inertia is computed with respect to the center of gravity of the scoped geometry.

5. Specify the component of interest using the **Reduction** property. Options include:

   - **X**, **Y** or **Z**: These options are available when the **Base Result** property is set to **Center of Gravity**.
   - **Ixx**, **Iyy**, **Izz**, **Ixy**, **Ixz**, **Iyz**: These options tensor component options are available when the **Base Result** property is set to **Moment of Inertia**.

   Note:

   Note the following for the **Component Reduction** category:

   - It is only available when the **Base Result** property is set to **Center of Gravity** or **Moment of Inertia**.
   - Point Mass, Distributed Mass, and Imported Condensed Parts are ignored.
   - Only supported for 3D Flexible Bodies.

6. When you solve, the read-only **Value** property displays the calculated mass, volume, center of gravity, or moment of inertia. You can parameterize (p. 2337) this value.

See the Understanding the Criterion Calculations (p. 2250) section for a description of the calculations used to obtain the criterion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Define Primary Criterion for a Harmonic Response Analysis

To specify a Primary Criterion object under the **Solution** object for a Harmonic Response analysis:

1. Select the **Solution** object and then select the **Primary Criterion** option from the Solution Context tab (p. 62), or right-click the **Solution** object and select **Insert** > **User Defined Criteria** > **Primary Criterion**.

2. Select the desired **Scoping Method**. Options include:

   - **Geometry** (default): Displays an associated **Geometry** selection property you use to select a geometric entity using the Vertex, Edge, Face, or Body selection options.

     Important:

     This scoping only supports 3D flexible bodies.

     Note:

     If specified, **Point Mass**, **Distributed Mass**, and **Imported Condensed Parts** are ignored.

   - **Named Selection**: Displays an associated **Named Selection** property you use to select an available user-defined Named Selection (p. 1057).

   - **Remote Point**: Displays an associated **Remote Point** property you use to select an available user-defined Remote Point (p. 1089).

   - **Boundary Condition**: Displays an associated **Boundary Condition** property you use to select either a Remote Force (p. 1552), Moment (p. 1574), or a Remote Displacement (p. 1674) (the three supported boundary conditions).

3. From the **Base Result** property, select the type of nodal vector field from which the extraction occurs. Property options for this analysis type include:

   - **Displacement**
   - **Velocity**
   - **Acceleration**

4. No action is required for the **Coordinate System** property. This property is read-only and set to **Global Coordinate System**.

5. Specify the component from which to extract node values using the **Component Reduction** property. Options include:

   - **X**/**Y**/**Z** (3D Only)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Magnitude**
- **Face Normal** (3D Face Selection Only)
- **Directional**

  If you specify **Directional**, the **Define By** property displays. Options for the property include:

  Component: When you select **Component**, the **X**/**Y**/**Z** (3D Only) **Component** properties display. Set the value of these properties as either 0 or 1 in order to define the direction(s) of interest. For example, specifying 1,0,0 for the components corresponds to a reduction in the X direction.

  Direction Selection: When you select **Direction Selection**, the **Direction** property displays. Use this property to specify a direction by selecting an entity on the model.

6. Using the **Spatial Reduction** property, specify a desired computation method to produce a scalar value from the vector field on multiple nodes for each frequency. Options include **Average** (default) and **Absolute Maximum**.
7. Specify the properties of the **Frequency Reduction** category. Properties include:
   - **Frequency Reduction**: This is a read-only property set to Average.
   - **Frequency Range**: You use this property to define a desired frequency range. This produces a scalar value from the vector field on multiple frequencies. Options include **Use Parent** (default) and **Manual**. The **Use Parent** option uses the entire frequency range of the analysis or you can use the manual option and specify a range. When you select the **Manual** option, the **Minimum Frequency** and **Maximum Frequency** properties display automatically so that you can enter values. These properties are read-only when you select the **Use Parent** option.
8. After you solve the analysis, the read-only properties, **Value** and **Discretization Error**, display the calculated displacement, velocity, or acceleration as well as an estimate of discretization error. The discretization error result enables you to consider the quality of the chosen frequency spacing. You can parameterize (p. 2337) this value.

See the Understanding the Criterion Calculations (p. 2250) section for a description of the calculations used to obtain the criterion.

Define Primary Criterion for a Modal Analysis

To specify a Primary Criterion object under the **Solution** object for a Modal analysis:

1. Select the **Solution** object and then select the **Primary Criteria** option from the Solution Context tab (p. 62), or right-click the **Solution** object and select **Insert** > **User Defined Criteria** > **Primary Criteria**.
2. Specify the setting of the **Base Result** property. Property options for this analysis type include:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Single Frequency** (default): Using this option, the application controls the i-th eigenfrequency that you have selected in the mode number. During the optimization the mode order may change, but the optimizer only focuses on the selected i-th eigenfrequency.
- **Robust Frequencies**: Using this option, the application controls the i-th eigenfrequency that you have selected in the mode number, but also takes into account the presence of the other eigenfrequencies per a weighted sum. The purpose of this option is to better manage the mode-crossing effect, that is, modes order can change drastically during the solution and as a result, cause the solver to prematurely converge and produce an undesirable design.

  When you select this option, the **Frequency Reduction** property displays. Options for this property include **Average** and **Smooth Minimum**.

3. Specify a **Mode Number** that corresponds to the eigenfrequency you wish to observe. The default setting is **1**. The entry range depends on the number of modes that you have requested in the upstream modal analysis, which has a default setting of **6**.
4. When you solve, the read-only **Value** property displays the calculated frequency or frequencies. You can parameterize (p. 2337) this value.

See the Understanding the Criterion Calculations (p. 2250) section for a description of the calculations used to obtain the criterion.

Define Primary Criterion for a Static Structural Analysis

To specify a Primary Criterion object under the **Solution** object for a Static Structural analysis:

1. Select the **Solution** object and then select the **Primary Criterion** option from the Solution Context tab (p. 62), or right-click the **Solution** object and select **Insert** > **User Defined Criteria** > **Primary Criterion**.
2. Select the desired **Scoping Method**. Options include:
   - **Geometry** (default): Displays an associated **Geometry** selection property you use to select a geometric entity using the Vertex, Edge, Face, or Body selection options.

     Important:

     This scoping only supports 3D flexible bodies.

     Note:

     If specified, **Point Mass**, **Distributed Mass**, and **Imported Condensed Parts** are ignored.

   - **Named Selection**: Displays an associated **Named Selection** property you use to select an available user–defined Named Selection (p. 1057).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Remote Point**: Displays an associated **Remote Point** property you use to select an available user–defined Remote Point (p. 1089).
- **Boundary Condition**: Displays an associated **Boundary Condition** property you use to select either a Remote Force (p. 1552), Moment (p. 1574), or a Remote Displacement (p. 1674) (the three supported boundary conditions).

3. From the **Base Result** property, select the type of nodal vector field from which the extraction occurs. Property options for this analysis type include:
   - **Displacement**
   - **Rotation**
   - **Reaction Force**
   - **Reaction Moment**
4. In the **Load Step Selection** category, specify a desired Load Step (p. 1418) for which to compute the criterion using the **Step** property.
5. Specify the component from which to extract node values using the **Component Reduction** property. Options include:
   - **X**/**Y**/**Z** (3D Only)
   - **Magnitude**
   - **Face Normal** (3D Face Selection Only)
   - **Directional**

     If you specify **Directional**, the **Define By** property displays. Options for the property include:

     Component: When you select **Component**, the **X**/**Y**/**Z** (3D Only) **Component** properties display. Set the value of these properties as either 0 or 1 in order to define the direction(s) of interest. For example, specifying 1,0,0 for the components corresponds to a reduction in the X direction.

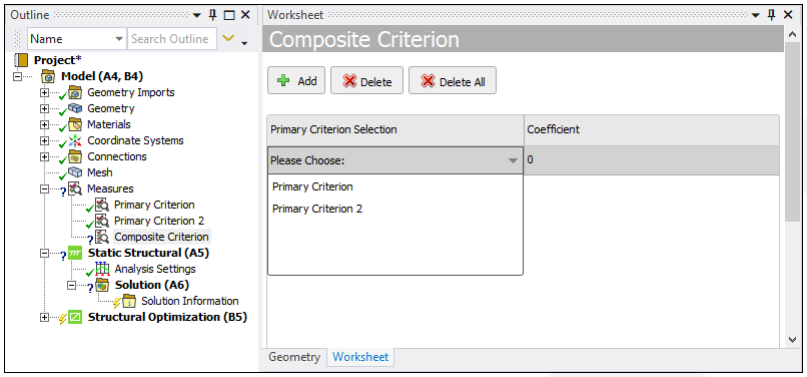
     Direction Selection: When you select Direction Selection, the Direction property displays. Use this property to specify a direction by selecting an entity on the model.
6. Using the **Spatial Reduction** property, specify a desired computation method to produce a scalar value from the vector field on multiple nodes. Options include **Average** (default) and **Absolute Maximum** for the **Displacement** and **Rotation** options of the **Base Result** property and **Sum** for the **Reaction Force** and **Reaction Moment** options.
7. The **Method** property currently supports only one option: **Discrete** (default).
8. When you solve, the read-only **Value** property displays the calculated displacement, rotation, force, or torque. You can parameterize (p. 2337) this value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

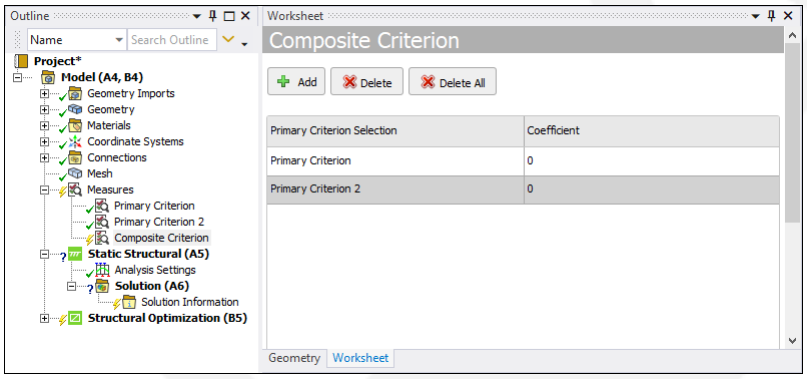
See the Understanding the Criterion Calculations (p. 2250) section for a description of the calculations used to obtain the criterion.

Combine Primary Criterion Objects

For one or more defined **Primary Criterion** objects, you can use the **Composite Criterion** option. When inserted, the **Worksheet** displays automatically. Using the table and its options, add the **Primary Criterion** objects you wish to combine. You need to add a row for each object. As needed, specify **Coefficient** values for your entries.



Once solved, the **Composite Criterion** result is displayed by the **Value** property.



Understanding the Criterion Calculations

The following topics describe the calculations for the methods and base results as well as the Composite Criterion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Measures Object

Mass

The criterion calculation:

$$Crit = M_{tot} = \sum_k M_k$$

Given (ρ), the volumetric mass of the body, and (Ω), the mass formula of the k-th body reads (continuous and discretized):

$$M_k = \int_{\Omega_k} \rho.dv = \sum_{e \subset \Omega_k} \rho_e V_e$$

Where V_e and ρ_e are the volume and resp. the volumetric mass of the e-th element.

Note:

The volume criterion uses the same formula with $\rho \equiv 1$.

In the context of topology optimization, the formula reads:

$$M_k = \int_{\Omega_k} \rho.dv = \sum_{e \subset \Omega_k} \alpha_e \rho_e V_e$$

Where α_e is the density parameter of the e-th element. This is a unit-less multiplier with a range of 0.0 to 1.0.

Center of Gravity

For Center of Gravity, you can choose the component of interest (x, y or z).

Given X = (x, y, z), a material point within the body, the formula reads (continuous):

$$CoG_{(0)} = \left(\frac{\sum_k \int_{\Omega_k} X.\rho dv}{\sum_k \int_{\Omega_k} \rho dv} \right) = \left(\frac{\sum_k N_k}{\sum_k M_k} \right)$$

The sum over the k-th body of interest.

After discretization:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$N_k = \int_{\Omega_k} X.\rho dv = \begin{pmatrix} \int_{\Omega_k} x.\rho dv \\ \int_{\Omega_k} y.\rho dv \\ \int_{\Omega_k} z.\rho dv \end{pmatrix} = \begin{pmatrix} \sum_{\in \subset \Omega_k} x_e \rho_e V_e \\ \sum_{\in \subset \Omega_k} y_e \rho_e V_e \\ \sum_{\in \subset \Omega_k} z_e \rho_e V_e \end{pmatrix}$$

Where:

$X_e = (x_e, y_e, z_e)$ is the centroid of the e-th element.

other quantities are defined above

Notes:

$CoG_{(0)}$ is given with respect to the global coordinate system.

for a coordinate system: $CoG_{(CSys)} = P\left(Crit_{(0)} - Origin_{(CSys)}\right)$ where P is the rotation matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Moment of Inertia

The Moment of Inertia is by nature a tensor. You can choose the component of interest (Ixx, Iyy, Izz, Ixy, Ixz, Iyz).

The full tensor with respect to the global coordinate system reads (continuous formula):

$$I_{(0)} = \begin{pmatrix} I_{xx} & I_{xy} & I_{xz} \\ I_{xy} & I_{yy} & I_{yz} \\ I_{xz} & I_{yz} & I_{zz} \end{pmatrix} = \begin{pmatrix} +\sum_k \int_{\Omega_k} (y^2 + z^2).\rho dv & -\sum_k \int_{\Omega_k} (xy).\rho dv & -\sum_k \int_{\Omega_k} (xz).\rho dv \\ -\sum_k \int_{\Omega_k} (xy).\rho dv & +\sum_k \int_{\Omega_k} (x^2 + z^2).\rho dv & -\sum_k \int_{\Omega_k} (yz).\rho dv \\ -\sum_k \int_{\Omega_k} (xz).\rho dv & -\sum_k \int_{\Omega_k} (yz).\rho dv & +\sum_k \int_{\Omega_k} (x^2 + y^2).\rho dv \end{pmatrix}$$

After discretization:

$$\begin{cases} I_{xx} = \sum_k \int_{\Omega_k} (y^2 + z^2).\rho dv \triangleq \sum_k \left[\sum_{e \in \Omega_k} (y_e^2 + z_e^2).\rho_e V_e \right] \\ I_{xy} = \sum_k \int_{\Omega_k} (xy).\rho dv \triangleq \sum_k \left[\sum_{e \in \Omega_k} (x_e y_e).\rho_e V_e \right] \\ \ldots \end{cases}$$

Note:

$I_{(0)}$ is given with respect to the global coordinate system (origin = 0).

With respect to the Center of Gravity (G) the formula now reads:

$$I_{(CSys)} = I_{(0)} - M_{tot}\left(\left\| \overrightarrow{OG} \right\| \bar{\bar{I}} - \overrightarrow{OG} \otimes \overrightarrow{OG} \right)$$

Where M_{tot} is the total mass of the body scoping.

And $\bar{\bar{I}} = \begin{pmatrix} 1 & & \\ & 1 & \\ & & 1 \end{pmatrix}$ is the 3 x 3 identity matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Harmonic Response Analysis

Primary Criteria

The goal of the **User Defined Criterion** feature is to define a singular scalar value that measures the mechanical performance of a design as a result of the following three-step reduction process:

1. Component reduction that converts a base-result (namely a complex-valued spatial vector-field, defined for every excitation frequency) onto a spatial scalar-field defined for every excitation frequency.
2. Spatial reduction that transforms the component-reduced field into a singular scalar value defined for every excitation frequency.
3. Frequency reduction that aggregates the spatially-reduced field into a singular scalar value.

Here is an example of the computation process for a displacement base result:

Component reduction returns a spatial scalar field, defined for every excitation frequency:

$k_n(\omega)$

Spatial reduction returns a singular value, defined for every excitation frequency:

$$Crit(\omega) = \frac{1}{\left(\int_{\Gamma_{scoping}} dv\right)} \int_{\Gamma_{scoping}} k_n(\omega).dv$$

After discretization, $Crit(\omega)$ generally reads:

$$Crit(\omega) \triangleq \frac{1}{\left(\sum_{n \in NodeSet} w_n\right)} \left(\sum_{n \in NodeSet} w_n k_n(\omega)\right)$$

Frequency reduction returns a singular scalar value, the criterion:

$$Crit = \frac{1}{\int_{\omega_{\min}}^{\omega_{\max}} d\omega} \int_{\omega_{\min}}^{\omega_{\max}} Crit(\omega).d\omega$$

After discretization, the criterion reads:

$$Crit = \frac{1}{\omega_{\max} - \omega_{\min}} \sum_{j \in FrequencySet} Crit(\omega_j).d\omega_j$$

where:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

NodeSet is the set of nodes defined by the scoping.

FrequencySet is the set of frequencies defined by the Frequency Reduction.

$[\omega_{\min}, \omega_{\max}]$ is the Frequency Range of the criterion:

$\begin{cases} \text{If the Frequency Range property is set to Use Parent, then the frequency range of the Harmonic R} \\ \text{If the Frequency Range property is set to Manual, you can set a subinterval of the Harmonic Resp} \end{cases}$

$\omega \in [\omega_{\min}, \omega_{\max}]$ is the excitation frequency.

$k_n(\omega)$ is the quantity obtained by component reduction.

Considering the displacement as a base result, its complex harmonic amplitude $u_n(\omega)$ is a complex-valued vector field that reads:

$$u_n(\omega) = \begin{bmatrix} u_{n,x}(\omega) \\ u_{n,y}(\omega) \\ u_{n,z}(\omega) \end{bmatrix} = \begin{bmatrix} \mathrm{Re}(u_{n,x}(\omega)) \\ \mathrm{Re}(u_{n,y}(\omega)) \\ \mathrm{Re}(u_{n,z}(\omega)) \end{bmatrix} + i \begin{bmatrix} \mathrm{Im}(u_{n,x}(\omega)) \\ \mathrm{Im}(u_{n,y}(\omega)) \\ \mathrm{Im}(u_{n,z}(\omega)) \end{bmatrix}$$

Then the different component reductions are:

$$\begin{cases} \text{if [Component Reduction=X], then } \mathrm{k}_n(\omega) = \|u_{n,x}(\omega)\| = \sqrt{\mathrm{Re}(u_{n,x}(\omega))^2 + \mathrm{Im}(u_{n,x}(\omega))^2} = \text{harmoni} \\ \text{if [Component Reduction=Y], then } \mathrm{k}_n(\omega) = \|u_{n,y}(\omega)\| = \sqrt{\mathrm{Re}(u_{n,y}(\omega))^2 + \mathrm{Im}(u_{n,y}(\omega))^2} = \text{harmon} \\ \text{if [Component Reduction=Z], then } \mathrm{k}_n(\omega) = \|u_{n,z}(\omega)\| = \sqrt{\mathrm{Re}(u_{n,z}(\omega))^2 + \mathrm{Im}(u_{n,z}(\omega))^2} = \text{harmoni} \\ \text{if [Component Reduction=magnitude], then } \mathrm{k}_n(\omega) = \sqrt{\|u_{n,x}(\omega)\|^2 + \|u_{n,y}(\omega)\|^2 + \|u_{n,z}(\omega)\|^2} = \text{har} \\ \text{if [Component Reduction=Normal Face], then } \mathrm{k}_n(\omega) = (\|u_{n,x}(\omega)\|.n_x + \|u_{n,y}(\omega)\|.n_y + \|u_{n,z}(\omega)\| \end{cases}$$

w_n is the weight of each node of the *NodeSet*.

This parameter is related to the following **Spatial Reduction** when the property is set to **Average**:

$\begin{cases} \text{if [Spatial Reduction=Average, and Method=Discrete], then } w_n = 1, \text{ same weight for each node.} \\ \text{if [Spatial Reduction=Average, and Method=Continuous], then } w_n = d\Gamma_n, \text{ the volume of influence} \end{cases}$

(Based on scoping, $d\Gamma_n$ can be the volume of influence, the surface of influence or the length of influence).

When the **Spatial Reduction** property is set to **Absolute Maximum**, the criterion is defined as:

$$Crit(\omega) = \max_{n \in NodeSet} |k_n(\omega)|$$

However, in order for this calculation to be derivable, the **max** operator is approximated. The calculation uses the absolute values of the result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The **Frequency Reduction** property is a read-only property set to **Average**.

Note:

- The frequency reduction process naturally introduces discretization error in the criterion evaluation. The application calculates an estimate for this error. If it exceeds 10% of the criterion value, then a corresponding message appears in the Optimization Output (p. 1836).
- If the calculated displacement, velocity, or acceleration presented in the **Value** property is close to or equals zero, then the **Discretization Error** property is always set to 0%.

Modal Analysis

Primary Criterion

For the **Robust Frequencies** setting, the criterion reads:

$$J(\Omega) = \frac{1}{\sum_{i \geq i_0} w_i} \left(\sum_{i \geq i_0} w_i f_i \right)$$

$$w_i \equiv w_i\left(f_i\right)$$

Where:

- i_0 is the selected mode of interest (**Mode** property).

For the Average reduction frequency, the weight is the same for all frequencies, that is: $w_i \equiv 1$

For the **Smooth Minimum** setting, the weight is given by:

$$w_i \equiv \exp\left(-10. \left(\frac{f_i - f_0}{f_0} \right) \right)$$

This permits to approximate the min value while staying derivable.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Static Structural Analysis

Primary Criterion

The goal of the User Defined Criterion feature is to define a singular scalar value that measures the mechanical performance of a design as a result of the following two-step reduction process:

1. Component Reduction that converts a base-result (namely a spatial vector-field) onto a spatial scalar-field.
2. Spatial Reduction that transforms the spatial scalar-field onto a singular scalar value.

Here is an example of the computation process for a displacement base result:

- Component reduction returns a spatial scalar field:

  $\left(k_n^i\right)$

- Spatial reduction returns a singular value, the criterion (Crit):

$$Crit = \frac{1}{\left(\int_{\Gamma_{scoping}} dv\right)} \int_{\Gamma_{scoping}} k_n^i . dv$$

After discretization, the criterion generally reads:

$$Crit = \frac{1}{\left(\sum_{n \in NodeSet} w_n\right)} \left(\sum_{n \in NodeSet} w_n k_n^i\right)$$

Where:

- *NodeSet* is the set of nodes defined by the scoping.
- *i* is the specified load step (**Current Step Number** property).
- $\left(k_n^i\right)$ is the quantity obtained by component reduction.

Considering the displacement as a base result, the different component reductions are:

$$\begin{cases} \text{if [component Reduction X], then } k_n^i = u_{n,x}^i = \text{x displacement of the n-th node, i-th load step.} \\ \text{if [component Reduction X], then } k_n^i = u_{n,x}^i = y \text{ displacement of the n-th node, i-th load step.} \\ \text{if [component Reduction X], then } k_n^i = u_{n,x}^i = z \text{ displacement of the n-th node, i-th load step.} \\ \text{if [component Reduction X], then } k_n^i = u_{n,x}^i = \text{magnitude displacement of the n-th node, i-th load step.} \\ \text{if [component Reduction X], then } k_n^i = u_{n,x}^i = \text{displacement along the outward normal of the n-th node, i-} \end{cases}$$

w_n is the weight of each node of the NodeSet. This parameter is related to the following **Spatial Reduction** when the property is set to **Average**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

$$\begin{cases} \text{if [Spatial Reduction = Average and Method = Descrete] then } w_n = 1, \text{ the same weight for each node.} \\ \text{if [Spatial Reduction = Average and Method = Continuous] then } w_n = d\Gamma_n, \text{ the volume of influence of th} \end{cases}$$

(Based on scoping, $d\Gamma_n$, can be the volume of influence, the surface of influence, or the length of influence).

When the **Spatial Reduction** property set to **Absolute Maximum**, the Criterion is defined as:

$$Crit = \max_{n \in NodeSet} \left| k_n^i \right|$$

However, in order for this calculation be derived, the maximum (max) criterion is approximated. The calculation uses the absolute values of the result.

Composite Criterion

Using the **Composite Criterion** object, you combine several primary criteria into a single scalar value. The composite criteria is computed as a weighted sum of Primary Criteria.

$$Crit = \sum_i \omega_i C_i$$

Where:

$Crit$ is the composite criterion value.

ω_i, C_i are respectively the weights and primary criteria values.

Result Utilities

The following topics related to result utilities are covered in this section.

Automatic Result Creation for All Result Sets
Adaptive Convergence
Animation
Capped IsoSurfaces
Dynamic Legend
Exporting Results
Generating Reports
Local Minimum and Maximum Probes
Renaming Results Based on Definition
Results Legend
Results Tab
Solution Combination

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Automatic Result Creation for All Result Sets

For a solved result in the tree **Outline** the contextual (right-click) menu **Create Results at All Sets** enables you to automatically create one or more results, that are grouped, for all of the available result sets for the selected result item. The associated result sets are read from the solver generated result file.

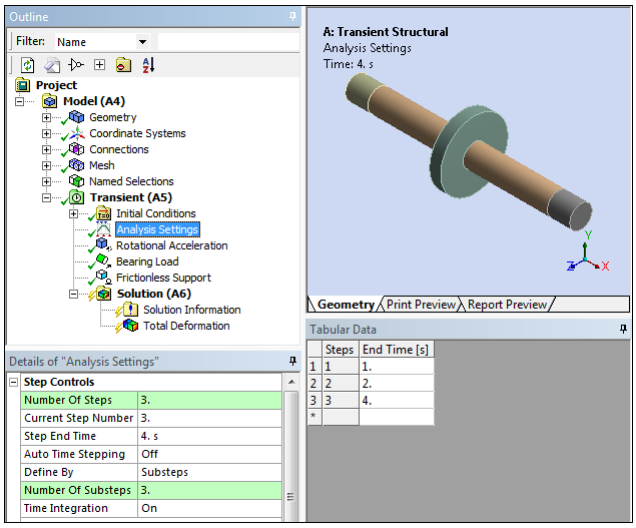
Note:

This feature is not supported for:

- Explicit Dynamics, Response Spectrum, Random Vibration, or Topology Optimization analyses
- Probe Results
- Fracture Tool
- Fatigue Tool
- Composite Failure Tool

Example Application

The following example illustrates the process for using this feature. This example includes multiple load steps, as shown in the first image.



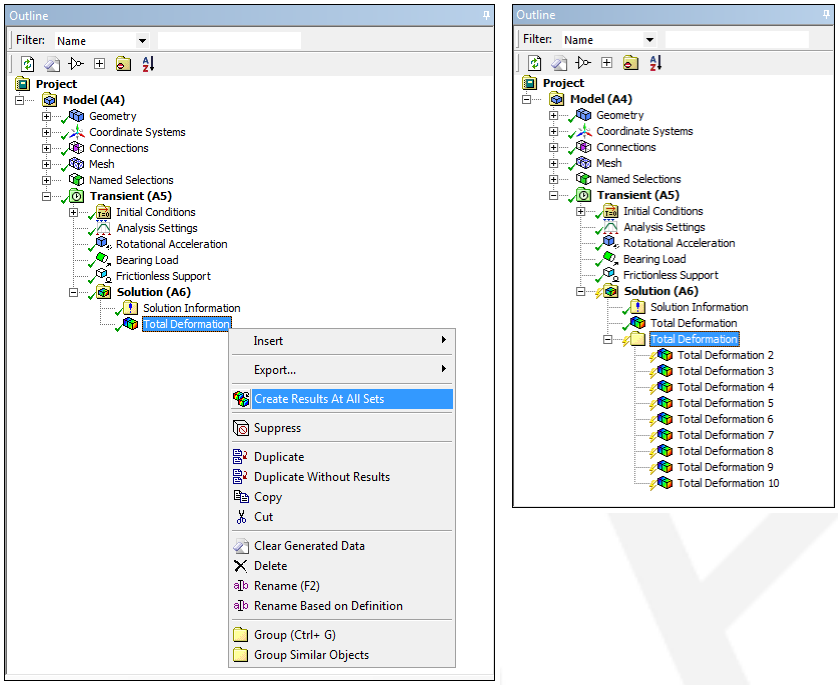
Once solved, the **Tabular Data** is as follows.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Tabular Data

	Time [s]	Minimum [mm]	Maximum [mm]
1	0.33333	1.4091	70.417
2	0.66667	4.9381	246.74
3	1.	6.9464	347.08
4	1.3333	4.2822	213.98
5	1.6667	0.58192	29.098
6	2.	1.4924	74.59
7	2.6667	5.5037	275.02
8	3.3333	1.5145e-002	0.76495
9	4.	3.7565	187.7

You select a desired result, right-click, and select **Create Results at All Sets**. The application automatically creates a result for each time point.



Adaptive Convergence

See the Using Adaptive Convergence (p. 1852) topic in the *Understanding Solving* section of the Ansys Mechanical User's Guide.

Animation

Select a desired topic:

Animation Options

Animation Behaviors

Animation with Attached Camera

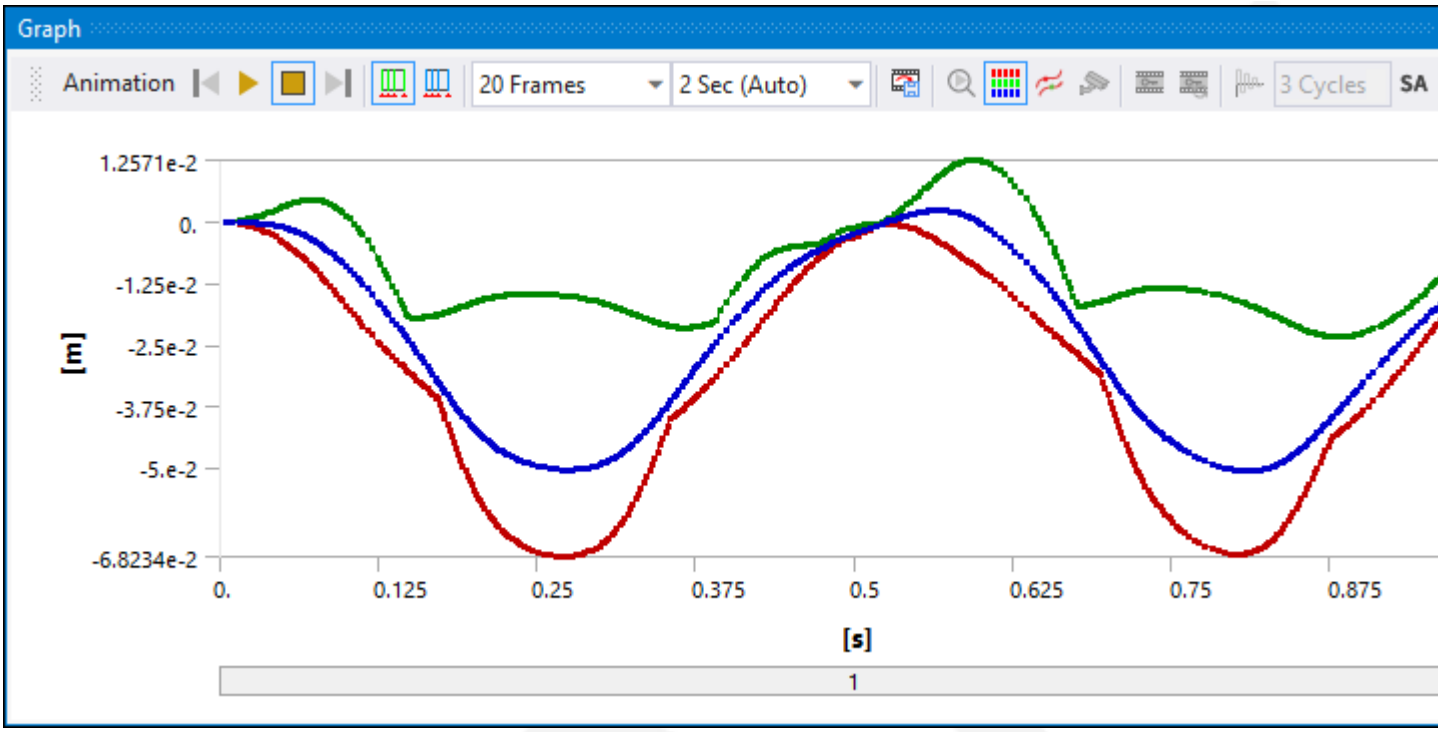
Accelerated Animation

Animation Options

The **Animation** feature displays in the **Graph** window when you select a result object in the Mechanical application. Here is an example of the **Graph** window with a result object selected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

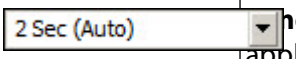
Published: 2023-10-23



The specific animation features are presented below.

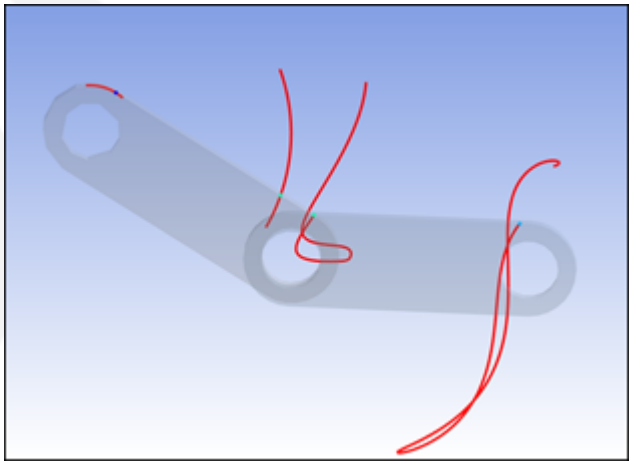
Option/Control	Description
	Play: Initiates a new animation. For the **Standard Animation (SA)** setting, the application becomes locked while the animation frames are generated. You cannot perform any actions or make any selections. However, you can abort the generation process using the **[ESC]** key. This unlocks the application and stops the generation process. Any generated frames continue to animate. For the **Accelerated Animation (AA)** setting, the **[ESC]** key does not apply and the application does not lock so you can both pause and/or stop the animation during the generation process. **Note:** Additionally, for the **Standard Animation (SA)** setting, you can change the default display behavior by setting the **Animation Draw Option** of the Graphics (p. 228) category of the **Options** dialog to **No**. Using this optional display behavior, the application produces the animation in its entirety before presenting it. With this option active, the application does not support the abort action (**[ESC]** key).
	Pause: Pauses an existing animation. Choosing **Play** after **Pause** does *not* generate new animation frames. When the animation is paused, as you move

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option/Control	Description
(same toolbar location as **Play**)	the cursor across the graph, the cursor's appearance changes to a double horizontal arrow when you hover over the current frame indicator. With the cursor in this state, you can drag the frame indicator to define a new current frame. The result graphic will update accordingly.
	Stop: Halts a result animation. Choosing **Play** after **Stop** generates new animation frames.
	Previous Frame and **Next Frame**: These two buttons reverse or advance the animation frame by frame.
	Distributed: For static analyses, frames display linearly interpolated results. Frame 1 represents the initial state of the model and the final frame represents the final results calculated by the solver. For stepped and transient analyses, the frames in **Distributed** mode are distributed over a time range selected in the graph.[a]
	Result Sets: (available only for stepped and transient simulations) Frames represent the actual result sets that were generated by the solver.[a]
	mes: Select the number of frames in the animation. This setting applies the display of your animation and to any exported animations, except the GIF file format.
	**e**: Select the desired amount of time for the entire animation. This setting applies the display of your animation and to any exported animations, except the GIF file format.
	Export Video File: This option enables you to save your animation. Supported video file types include AVI, MP4, WMV, and GIF files. You can specify a desired number of frames and time when exporting an animation using the **Frames** and **Time** options above. The default settings for **Frames** and **Time** are **20** and **2**, respectively. This gives you a default frame rate of 10 frames per second. Note that if you specify a high number of frames with a low **Time** setting, and therefore a very rapid frame rate, the quality of the exported animation is affected. **Note:** • When exporting, make sure that you keep the Mechanical module window in front of other windows until the exporting is complete. Opening other windows in front of the module window before the exporting is complete may cause those windows to be included in the file capture. • The GIF file format does not support the Workbench Gradient settings (the default setting). For this file format, the application automatically changes the Workbench appearance setting to Uniform. As a result, exported GIF files have a plain background compared to exported videos.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option/Control	Description
	• You can use the **Options** dialog to enable the ability to export AVI animation files using Microsoft Windows API on the Windows platform. See the description for the **Animation Export** category in the Graphics (p. 228) topic of the Specifying Options (p. 217) section for more information.
	Zoom to Fit Animation: When turned on, Mechanical loops through all the time steps to compute an auto scale factor that will accommodate the displacement for a full range of time steps and makes sure that the animation fits properly (is scaled) within the Geometry window. For more information on auto scale factors, see the Result Context Tab (p. 62) topic in the Context Tab section. When turned off, Mechanical uses the current result scale factor for animation. While this option results in increased animation speed, you may experience some distortion if the current display time has a displacement that is much smaller than the peak displacement through which the animation occurs.
	Update Contour Range at Each Animation Frame: By default, the animation feature displays the global range of result values over all of the available frames. When you select this option, you animate the result on a frame by frame basis. That is, the animation changes so that at each frame, the entire range of result values for that frame (only) is used. Therefore, for each frame, the application displays the minimum and maximum values for the result, at that frame, for the contour values of the legend and for the contour result. As a result, the legend dynamically changes from frame to frame and the result contours always display the full range of colors from blue (or the color representing the minimum value on the legend) to red (or the color representing the maximum value on the legend).
	This option is only available for 1) Total Deformation and Direction Deformation results that are 2) scoped to one more nodes during 3) either a Static Structural or a Transient structural analysis that includes at least two result sets. When enabled, your animation will include traces of the deformations based on the selected nodes. 

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option/Control	Description
	Animation with Attached Camera: Visualization technique that makes a selected rigid body appear to be fixed during the animation with all other bodies moving relative to it. See the usage steps in the Animation with Attached Camera (p. 2266) topic below.
	This **Keyframe Animation** option becomes available once you have opened the Keyframe Animation (p. 255) window using the option on the Graphics toolbar (p. 97). When selected, your result animates according to the defined keyframes. The feature requires that you have defined at least two keyframes. Selecting the check box beside this option, which activates by default when the you select the keyframe option in Animation tool, synchronizes the frames of your original animation to your defined keyframes.
	Damped Modal Animation: This option turns on time decay animation of complex modes for a Modal analysis (p. 515) with applied damping. This button is not available (grayed out) for any of the following: • Any analysis type other than modal. • Any modal analysis whose **Damped** setting (under **Solver Controls** (p. 1428)) is set to **No**. • Any modal analysis whose **Damped** setting is set to **Yes**, and whose **Solver Type** is set to **Reduced Damped**, and **Store Complex Solution** is set to **No**. Set the number of cycles to be used in the time decay animation in the adjacent **Cycles** property.
	This option is for display only. It indicates the currently selected animation mode. Options include either **SA** for Standard Animation or **AA** for Accelerated Animation. See the Accelerated Animation (p. 2268) section for more information.

[a] For stepped and transient simulations, as you move the cursor across the graph, the cursor's appearance changes to a scope icon for solved solution points.

Beta Option: Currently, there is a beta capability available for exporting a video of a result animation.

Animation Behaviors

Depending upon the type of simulation that you perform, the behavior of the resulting animation varies.

For a static analysis with one time step or a Modal analysis, the animation progresses forward and then backwards in a continuous motion.

For simulations that have more than one associated time or step range, the animation begins at the initial time or step value, progresses to the final set, and then stops and starts at zero again (forward animation only).

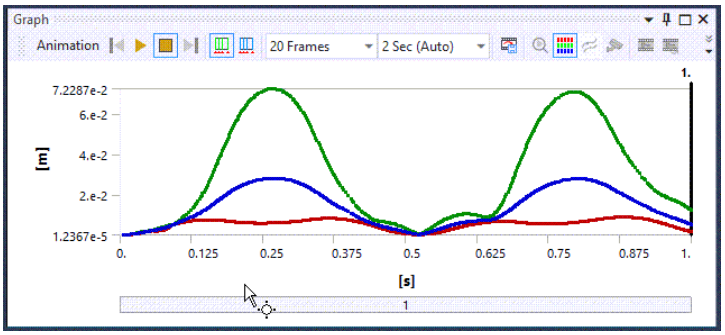
For transient and stepped simulations that have an associated time or step range, the animation begins at the initial time or step value, progresses to the final set, and then stops and starts at zero again. It does not traverse backward as it does for static simulations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

As illustrated below, you may also select a specific time period to animate that is a subset of the total time. To do so, drag the mouse through the time period in the graph. The selected time period turns blue. Click the **Play** button to animate only through that period. While that specific period is playing, you can right-click the mouse to receive the options to **Pause**, **Stop**, or to **Zoom To Range**, which expands the defined period across the entire graph.

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.



Note:

- In older releases, the application first processed result animations and then displayed them instead of displaying the animation as it is being processed. As needed, you can revert to the previous display method by setting the **Animation Draw Option** to **No** in the Graphics (p. 228) category of the preferences dialog.
- In the event your result file includes a result set with no values, the animation ignores that result set.
- In a dynamic analysis, probe animation for joints is only supported if there is at least one rigid body. See Probes (p. 1946).
- For a Modal analysis, rigid bodies are not currently auto-scaled during animations and as a result may not display accurately.
- When animating mode shapes with color contours, be aware that the shapes are animated by scaling the displacements from 1 to 0 to -1 to 0 to 1 in a full cycle, but the corresponding contours are animated by scaling their values from 1 to 0 to 1 to 0 to 1 over the same cycle. You can correct this display limitation by setting a variable (p. 247) (**`TreatModalAsComplex`** = **`1`**)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Animation with Attached Camera

The attached camera allows the animation of rigid bodies in such a way that the selected body appears not to move during the animation and all others move relative to it. The animation with the attached camera is strictly a visualization technique and not a post processing computation.

Note:

The attached camera can only be used for rigid bodies in a Rigid Dynamics, Static or Transient Structural analyses, and bodies that have a dimensionally reduced mesh in Explicit Dynamics analyses.

To use the attached camera:

1. Click the result in the Outline that you want to animate.
2. Click the attached camera icon in the animation toolbar to enable the feature. The icon will remain outlined when enabled.
3. The body picking filter will be selected by default when the attached camera is active. Pick the body that you want to attach the camera to (the selected body will turn green).
4. At this point, either you can click the attached camera icon in the mini toolbar, or right-click the body and choose Select Body for Attached Camera from the menu. The body will turn pink.

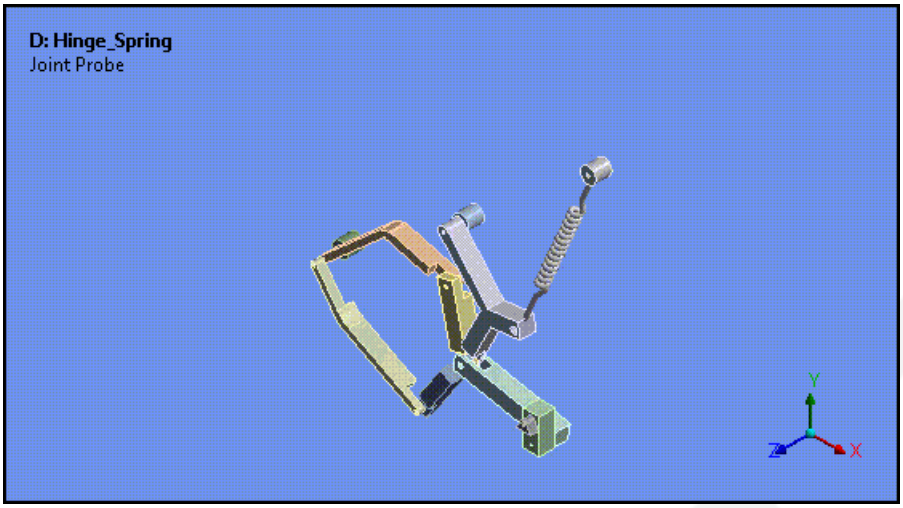
   Note:

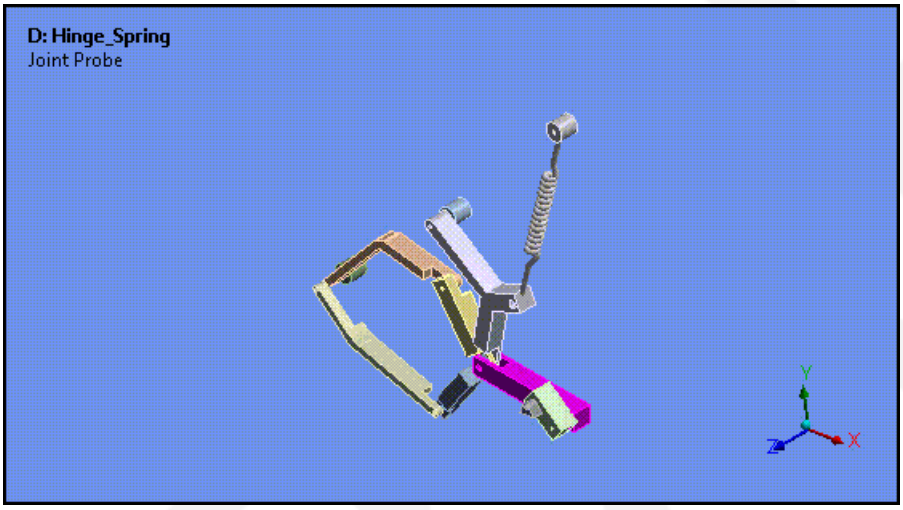
   These two options will only appear if the selected body is a valid candidate for the attached camera animation.

5. Reorient the model in the graphics window as you want to see it during the animation.
6. Play the animation. The selected body (pink) appears fixed. All other bodies move relative to it. Compare below the regular animation (top) and the attached camera animation (bottom).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





7. To unselect the body you attached the camera to, click the selected body and click the attached camera icon in the mini toolbar, or right-click the body and choose Select Body for Attached Camera from the menu.

8. To deactivate the attached camera, click the icon again (the outline will disappear when it is disabled).

Note:

Limitations:

- When you try to pick a body in a result, the transient highlighting will follow the original geometry. This may not align with the displayed position of the result (which is usually different from the initial timestep and often the position at the last timestep). In order to pick the body aligned with the original geometry, Ansys recommends retrieving the result closer to the initial timestep.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Additionally, you can select a body for the attached camera by clicking Geometry in the Outline while the animation is stopped and picking the body in the Graphics window. Note that you will not see the body turn pink until you click the result in the Outline.

- For analyses that are not pure Rigid Dynamics (for example Explicit Dynamics containing bodies with dimensionally reduced mesh and mixed Rigid Dynamics containing flexible bodies) Ansys recommends changing the result **Deformation Scale Factor** to 1.0 (True Scale).
- Once the animation starts playing with the attached camera enabled, the orientation of the camera cannot be modified, that is, Rotate, Pan and Zoom will not work.
- If you attach the camera to the same body that is scoped to a result probe, the body will not turn pink; it will keep the color of the result scoping.

Accelerated Animation

If you are using a Windows system that supports OpenGL 4.3, you can use the accelerated animation feature. This feature processes result animations much faster.

Important:

Although released, additional development is being made to the feature. Once active, you will see that accelerated animation, designated **Accelerated (Beta)**, does not currently support all display options. Support in this regard is ongoing.

Application

You activate this feature using the **Animation Mode** property in the **Options** dialog (**Graphics** (p. 228) category). The property includes the following options:

- **Program Controlled** (default): The application selects the animation mode based on the given analysis to balance performance and display features of the animation. This is the recommended setting.
- **Standard**: This setting uses the traditional animation process. It supports all display options but does not include advanced processing capabilities. This is an appropriate setting for straightforward models.
- **Accelerated (Beta)**: This setting offers improved processing capabilities. For this setting, certain display options are still being developed. In addition, the availability of the option does not require you to turn on the beta option in Workbench. This is a desirable setting for complicated models.

Note:

For the accelerated mode:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

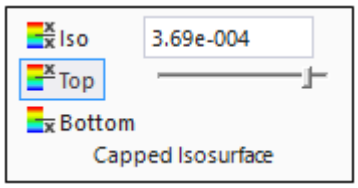
Published: 2023-10-23

– The application displays a progress bar of the processing status of the animation. For large, complicated models, this enables you to predict how long it will take to create the animation. The progress feature includes an option to cancel the operation if desired.

– If the application detects that this setting cannot support an animation given its development status, it will automatically revert to the **Standard** setting.

Capped IsoSurfaces

The **Capped IsoSurfaces** group represents a set of all points that equal a specified result value within the range of values for the result with additional features. This option provides three display selections. A display based on all points of a specified result, all points equal to and less than the specified result, and all points equal to and greater than the specified result value. You select this option from the **Geometry** drop-down menu on the Result Context Tab (p. 62). The option has a corresponding ribbon group as shown below.



Limitations

- For this view, when 1) you scope a result to a surface body (shell elements), and 2) the **Thick Shells and Beams** option is deactivated, and 3) the **Position** property of the result is set to **Top/Bottom**, application shows only the Top side stresses and strains. In order for these view options to correctly display result values, make sure you select the **Thick Shells and Beams** option.
- Because of internal tolerance calculations, there are times when the options of this view feature may present color inconsistencies between what is shown on the results legend versus what is displayed in the **Geometry** window.

Ribbon Options

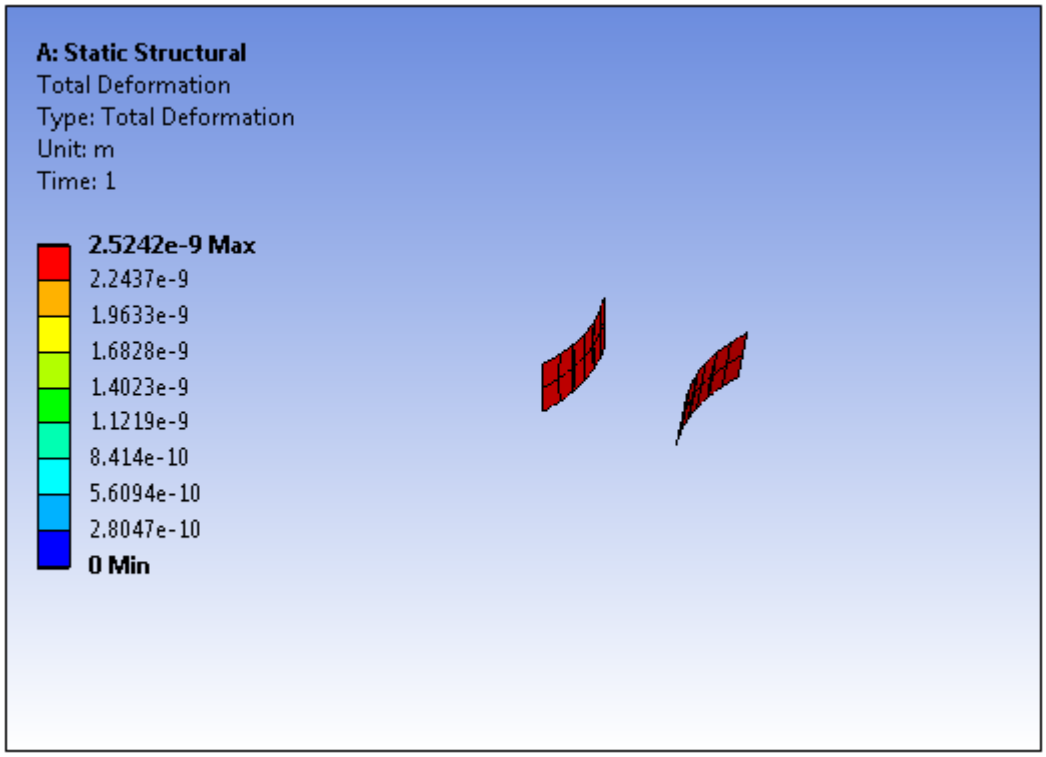
The value for the isosurface is set by the slider or entry field (specified result value). The slider represents the range from minimum to maximum for the selected result. Slider setting changes require you to click in the **Geometry** window to redisplay the result values in the legend.

The three options control the display, and include:

Iso

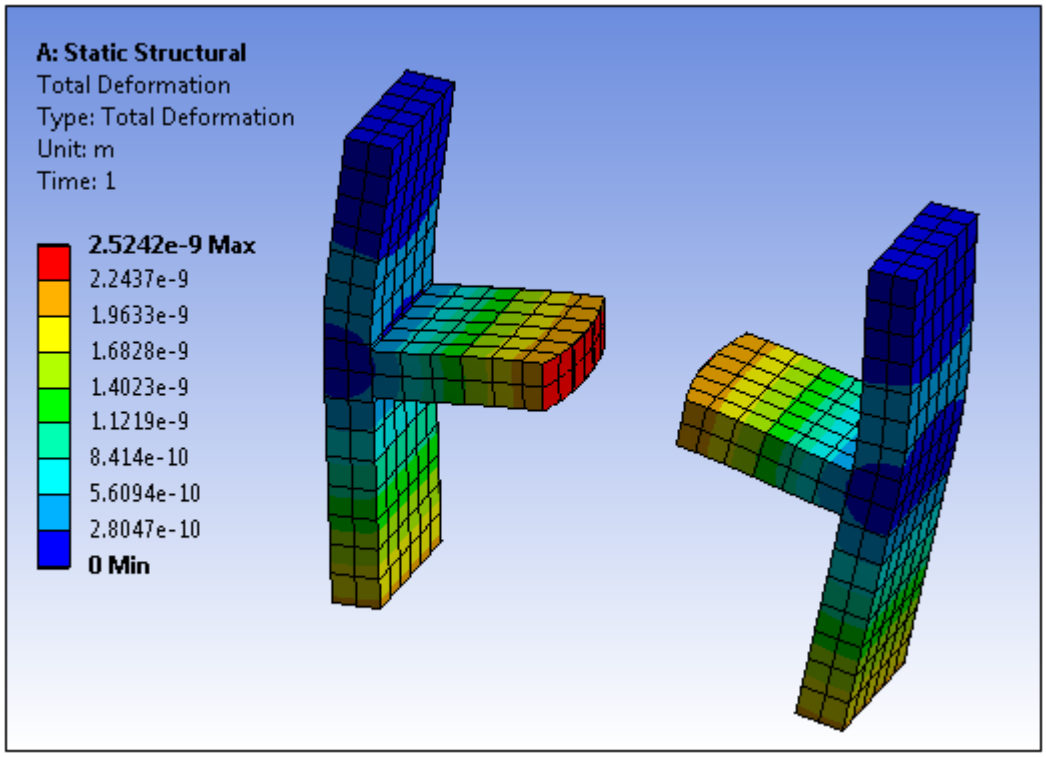
This option displays isosurface values equal to the specified slider or entry field value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Top

This option displays isosurface values that are less than the specified slider or entry field value.

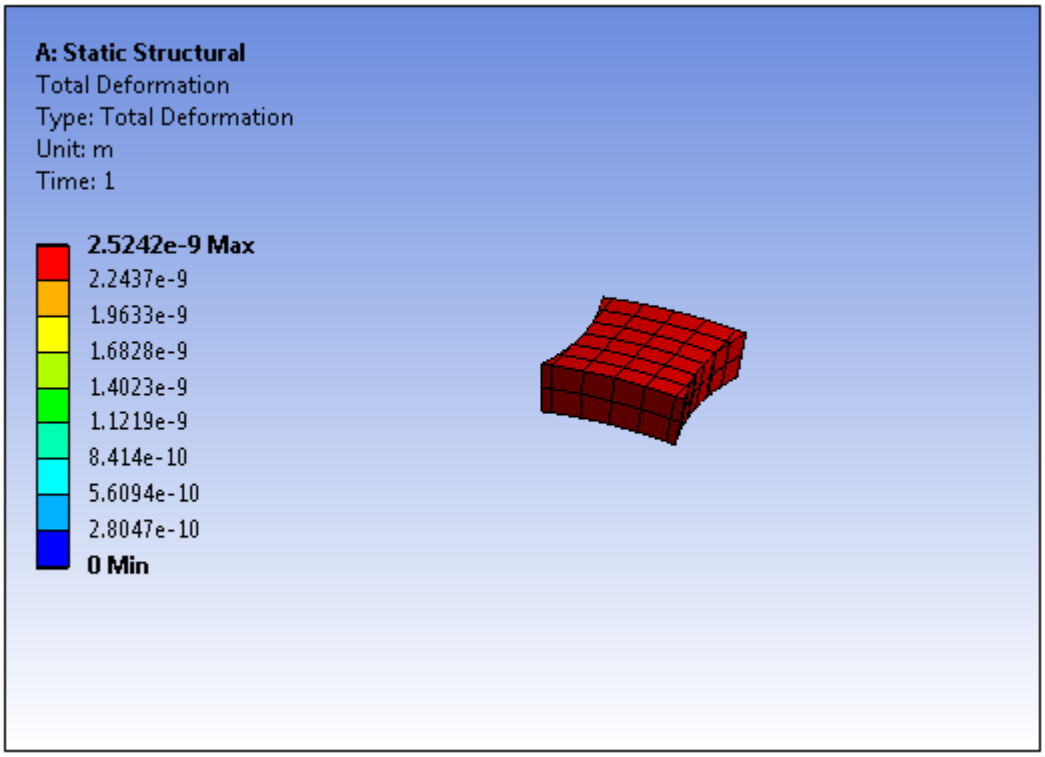


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Bottom

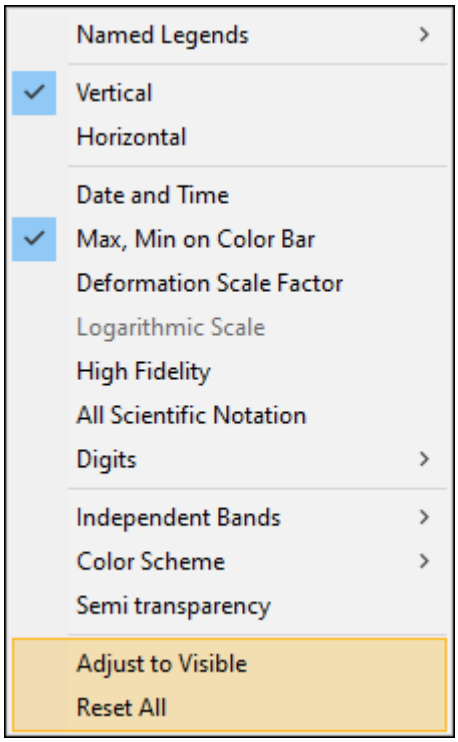
This option displays isosurface values that are greater than the specified slider or entry field value.



Dynamic Legend

When you place the cursor over a results legend content, the dynamic legend feature is available through the context (right-click) menu option Adjust to Visible. If you hide bodies/parts or create a Section Plane (p. 294) on a result, this feature updates the contour to only consider elements that become visible.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

This dynamic legend behavior does not apply to **Probe** annotations.

Minimum and Maximum Values

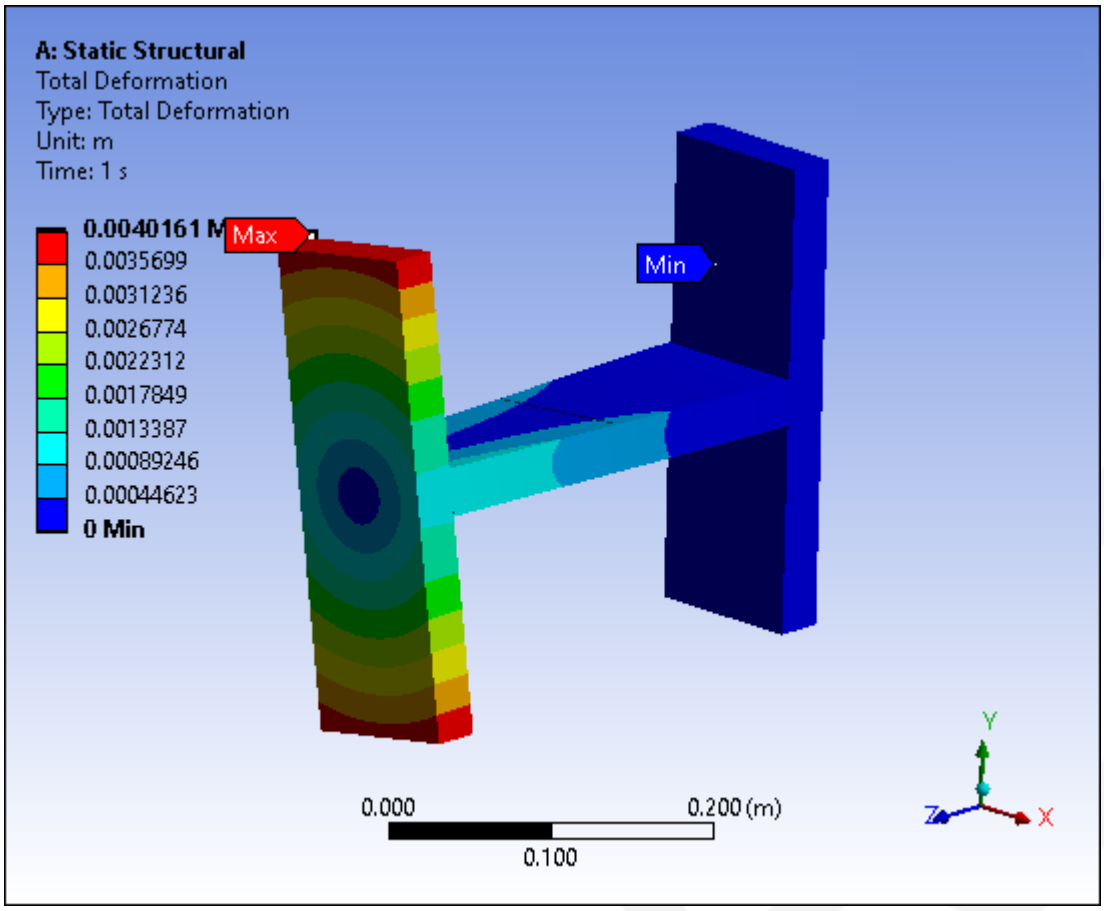
When you select the **Adjust to Visible** option for a sliced body, Mechanical repositions the minimum and maximum annotations to the lowest and highest result values displayed on the sliced body.

Example

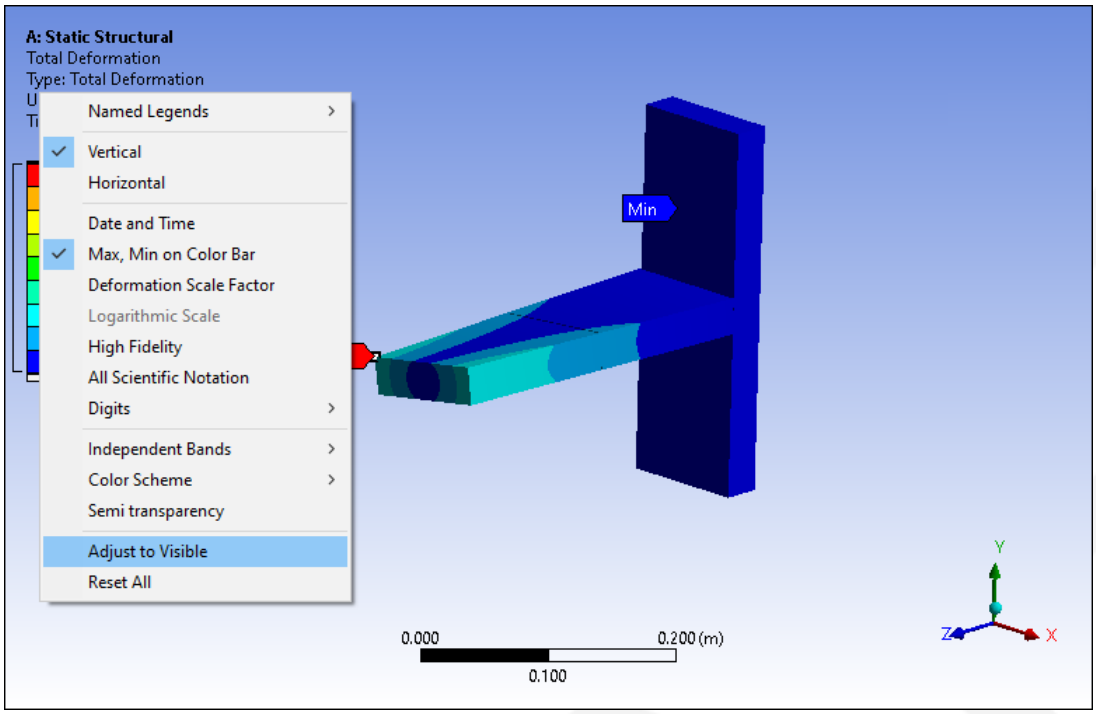
The sequence below is an example for a hidden body. Once you hide the body, select **Adjust to Visible** option to update the legend and view the result ranges for the visible elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

When you are viewing a result using the **Capped Isosurfaces** option (via the **Geometry** (p. 67) drop-down menu on the Result Context Tab (p. 62)), and the minimum or maximum value of a result occurs on the inside of your geometry and not on the exterior, the **Adjust to Visible** option will not display expected contour colors.

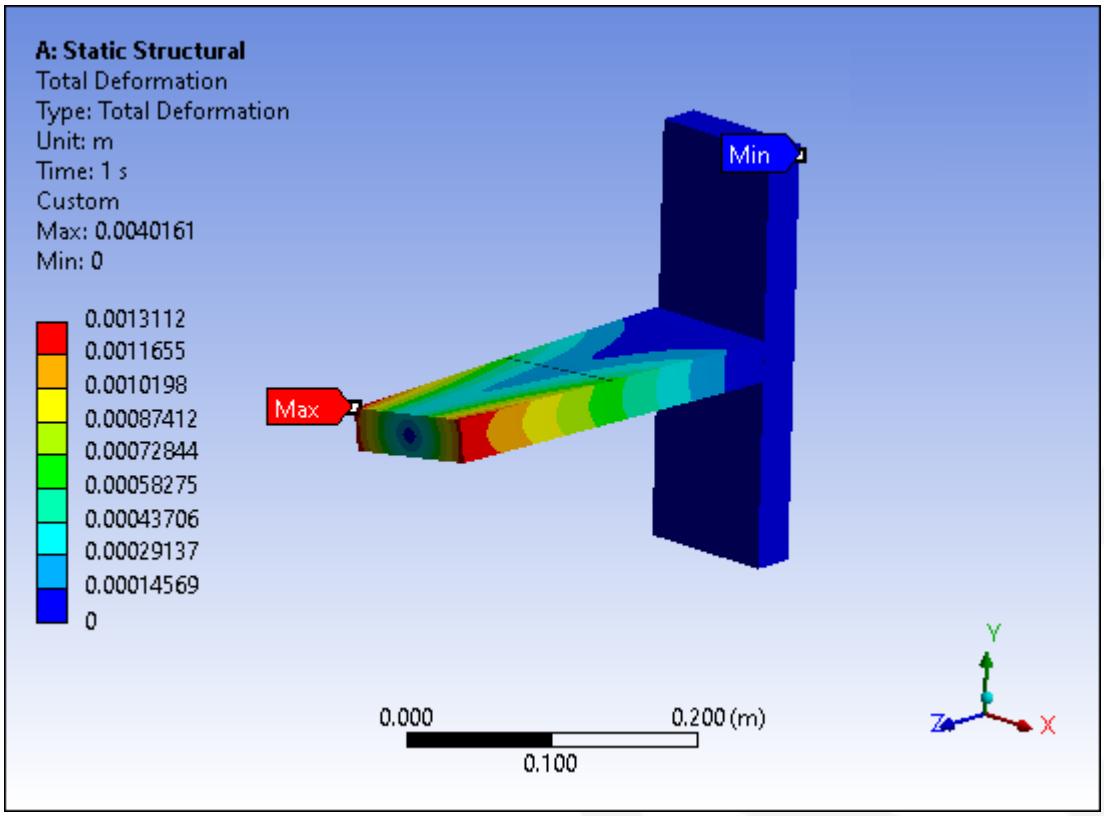
For the visible elements, the legend displays new content for the following:

- Legend colors and values
- **Min** and **Max** values
- Annotations for all values

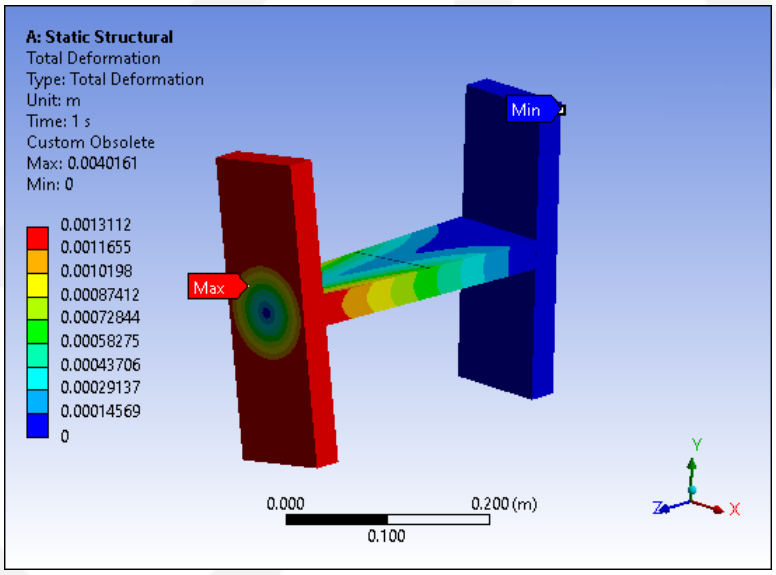
In addition, the label "Custom" is added to the legend to indicate the change to the display.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



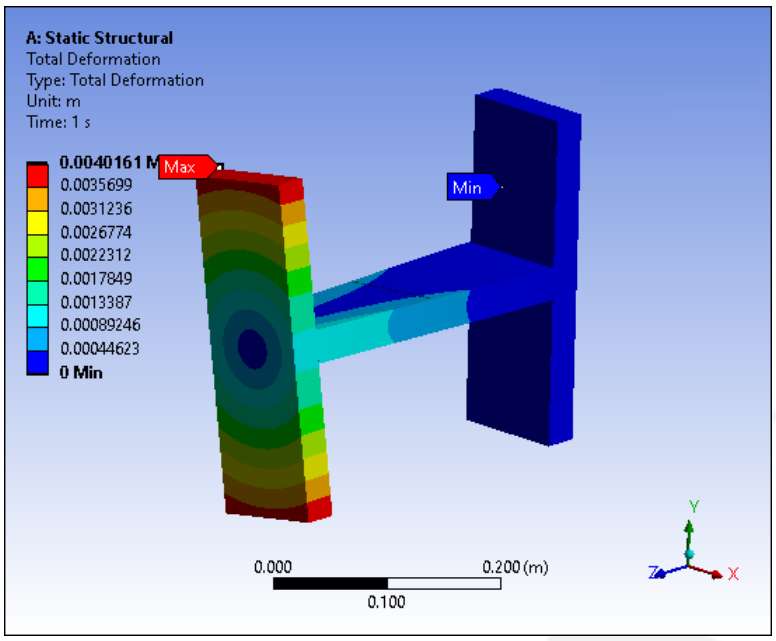
A change to the visibility of a part, displaying all parts again for example, or removing parts from the current display, adds the label "Custom Obsolete" to the legend annotation information. This indicates that the custom display is no longer accurate. It is displaying the values of the previous display (legend colors and values, the **Min** and **Max** values and annotation location, etc.), shown above, associated with a part being hidden. You need to select the **Adjust to Visible** option again to re-display the contours or select the **Reset All** option to display the original contours.



To restore the legend display for the entire body after you disable the slice or hide command, right-click the legend, and then click **Reset All** to view the result ranges for the entire body. If you do not

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

reset the legend to show result range for the entire body after disabling the slice or hide command, Mechanical displays the out of range values with colors not included in the legend.



Exporting Results

To export the data of a contour result, right-click the desired result object, select Export, and specify a file name and type. Supported file types include Text (`.txt`) and Excel (`.xls`). Text format is the default file format. However, the application always creates and displays an Excel file even when you save the data in text format. The exported file includes headers that display the node ID or element ID, and the result name and unit. Note that the application may **not** sort the node/element ID column.

You can prepend header information listed below using the **Show Additional Header** preference of the Export (p. 222) group of the **Options** dialog, you can prepend the exported file with the following header information:

- Result File Name
- Environment Name
- Result Name
- Result Type
- Unit
- Coordinate System
- Time

Multi-Body Scoping and Duplicate Result Values

The application processes and exports result data one body at a time. For result contours scoped to more than one body that include shared nodes, the export process may produce multiple result listings at each shared node. Depending on the type of result, results values on shared nodes for different bodies may be identical (and redundant duplicate values) or different. For example, note the differences of the following result types scoped to different bodies with share nodes:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Degree of freedom results, such as temperature or displacement, produce *identical* results.
- Element-nodal results, like stress or strain, produce *different* result values at each shared node.

The application offers a preference, **Remove Duplicate Nodes**, of the Export (p. 222) group of the Options dialog that removes duplicate nodes within a body. However, this may not address all circumstances. For the following results, the application removes identical duplicate result values from the export file based on property settings or element type:

- Element-nodal results and reaction forces (enforced motion) on planar elements because there is no variation across the thickness.
- Duplicate element-nodal result values, but only if you set the **Average Across Bodies** property to **Yes**.
- Shell elements unless you set the **Position** property to the **Top/Bottom** option.
- Elemental-nodal reactions on shell elements.
- Duplicate element-nodal result values for solid elements, but only if you set the **Average Across Bodies** property to **Yes**.
- Duplicate element-nodal result values for shell elements unless you set the **Position** property to the **Top/Bottom** option.

Contact Results

For unaveraged element- or node-based contact results or elemental-based contact result results, the element IDs in the exported file correspond to the solid elements with which the contact elements share a face.

Node Results

In addition to a general export of all contour result data, you can also manually select results for one or more nodes and export data for those nodes only. To export node results:

1. Select the desired result object.
2. Select the **Node** filter on the Graphics Toolbar (p. 97) and pick nodes on the result plot. You can also use a selection tool form the **Select Mode** menu (Box, Lasso, etc.). See the Selecting Nodes (p. 269) section for more information.
3. In the Geometry window, right-click and select the **Export Node Results** option. You are prompted to save the data in `.txt` format. You may also select to save the data as an Excel file. Once you save the file, an Excel file automatically opens providing the node numbers and the corresponding result data.

Note:

The following do not support node-based export:

- Path Results (p. 1940), Surface Results (p. 1959), and Crack-based results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

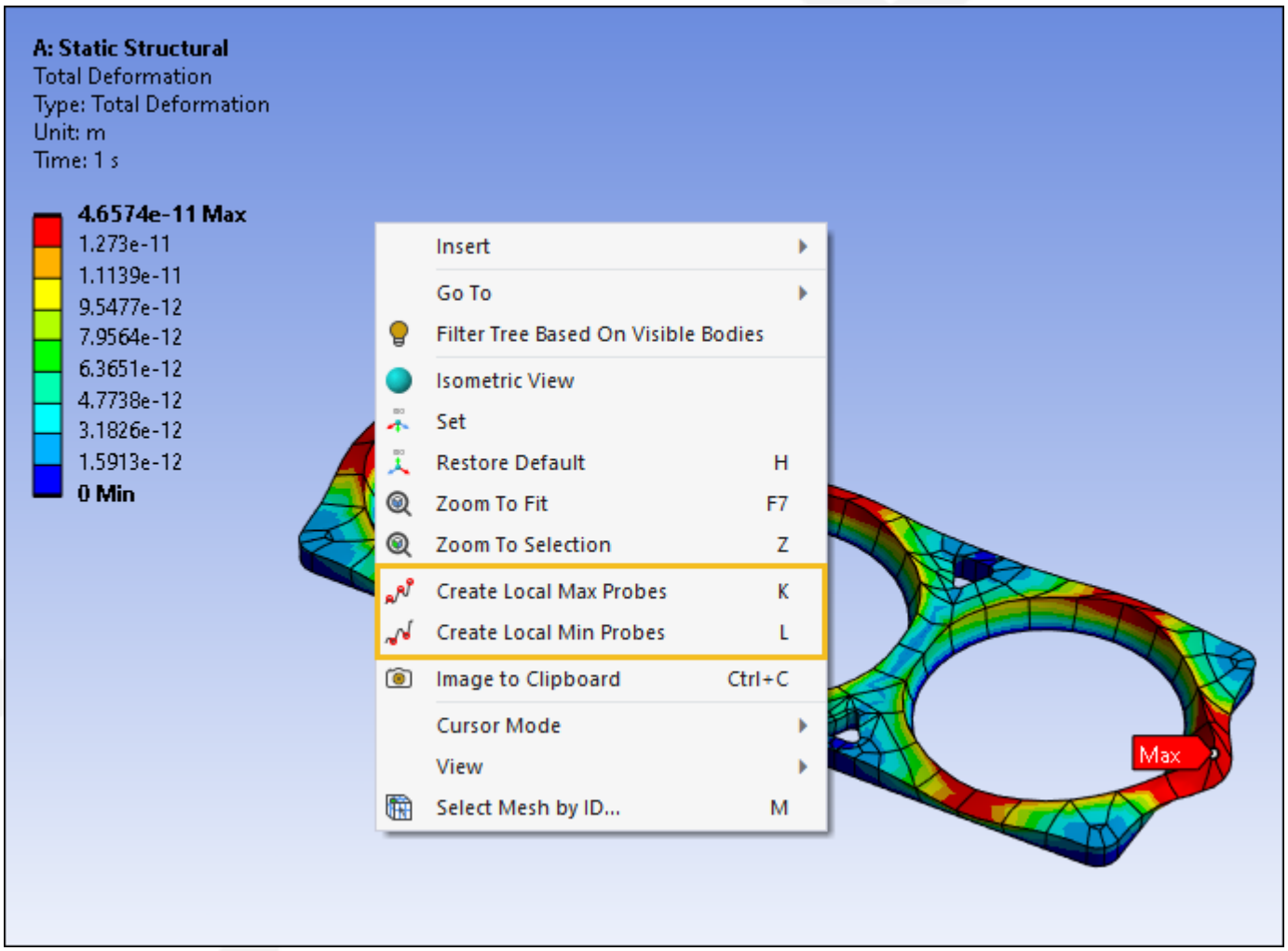
- Results scoped to elements or element-based named selections.

Generating Reports

See the Report Preview (p. 174) section.

Local Minimum and Maximum Probes

For nodal results such as displacements, and element nodal results such as stresses and strains, the application provides the context menu (right-click) options **Create Local Max Probes (K)** and **Create Local Min Probes (L)**, as illustrated below. The options are available on the menu when the **Geometry** window has focus. As shown on the context menu, the **K** and **L** keys also activate the options. Once you have probes labels displayed, re-selecting these options replots your result data or updates the locations and values if you have made a change.



Go to a section topic:

- Requirements (p. 2279)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Probe Label Display (p. 2279)
- Display Options (p. 2282)

Requirements

Note the following requirements for this feature:

- Only displays labels on active (not suppressed or hidden) parts.
- Can only display result values available on exterior faces.
- Does not support results scoped to either a **Path** or **Surface**.
- Does not support beam element structure.

Note:

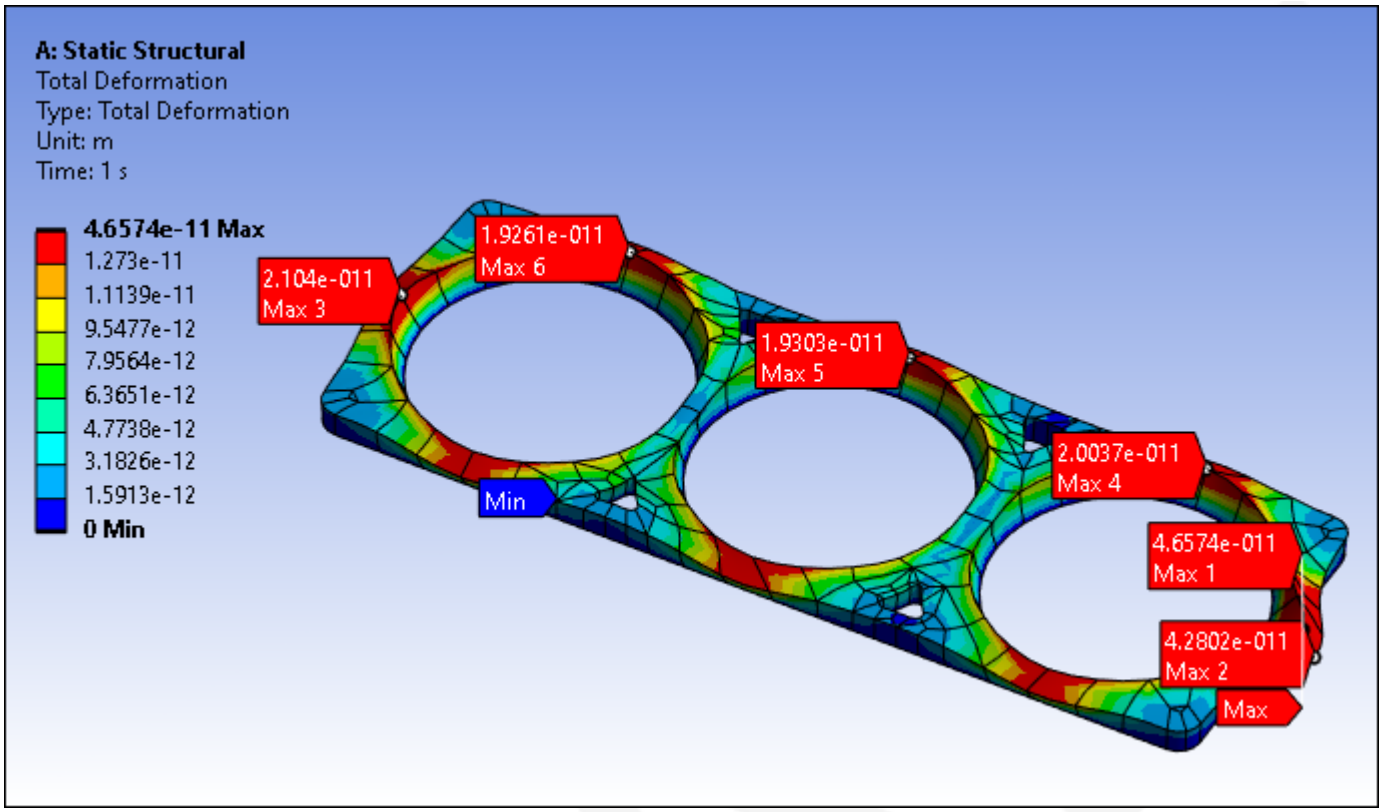
Currently there is a beta option to Create Local Volume Min/Max Probes. Follow the link to review the documentation on this beta feature.

Probe Label Display

When activated for a selected result, the application creates probe labels at local maximums and minimums across the result's scoping. The application labels a node as a local maximum if all adjacent nodes (immediate neighbors) have smaller result values than itself. Similarly, a node is a local minimum if all adjacent nodes have larger values than itself. Specifically, a node can be a local maximum or minimum even if there are nodes on the model that have larger/smaller values than itself if they are not an immediate neighbor. For example, given a 2D model, in these sequences: 80, 90, 100, 90, 70, and 30, 40, 50, 40, 30, the application labels the values 100 and 50 as maximums because their neighbors (to left and right) have smaller values.

For the supported result types, note that the **Max 1** and **Min 1** probes are equal to the global min/max probe labels displayed using the **Maximum** and **Minimum** buttons on the Results (p. 71) Context tab. The example below shows maximum local probe labels. You can think of each label as a maximum for its immediate area on the model. The application creates labels only for the largest maximas (and smallest minimas) across the model, up to the limit defined by the preference.

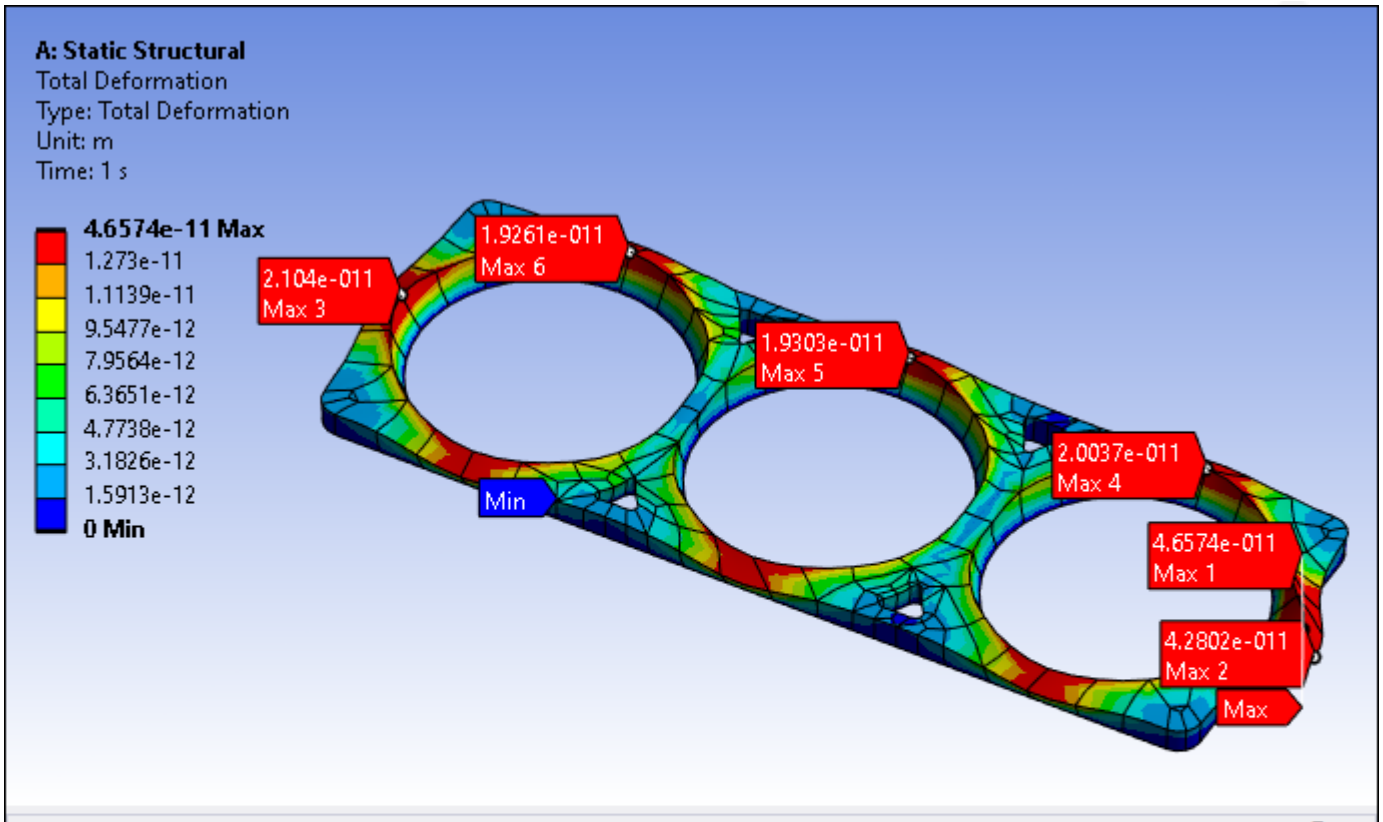
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



The number of probe labels displayed is based on the amount of result data. The default value for this display is six (6) labels. As desired, you can change this default setting of the **Number of Local Min/Max Probes** controls in the Graphics (p. 228) preference of the Options (p. 217) dialog. You can specify the number of labels to display (for each - Min and Max) from between 1-20.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

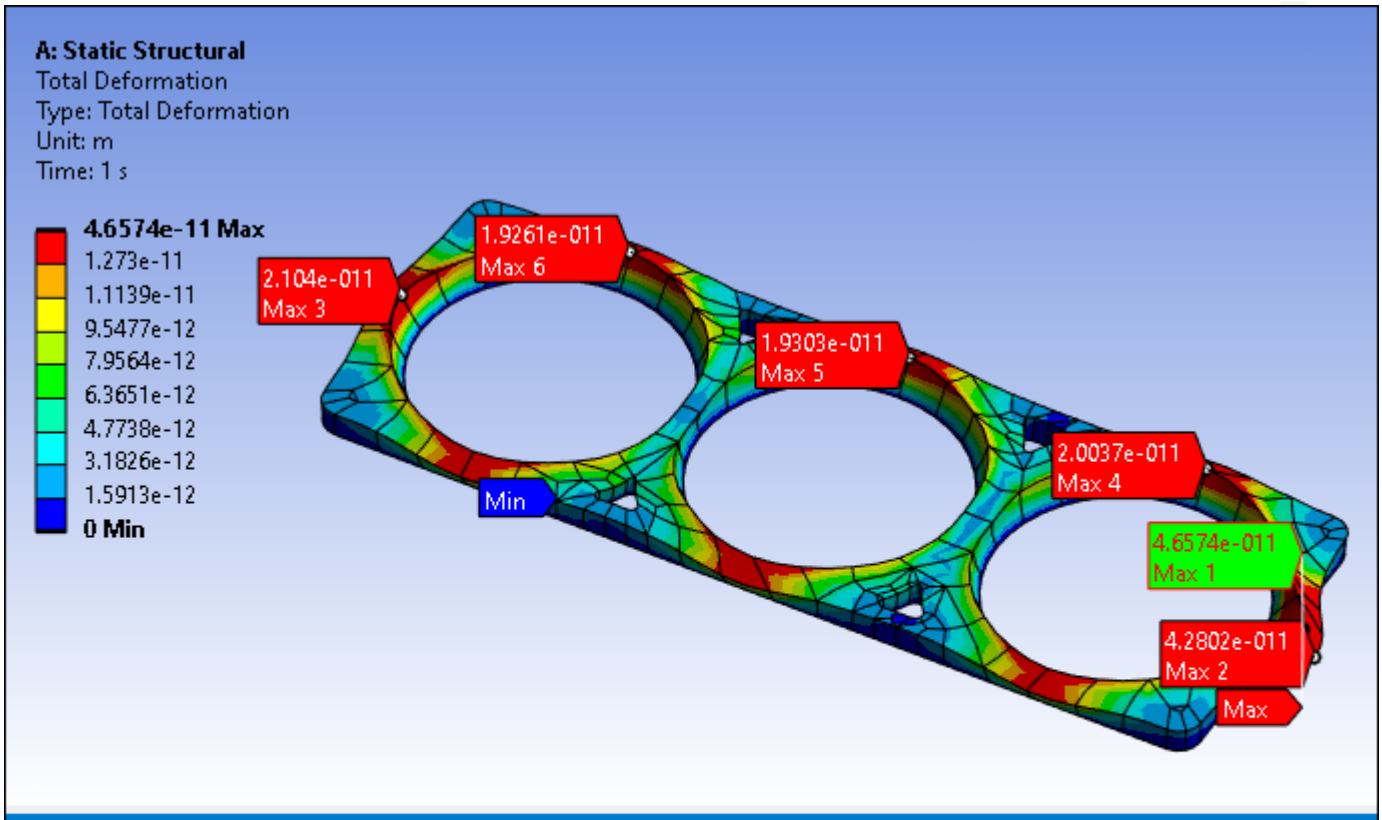


Graphics Annotations

Type	Value	Note	Unit	Location X	Location Y	Location Z
Local Max	1.9261e-011	Max 6	m	-0.032117	0.017089	0.003784
Local Max	1.9303e-011	Max 5	m	0.000009	0.017100	0.003770
Local Max	2.0037e-011	Max 4	m	0.034187	0.017167	0.003840
Local Max	2.104e-011	Max 3	m	-0.049426	-0.000002	0.003845
Local Max	4.2802e-011	Max 2	m	0.055799	-0.000132	0.001572
Local Max	4.6574e-011	Max 1	m	0.054372	-0.000053	0.004693

Note that the application automatically displays the **Graphics Annotations Window** (p. 202) along with the probe labels. Selecting a table cell in the **Value** column or a cell of the **Association** column or selecting the annotation label in the **Geometry** window, highlights the annotation in the table as well as in the **Geometry** window, as illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Graphics Annotations

Type	Value	Note	Unit	Location X	Location Y	Location Z
Local Max	1.9261e-011	Max 6	m	-0.032117	0.017089	0.003784
Local Max	1.9303e-011	Max 5	m	0.000009	0.017100	0.003770
Local Max	2.0037e-011	Max 4	m	0.034187	0.017167	0.003840
Local Max	2.104e-011	Max 3	m	-0.049426	-0.000002	0.003845
Local Max	4.2802e-011	Max 2	m	0.055799	-0.000132	0.001572
Local Max	4.6574e-011	Max 1	m	0.054372	-0.000053	0.004693

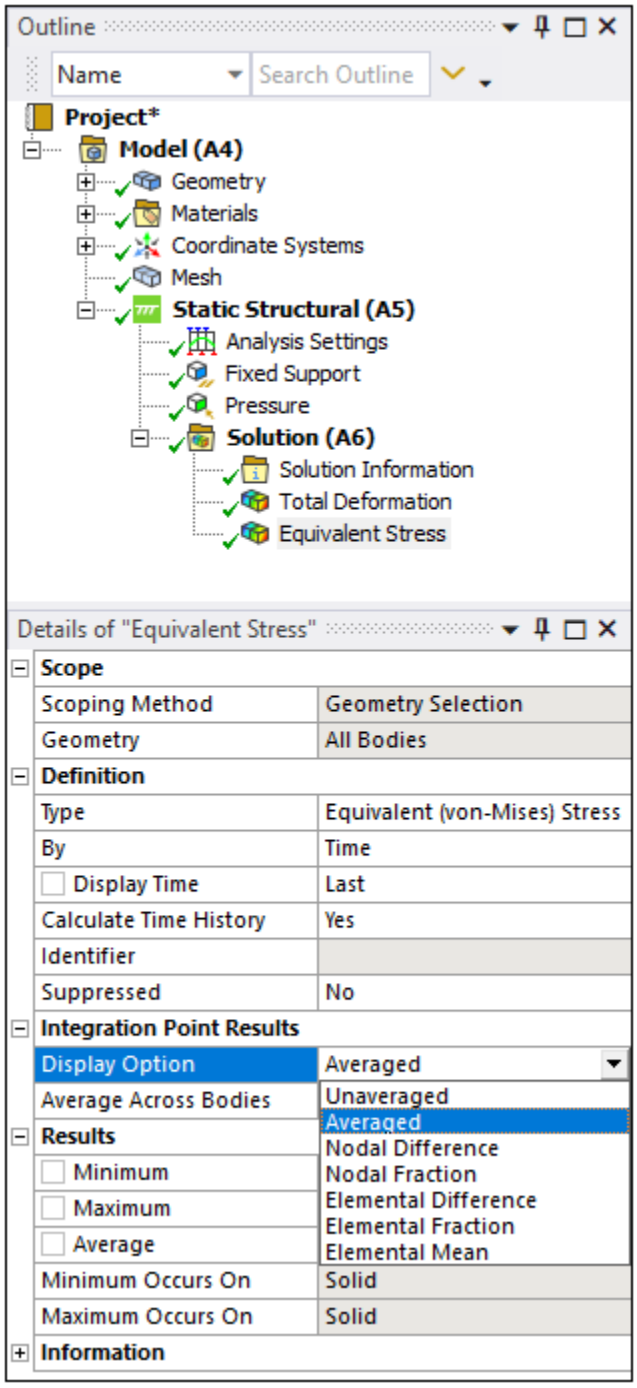
In addition, as you use this feature to display labels from multiple results, the data is added to the **Graphics Annotations** window. Therefore, you may see more than one table entry with the same **Note** value (for example, **Max 6**). The **Associated** column displays the result object that corresponds to the row. In order to remove the labels from your **Geometry** window display, you need to delete them individually or all together using the delete options of the **Graphics Annotations** window.

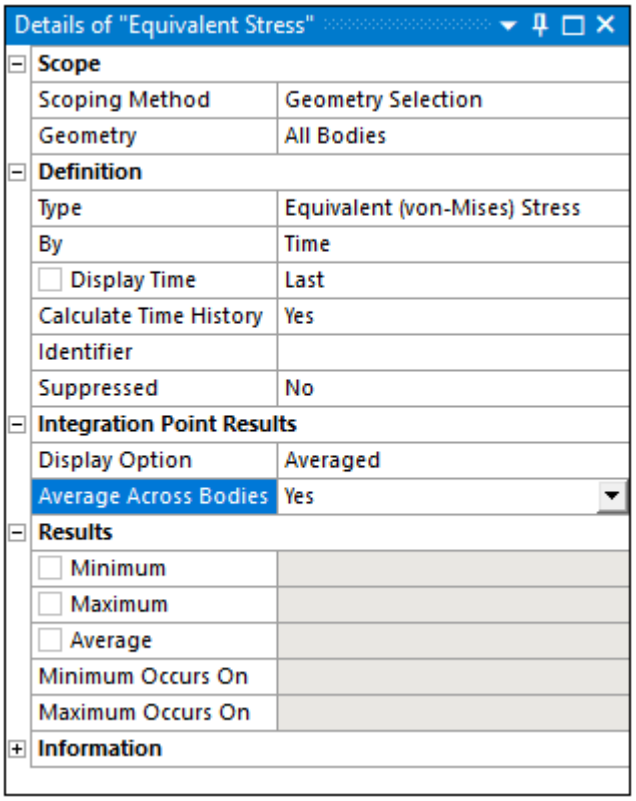
Display Options

These options support nodal results and averaged element nodal results. For element nodal results, you can change the setting of the **Display Options** property (**Integration Point Results** category) in the **Details** view of the selected result. These maximum and minimum probe options support the following display options only: **Averaged** (default), **Nodal Difference**, and **Nodal Fraction**. If you select the option **Averaged**, you can also set the **Average Across Bodies** property to **Yes** to enhance the display of minimum and maximum probes if they are located at shared nodes of multiple bodies. See the Averaged vs. Unaveraged Contour Results (p. 1981) section of the Help for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23





Renaming Results Based on Definition

The option **Rename Based on Definition** is available when you right mouse click any result (under **Solution** objects), or any **Result Tracker** (p. 1844) (under **Solution Information** (p. 1833) objects). When you choose this option, the Mechanical application automatically renames the result or **Result Tracker** based on the selected parts (for example, **Temperature** can be renamed to **Temperature - Tube**, or **Directional Deformation** can be renamed to **Directional Deformation - All Bodies**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Results Legend

By default, the legend for a selected result displays basic information, including:

- Result Name and Type
- Units
- Step End Time
- Time and date

Review the following topics in order to customize and manage the annotations and content of your result legend:

- Customizing Legend Content (p. 2284)
- Creating and Managing Legend Schemes (p. 2285)
- Customizing the Contour Range (p. 2286)
- Customization Limitations (p. 2286)

Customizing Legend Content

Using the right-click (context) menu, you can customize the legend's content.

- **Named Legends**: a name can represent the following data:
  - Number of contours
  - Color scheme
  - Color overrides per band
  - Value break per break, either automatic or numeric
- **Vertical**: view the result contours color spectrum vertically (default).
- **Horizontal**: view the result contours color spectrum horizontally.
- **Date and Time**: toggle Date and Time on and off.
- **Deformation Scale Factor**: For deformation results, this option turns on (or off) a legend annotation of your scaling menu (p. 63) selection.
- **Max, Min on Color Bar**: shows extremes when checked. If unchecked, they appear in the title book.
- **Logarithmic Scale**: displays result values.
- **High Fidelity**: Use this option when color display issues (p. 2286) occur on the model due to customization. For example, when you are interactively probing a contour result that has a customized legend and the interval between adjacent bands is much smaller (at least less than

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

one thousandth) compared with the full min-max range, this option replots and improves the synchronization of the values of the probes with the colors on the model as well as the corresponding color/range in the legend.

- **All Scientific Notation**: displays result values.
- **Digits**: specifies the number if significant digits for result values. The default is 3. Options include 2 through 8.
- **Independent Bands**: Use to set the alarm color representing the maximum/minimum contour range. The following choices are available:
  - **None** (default)
  - **Top**
  - **Bottom**
  - **Top and Bottom**
- **Color Scheme**: used to change the color spectrum. The choices available are:
  - **Rainbow** (default)
  - **Reverse Rainbow**
  - **Grayscale**
  - **Reverse Grayscale**
  - **Reset Colors**
- **Semi transparency**
- **Adjust to Visible**
- **Reset All**

Creating and Managing Legend Schemes

The Named Legends menu option provides additional options that enable you to save a customized legend scheme. The **New** option saves the current legend and places it in the list of available legends available from this menu option. Any changes to the legend are included. You can create an independent variation of a named legend by choosing **Unnamed** or **New**. The option **Unnamed** is the default. The **Unnamed** option indicates that the legend can be edited independently.

The **Manage** option opens a dialog that lists all available legends and it includes additional capabilities:

- **Import**: Import a previously created legend scheme.
- **Export**: Export a legend scheme.
- **Rename**: Rename the current legend scheme.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Delete**: Delete the current legend scheme.

Customizing the Contour Range

Placing the mouse pointer over a contour color of the legend and right-clicking the mouse provides the following additional context menu options:

- **Custom Color**: a pop-up color appears when you right-click a color band. The same color can be used for more than one band.
- **Automatic Color**: the default color is restored.

Placing the mouse pointer over a value on the legend and right-clicking the mouse provides the following additional context menu options:

- **Edit**: Enables you to enter a custom value. Double-clicking a value also performs this edit action.
- **Automatic Value**: The value calculated by the program.

In addition, selecting a contour color or a value on the legend displays plus (+) and minus (-) buttons. These buttons enable you to set the number of bands between the bottom and top of the contour. The number of bands can range from 4 to 14.

Note:

On some computers, you may encounter entry issues when manually editing the legend. If you experience this issue, you can set the environment variable **`ANS_USE_OPENGL_LEGEND_EDIT`** to **`1`**. However, this editing mode presents certain limitations, such as unrelated hotkeys activating while you make entries and entries extending outside of the legend entry field.

Customization Limitations

If you customize the legend between adjacent bands and the interval is very small, for example, less than one thousandth of the entire range, the contour colors may not correctly reflect the ranges in the legend. A result is illustrated below.

Determining the difference between the maximum and the minimum, you get: 46.592 - 0.00032942 = 46.592. The interval between the customized interval is: 0.01 - 0.0087912 = 1.2088 e-3. This is the interval between each of the subsequent color bands (orange through blue). Dividing the first interval by the interval of the subsequent bands: 46.592/1.2088 e-3 = 38,543. The first interval is nearly 40,000 times greater than the subsequent intervals. This is a clear example of the issue of legend customization involving very small intervals.

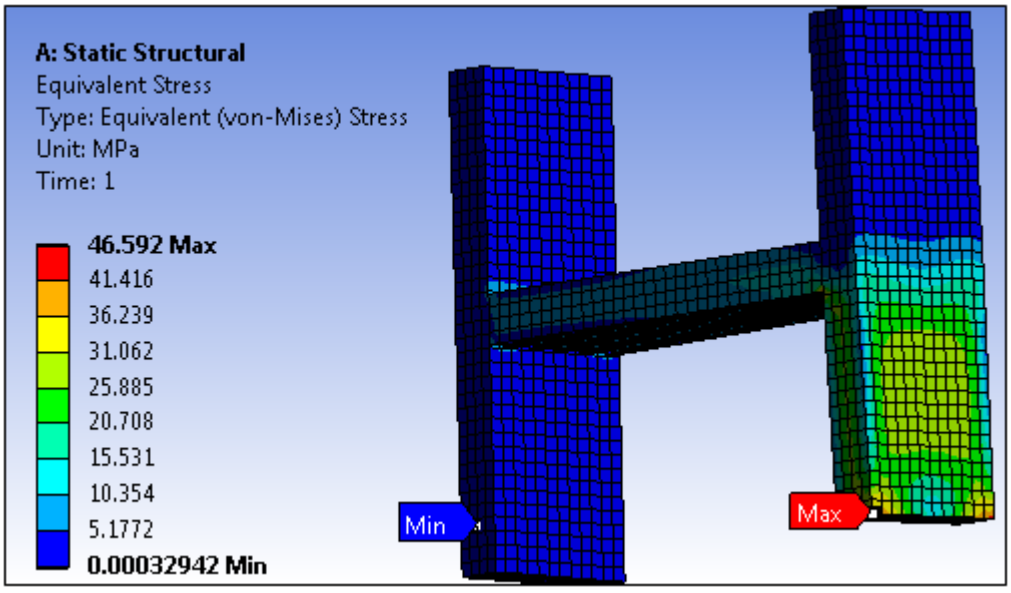
Therefore, customizing the legend at the second contour color with a value of 0.01, the coloring plotting experiences a limitation and the model becomes entirely red. The legend values are correct, but the coloring of the model becomes inaccurate.

Displaying the context menu and selecting **High Fidelity** replots the model's coloring accurately, as shown in the final image.

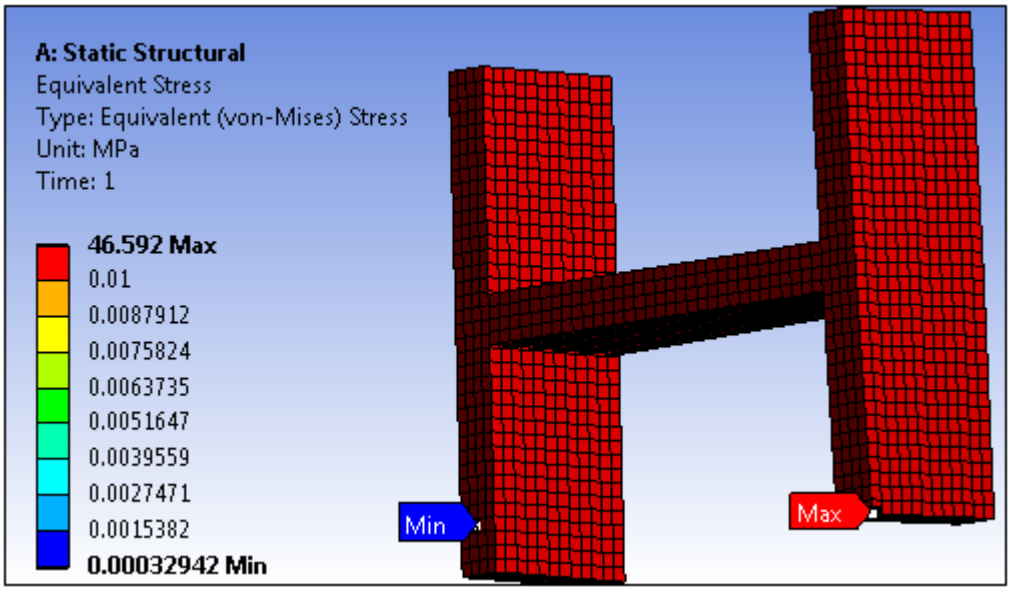
Original Contours

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

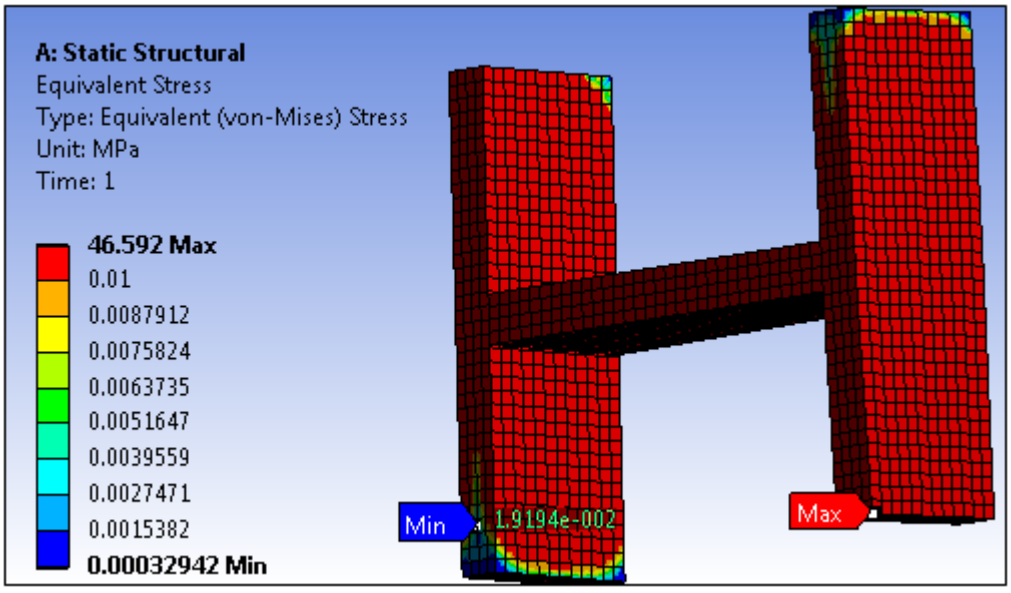


Range Customized



High Fidelity Synchronization

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Results Tab

Refer to the Result Context Tab (p. 62) topic in the Context Tabs (p. 47) section.

Solution Combination

You use the Solution Combination object/feature to combine load cases, called "base cases" in the application, to generate results that display the effects of the combination of the load cases. The application supports two combination methods:

- **Multiple**: You use this method to combine (sum) multiple base cases.
- **Envelope**: You use this method to display the highest (maximum) or lowest (minimum) result values, among the chosen base cases, at each node/element.

Go to a section topic:

- Supported Analysis Types (p. 2289)
- Define Solution Combination (p. 2289)
- Import/Export Worksheet (p. 2295)
- Interpreting Worksheet Content (p. 2295)
- Supported Result Types (p. 2296)
- Examining Results (p. 2297)

Important:

Review the next section, Solution Combination Process Requirements and Conditions (p. 2300), to ensure your understanding of the feature's operations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Supported Analysis Types

This feature supports the combination of the following analysis types:

- Harmonic Response
- Random Vibration
- Response Spectrum
- Static Structural
- Transient Structural

Define Solution Combination

1. **Define Simulation**

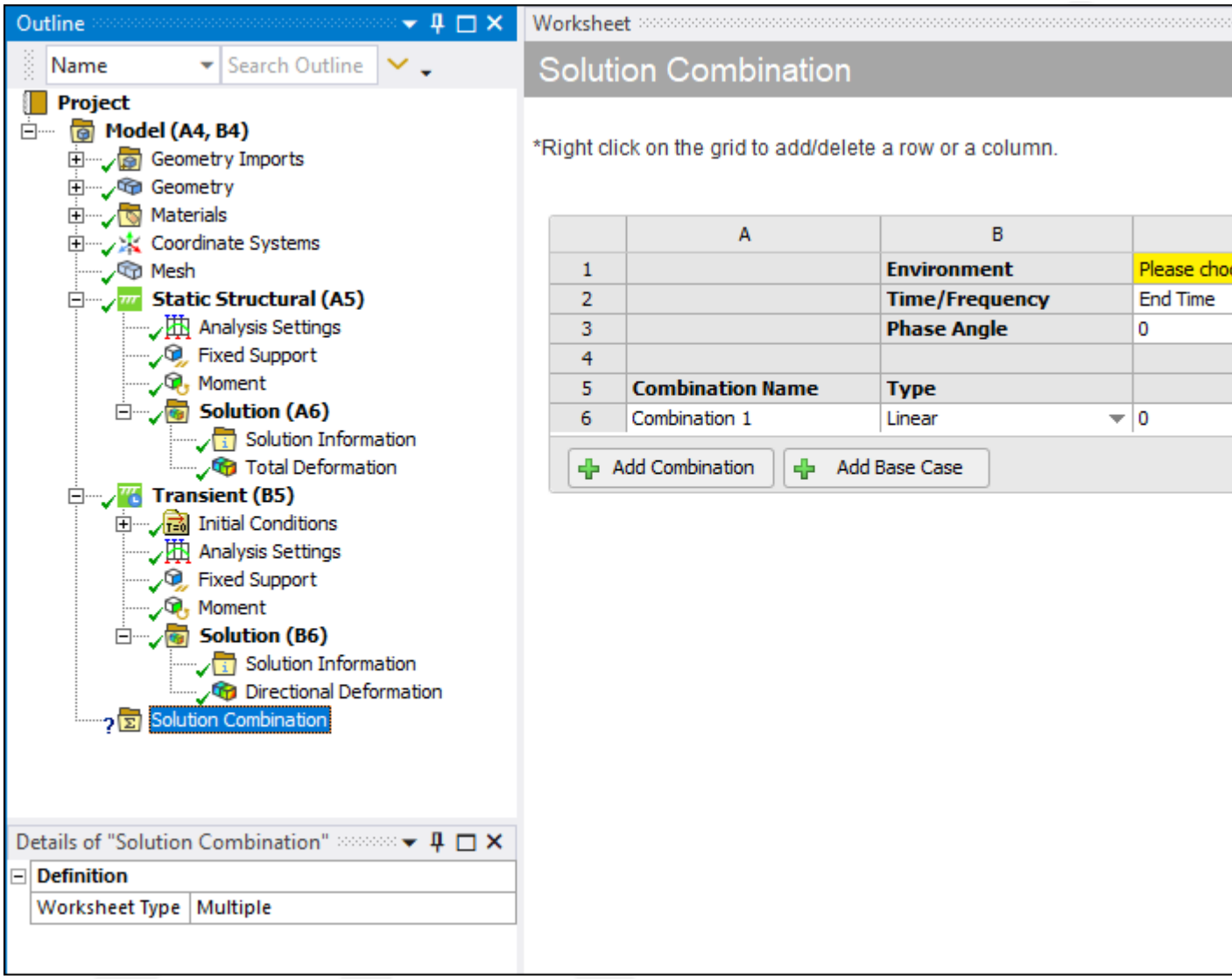
   Define your environments and solve the simulation.

2. **Specify Object**

   From the Model object, either select the **Solution Combination** option on the Model Context (p. 48) tab. You can also right-click the **Model** object or in the Geometry window and select **Insert** > **Solution Combination**. The object is inserted and the **Worksheet** displays.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



3. Specify a combination method using the **Worksheet Type** property. Options include **Multiple** (default) and **Envelope**.

4. **Define Base Cases**

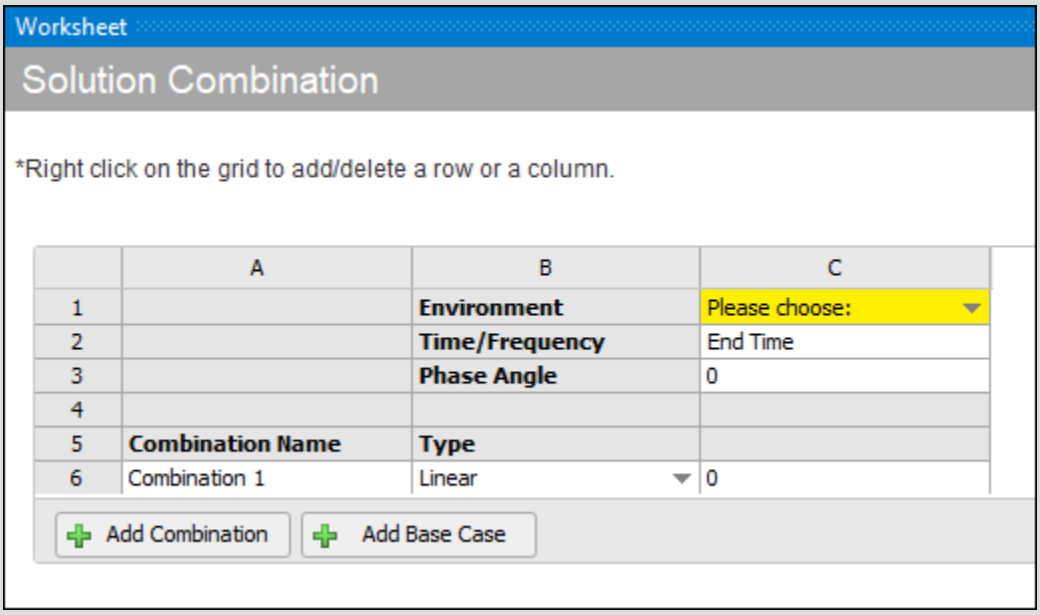
   Select one of the following procedures based on the specified method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Multiple Method

As illustrated below, the application inserts a base case (C column) by default. The **Environment** setting is not specified by default (yellow highlight). To add additional base cases, select the **Add Base Case** button or use the available context (right-click) menu.



You define a base case using the following fields:

- **Environment**: Select the desired input environment from the drop-down menu. Supported environments include: Harmonic Response, Random Vibration, Response Spectrum, Static Structural, and Transient Structural.
- **Time/Frequency**: Enter the time step for the base case (for Static and Transient analyses) or the Frequency (for Harmonic Response analysis).
- **Phase Angle**: Enter a phase angle for the base case (for Harmonic Response analysis only)

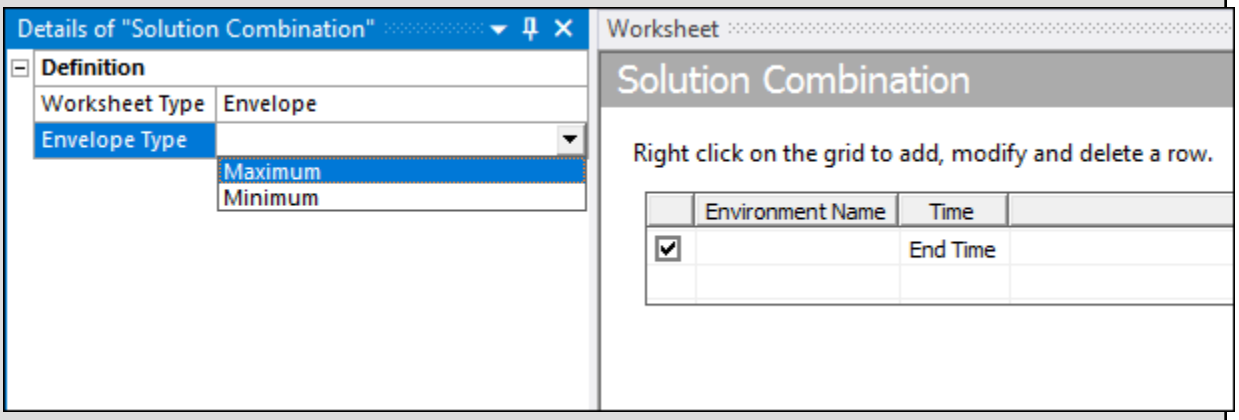
Note:

Time is always defined in seconds. **Frequency** is always defined in Hz. And **Phase Angle** is always defined in degrees. No other units or conversions are supported for the **Solution Combination** Worksheet.

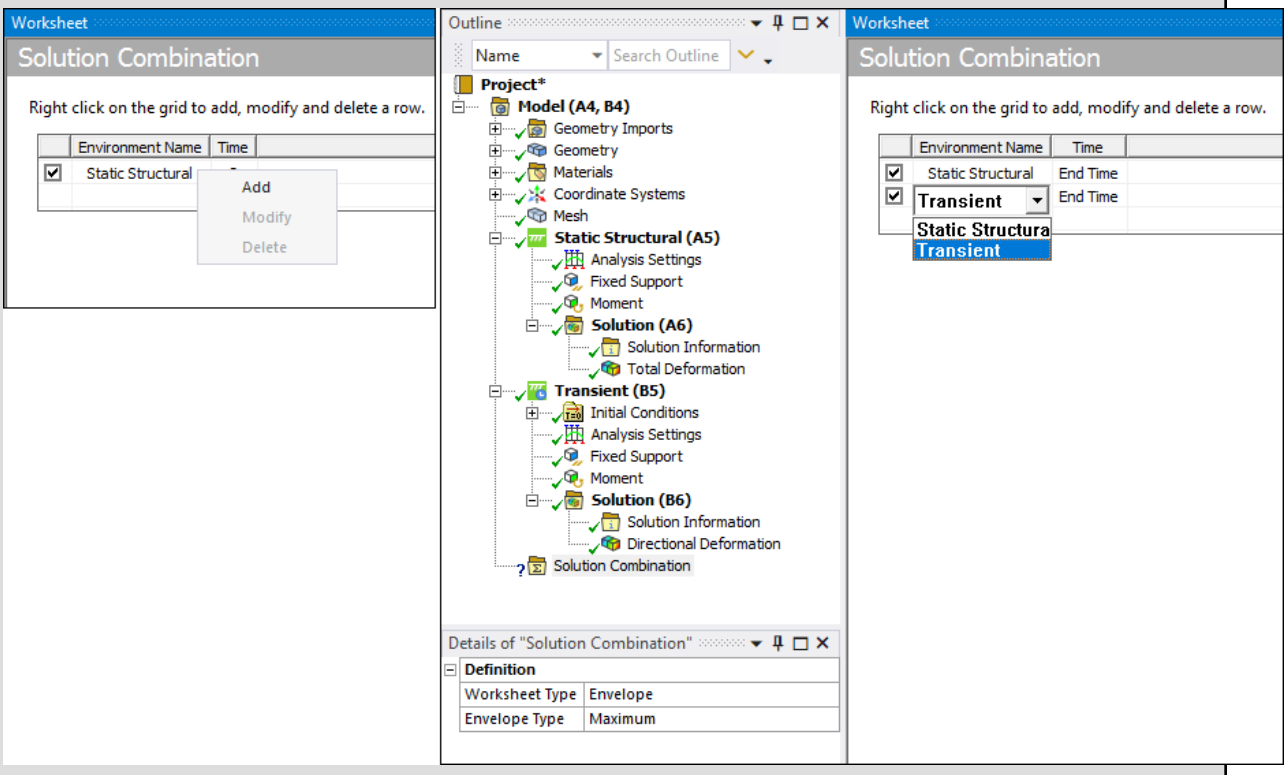
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Envelope Method

When you select the **Envelope** method, you need to also specify the **Envelope Type** property. Options include **Maximum**, **Minimum**, **Maximum Load Case**, and **Minimum Load Case**.



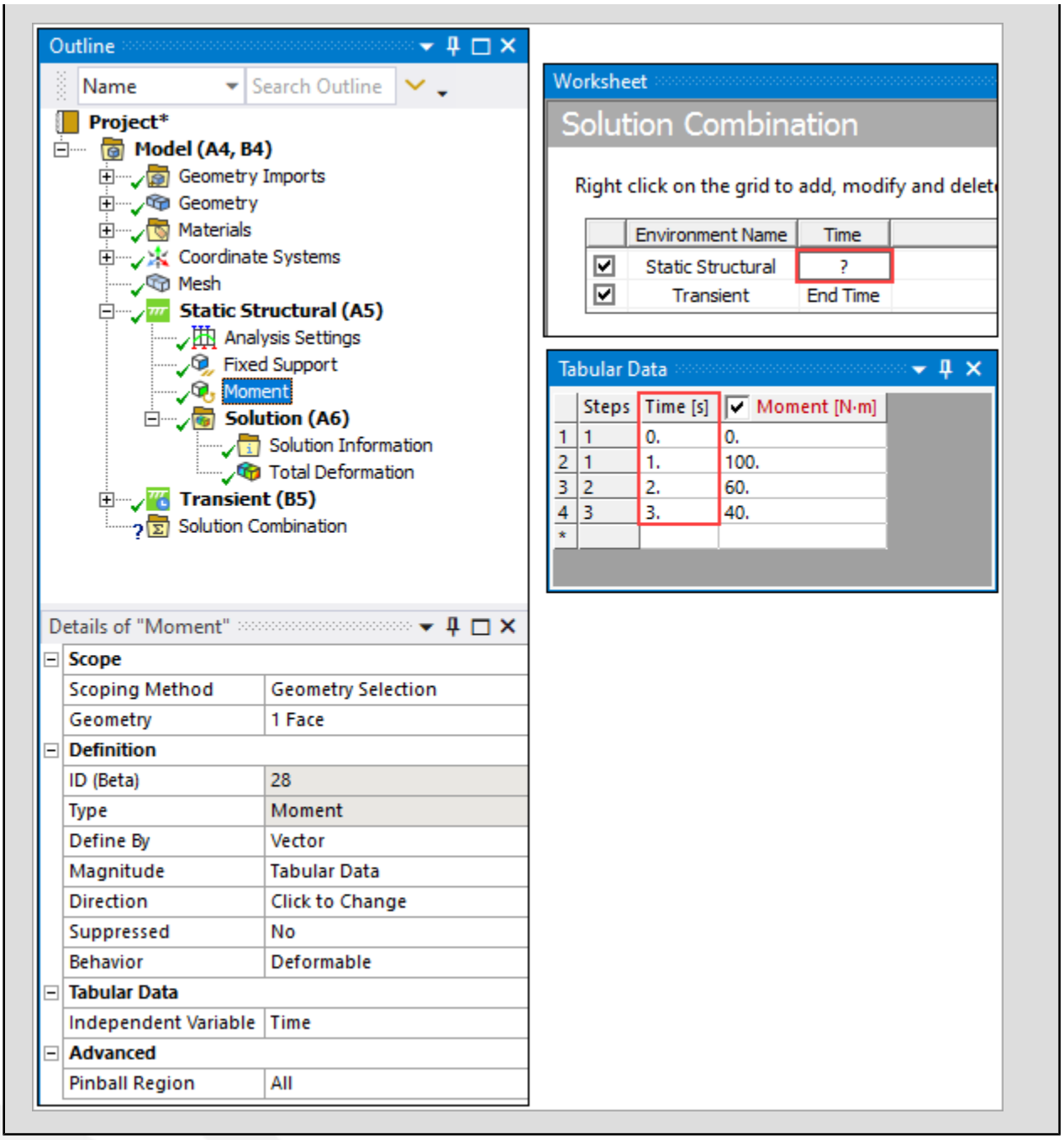
Select environments from the **Environment Name** drop-down list in the worksheet. A worksheet includes one row by default. Right-click in the worksheet to display the context menu to **Add** (**Modify** or **Delete**) rows.



Specify a desired load step for the environment in the **Time** column. The following example shows the available times (for each Step) for the Moment load in the structural environment. You select from these time values. The default entry **End Time** is equal to the last load step. You can also enter the specific last step (3 below). You can return the setting to "End Time" by entering 0 (zero) in the time field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

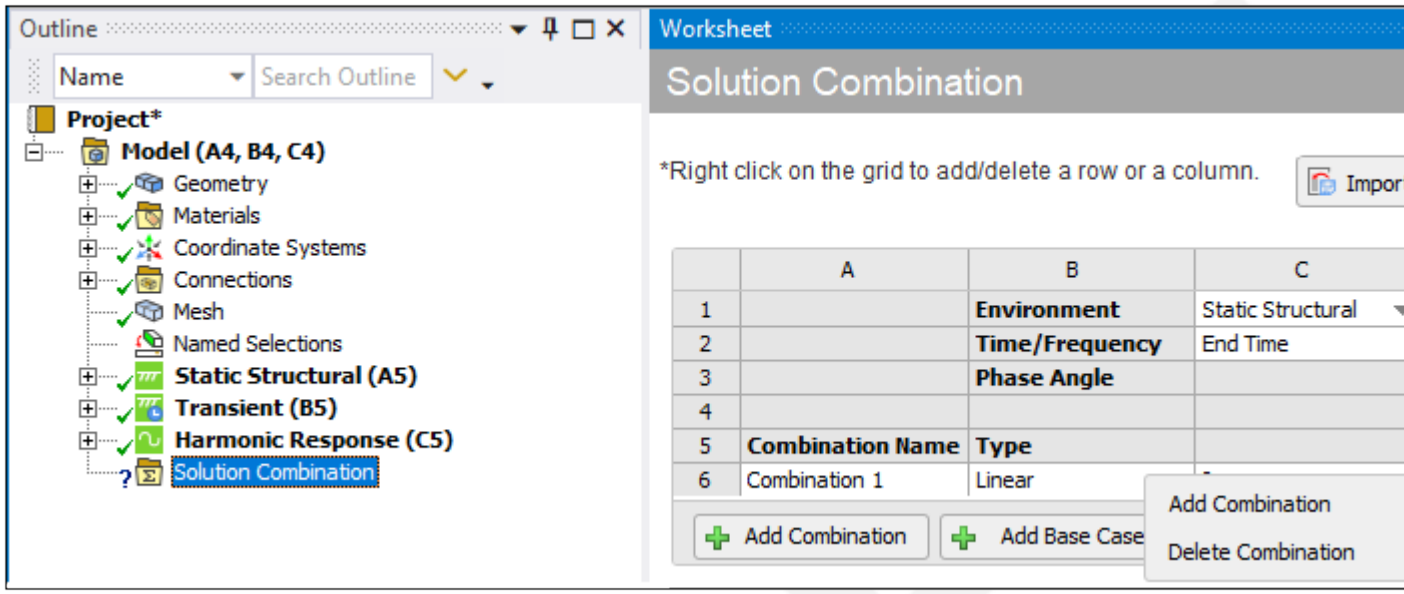
Published: 2023-10-23



5. **Define Load Combinations (Multiple Method)**

When using the **Multiple** method, the application inserts a Combination by default ("Combination 1"), as illustrated below. To add additional Combinations, select the **Add Combination** button or use the available context (right-click) menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



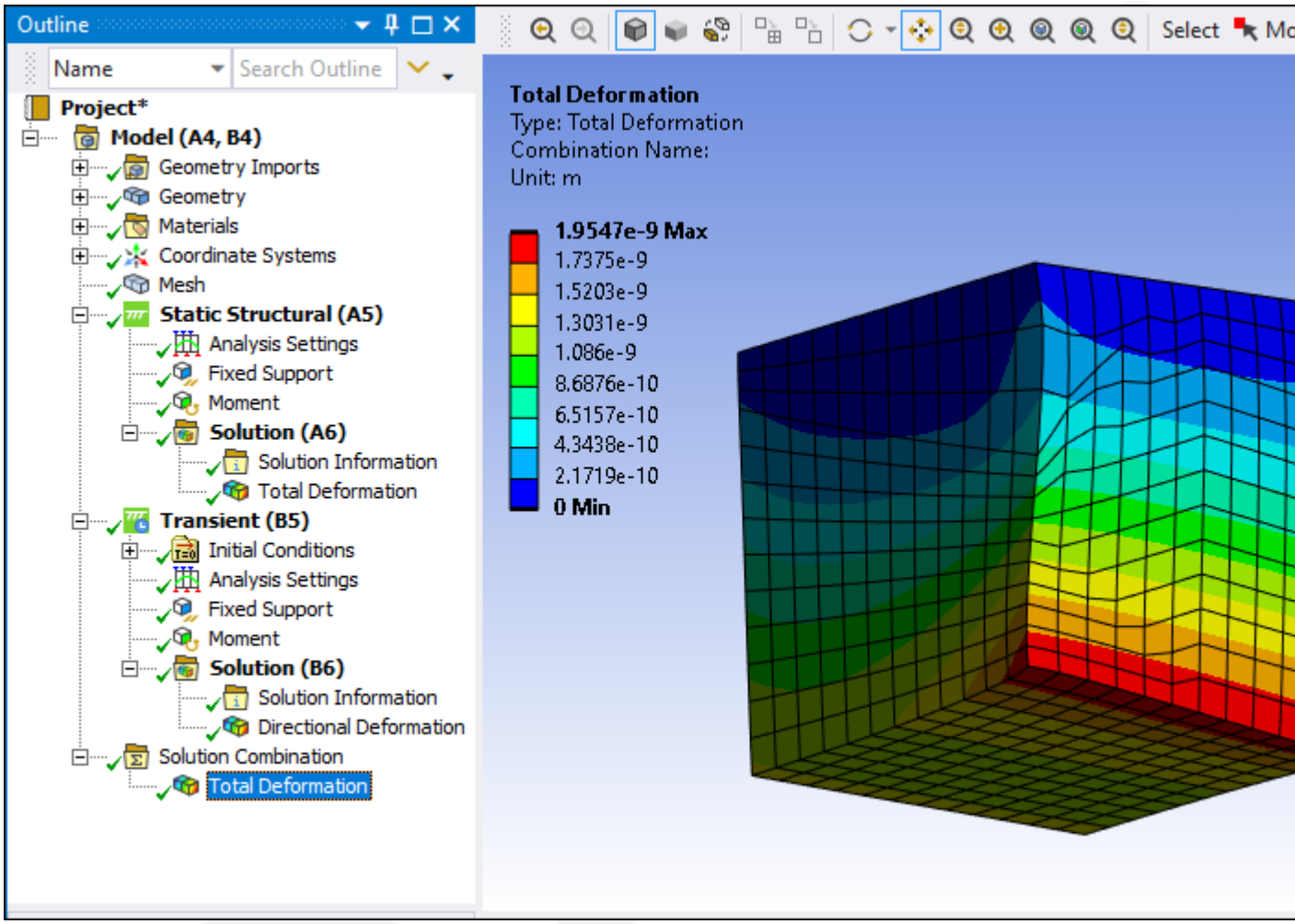
You define a Combination using the following fields:

- **Name** (optional): This field defaults to Combination 1, 2, 3, etc.
- **Type**: Specify you combination as either **Linear** (default) or **SRSS**. For these options:
  - **Linear**: Simple addition after multiplying the coefficients with the base results.
  - **SRSS**: Square Root of the sum of squares (SRSS) after multiplying the coefficients with the base results.
- **Coefficient**: Enter the coefficient or scale factor that you'll multiply the corresponding base case by.

6. Specify a desired result or results and solve the analysis. You need at least one result. When you solve, the base cases are evaluated and the result provides contours of the combined environments. An example of a result using the **Envelope** method is shown below. It displays all of the highest or the lowest values (minimums or maximums) from each environment.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



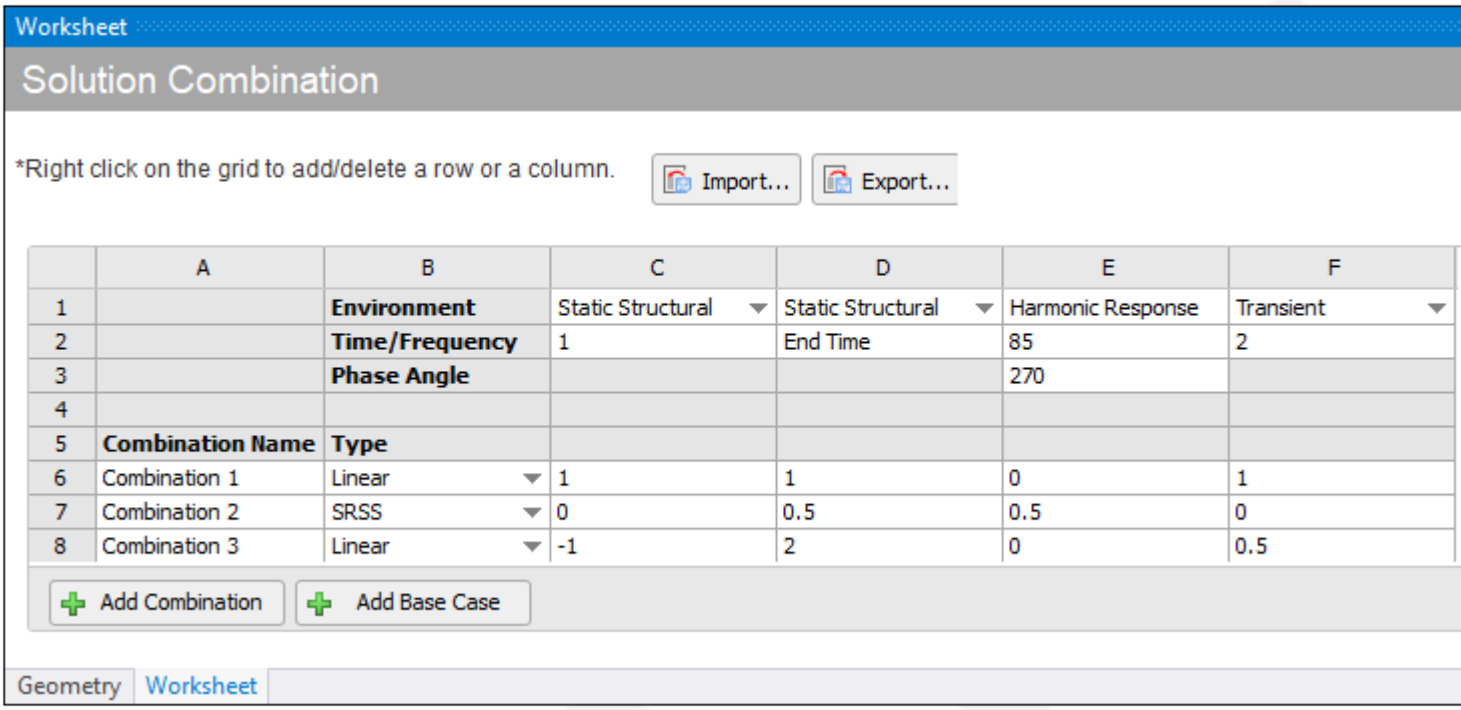
Import/Export Worksheet

Using the **Export** option of the **Solution Combination** Worksheet, you can save the settings that you create as a Comma Separated Value (`.csv`) file. Using the **Import** option, you can import these saved files into an analysis.

Interpreting Worksheet Content

Using the **Worksheet**, you can set up as many combinations as you wish and define different coefficients and input base cases.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



	A	B	C	D	E	F
1		**Environment**	Static Structural	Static Structural	Harmonic Response	Transient
2		**Time/Frequency**	1	End Time	85	2
3		**Phase Angle**			270	
4						
5	**Combination Name**	**Type**				
6	Combination 1	Linear	1	1	0	1
7	Combination 2	SRSS	0	0.5	0.5	0
8	Combination 3	Linear	-1	2	0	0.5

You interpret the base cases (columns) as:

- Base Case C: Results from Static Structural environment at Time Step **1**.
- Base Case D: Results from Static Structural environment at **End Time**.
- Base Case E: Results from Harmonic Response environment with a **Frequency** of 85 Hz and a **Phase Angle** of 270°.
- Base Case F: Results from Transient environment at Time Step **2**.

You interpret the Combinations (rows) as:

- Combination 1 (Linear): (1×C) + (1×D) + (1×F)
- Combination 2 (SRSS): sqrt($(0.5\times D)^2 + (0.5\times E)^2$)
- Combination 3 (Linear): (-1×C) + (2×D) + (0.5×F)

Supported Result Types

Once you define Worksheet content, you can define results under the **Solution Combination** object. Supported results include:

- Stresses
- Elastic Strains
- Deformations
- Fatigue Tool
- Contact Tool
- Beam Tool (Static and Transient Structural only)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Beam (Static and Transient Structural only)
- Stress Tool
- User Defined Results (review requirements (p. 2302))

Note:

When using a **Fatigue Tool** with **Solution Combination**, the desired combination is selected using the **Combination** property on the **Fatigue Tool**. All results added underneath the **Fatigue Tool** respect this selection.

Examining Results

Even though the **Solution Combination** object enables you to define and solve multiple combinations, you can only view the contour results for a single combination. However, you can select the combination for which you wish to view the results. For the supported result types, the **By** property in the Details view enables you to specify all combinations or you can specify a specific combination (as well as the additional options described below). You can also select a desired combination from **Tabular Data**.

Using the Options of the By Property

The **By** property includes the following options:

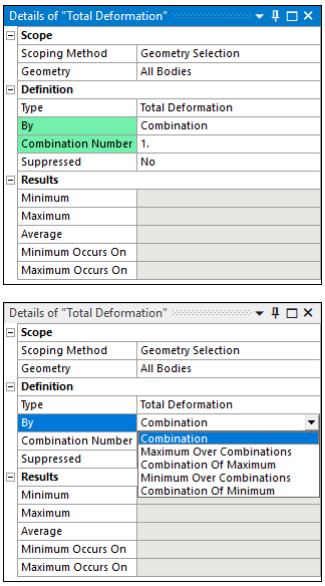
- **Combination** (default): For this setting, there is an associated property: **Combination Number**. Using the **Combination Number** property, and based on the combinations defined in the Worksheet, you can specify a desired Combination for result evaluation. The default setting for the **Combination Number** property is **Last** (last result set).

  Note:

  For the **Fatigue Tool**, the **By** property is not available. Only the **Combination Number** property is available, enabling you to specify a desired Combination.

- **Maximum Over Combinations**/**Combination of Maximum**: Each node, element, or sample point is swept through the Combinations to find its maximum result. Either the result itself is reported or the Combination at which the peak occurred is reported.
- **Minimum Over Combinations**/**Combination of Minimum**: Each node, element, or sample point is swept through the Combinations to find its minimum result. Either the result itself is reported or the Combination at which the minimum occurred is reported.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



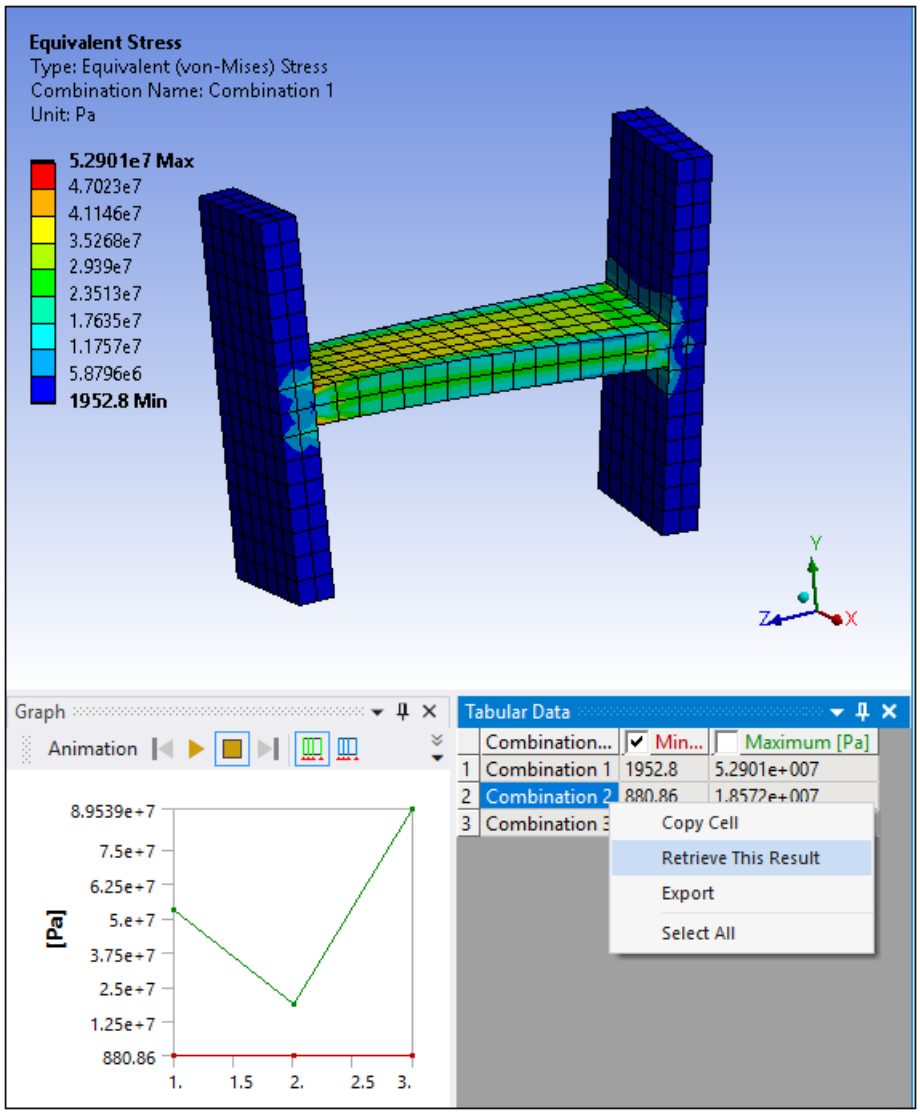
Using Tabular Data

The **Tabular Data** window displays a row for each combination defined in the **Worksheet**. Each row corresponds to a defined combination so you can right-click a row and select the **Retrieve This Result** option to view the contour for the desired combination.

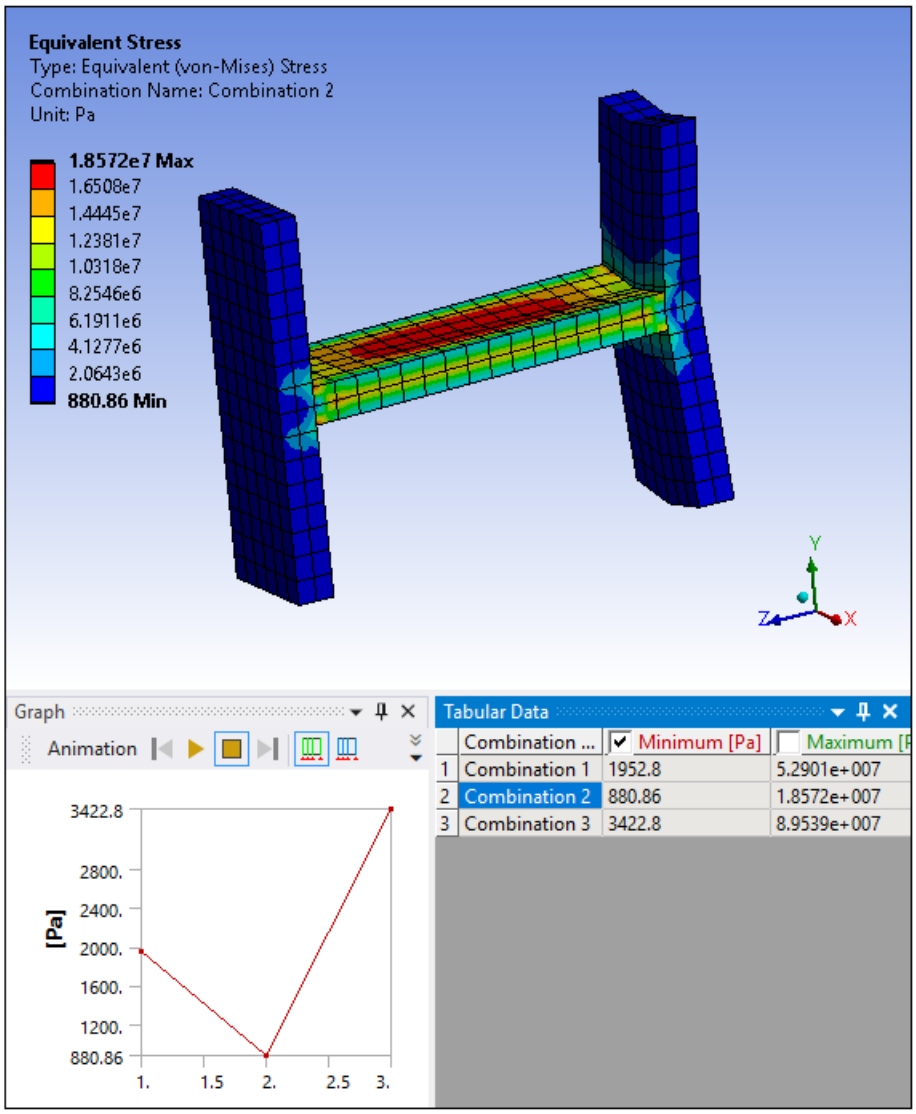
DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Solution Combination Process Requirements and Conditions

Review the following requirements and conditions. Go to a section topic:

- Order of Operations (p. 2301)
- Coordinate Systems (p. 2301)
- Stress and Strain Results for Random Vibration and Response Spectrum (p. 2301)
- Averaging (p. 2301)
- Equivalent Strains (p. 2301)
- User Defined Results (p. 2302)
- Update Project (p. 2302)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Order of Operations

The application performs the **Solution Combination** calculations in the following order of operations:

- Component results for base cases (this may involve temporal interpolation, such as **Time** = 1.5)
- Linear/SRSS combination on these component results.
- Spatial interpolation (such as scoping to Paths, Surfaces, etc.).
- Computed results (such as Equivalent Stress) from the combined component results.

These operations are generally not interchangeable or commutative.

Coordinate Systems

Based on the results you add under the **Solution Combination** object, you can specify a user-defined Coordinate System for the result. The default setting is **Global Coordinate System**. The combination is calculated for the result based on this **Coordinate System** setting.

Important:

Random Vibration and Response Spectrum analysis results are always calculated using the **Solution Coordinate System** (p. 1975). You must consider this when combining these results with results using another Coordinate System.

Stress and Strain Results for Random Vibration and Response Spectrum

When you specify a Random Vibration or a Response Spectrum analysis as a base case in the **Worksheet**, only **Normal** and **Shear** results are available (in the **Stress** and **Strain** result drop-down menus) for combination.

Averaging

For results, when you set the **Averaging** property to **Yes**, the application always uses the **entire body** for the averaging, regardless of the specified scoping. That is, if you scope a result to an edge or element face and specify averaging, the application uses the entire body of the scoped entity.

Equivalent Strains

Equivalent strains (including elastic, thermal, plastic, creep, total, and total-plus-thermal equivalent strains) are read from the result file and used directly in the linear combination formulation. The components (X, Y, Z, XY, YZ, XZ) are not used. Therefore, using the equivalent strains from the result file may lead to unexpected (or possibly negative) values.

Note:

This behavior only applies to Equivalent Strains. Other calculated results, such as Equivalent Stresses and Total Deformations, are calculated by first combining the components and then following the order of operations specified above.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

User Defined Results

The **Solution Combination** feature supports the use of User Defined Results (p. 2192) to query result file data that is not exposed through customary results. See the User Defined Results for the Mechanical APDL Solver (p. 2203) section for additional information.

User-defined results can perform basic mathematical operations; however, this option cannot reference another solved result that uses the same Identifier.

Important:

The **User Defined Result** under the **Solution Combination** object follows an order of operations different from customary (non-user-defined) results. Therefore, if you query a result, such as SEQV, that is also available as a customary result, you may observe slight differences between the two.

Note:

Note the following for a **User Defined Result** included under the **Solution Combination** object:

- The application does not support the **Display Option** property, it is a read-only property set to **Average**. Furthermore, the **Output Unit** property is not available, therefore, results display an **Average** quantity instead of a **Total**.
- Any mathematical operations specified in the **Expression** property apply to the individual base cases only, not the combined result.

Update Project

The **Update Project** option on the Workbench **Project Schematic** does not solve a **Solution Combination**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

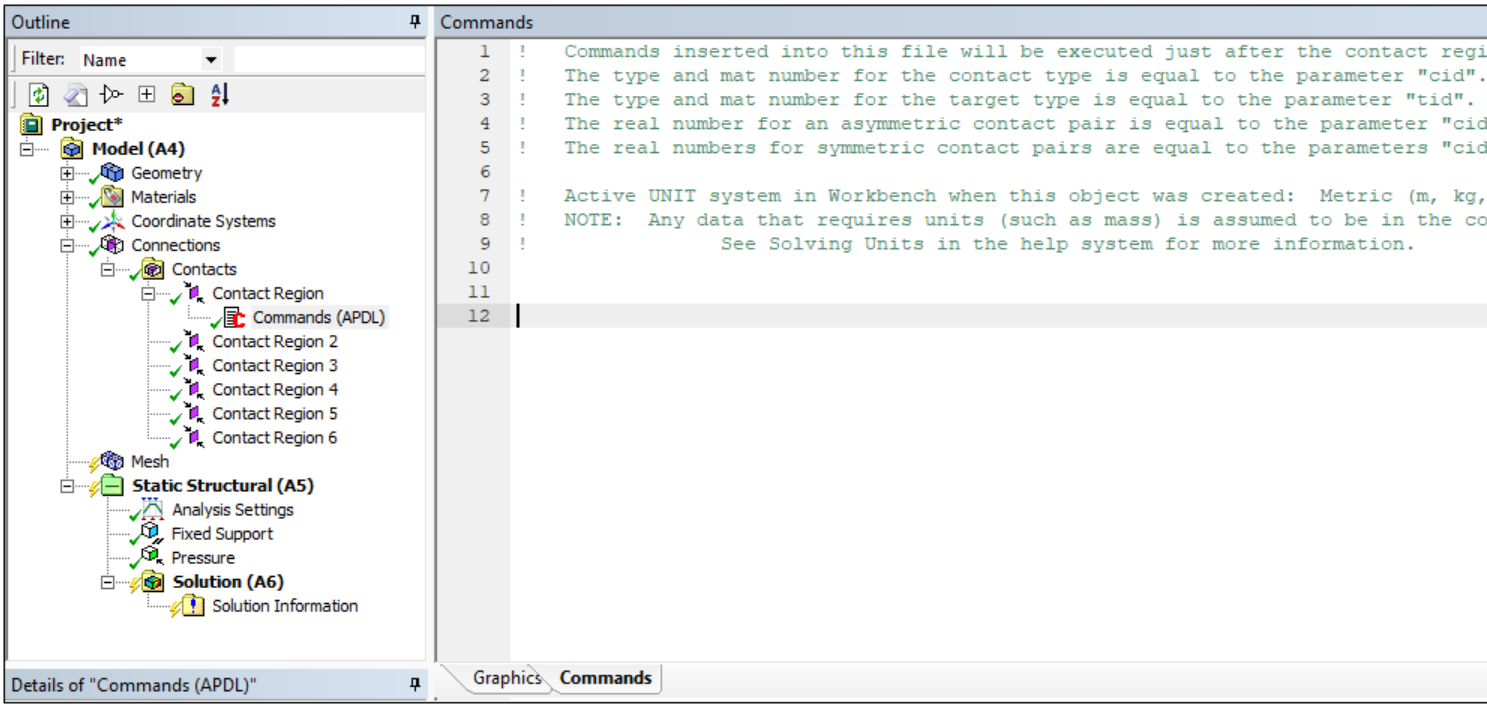
Published: 2023-10-23

Using the Command Object

Overview and Application

The Commands (APDL) object enables you to execute Mechanical APDL commands in the Mechanical application. Insert this object by first selecting the desired and supported parent object and then selecting the **Commands** option from the **Home** tab or by right-clicking on the parent object or in the **Geometry** window and selecting **Insert** > **Commands**.

Once inserted, the **Worksheet** automatically displays, as illustrated below. You use this window to make Mechanical APDL command entries. As shown, the **Worksheet** contains default instruction information. The default information displayed changes depending upon the parent object. The example shown below appears for a **Commands (APDL)** object inserted under a **Contact Region** object.



Reading and Writing Command Object Instructions

When you initiate a solution in Mechanical, all simulation data and instructions are read-into the `ds.dat` file and then processed, including the commands included in a **Commands (APDL)** object. However, because Mechanical does not read and process the `ds.dat` file, you may see that the execution of **Commands (APDL)** object instructions may not match the simulation processing of the solver as expected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Command Support Limitations

It is important to note that not all Mechanical APDL commands are available for use in Mechanical. Here are two examples of unsupported commands that have been experienced in the past:

- The **RAPPND** command, that physically alters the result file.
- The **On** setting for the **Nrmkey** field of the **MODOPT** command (to normalize mode shapes).

Caution:

Using commands that change the numbering of nodes or the numbering of elements or change how nodes are arranged on elements will most likely cause post processing errors.

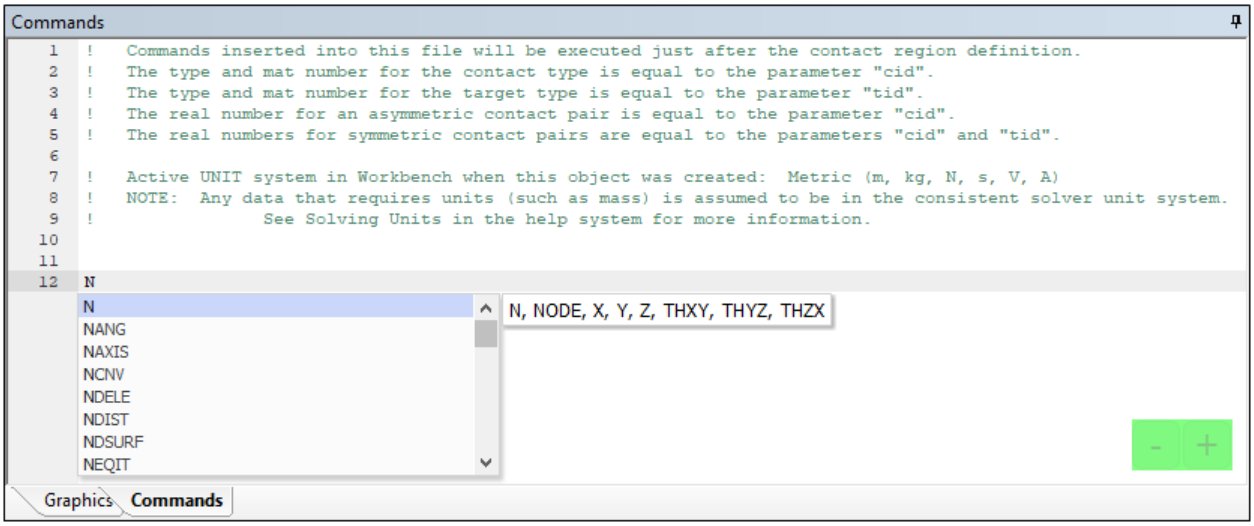
Additional Topics

The following additional topics are covered in this section:

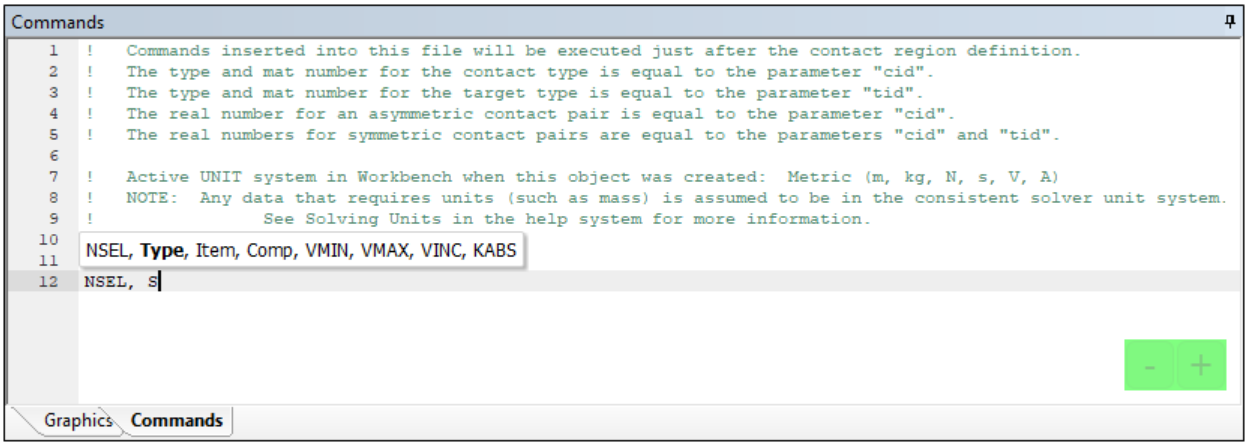
Command Entry and Mechanical APDL Entry Options
Commands (APDL) Object Properties
Commands (APDL) Object Post Processing Specifications
Commands (APDL) Objects and the Mechanical APDL Solver
Commands (APDL) Objects and the Rigid Dynamics Solver
Commands (APDL) Objects and the LS-DYNA Solver

Command Entry and Mechanical APDL Entry Options

To make an entry you must first place your cursor in an available blank line. When you have the Mechanical APDL solver selected, a drop-down appears displaying supported commands (based on the entered character) as well as a tooltip banner of supported command arguments. As you continue your command entry, the tooltip banner is maintained in the window. These features are illustrated below.

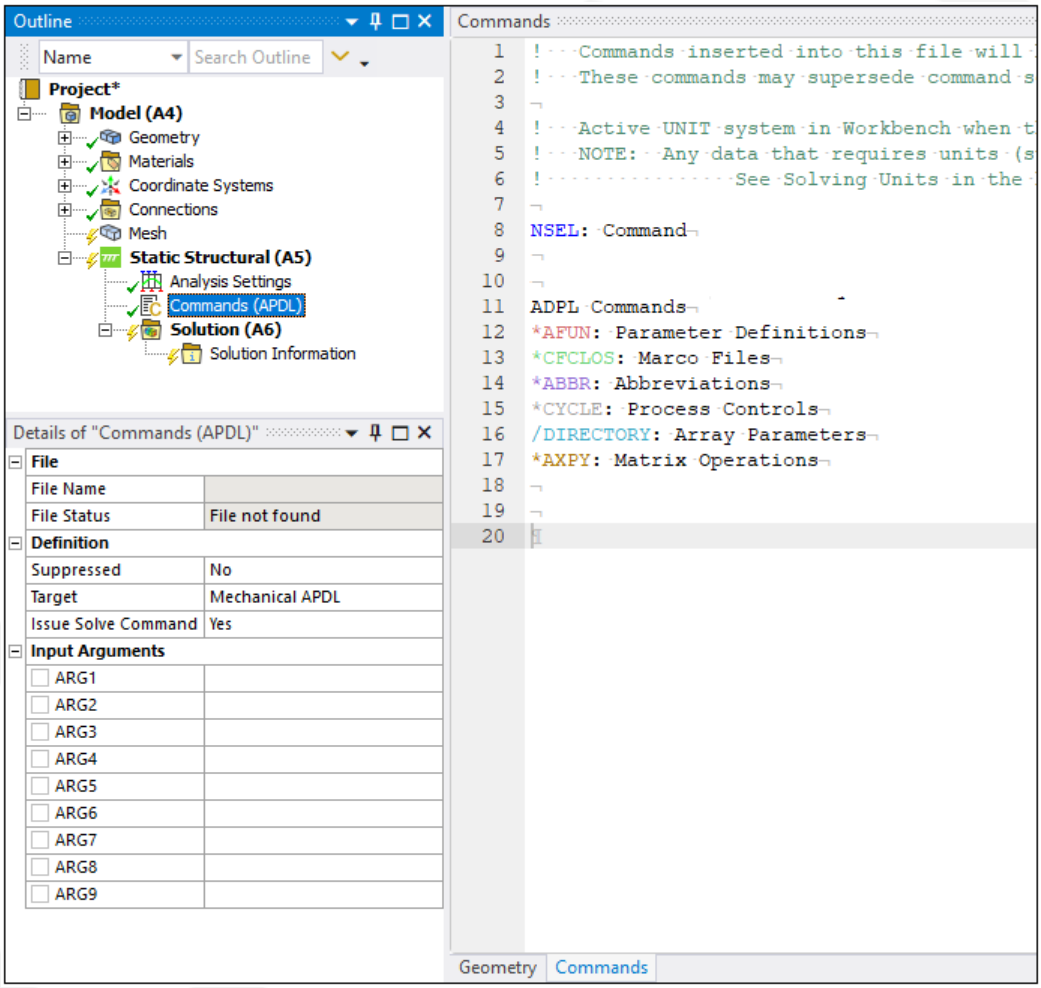


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



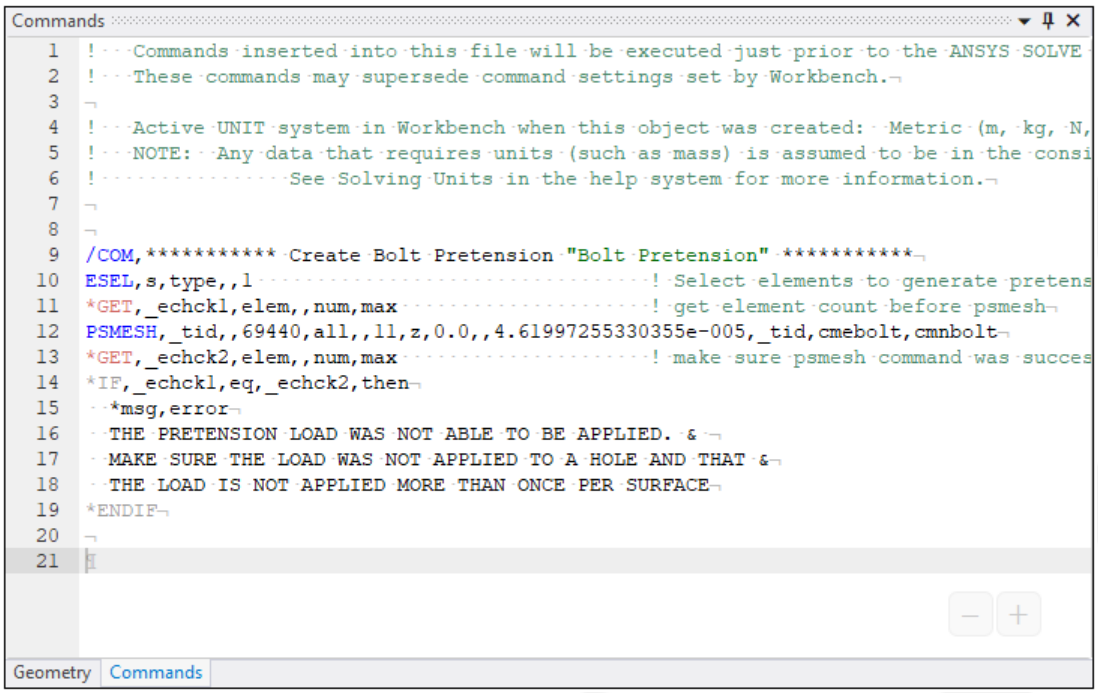
Additional **Worksheet** options include:

- Automatic syntax color highlighting for scripting commands, as illustrated below, including Commands, Keywords, fixed values, variables, parameters, etc. (Mechanical APDL or Ansys Rigid Dynamics solvers only).



Example Command Entry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



- A zoom in/out option available in the lower right corner of the **Worksheet** (highlighted above).
- A search feature accessed using the key combination **[Ctrl]**+**[F]**. The displayed dialog enables you to perform a text/character search on your command entries. This features also supports search and replace.

Furthermore, you can change default display settings and behaviors (use of Interactive Tooltips, font size, etc.) of the **Worksheet** using the preferences available in the **Options** dialog (p. 217) under the **Commands** category (p. 244).

Important:

This object supports command entry for other supported solvers.

- For the Rigid Dynamics solver, commands are expressed in Python. See the **Commands (APDL)** Objects and the Rigid Dynamics Solver (p. 2321) section for additional information.
- For the Explicit Dynamics solver, commands are expressed as keywords with comma separated arguments. See Commands Objects in Explicit Dynamics for additional information.
- For the LS-DYNA solver, commands are expressed as LS-DYNA keywords. See **Commands (APDL)** Objects and the LS-DYNA Solver (p. 2321) for additional information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- See the appropriate external documentation for command entries for any other supported solver type.

Refer to the object reference page for the Commands (APDL) object for information about the valid objects under which you can insert single or multiple **Commands (APDL)** objects.

Commands (APDL) Object Properties

The main properties of the command feature, regardless of the selected solver include:

- Solver Target (p. 2307)
- Issue Solve Command (p. 2307)
- Input Arguments (p. 2308)

Solver Target

The **Target** property in the Details view of a **Commands (APDL)** object provides a drop-down list of solver selections (Mechanical APDL, Samcef, ABAQUS, etc.). The text displayed in the **Worksheet** for the selected solver varies. If you change the solver selection, the application prompts you that a Target property change replaces the Worksheet text and gives you the option to replace or not to replace the text.

When displayed, the **Target** property is set according to the following situations:

- If all the environments in the tree have the same solver target then the **Commands (APDL)** object is tied to that solver target.
- If there is a mix of solver targets in the tree, the **Target** property is left empty and you must assign a solver target.

Important:

In order to send command entries to the solver, the solver specified for the environment being must match the solver specified in the **Commands (APDL)** object.

Issue Solve Command

This property is only available for a **Commands (APDL)** object inserted under the Environment object for analyses that include load steps (Static or Transient). This property enables you to issue the **SOLVE** command for a load step (or steps). Property options include **Yes** (default) and **No**.

This option supports all analysis types that use the Mechanical APDL Solver. It may be particularly useful for additive manufacturing process simulations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Input Arguments

Input arguments are available on all **Commands (APDL)** objects. There are nine arguments that you can pass to the Mechanical APDL application macros. Numerical values only are supported. Input Arguments are editable on the Details view of a **Commands (APDL)** object under **Input Arguments** and listed as **ARG1** through **ARG9**. If you enter a numerical value, including zero, for an argument, that value is passed along to the Mechanical APDL application. If you leave the argument value field empty, no argument value is passed for that specific argument.

Note:

If you are calling a user defined macro from within a **Commands (APDL)** object, be aware of the macro's location on the disk to make sure the macro is able to be located during the solution. Refer to the /PSEARCH command description located in the Mechanical APDL application *Command Reference* within the Mechanical APDL Help for more information.

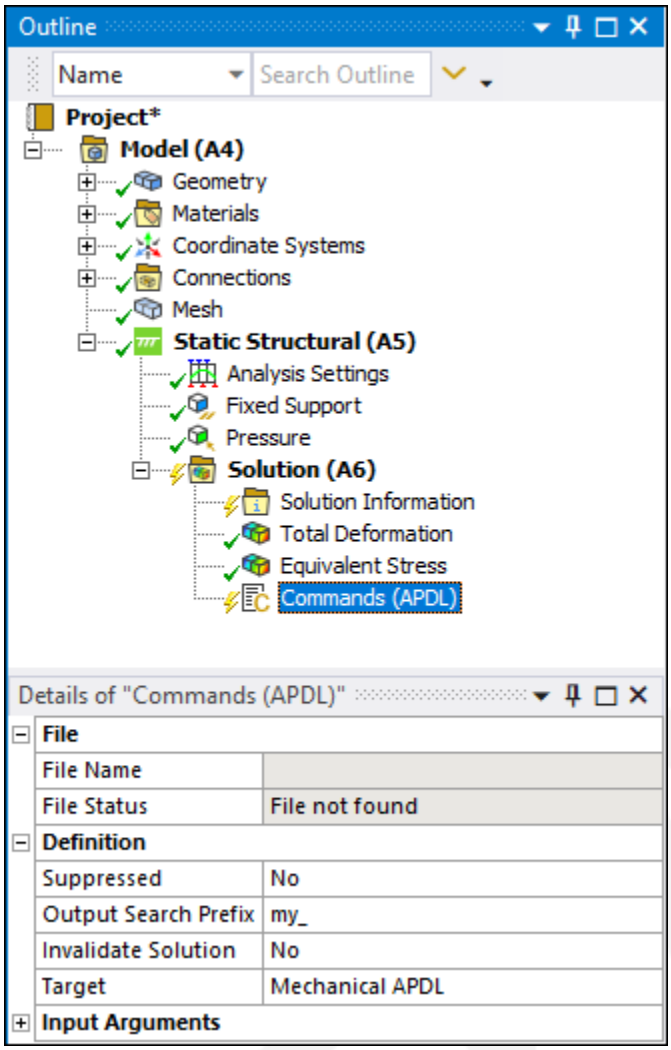
Commands (APDL) Object Post Processing Specifications

The Commands (APDL) object can perform post processing actions when inserted under the Solution object.

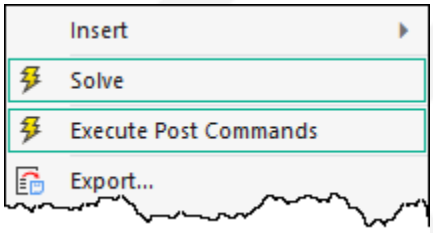
As illustrated below, when your analysis is solved and you insert a **Commands (APDL)** object, the **Solution** object changes to a ready state (lightning bolt icon) but the results maintain their successful evaluation (green check mark).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



From here, using one of the following options, you can:

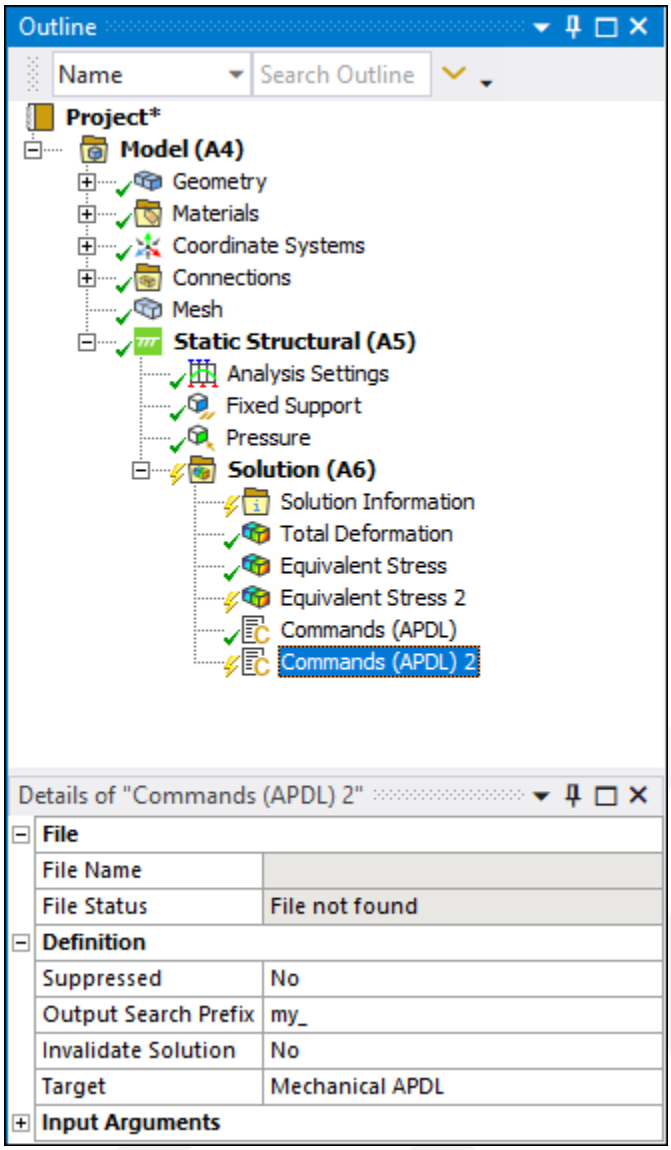


1. Resolve using the **Solve** option. This action processes all commands objects and solves any unsolved results.

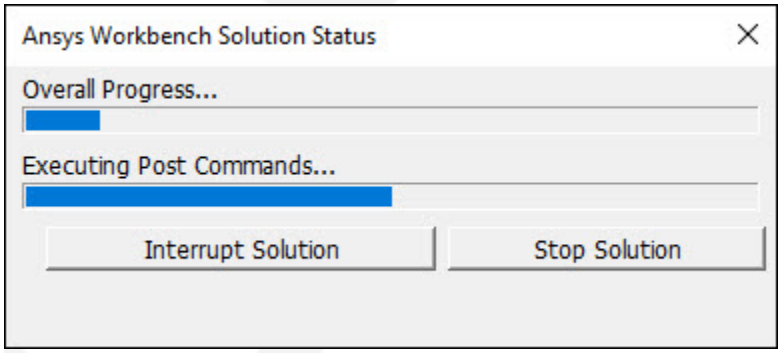
2. Resolve using the **Solve** option with the Invalidate **Solution** property to **Yes**. This action invalidates all existing results and reprocesses the entire solution.

3. Use the context (right-click) menu option **Execute Post Commands**. This option enables you to quickly process the selected commands object(s) only. This action does not evaluate any unsolved results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Also note that, for the appropriate option executed, if you display the status/progress dialog box, it shows that the solution is only processing newly specified commands.



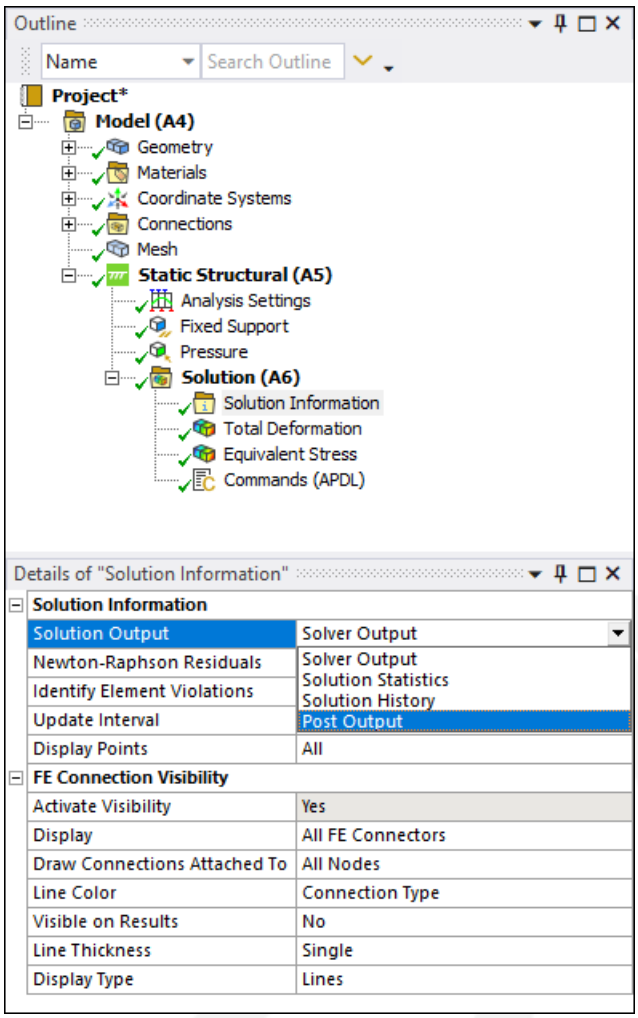
Post Output File

The post command entries generate a new and independent solution output file, `post.dat`. The post.dat file contains only the content of unsuppressed command objects. The output file can be viewed

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

in the Worksheet for the **Solution Information** object by setting the **Solution Output** control to **Post Output**, as shown below.



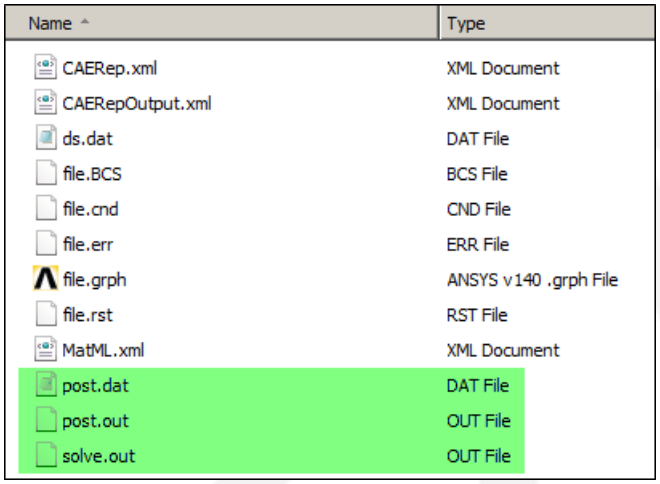
Note:

- This post processing solution *only* happens if changes or additions are made to the **Commands (APDL)** object of a **Solution** object for an otherwise solved environment. If the solution is unsolved or obsolete for some other reason, then the commands are executed as part of the normal solving process.
- Existing and post processed results are available for use with any subsequent linked analyses.
- When using this mode, Mechanical APDL runs all commands including the ones that may have existed as a part of the regular solve. Some commands may require certain variables or parameters to be active for execution or to produce correct results. As a result, it may be necessary to resume the Mechanical APDL db file by making sure that the **Analysis Settings>Analysis Data Management>** (p. 1484)**Save MAPDL db** option is set to **Yes** prior to restarting the entire solution.
- The solve mode is always In Process (p. 1803).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- If the command snippet is inserted or edited after a full or partial solution, then you can issue post-processing commands for the available result sets of the solution. The solution executes without incurring the cost of a full solve, as it sends only the post commands and will generate `post.out` as a post output file.
- If you are using the Mechanical APDL post processing command, **SET**, and you are not executing a complete re-solve of the solution or if you do not resume a database file (`file.db`) first, the default result file name is `file.rst`. Therefore, if your result file name is not `file.rst`, you need to implement the **FILE** command before any **SET** commands in order to open the proper file. An example is a thermal analysis that uses the file name `file.rth`. In this case, either a **FILE,file,rth** command or a **RESUME** command (if the `file.db` was saved during the full solution) is needed.

Note that the generated Output files are written to the Solver Files Directory and are named accordingly. An example of the directory is shown below.



Commands (APDL) Objects and the Mechanical APDL Solver

The commands issued by the **Commands (APDL)** objects affect the solution, however; they do not alter settings within Mechanical. This section describes how the following characteristics and requirements apply to **Commands (APDL)** object when used with the Mechanical APDL solver.

- Text and Units (p. 2313)
- Step Selection Mode (p. 2313)
- Point Selection Mode (p. 2314)
- User Convenience Parameters (p. 2315)
- Output Parameters: Using Parameters Defined in Solution Command Objects (p. 2315)
- Mechanical APDL Application Plots in Workbench (p. 2318)
- Commands (APDL) Objects and Materials (p. 2319)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Mechanical and Mechanical APDL Applications Conflicts (p. 2320)

Text and Units

Command text *cannot* contain characters outside of the standard US ASCII character set due to the fact that this text will propagate into the Mechanical APDL application input files and must follow the rules set aside for the Mechanical APDL application commands and input files. Use of languages other than English for the command text may cause erratic behavior. The Mechanical APDL application commands should *not* be translated.

Make sure that you use consistent units of measure throughout your simulation. The application does not convert units of measure in the **Commands (APDL)** objects. Unit-dependent entries do not get converted if you make unit system changes in the analysis. Changes of this nature can lead to inaccurate results.

Commands (APDL) object input for magnetostatic analyses must be in MKS units (m, Kg, N, V, A).

Step Selection Mode

The **Step Selection Mode** property displays when you:

- Have a **Commands (APDL)** object inserted under the environment and have **Steps Controls** (p. 1418) defined.

  or...

- Are performing a Coupled Field Harmonic, Harmonic Acoustics, or a Harmonic Response analysis with the **Multiple Steps** property (**Analysis Settings** > **Step Controls**) set to **Yes**.

  or...

- Are performing an MSUP Harmonic Response or Transient analysis (standalone or linked).

  or...

- Are performing a Random Vibration analysis.

Stepped or Multiple Step Analyses

For stepped analyses, or multiple step Coupled Field Harmonic, Harmonic Acoustics, or a Harmonic Response analyses, this property enables you to specify which sequence steps are to process the **Commands (APDL)** object. The choices are: **First**, **Last**, **All**, and **By Number**. If you select the **By Number** option, the property **Step Number** displays. You use this property to specify the step during which your command(s) will execute.

Solution Phase

For the following analysis types, the application executes the command based on the solution phase you specify.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Analysis Type	Steps to Process Command
MSUP Harmonic Response	• **All** • **Modal Solution (Standalone MSUP)** • **Harmonic Solution - All** • **Harmonic Solution - By Number (Multi Step Analysis)** • **Harmonic Expansion**
MSUP Transient	• **All** • **Transient Solution - Zero** • **Transient Solution - First** • **Transient Solution - Last** • **Transient Solution - All** • **Transient Solution - By Number** • **Transient Expansion**

Random Vibration

For a **Random Vibration** analysis, this property provides the following options:

- **Random Vibration Solution**: The application writes the **Commands (APDL)** object commands to the solver input file prior to the **SOLVE** command.
- **Participation Factors Calculation**: The application writes the **Commands (APDL)** object commands to the solver input file prior to the **PFACT** command (participation factors calculation) for **each load**.

Note:

If your analysis does not include any loads when you set the **Step Selection Mode** property to **Participation Factors Calculation**, the **Command (APDL)** object does not write any instructions to the solver input file.

Point Selection Mode

The **Point Selection Mode** property displays in the Details view of a **Commands (APDL)** object that is inserted under a Modal environment when the **Campbell Diagram** property is set to **On** (**Analysis Settings** > Rotordynamics Controls (p. 1490)). This property enables you to specify which Modal Points, as they correspond to the Campbell Diagram, are to process the **Commands (APDL)** object. Options include:

- **First**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Last**
- **All** (default)
- **By Number**

  The **By Number** property enables you to choose specific Campbell Diagram solve points. When you select **By Number**, the **Point Number** also displays. Using this additional property, you specify individual and/or intermediate Campbell Diagram solve points to execute command(s).

  The additional property **Point Number** displays when you select the option **By Number**. You use this additional property to specify the **Campbell Diagram** solve point your command(s) will execute.

  The **Point Number** value cannot exceed the **Number of Points** value specified in the Rotordynamics Controls (p. 1490). If you specify a **Point Number** that exceeds the **Number of Points** value, the application defaults to the highest solve point available.

User Convenience Parameters

When a project is saved in workbench, the application's project file management (p. 1859) creates a directory/folder structure. The generated folders house a variety of files, such as input or result files. As a part of this structure, there is a folder created that is named **user_files**.

The Mechanical APDL solver input file, **ds.dat**, includes the following parameter (variable):

_wb_userfiles_dir(1)

The value of this parameter equals the path to the **user_files** directory.

You can use this parameter with the Commands (APDL) object and perform file operations in the Mechanical APDL language. For example, by specifying this parameter, you can copy result files to the **user_files** directory.

For a more specific example, accessing external user macros located in this directory might be done using the following Mechanical APDL command:

```
/INPUT, '%_wb_userfiles_dir(1)%file_aqld1001.dat'
```

For additional information on the Mechanical APDL Command language, see the Mechanical APDL *Command Reference*.

Output Parameters: Using Parameters Defined in Solution Command Objects

When you insert a **Commands (APDL)** object under the **Solution** object, the **Details** pane includes the property: **Output Search Prefix**. You use this property to specify a desired prefix to create searchable variables/parameters in result files.

When defining a parameter that includes a **Output Search Prefix**, you must:

- Use the equals (=) symbol to specify your parameter (**`my_parameter = 1`**). Or, you can...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Specify your parameter as an input argument for a *set or *get command.

Note:

When you are entering a large number of output parameters, you could experience lag in the **Command** pane worksheet. To counter this condition, Ansys recommends that you create a complete command entry in text file format and then either import the file or copy and paste the complete entry into the worksheet.

Range Entries and Do Loops

The above entries can be created within Do...Loop statements. And, within the Do Loop, you can make use of variables inside percent (%) symbols to easily define a range of parameter values. When creating the output parameters containing variables inside of percent (%) symbols, the following rules apply:

1. The variable inside of the symbols must be set using:
   - The equals sign (variable = integer value).
   - *SET, variable, integer value.
   - The iterator of a supported Do Loop (nested do loops are also supported).

   When creating the output parameter, the (%) symbols and the variables inside them are replaced by the value the variable has been set to using equal (=) symbol or *SET command, or if the variable is the iterator of a Do Loop, a range of parameters are created according to the limits of the Do Loop.
2. Do loops with positive integers as upper and lower bounds and integer increments are supported if:
   - The loop's upper bound greater than/equal to the loop's lowed bound and the loop's increment > 0.

     Or...
   - The loop's upper bound less than/equal to the loop's lowed bound and the loop's increment < 0.
3. The Do Loop's upper and lower bounds can be parameters which are set via (variable = integer value), or (*SET, variable, integer value).

Results

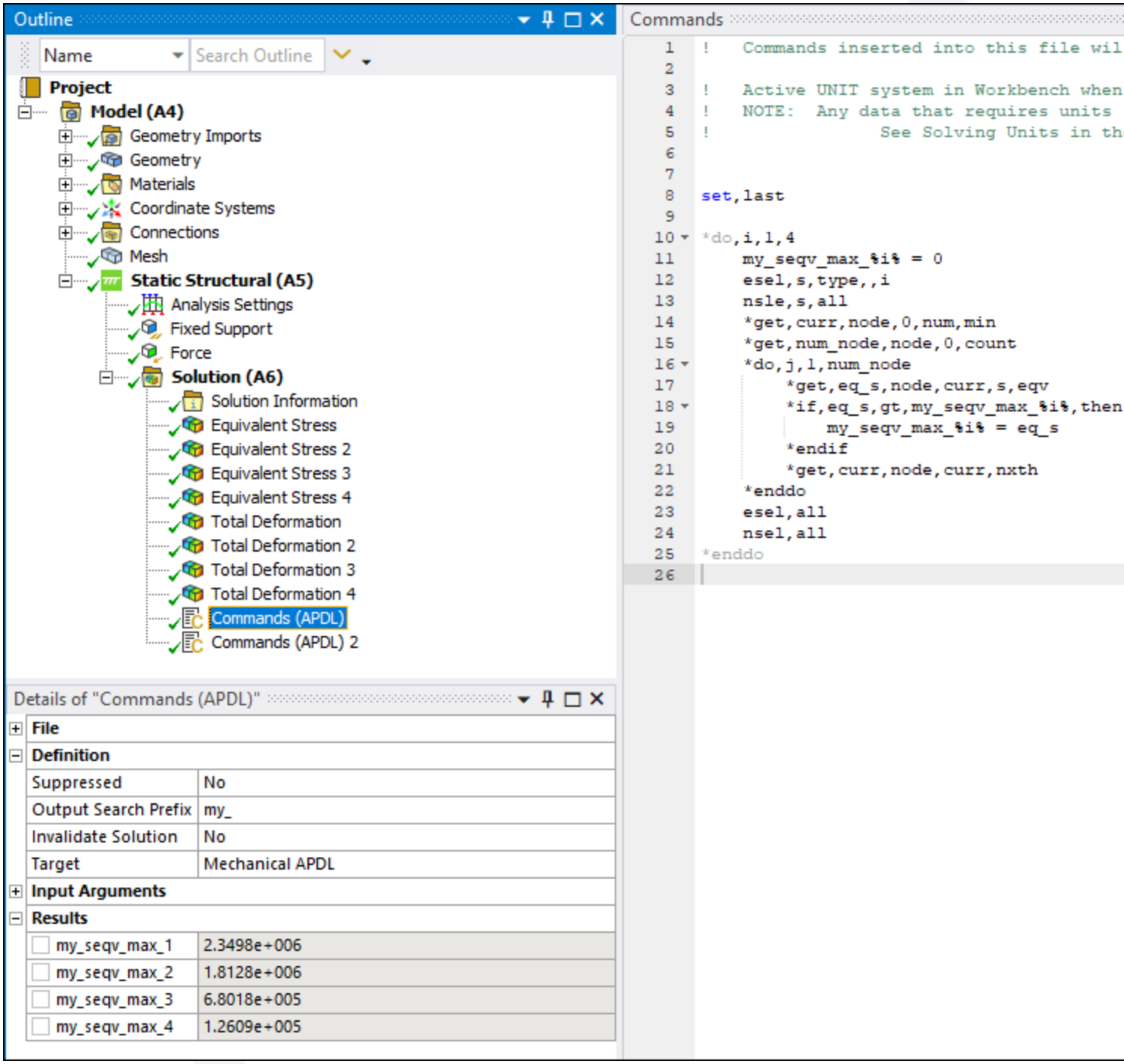
When you create an entry in the **Command** pane (worksheet) using a **Output Search Prefix** and a parameter, the application automatically:

1. Displays your new prefix/parameter in the **Results** category of the **Details** pane.
2. Searches the result file for matches (after the solution process) and then displays the results values. An example is illustrated below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

This example uses the default **Output Search Prefix** property entry, "my_". If you change the prefix at any time, the application automatically rescans the result file for matches.



Note:

- If you have parameterized an output parameter in the **Commands APDL** object, you cannot edit the command text. You need to remove the parameters to edit the text
- You can use generated parameters in Design Exploration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Mechanical APDL Application Plots in Mechanical

You can view Mechanical APDL application plots in Mechanical that result from using **Commands APDL** objects. The Mechanical APDL application plots are returned from Mechanical APDL to display in the **Worksheet**. This feature is useful if you want to review result plots that are available in the Mechanical APDL application but not in Workbench, such as unaveraged stress results or contact results only on a particular region.

To View the Mechanical APDL Application Plots in Mechanical:

1. Create one or more **Commands APDL** objects.
2. Direct plot(s) to PNG format.
3. Request plots in the **Commands APDL** objects.
4. Make sure that there is at least one **Commands APDL** object under **Solution** in the tree.
5. **Solve**. Requested plots for all **Commands APDL** objects are displayed as objects under the first unsuppressed **Commands APDL** object that appears below **Solution**.

Note:

The Mechanical APDL application PowerGraphics mode for displaying results is not compatible with **Commands APDL** objects. No results will be produced in this mode. If your command list includes the PowerGraphics mode (/GRAPH,POWER), you must switch to the Full mode by including /GRAPH,FULL at the end of the list.

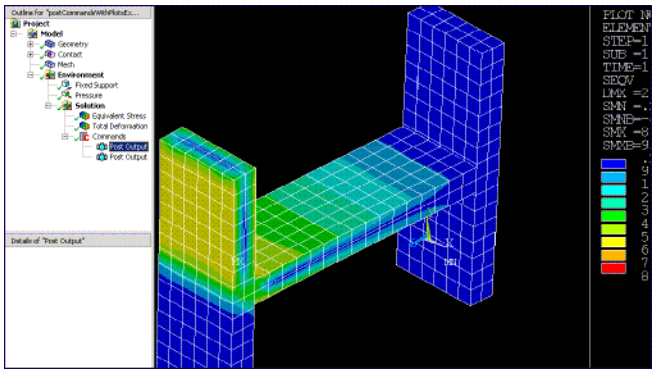
Presented below is an example of a **Commands APDL** object used to create two plots, one for unaveraged stress, and one for element error.

```
!   Commands inserted into this file will be executed immediately after the Ansys /POST1 command.
!   If a SET command is issued, results from that load step will be used as the basis of all
!   result objects appearing in the Solution folder.

set,last      ! read the last data set
/show,png     ! output to png format

/gfile,650    ! adjust size of file

/edge,1,1     ! turn on element outlines
/view,,1,1,1  ! adjust view angle

ples,s,eqv    ! plot unaverage seqv
ples,serr     ! plot element error
```

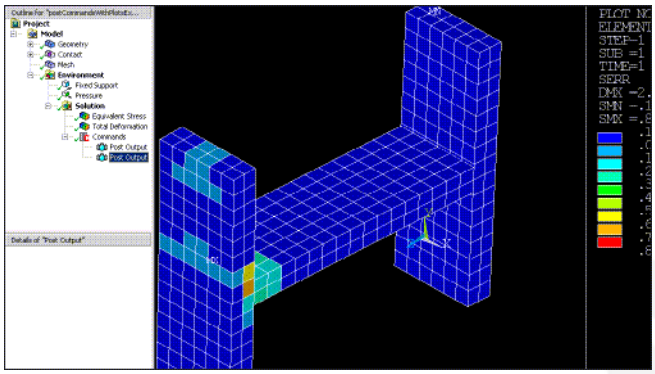
The Mechanical APDL application plots are shown below.

Unaveraged Stress Result:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Element Error Result:



Commands APDL Objects and Materials

When using **Commands APDL** objects:

- Do not change the material IDs for elements. This will cause the results retrieval form the Mechanical APDL application to Workbench to malfunction.
- Instead of adding one large **Commands APDL** object to change all of the materials, add individual **Commands APDL** objects under each part. That way you will be able to reference the **`matid`** in the **Commands APDL** object for the material ID of the elements that make up the part. You will also only need to enter the adjusted coefficient of thermal expansion and not the other materials.
- Use the **Worksheet** (p. 192) view of the **Geometry** object to determine which materials are assigned to specific parts.
- Click the right mouse button on a selected item in the **Worksheet** view, then choose **Go To Selected Items in Tree** (p. 192) to add **Commands APDL** objects.
- Copy and paste **Commands APDL** objects from one part to another that have the same material assignment.

Note:

When specifying the same material to multiple bodies using Material Assignment object, the application can no longer identify the bodies using the material identifier (**`matid`**) in the solver input file. In this case, you can use the **`typeids`** list to identify a body. The identifier **`typeids`** is a one-dimension array parameter that you can use to access the type numbers

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

for a body. You can access type numbers using a subscript (enclosed in parentheses) to identify the required item of the array. For example, to access the first type number for the body use `typeids(1)`.

Mechanical and Mechanical APDL Applications Conflicts

In order to utilize the **Command** object feature and execute Mechanical APDL application commands in Mechanical, you must have a thorough understanding the use of Mechanical APDL Commands. Moreover, you need to understand that there are times when Mechanical APDL commands may conflict with the internal settings in Mechanical and as a result adversely affect your solution values.

The following are some common conflicts that could arise:

- If you define only linear elastic properties in Engineering Data, it is possible to use the Mechanical APDL application commands in a **Commands APDL** object to override the material properties defined in Engineering Data or even change the linear elastic material model to a nonlinear material model, such as adding a bilinear kinematic hardening (BKIN) model. In this case, the solution uses the BKIN model defined in the **Commands APDL** object. However, since Mechanical is unaware of the nonlinear material specified by the **Commands APDL** object, nonlinear solution quantities such as plastic strain will not be available for postprocessing.
- The Mechanical APDL application applies nodal boundary conditions in the nodal coordinate system. For consistency, Mechanical must sometimes internally rotate nodes. As a result, any node-based boundary conditions defined in a **Commands APDL** object will be applied in the rotated nodal coordinate system.
- Commands support the definition of Mechanical APDL arguments via the settings of the properties **ARG1** through **ARG9**. Once a value for one of these arguments is set, it will be retained for the remainder of the Mechanical APDL solve run unless explicitly set to zero in the Commands text.
- Mechanical assumes certain behaviors with respect to Mechanical APDL solution file names and locations. Therefore, unexpected behavior could occur if Mechanical APDL commands such as **/RENAME**, **/FILNAME**, and **/ASSIGN** are used in a **Commands APDL** object which alters the solution files.

Using Mechanical APDL Commands in the Mechanical application requires a working knowledge and experience with Mechanical APDL Commands and therefore, it is your responsibility to make sure that any command that you are issuing does not conflict with any existing Mechanical application requirement.

Important:

It is important to note that not all of the Mechanical APDL Commands are available for use in Mechanical. For example, the command **RAPPND**, that physically alters the result file is not currently supported. Furthermore, using commands that change the numbering of nodes or the numbering of elements or change how nodes are arranged on elements will most likely cause post processing errors.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Commands (APDL) Objects and the Rigid Dynamics Solver

The following information applies to **Commands (APDL)** objects used with the Rigid Dynamics solver. Their use is very similar to **Commands (APDL)** objects used in the Mechanical APDL solver, but their behavior may differ. This section highlights these differences.

The Rigid Dynamics solver commands are based on Python and follow the Python syntax. See Command Reference for Rigid Dynamics Systems (p. 561) for a complete list and descriptions of commands available with the Rigid Dynamics solver.

Input Arguments:

As with the Mechanical APDL solver, the Rigid Dynamics solver handles up to nine input arguments (see **Commands (APDL)** Object Properties (p. 2307)). The are available in the Rigid Dynamics commands using variables `_arg1` to `_arg9`.

Output Parameters: Using Parameters Defined in Solution Command Objects

As with the Mechanical APDL solver, **Commands (APDL)** objects at the solution level can be used to retrieve values such as output parameters. Their use is similar to Mechanical APDL (see **Commands (APDL)** Objects and the Mechanical APDL Solver (p. 2312)) except for the following differences:

- The Rigid Dynamics solver is case sensitive.
- Unlike Mechanical APDL, it is not possible to perform post-only solve. Modifications to **Commands (APDL)** objects at the **Solution** level require a full solve.

Commands (APDL) Objects and the LS-DYNA Solver

The following information applies to **Commands (APDL)** objects used with the LS-DYNA solver. When you select a Commands object from the Mechanical user interface, it inserts a **Keyword Snippet (LS-DYNA)** object into the LS-DYNA system. The use of Keyword Snippet objects is very similar to **Commands APDL** objects used in the Mechanical APDL solver, but their behavior may differ. This section highlights these differences.

The LS-DYNA solver commands are based on the LS-DYNA language and follow the LS-DYNA syntax. See Keyword and Theory Manuals for a complete list and descriptions of keywords available for use with the LS-DYNA solver.

Input Arguments

As with the Mechanical APDL solver, the LS-DYNA solver accepts up to nine input arguments (see **Commands (APDL)** Object Properties (p. 2307)) per command object. They are available in the **Details** panel of the **Keyword Snippet (LS-DYNA)** object under **Input Arguments**, fields **ARG1** to **ARG9**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Syntax for Using Parameters Inside Command Objects

Parameters can be referenced anywhere in the input by placing an ampersand ("&") immediately preceding the parameter name. If a minus sign ("−") is placed directly before "&" with no space ("−&") the sign of the numerical value is switched.

Output Parameters: Using Parameters Defined in Solution Command Objects

LS-DYNA does not support **Commands (APDL)** objects at the Solution level.

Parameters used with the LS-DYNA solver are case sensitive.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Using Python Code

The Python Code object enables you to execute python code in response to events occurring in Mechanical for a given workflow. When inserted, an editor pane displays automatically and includes features such as auto-completion and syntax highlighting. The events related to Python code execution can be split into two main categories:

1. Use python API to query information about the data model and inject MAPDL commands into the solver input file at different points in the construction of the input file.
2. Execute python code based on events in Mechanical.

Jump to a topic using the following links:

Using Python Code for MAPDL Solver Input

Using Python Code for Event Based Code Execution

Resuming Databases with Python Code Objects

Sharing Scope

Working with the Property Provider

Note:

For security purposes, the application does not automatically connect **Python Code** (object) callbacks when you launch Mechanical. You must first activate the **Connect/Run Python Code Objects when Mechanical is Launched** preference contained in the Mechanical category of the Workbench **Options** dialog (**Tools** > **Options** > **Mechanical**).

Python code callbacks can execute code on a host system with the same privileges as Mechanical. Therefore, when you open a project that includes a **Python Code** object, and that object contains unreliable code, your system could become exposed to external threats (system scans, data deletion, installation of undesired software, etc.).

You can activate the above preference to automatically connect **Python Code** objects when opening projects, however, Ansys recommends that you keep this option unchecked if you receive and run projects from users other than yourself.

Important:

Also note that if the **Python Code** object includes scripts related to **Design Points**, you must activate the above option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

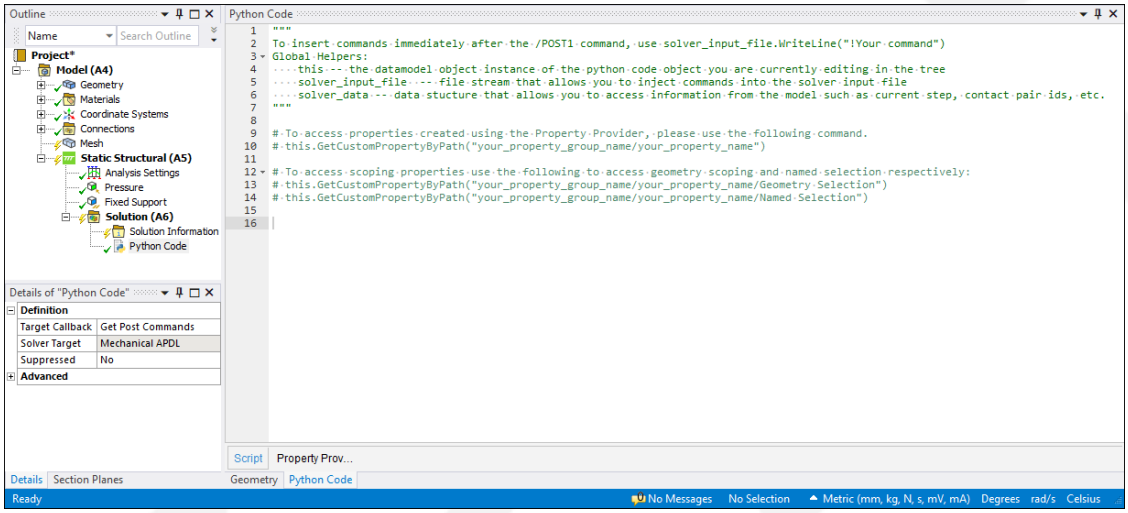
Published: 2023-10-23

Using Python Code for MAPDL Solver Input

Python Code object works similarly to how the existing Commands (APDL) object works, but it provides a way to interact with the solver input file using the MAPDL commands. Also like **Commands (APDL)** object, the location of the **Python Code** object in the Outline decides where the injected MAPDL is added to the solver input file. You can insert this object under the following **Outline** objects:

- **Body**
- **Contact Region**
- **Environment**
- **Solution**

Once inserted, a script entry pane displays, as illustrated below. As shown, the entry pane contains default information about what entries to make. This default information changes depending upon the parent object. For this example, the Solution object is the parent object.



As shown, the command inserted **`solver_input_file.WriteLine("Your Command")`**. This command will be inserted following the material definition in /POST1. Furthermore, it lists three global helpers that enable you to easily interact with pertinent information, including:

- **`this`**: A variable that you can utilize that corresponds to the current Python Code object.
- **`solver_input_file`**: This indicates the file handle you use to write MAPDL commands to the input file.
- **`solver_data`**: This is a structure that gives you access to information during the solution, such as MaxElementId , SolveType, etc.

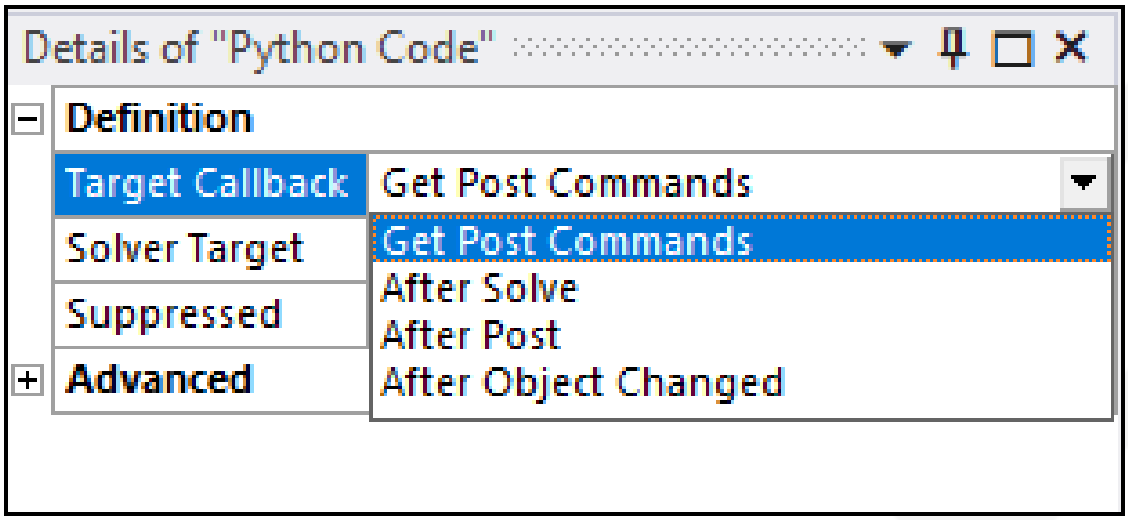
Using Python Code for Event Based Code Execution

In addition to executing Python code during the solution process, each Python object you insert into the Outline has a **Target Callback** property that specifies when the object's code should execute based

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

on Mechanical events. The following example of the Details pane shows the **Target Callback** property options for a **Python Code** object inserted under the **Solution** object.



Specifying Callbacks

The options contained in the **Target Callback** property is based on where in the tree you add the **Python Code** object. The application executes callbacks when their parent object is processed. If more than one **Python Code** object is contained under the same parent, the application executes each object.

The exception is the **Model** object. If a **Python Code** object is contained under the Environment object with the **Target Callback** property set to **After Object Changed**, the code is only executed if any object under the Environment changes. However, for a **Python Code** object that is contained under the **Model** object with the **Target Callback** property set to **After Object Changed**, the code is executed when any object in the system changes.

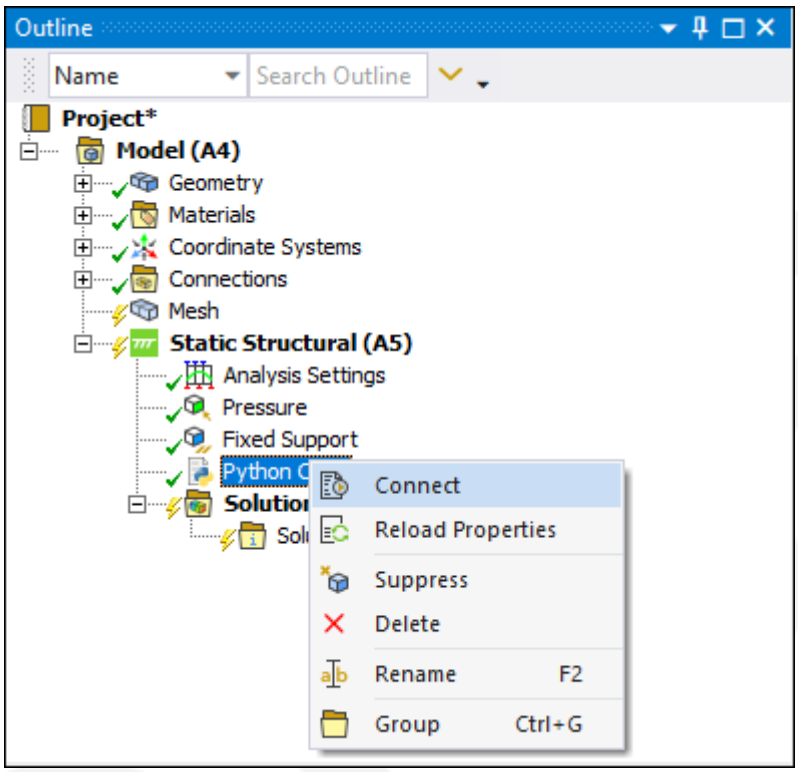
Parent Object	Target Callback Options
Model	After Mesh Generated After Geometry Changed After Object Changed
Body	Get Body Commands After Object Changed
Contact Region	Get Contact Commands After Object Changed
Environment	Get Solve Commands Before Solve After Object Changed
Solution	Get Post Commands After Solve

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Parent Object	Target Callback Options
	After Post After Object Changed

Connecting Callbacks

When the text of the Python Code object is modified, the object becomes under-defined as the Python code is no longer connected to the callback. It is necessary to connect the code changes once completed or the code will not be executed. Note that as soon as you start editing the text in the editor, the Python Code object will become under-defined and the Details pane will indicate that the Connected property of the Python Code object has become false. At this point, once you are finished making your code changes, use the **Connect** button in the context tab in the ribbon or use the right-click context menu on the object in the tree to Connect the Python Code object.



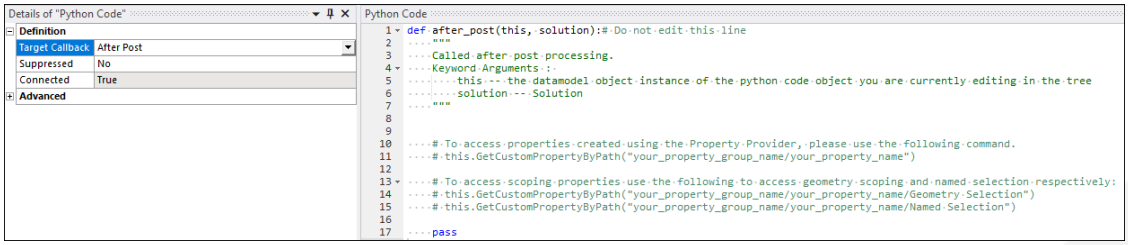
Example

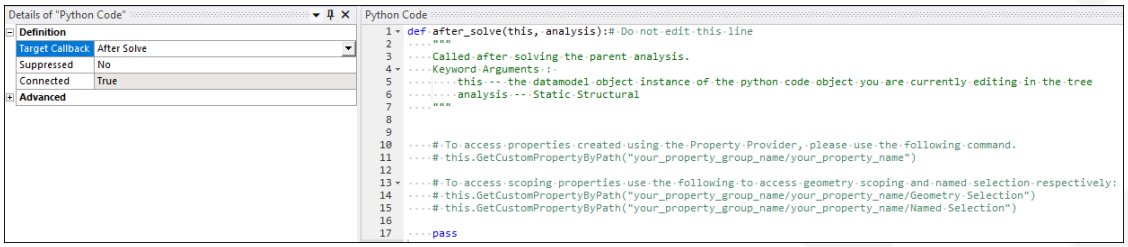
Here are some examples of the default content for a **Python Code** object inserted under the **Solution** object with the **Target Callback** property set to **After Post** and **After Solve**. The function name corresponds to the type of event.

At the top of the editor, you will see a function which will take in different arguments based on different callbacks. In the following examples, the "after" callbacks takes in "this" and "solution" as arguments to this function definition. Any code that you want to execute on the given event should be placed within the function definition replacing the pass.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



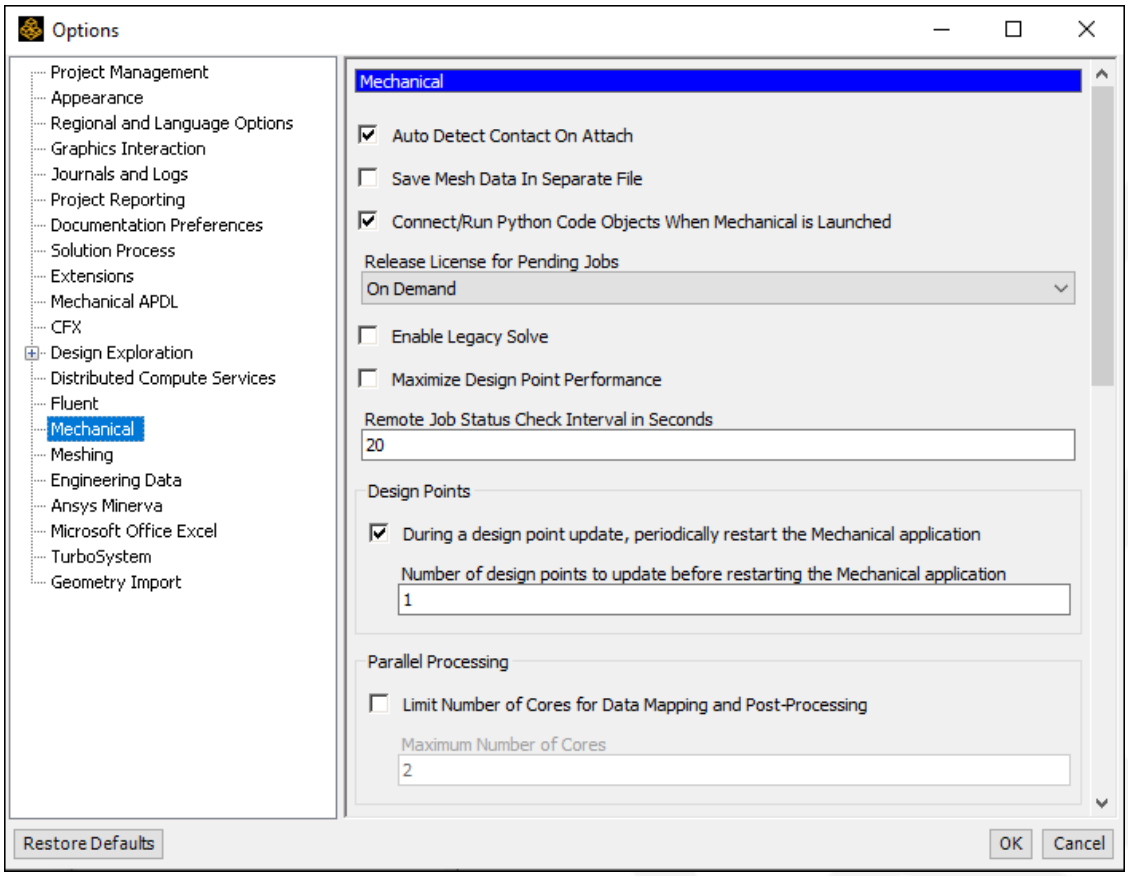


Resuming Databases with Python Code Objects

When you open a saved analysis that includes **Python Code** objects, by default, the objects display in an under-defined state. This is done so that callbacks are not automatically connected to the objects.

To change this default setting, go to the Workbench project page and open the **Options** dialog by selecting **Tools** > **Options**. Highlight the **Mechanical** option and then select the checkbox for the **Connect/Run Python Code Objects when Mechanical is Launched** property.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

For security purposes, the application does not automatically connect **Python Code** (object) callbacks when you launch Mechanical. You must first activate the **Connect/Run Python Code Objects when Mechanical is Launched** preference contained in the Mechanical category of the Workbench **Options** dialog (**Tools** > **Options** > **Mechanical**).

Python code callbacks can execute code on a host system with the same privileges as Mechanical. Therefore, when you open a project that includes a **Python Code** object, and that object contains unreliable code, your system could become exposed to external threats (system scans, data deletion, installation of undesired software, etc.).

You can activate the above preference to automatically connect **Python Code** objects when opening projects, however, Ansys recommends that you keep this option unchecked if you receive and run projects from users other than yourself.

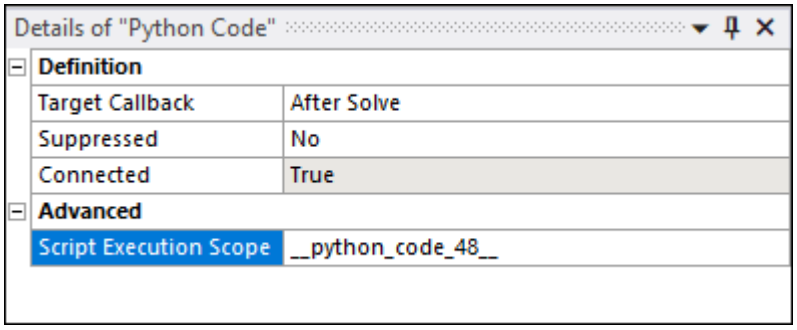
Important:

Also note that if the **Python Code** object includes scripts related to **Design Points**, you must activate the above option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

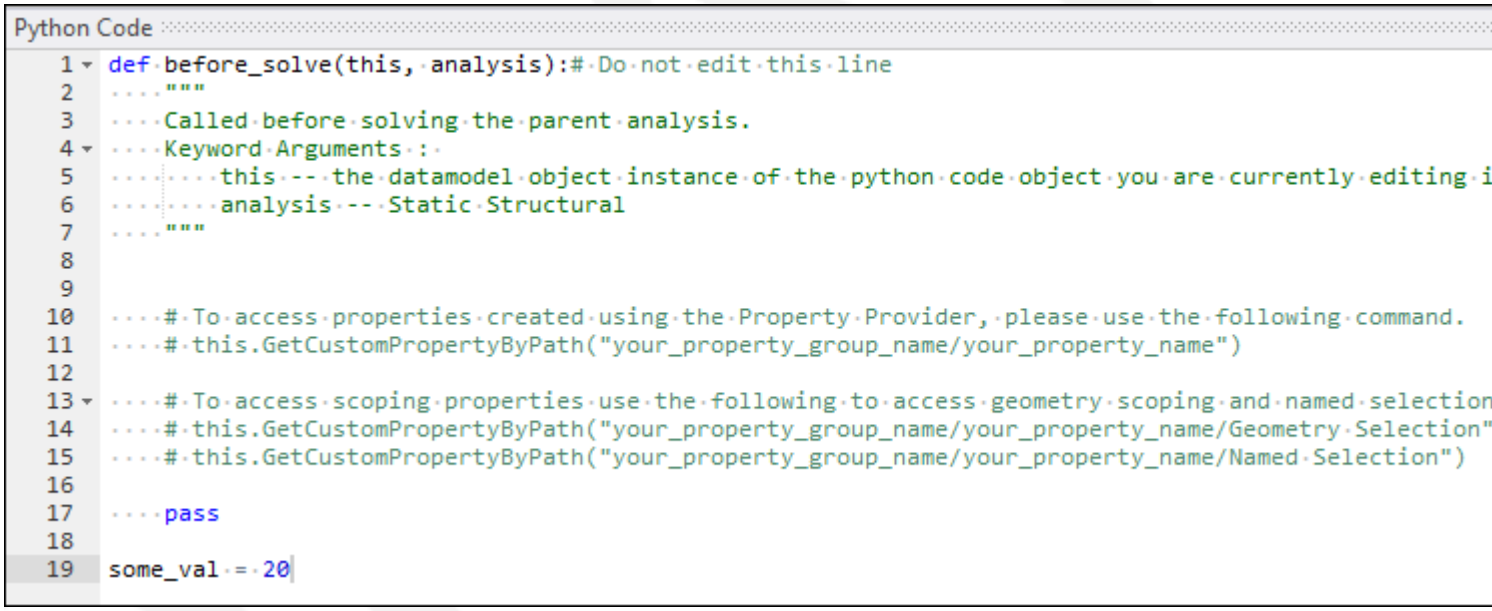
Sharing Scope

The **Script Execution Scope** property, illustrated below, enables you to share the scope in which the script runs between two or more python code objects.



As shown below, let's say you set the script execution scope property to "shared_scope" for two python code objects. Then you can define variables in one global scope and then use them in the other. The value of some_other_val in the second object will effectively be 20 because the scopes are shared between the two python objects.

Script for Python Object 1



Script for Python Object 2

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Python Code

```
def after_solve(this, analysis):# Do not edit this line
    """
    Called after solving the parent analysis.
    Keyword Arguments :
        this --- the datamodel object instance of the python code object you are currently editing i
        analysis --- Static Structural
    """


    # To access properties created using the Property Provider, please use the following command.
    # this.GetCustomPropertyByPath("your_property_group_name/your_property_name")

    # To access scoping properties use the following to access geometry scoping and named selection
    # this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Geometry Selection"
    # this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Named Selection")

    pass

some_other_val = some_val
```

Working with the Property Provider

The property provider will allow a user to add custom properties to the **Python Code** object. These properties will be available in the in the UI and will also be available through the scripting API.

Jump to a Property Provider topic:

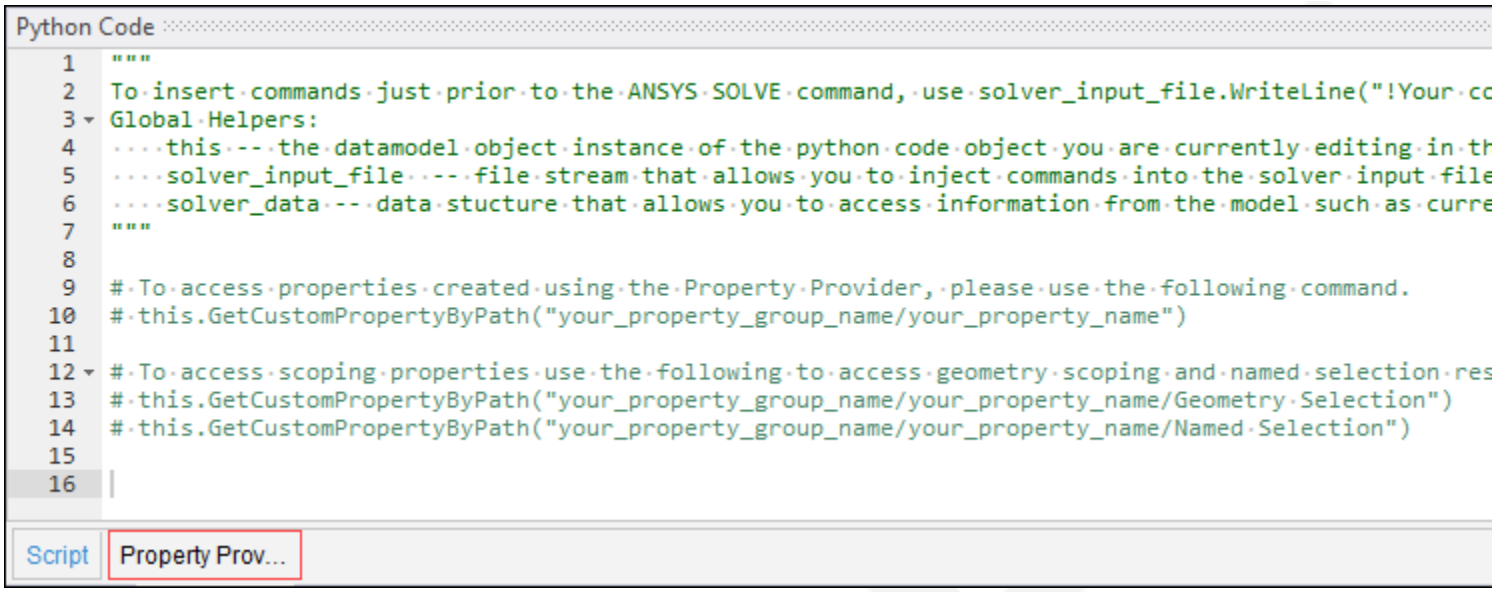
- Usage (p. 2330)
- Property Controls (p. 2331)
- Working with Properties (p. 2332)
- Controlling the Properties (p. 2334)
- Defining Callback Handlers (p. 2336)

Usage

To use the property provider, you simply set the PropertyProvider property on the Python Code object to an instance of a class that implements IPropertyProvider interface. There's a class template provided in the Property Provider tab in the Python Code UI. You can use this template as is to add custom properties or make changes as necessary to customize.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



The Provider class in the Property Provider does the following for you:

1. Implements the IPropertyProvider interface
2. Adds a helper AnsGroup class that groups the properties in the details view.
3. Exposes some other helper methods that allow you to:
   - Access the properties that you have created via scripting API.
   - Control different aspects of your properties, such as whether they are visible, read-only, etc.

Property Controls

You can create properties of four different control types.

- Expression
- Double
- ApplyCancel
- Options

These property control types can be further extended, and custom properties can be created using these base properties. An example of this is the scoping property, which you will see being added as an example in the property provider tab. You can access the python code that extends these base properties in "INSTALL_DIR/aisol/DSPages/Python/mech_templates/property_templates.py". These

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

property templates show how we can combine Options, and ApplyCancel property to create a scoping property that you may see in an object like Pressure.

Note:

The expression control is for a string property. It does not imply connection to Mechanical expressions.

Working with Properties

Let us work through an example to understand the various aspects of the Provider class in the provider tab.

Adding New Properties

Let us first start by adding new properties using the Provider class. The idea here is that you create groups using the provider instance, then you add properties to those groups, and you connect the provider instance back to the Python Code object. After you have written the code to do this, last step is to Reload the properties using option in the context tab in the Ribbon or right click context menu in the tree.

You will see some helper code in a function named reload_props in the property provider tab that shows how to create a group and add new properties to it.

```
def reload_props():
    #Create an instance of the provider.
    provider = Provider()

    # Add a group with the group name Group 1 to the provider
    group = provider.AddGroup("Group 1")

    # Use the group returned to add a property called Property 1 with control type Double
    prop = group.AddProperty("Property 1", Control.Double)

    # Double properties have a member called ValidRange that can be assigned a tuple that
    # controls the validity of the property
    prop.ValidRange = (1.0, 4.0)

    # Create an instance of a scoping property in the same group as the double property.
    # Here we pass in property type as a string because this type is a custom property
    # Created using templates that extend the base properties.
    scoping_prop = group.AddProperty("Scoping Property", "Scoping", "property_templates")

    # Add a group with the group name Group 2 to the provider
    group_2 = provider.AddGroup("Group 2")

    # add a double property to this group
    double_prop_2 = group_2.AddProperty("Property 2", Control.Double)

    #Finally, we set the PropertyProvider member on this to the provider instance. "this"
    # refers to the current Python Code object that you are working with.
    this.PropertyProvider = provider
```

Important:

Property and group names must be unique, this will come into play later when we access the properties from the property provider.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Removing Properties

To remove properties, you simply remove the code that is adding the property to the group in the provider and reload the properties again. In our example, let's remove "Group 2" and "Property 2," to do that we will remove the following lines from the reload_props() function.

```
#Add a group with the group name Group 2 to the provider
group_2 = provider.AddGroup("Group 2")
#add a double property to this group
double_prop_2 = group.AddProperty("Property 2", Control.Double)
```

Important:

This action will remove the properties that you have removed from your code but will also reset the other properties which means any values assigned to the properties will be lost when you reload.

Accessing the Properties

You can access the properties using the path to the property from the **Python Code** object. The path to a property is constructed by concatenating the property group name and property name, separated by a forward slash. For example, if you have a property named "Property 1," in group named "My Group", the path of this property would be "My Group/Property 1."

For example, if you want to access the double property's value, you would do the following:

```
prop_val = this.GetCustomPropertyByPath("Group 1/Property 1").Value
```

To get the value of the scoping property in the example, you must do things a bit differently because it's an extended property. The way the extended property is written, it uses the strings "Geometry Selection," "Named Selection," "Define By" in all scoping property instances and uses the property name provided to make the property names unique. For example, if you created a scoping property with name "Scope 1," to access the geometry selection property value of that property you would use the name "Group Name/Scope 1/Geometry Selection."

To access the geometry selection value of the scoping property that was created in the previous example, do the following:

```
geom_sel_val = this.GetCustomPropertyByPath("Group 1/Scoping Property/Geometry Selection").Value
```

Parameterization

You can make your custom properties parameterizable. However, this is only supported for Double properties. To make your properties parameterizable, you need to set the CanParameterize property on the property instance to True. As a default, this will turn the property into an input parameter. To make this property an output parameter, you must set the property ParamterType to Paramter-Type.Output. Taking the script from the first example if you wanted to make Property 1 an input parameter and Property 2 an output parameter you would make the following changes.

```
#Use the group returned to add a property called Property 1 with control type #Double
prop = group.AddProperty("Property 1", Control.Double)

#Will make prop an input parameterizable property
prop.CanParameterize = True
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#add a double property to this group
double_prop_2 = group2.AddProperty("Property 2", Control.Double)

#Will make double_prop_2 an output parameter
double_prop_2.CanParameterize = True
double_prop_2.ParameterType = ParameterType.Output
```

Controlling the Properties

You can use the callback functions exposed in the Provider class to control certain aspects about your custom properties. The callback functions are there to simply help you, you can override these with your own functions for your properties.

These callback functions are connected to the property delegates in the AddProperty function in the AnsGroup class.

```
#set the delegates to property provider functions
prop.IsValidCallback = self.provider.IsValid
prop.IsReadOnlyCallback = self.provider.IsReadOnly
prop.IsVisibleCallback = self.provider.IsVisible
prop.GetValueCallback = self.provider.GetValue
prop.SetValueCallback = self.provider.SetValue
```

IsValidCallback

The IsValid callback allows you to control the validity of your property based on some conditions. This validity will also control the state of your object. If your property is in invalid, your Python Code object will show as under-defined in the Tree. The invalidity of your object will also be indicated by the property being highlighted yellow in the details pane.

This callback could be something like a valid range as seen in the screen shot below.

```
    def IsValid(self, prop):
        """
        Called when checking the validity of a property, with the prop
        """

        #for double property use the ValidRange property to check vali
        if(isinstance(prop, DoubleProperty)):
            return prop.ValidRange[0] <= prop.Value and prop.ValidRang

        return True
```

You will see the definition above in the Provider class. In case the instance of the property is a DoubleProperty the ValidRange property on the property instance is used to decide whether the property is valid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

IsReadOnlyCallback

The IsReadOnlyCallback allows you to control whether your property should be read-only or not in the details pane. This is useful if you want your property be something that just provides some information. Similar to the IsValid callback a template for this is provided as a function in the Provider class as seen in the screenshot below:

```
def IsReadOnly(self, prop):
    """
    Called when checking if a property should be readonly, wi
    """

    return False
```

The above callback as a default returns False, however can be customized. For example, if you wanted to make property with the name "Property 1" to be read-only. The following code can be added to the function definition to achieve that. The new definition will look like this:

```
def IsReadOnly(self, prop):
    """
    Called when checking if a property should be readonly, with
    """

    if prop.Name == "Property 1":
        return True

    return False
```

IsVisible Callback

The IsVisble callback allows you to control whether your property shows up in the details view. You return a boolean value based on some conditions from the function in the Provider class.

SetValue Callback

The SetValue callback allows you to control what the internal value of the property is based on what value the property is being set to. For example, if you want to put an upper limit on what the value for a DoubleProperty should be, you can do this via validating the property. However, you can also use the SetValue callback, where if the user enters a value larger than the upper limit, you set the internal value to the upper limit. Let the upper limit be 20, if the user sets the value to 21, you can force the value to be set to 20. The following code can accomplish this for a property with the name "Property 1":

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

```
def SetValue(self, prop, val):
    """
    Allows you to override the setter of the Value property on the prope
    Keyword Arguments:
        prop -- property of which the value is being set
        val -- the value that was set
    Returns:
        The value that the Value property should be set to
    """

    if prop.Name == "Property 1" and val > 20:
        return 20
    return val
```

GetValue Callback

The GetValue callback allows you to control what the value of the property is when queried using the Value property on the property instance.

Defining Callback Handlers

You can define your own functions that handle these callbacks. For example, you create prop1 which is of type Double and you want to user your own function to handle the IsValid callback. After you get the instance of your property from the AddProperty method, you set the IsValid delegate to your function definition. Your function signature should match the function signature in the Provider class except the self argument. Here is what this would look like in the Property Provider tab:

```
def my_is_valid(val):
    # your validity code here
    return True
prop1 = group.AddProperty("Property 1", Control.Double)
prop1.IsValidCallback = my_is_valid
```

The example above, creates prop1 which has a custom IsValidCallback handler “my_is_valid” which validates whether the value of the property is equal to 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Setting Parameters

The term *Parameters* in the Mechanical application includes CAD parameters and engineering parameters (pressure magnitude, maximum stress, fatigue life, dimension of a part, material property type, Young's modulus, and others).

While engineering parameters are indicated simply by clicking the parameter box (p. 128) in the Details Pane (p. 119), CAD Parameters (p. 2340) must be given some extra attention, both in the CAD package and in the Mechanical application.

The **Parameter** tab collects all specified parameters and lists them in the **Parameter** tab grids for later use and/or modification.

Also see the Specifying Parameters (p. 2337) section for additional information.

Specifying Parameters

The Details Pane (p. 119) in the application window provides check boxes for items that may be parameterized. The following Details pane images illustrate parameter definition for typical objects in the Mechanical application:

Part Object (p. 2337)

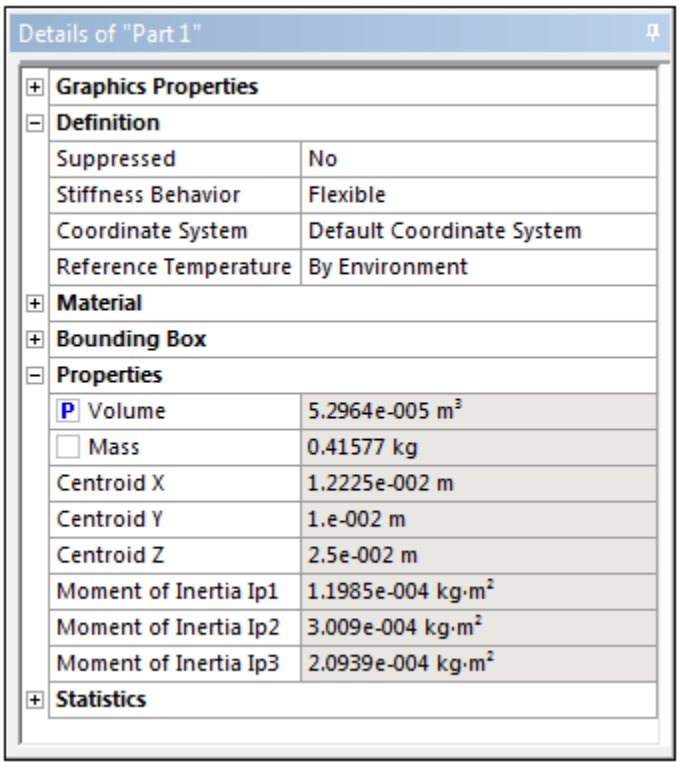
Force Object (p. 2338)

Stress Object (p. 2338)

Part Object

The details of a part object:

DRAFT

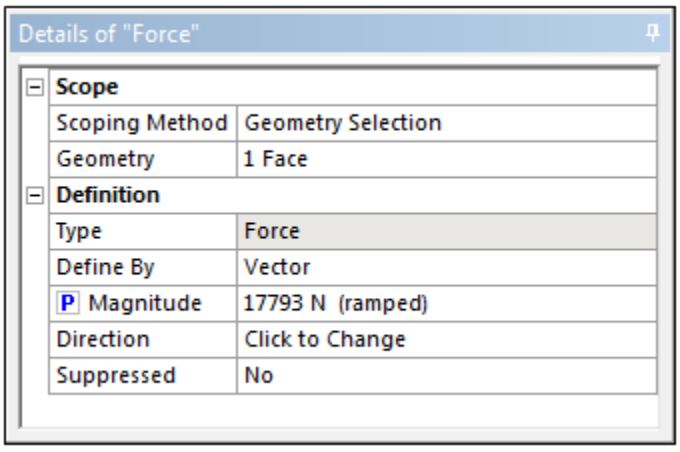


Details of "Part 1"	
Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Bounding Box	
Properties	
P Volume	5.2964e-005 m^3
Mass	0.41577 kg
Centroid X	1.2225e-002 m
Centroid Y	1.e-002 m
Centroid Z	2.5e-002 m
Moment of Inertia Ip1	1.1985e-004 $kg \cdot m^2$
Moment of Inertia Ip2	3.009e-004 $kg \cdot m^2$
Moment of Inertia Ip3	2.0939e-004 $kg \cdot m^2$
Statistics	

A **P** defines the **Volume** as parameterized.

Force Object

The details for a Force object:



Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Vector
P Magnitude	17793 N (ramped)
Direction	Click to Change
Suppressed	No

The **Magnitude** of the force is parameterized.

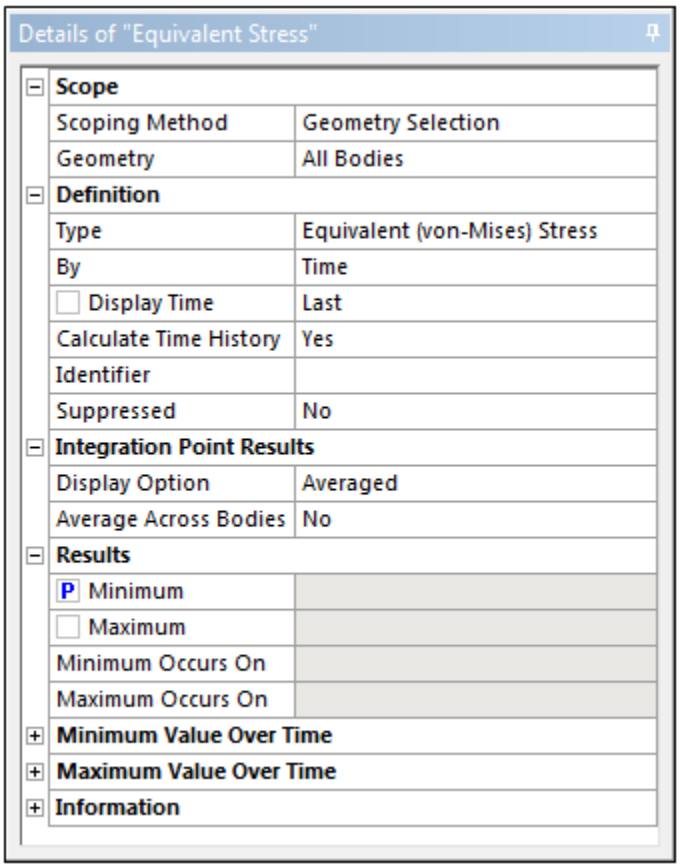
Other details, such as the **Geometry**, **Define By** and **Direction** cannot be parameterized.

Stress Object

The details for a Stress object.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



A **P** appears next to the selected output parameters.

The **Minimum** is selected as an output parameter.

The **Maximum** is not selected as an output parameter.

Parameter Restrictions

If an object has a parameterized field, and that object definition is changed in a way that makes that parameterization non-meaningful, the parameterization will be removed by the program. Some examples include:

- A material in Engineering Data has a parameterized density, and then the user suppresses the material.
- A result in the Mechanical application is scoped to a face and has a parameterized maximum value, and then the user re-scopes the result to a different topology.

Note:

If you suppress an object, no parameter boxes will be shown for any property on that object. If you parameterize the **Suppressed** property on an object, no parameter boxes will be shown for any other property on that object, regardless of whether or not the object is suppressed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

CAD Parameters

CAD parameters are a subset of the application parameters. As the name implies, CAD parameters come from a CAD system and are used to define the geometry in the CAD system. Although each CAD system assigns its parameters differently, the Mechanical application identifies them via a key (**ans; ds** or **ANS: DS**). This identifier can appear either at the beginning or the end of the parameter name and does not need to be separated from the name with an underscore or any other character. By identifying the parameters of interest you can effectively filter CAD parameter exposure. Any of the following examples are valid CAD parameter names using **ANS; DS** or **ans: ds** as the key:

- DSlength
- widthds
- dsradius

ANS; DS is the default key for importing CAD parameters into the application. You can change this default via the Personal Parameter Key option on the Geometry Preferences.

Note:

If you change the key phrase to nothing all parameters are exposed.

CAD parameters must be assigned correctly in the CAD system in order to be imported. Refer to your CAD system instructions for detailed information on assigning these parameters. Some system specific notes are included here for your convenience. Remember that these are all actions that must be performed in the CAD system *before* importing the model. CAD systems include:

- Autodesk Inventor (p. 2340)
- CATIA V5 (p. 2341)
- Creo Parametric (formerly Pro/ENGINEER) (p. 2341)
- NX (p. 2341)
- Solid Edge (p. 2341)
- SOLIDWORKS (p. 2341)

Autodesk Inventor

After a part is open in Inventor, click **Tools> Parameters**. In the **Parameters** dialog box, click a parameter name under the **Parameter Name** column, modify the parameter name to include **ans; ds** at either the beginning or end of the name and click **Enter**. Click **Done** to close the **Parameters** dialog box.

For detailed information, see CAD Integration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

CATIA V5

After a part is open in CATIA V5, click **Tools> Formula**. In the Formulas dialog box, select the desired parameter in the scrolling list. In the "Edit name or value of the current parameter" field, modify the parameter name to include **ans; ds** at either the beginning or end of the name, then click **OK** or **Apply**.

For detailed information, see CATIA V5 Associative Geometry Interface (*.CATPart, *.CATProduct) in the CAD Integration section of the product help.

Creo Parametric (formerly Pro/ENGINEER)

In Creo Parametric, modify the parameter name by selecting the feature it belongs to, right-click **Edit**. Creo Parametric will then display all dimensions (parameters) for the selected feature. If the model shows numeric values, then select **Info> SwitchDims** so that the names are text based instead of numeric. Next, select the dimension/parameter you wish to rename, it will turn red when selected. Then hold down right-click until a menu appears and there select **Properties**. The Dimension Properties dialog box will appear, select the **Dimension Text** tab. Here you can give the dimension a new name, also be sure to change the @D to @S (case sensitive) before completing the modification by clicking **OK**.

For detailed information, see Creo Parametric Associative Geometry Interface (*.prt, *.asm) in the CAD Integration section of the product help.

NX

After a model is opened in NX, click **Application** > **Modeling** and **Tools** > **Expression**. In the **Edit Expressions** dialog box, select the expression with the variable name that you want to rename and click **Rename**. Change the expression name in the **Rename Variable** dialog box to include **ans; ds** (or whatever matches your parameter key preference) at either the beginning or end of the name and click **OK**. Click **OK/Apply** to close the **Edit Expressions** dialog box.

For detailed information, see NX in the CAD Integration section of the product help.

Solid Edge

After a model is opened in Solid Edge, click **Tools> Variables...** If the dimensions (type Dim) are not shown in the **Variable Table** dialog box, click the **Filter** button for the **Filter** dialog box. Highlight both `Dimensions` and `User Variables` under the **Type** column; select `Both` under the **Named By** column and select `File` under the **Graphics in** column. Then click **OK**. Click the name of a dimension (under the **Name** column), modify the dimension name to include **ans; ds** at either the beginning or end of the name and click Enter. Close the **Variable Table** dialog box.

For detailed information, see Solid Edge in the CAD Integration section of the product help.

SOLIDWORKS

In SOLIDWORKS, open the part and then click the part or on the feature in the tree. Then right-click the dimension on the model, open the **Properties** dialog box, and edit the name of the dimension.

For detailed information, see SOLIDWORKS in the CAD Integration section of the product help.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Productivity Tools

The Mechanical application includes several features designed to help you create, navigate, and manage data in complex databases where a large number of objects are present. These features include tags, tree filtering, and the object generator.

This section examines the following topics:

- Generating Multiple Objects from a Template Object (p. 2343)
- Tagging Objects (p. 2349)
- Filtering the Tree (p. 114)

Generating Multiple Objects from a Template Object

You can use the Object Generator to make one or more copies of a template object, scoping each to a different piece of geometry. Almost any tree object that supports the "Duplicate" function can be used as a template.

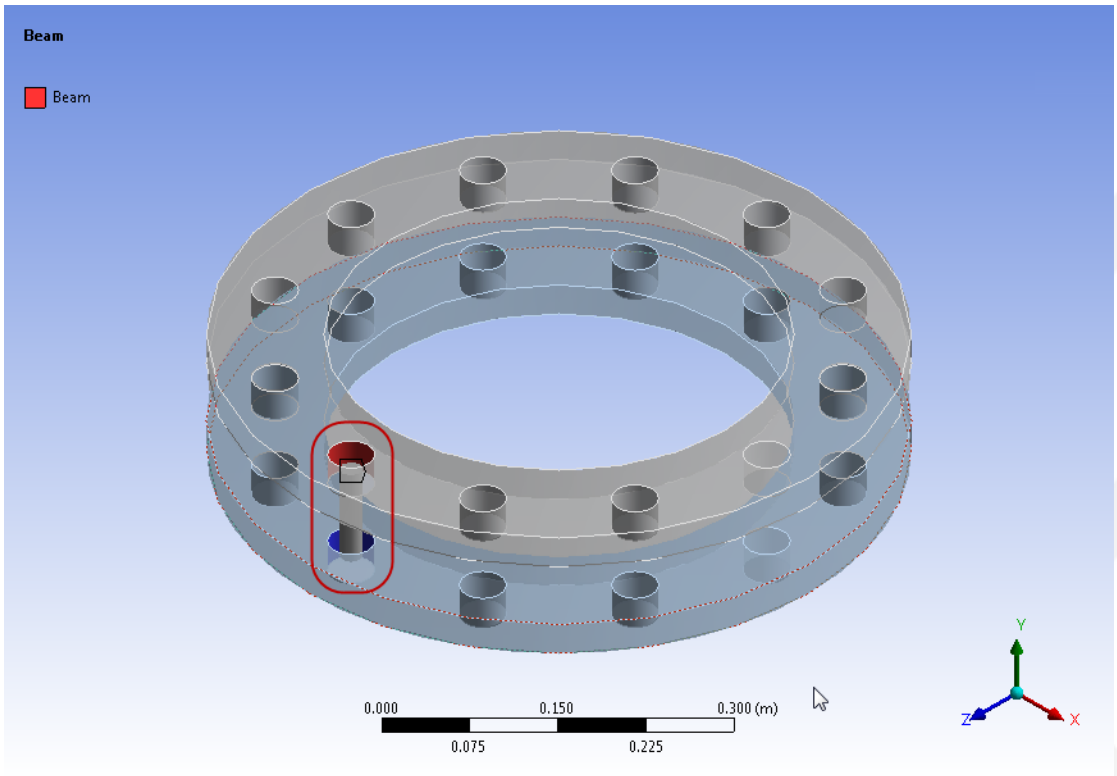
To use the Object Generator, you define a tree object to be copied, select the geometry to which it should be copied, and generate from the Object Generator. The original tree object is copied to all of the selected geometry, with all details from the original object maintained. You have the option of adding a common prefix and/or tag to the name of all generated objects.

- If your object must be scoped to more than one geometry set, you have a choice for how that scoping is handled.
- For objects with locations, such as remote points, you can choose to move the location to the centroid of the new geometry, or leave the location unchanged.
- If the geometry from the template object is part of the target geometry selection set, you can choose to ignore or include it.
- For any connections requiring two sets of geometry, you specify one named selection for each side of the connection. The Object Generator will then generate a connection between any geometry on each side which falls within a specified distance.
- Since end releases require a vertex and an edge, you can specify named selections for the vertices and edges. The Object Generator will then generate an end release for every specified vertex with an edge in the specified set of edges.
- Parameterized properties do not retain their parameterization in the generated objects.

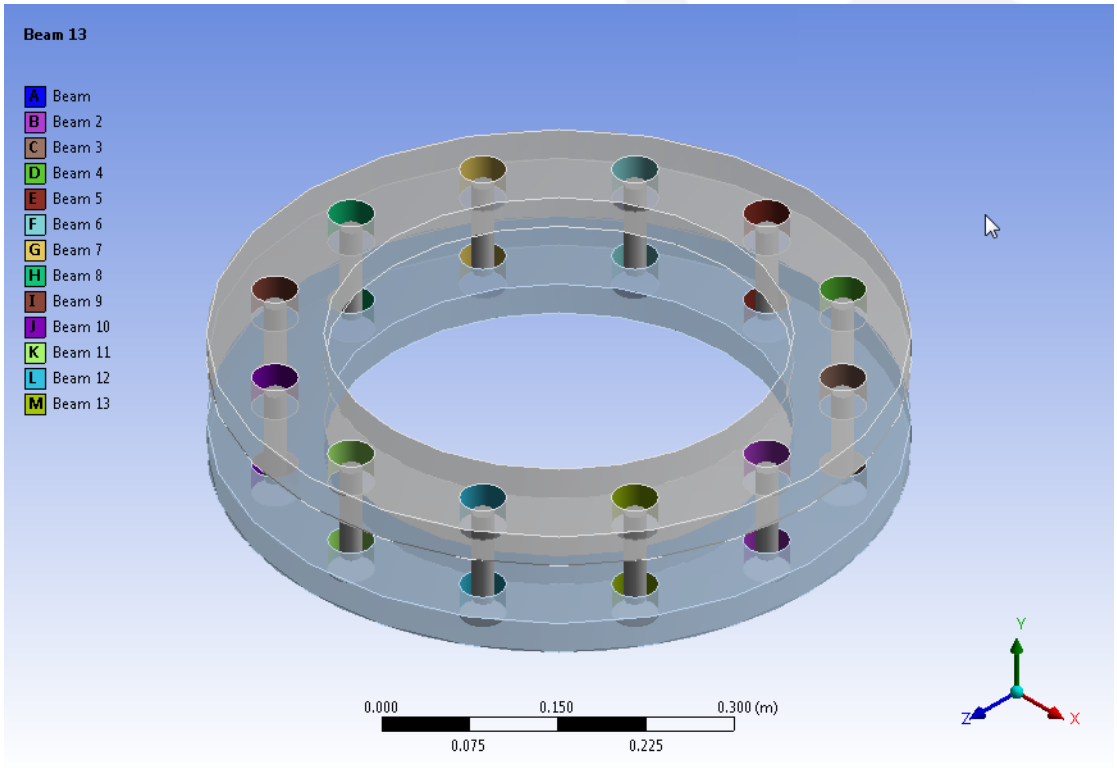
Example 10: Generating Clamping Bolts

For example, you have two retaining collars with one clamping bolt defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



You can use the object generator to generate the other bolt connections.



Generating an Object

To use the Object Generator:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

1. In the standard toolbar, click the **View Object Generator** button to view the **Object Generator** window.

2. In the **Tree Outline**, select the tree object to be copied. Define any details you want included in the generated objects.

3. In the **Geometry** window, select the geometry to which the tree object should be copied in the **Object Generator** window.

4. In the **Object Generator** window, select any required options.

   The options vary, depending on the selected object. Possible options are:

Option Name	Shown for Object Type	Description
Scope to	• General objects supporting one geometry selection • Connection objects	When multiple geometry items are selected, you have several options for scoping the new object. • **Each Entity**: Scope one new object to each geometric entity selected. • **Adjacent Entities**: Scope one object to all groups of adjacent selected entities. This is the default. • **All Entities by Part**: Scope one new object to all selected geometric entities on each part. **Note:** If none of the selected topologies are adjacent, then both options will work in a similar manner.
Ignore Original	All	If the geometry for the original object is part of the target selection set, this option directs the Object Generator to ignore the original and scope new objects only onto geometry not scoped to the original object. This option is selected by default. If you clear this option, the Object Generator copies new objects to all specified geometry, including that of the original, if selected. Note that this may result in duplicate objects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option Name	Shown for Object Type	Description
Name Prefix	All	If you want all generated objects to have a common name prefix, enter the desired prefix in the **Name Prefix** field.
Apply Tag	All	If you want to apply a label to all generated objects, enter a tag name in the **Apply Tag** field. Tags can be used to filter your tree. For more information on tags, see Tagging Objects (p. 2349).
Relocate	When relocation is possible, applies to: • General objects supporting one geometry selection • Connection objects	For objects with locations, such as remote points, you have an option for how to handle the location of the generated objects. • **Yes**: For generated objects, the remote point will be the centroid of the new geometry. This is the default. • **No**: Leave the location of the remote point as is when generating the new objects.
Generate from	If named selections are defined, applies to general objects supporting one geometry selection	Select whether to use the geometric entities selected in the **Geometry** window (Current Selection) or a named selection.
Reference	Springs	Select the named selection to use as the Reference side of the connection. You specify the other side using the **Mobile** option, then specify the lower and upper boundaries of the distance between sides to generate connections.
Mobile	Springs	Select the named selection to use as the Mobile side of the connection. You specify the other side using the **Reference** option, then specify the lower and upper boundaries of the distance between sides to generate connections.
Primary	Mesh connections	Select the named selection to use as the Master side of the connection. You specify the other side using the **Secondary** option, then specify the lower and upper boundaries of the distance between sides to generate connections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Option Name	Shown for Object Type	Description
Secondary	Mesh connections	Select the named selection to use as the Slave side of the connection. You specify the other side using the **Primary** option, then specify the lower and upper boundaries of the distance between sides to generate connections.
Contact	Contacts	Select the named selection to use as the contact side of the connection. You specify the other side using the **Target** option, then specify the lower and upper boundaries of the distance between sides to generate connections.
Target	Contacts	Select the named selection to use as the target side of the connection. You specify the other side using the **Contact** option, then specify the lower and upper boundaries of the distance between sides to generate connections.
Minimum	Connection objects	For connections, specify the lower boundary. The Object Generator will generate a connection between any geometry on each side which falls within the specified distance. The distance is defined as the distance between the centroid of one geometric selection and the centroid of another geometric selection.
Maximum	Connection objects	For connections, specify the upper boundary. The Object Generator will generate a connection between any geometry on each side which falls within the specified distance. The distance is defined as the distance between the centroid of one geometric selection and the centroid of another geometric selection.
Edges	End releases	For end releases, select a named selection that encompasses edges for which you want to generate objects. The Object Generator will generate an end release for every specified vertex specified in **Vertices** if it has an edge in the specified set of edges.
Vertices	End releases	For end releases, select a named selection that encompasses vertices for which you want to generate objects. The Object Generator will generate an end release for every specified vertex if it has an edge in the specified set of edges specified in **Edges**.
Source	Mesh method control	Select the named selection to use as the Source. Source appears in the Object Generator

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Option Name	Shown for Object Type	Description
		window for the Sweep and MultiZone mesh methods only. Specifying a Source is optional. You specify the target using the **Target** option.
Target	Mesh method control	Select the named selection to use as the Target. Target appears in the Object Generator window for the Sweep mesh method only. Specifying a Target is optional. You specify the source using the **Source** option.
High	Mesh match control	Select the named selection to use as the high side of the match control. You specify the other side using the **Low** option. The Object Generator will not assign a coordinate system. You must assign a coordinate system manually. **Note:** The software does not match the high and low sides on a part-by-part basis. Depending on the geometry and specified criteria, the high face and low face for a generated match control may not belong to the same part.
Low	Mesh match control	Select the named selection to use as the low side of the match control. You specify the other side using the **High** option. The Object Generator will not assign a coordinate system. You must assign a coordinate system manually. **Note:** The software does not match the high and low sides on a part-by-part basis. Depending on the geometry and specified criteria, the high face and low face for a generated match control may not belong to the same part.
Boundary	Mesh inflation control	Select the named selection to use as the inflation boundary.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

5. Click **Generate** to copy the selected tree object to the selected geometry.

Tagging Objects

For complex models, it may be difficult to keep track of all of the objects in your tree. With tags, you can mark objects in the tree with meaningful labels, which can then be used to filter the tree. For more information on filtering, see Filtering the Tree (p. 114).

Tags are managed through the **Tags** window. To view this window, click the **Tags** button in the **Graphics** toolbar.

This section covers the following:

Creating Tags
Applying Tags to Objects
Deleting a Tag
Renaming a Tag
Highlighting Tagged Tree Objects

Creating Tags

To create a tag and apply it to the currently-selected tree object:

1. In the **Tree Outline**, select an object.
2. In the **Tags** window, click the **Add a Tag** icon.

   The **Add New Tag** window appears.
3. Enter a name for the tag and click **OK**.

   The tag is listed in the **Tags** window. The check box is selected to indicate that it applies to the selected object in the tree.

Applying Tags to Objects

Once you have created tags, you can apply those tags to other objects in the tree.

To apply a tag to a tree object:

1. In the **Tree Outline**, select an object.
2. In the **Tags** window, select the check box for all tags you want to add to that object.

You can now use these tags to filter the tree. For more information on filtering the tree, see Filtering the Tree (p. 114).

Deleting a Tag

To remove a tag:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

1. In the **Tags** window, select one or more tags.
2. Click the **Delete Tag(s)** icon, or right-click the **Tags** window and select **Delete Tag(s)**.

Renaming a Tag

To rename a tag:

1. In the **Tags** window, select a tag.
2. Click the **Rename Tag** icon.

   The **Rename Tag** window appears.
3. Enter a name for the tag and click **OK**.

   The new tag name is listed in the **Tags** window.

Highlighting Tagged Tree Objects

Once a tag is applied to objects in the Tree Outline, you can highlight all of the objects with a selected tag.

You can search for objects that apply to one or more tags. When you select multiple tags, you have several options. You can search for objects that contain *any* of the selected tags, or you can search for objects that contain *all* of the selected tags.

To highlight objects:

1. In the **Tags** window, select one or more tags
2. Right-click the **Tags** window and select one of the following options:

   - **Find items with selected tag**: Available when only one tag is selected, this option highlights all tree objects with the selected tag.
   - **Find items with all selected tags**: Available when multiple tags are selected, this option highlights all items that contain every one of the selected tags.
   - **Find items with any selected tags**: Available when multiple tags are selected, this option highlights all items that contain one of the selected tags.

   Tree objects matching the selected number of tags are highlighted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical: CAD System Information

For detailed CAD-related information specific to the Ansys DesignModeler application and Ansys Workbench, see the CAD Integration section of the product help. When accessing the **Ansys Workbench Help** from the Help menu, click the **Contents** tab and open the **CAD Integration** folder in the hierarchical tree.

The CAD Integration section includes topics about:

- Overview
- Geometry Interface Support for Linux and Windows
- Project Schematic Presence
- Mixed import Resolution
- CAD Configuration Manager
- Named Selection Manager
- Caveats and Known Issues
- Installation and Licensing
- File Format Support (with information specific to the Mechanical application)

ACIS	Inventor
AutoCAD	JT Open
BladeGen	Monte Carlo N-Particle
CATIA	NX
Creo Elements/Direct Modeling	Parasolid
Creo Parametric (formerly Pro/ENGINEER)	Solid Edge
Ansys DesignModeler	SOLIDWORKS
GAMBIT	SpaceClaim
IGES	STEP

- SpaceClaim Related to CAD Integration
- Frequently Asked Questions
- Troubleshooting

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- Glossary
- Updates

Mechanical application topics:

General Information (p. 2352)

General Information

Body Filtering Property

There are four body filtering properties: **Process Solid Bodies**, **Process Surface Bodies**, **Process Line Bodies** and **Mixed Import Resolution**. Their value is set in the Project Schematic and they determine what bodies will get imported to the Mechanical application (p. 1). The default setting is:

Yes for Solid and Surface Bodies,

No for Line Bodies and,

None for Mixed Import Resolution.

Material Properties

The CAD system interfaces will process only the isotropic material type.

Multiple Versions of CAD Systems

For most CAD applications, you cannot import a geometry that was created in a newer version of the application if you are currently configured for an older version. When you have both a newer and an older version installed, Mechanical utilizes the version currently configured to import geometry.

For example, assume that you have both Creo Parametric 4.0 and Creo Parametric 6.0 installed and that your computer is configured for the 4.0 version. If you attempt to import geometry created in Creo Parametric 6.0 from the Project Schematic, the configured 4.0 version will not be able to load the geometry and the import will fail. For some CADs (e.g. AutoCAD) the last version run is the one that will be used to execute an import and is essentially the configured version.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Mechanical Troubleshooting

General Product Limitations (p. 2353)
Problem Situations (p. 2357)
Recommendations (p. 2383)

General Product Limitations

This section describes limitations that can be present in the Mechanical application during certain conditions.

Go to a section topic:

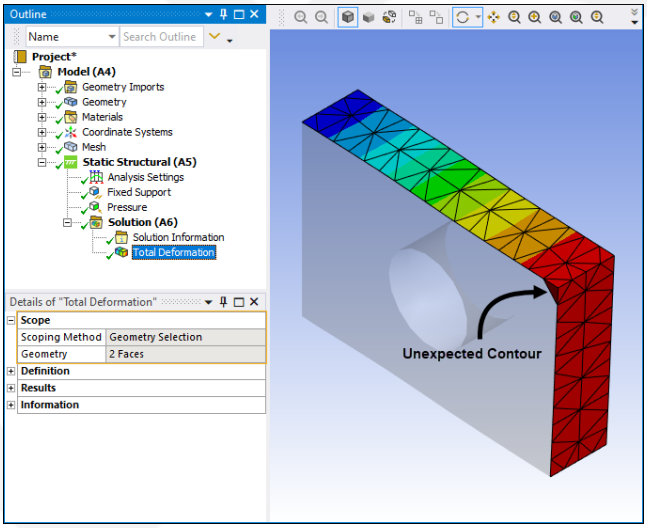
- General Limitations (p. 2353)
- Linux Specific Limitations (p. 2356)

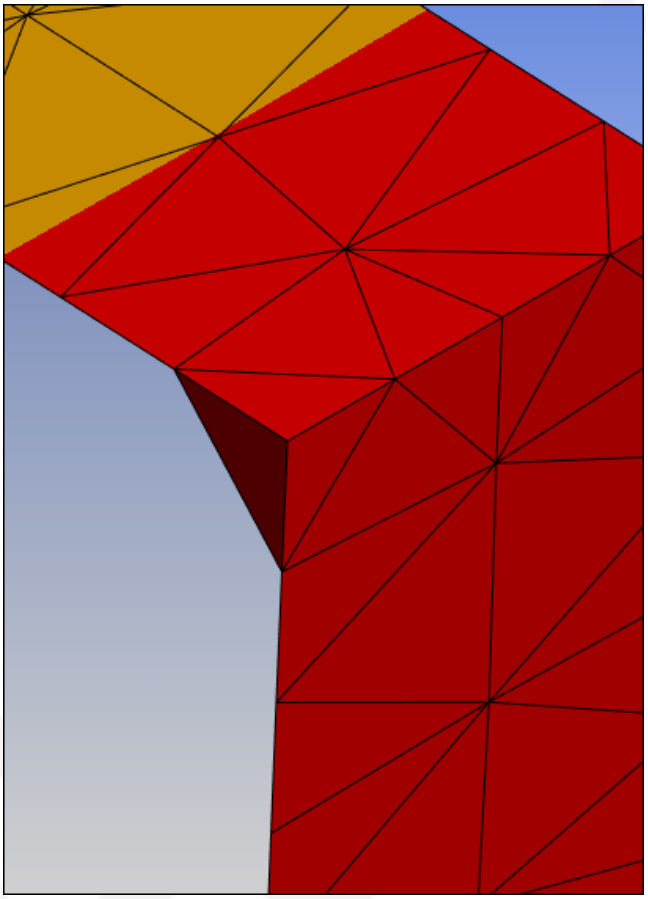
General Limitations

Unexpected Element Face Contour Display

For nodal results and averaged element-nodal results, the application's graphics processor has a known issue for models meshed with low-order elements, that is, linear elements or elements with no mid-side nodes. The application may apply color contours to an element face that is not included in the original scoping of the result. Two examples of this issue are shown below.

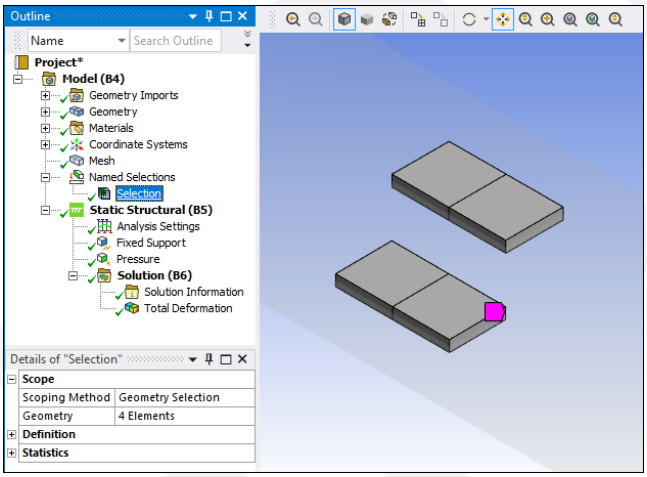
Result Scoped to Two Faces | **Contour Displayed on Un-scoped Element Face**

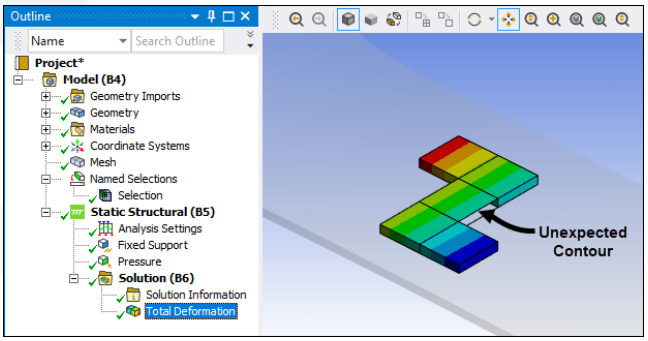




Named Selection Element Scoping

Contour Displayed on Un-scoped Element



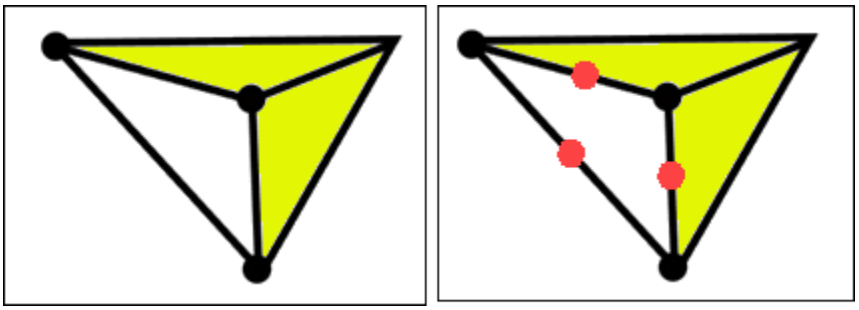


Workaround

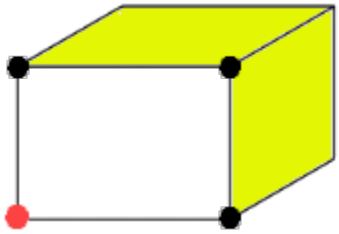
The black nodes illustrated below belong to the yellow faces, included in the original scoping, however, they also belong to the element face (white) in question. Result data is available for each node, and as a result, the application draws contours on all element faces, including the element face not included in the original scoping. Specifying a high-order element, which includes mid-side nodes (red nodes) on the element face in question and that do not have result data, can address the display issue. The presence of the mid-side node on the exterior element face makes sure that no contour is drawn on that element face.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



Also note that a quadrilateral element corrects this scenario as well.



Numerous Objects in Outline

If your analysis includes an exceptionally large number of objects, the **Outline** content may appear incomplete. The application has a display limitation of 65,536 objects. If the number of objects exceeds this threshold, any additional objects are not displayed.

Tabular Data Results

For results displayed in **Tabular Data** window, if **0** (zero) displays for both the **Minimum** and **Maximum** values of a row, the result set may not contain result data. You can use the **Retrieve This Result** option (via right-click on table row), to view result sets in order to determine if any data exists for the set. If no data is available, the result contours in the **Geometry** window display as fully transparent.

Color Coding

Color coding may change or disappear when displaying shared topologies using both the **By Connection** (p. 84) edge coloring and the **Section Plane** (p. 294) features. The **By Connection** option displays connectivity color coding, while the **Section Plane** feature is used to cut or slice the model to view its interior.

Mechanical 14.0 or Previous on Linux

When you are running Mechanical version 14.0 or any previous version on a Linux system, you may experience graphical distortions when animating results, rotating an animation of results, as well as zooming in and/or out on your results.

Application Delays

When running a Mechanical session on the Linux operating system, if you encounter a delay while meshing or launching the Mechanical application or even a crash of the Workbench application, a potential solution is to create a new session with the **Hardware Acceleration** option turned off (display the OpenGL tab and deselect the **Hardware Acceleration** option).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Resuming Projects on Linux

If you are resuming a project in Mechanical on the Linux platform, there is a restriction that the path to the project, as well as the project name, include ASCII characters only, otherwise, the project will not open.

Nice DCV

When using Nice DCV to remotely connect to a Linux machine running Mechanical, you may encounter display issues when using the **Report**, **Print Preview**, or **Image to File** features. To correct this issue, it is necessary to turn off DCV in the Nice DCV control panel, close and reopen Mechanical, and then generate the **Report**, **Print Preview**, or **Image to File**. Once complete, close the application, turn DCV back on, and then reopen the application once again.

Nahimic Audio Driver

If you are running the Nahimic audio driver on your system, the Mechanical application could unexpectedly close when performing certain tasks, such as exporting an image of the Geometry window. Disabling the driver should address the issue. The Nahimic application is not provided by or related to any Ansys products.

ANSWAIT Environment Variable

Mechanical no longer supports the ANSWAIT environment variable, which is used to enable license queuing in some Ansys products.

Linux Specific Limitations

- Make sure that the file system server hosting your working directories ($HOME, solver files, /tmp, etc.) associated with your analysis, has file locking support enabled.
- The application does not support copying and pasting text between a Linux Mechanical session and a Linux-based application, such as Xterm.
- When you are running Mechanical, DesignModeler, or the Meshing editor on the Linux platform, the following error message indicates that there is insufficient storage available to complete the current operation.

  RPCSS ServiceMain failed with 14 (0000000e)

  Not enough storage is available to complete this operation.

  This is likely a result of an entry in the "/etc/hosts file" that is over 500 characters long. If possible, shorten the alias lists of this entry to correct the issue.

- In the Editable drop-down controls, after you mouse over an item in the list, press the "enter" key to select it, or directly type the value in the edit box and press the "enter" key. The following features have been disabled on the Linux platform:
  - Body Views for Connections
  - View Ports for graphical display
- There are a few Linux platform behavior limitations when working with the Customize (p. 143) dialog:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

  - The label of the Tabs and the Close button does not display.
  - The prompt that a key combination you are defining has a conflict does not display.
  - The feature to create a new toolbar is not available.

- Depending on your Linux machine configuration and setup, you could experience issues with certain graphics features that require off-screen rendering, for example, exporting images and animations at resolutions different than the current window resolution. Issues may include:
  - Incorrect graphical output, such as a black image, an image containing just the graphics window's background color, or no image at all.
  - System crashes.
  - X server deadlocking.

  These kinds of issues could stem from the Mesa software drivers available on the system, especially if the machine does not have a properly configured dedicated GPU or if the machine is accessed remotely without hardware accelerated graphics.

  Setting the environment variable **`ANS_WB_FORCE_EGL`** = **`1`** may resolve the issue.

- If you are using CentOS 8.1 in combination with Nice DCV and menu arrows appear to display as a "1". You may be missing fonts for the docking pane buttons **Menu**, **Auto Hide**, **Maximize**, and **Close** and you are displaying locally (even with VNC and DCV), you will need to add the font path to your XWindows display server.

  Use the command **`xset +fp $INSTALL_DIR/aisol/CommonFiles/Fonts/linx64`** on your XWindows display server so your XWindows server can find the font files.

- For Linux Workbench running on RedHat/CentOS 8.1, if you experience a fatal error when exiting Mechanical, DesignModeler, or the Meshing Editor, set the environment variable **`MWDONT_XCLOSEDISPLAY=1`**.

Problem Situations

This section describes how to deal with the following problems:

A Linearized Stress Result Cannot Be Solved.
A Load Transfer Error Has Occurred.
A Master Node is Missing from the Condensed Part
Although the Exported File Was Saved to Disk
Although the Solution Failed to Solve Completely at all Time Points.
An Error Occurred Inside the SOLVER Module: Invalid Material Properties
An Error Occurred While Solving Due To Insufficient Disk Space
An Error Occurred While Starting the Solver Module
An Internal Solution Magnitude Limit Was Exceeded.
An Iterative Solver Was Used for this Analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

At Least One Body Has Been Found to Have Only 1 Element
At Least One Spring Exists with Incorrectly Defined Nonlinear Stiffness
Animation Does not Export Correctly
Application Not Closing as Expected
Assemblies Missing Parts
Cannot Undo Node Move
CATIA V5 and IGES Surface Bodies
Constraint Equations Were Not Properly Matched
Element n Located in Body (and maybe other elements) Has Become Highly Distorted
Error Inertia tensor is too large
Equivalent Creep Strain Ratio has Exceeded the Specified Limit Value
Failed to Load Microsoft Office Application
Illogical Reaction Results
Large Deformation Effects are Active
Missing fonts for the Docking Pane Buttons (Linux Platform Only)
MPC equations were not built for one or more contact regions or remote boundary conditions
One or More Contact Regions May Not Be In Initial Contact
One or more MPC or Lagrange Multiplier formulation based contact may have conflicts
One or More Parts May Be Underconstrained
One or More Remote Boundary Conditions is Scoped to a Large Number of Elements
Problems Unique to Background (Asynchronous) Solutions
Problems Using Solution
Proxy Server Environment Variable
Remote Points with Overlapping Geometry Selections are not Recommended within a Condensed Part
Running Norton AntiVirusTM Causes the Mechanical Application to Crash
The Correctly Licensed Product Will Not Run
The Deformation is Large Compared to the Model Bounding Box
The Initial Time Increment May Be Too Large for This Problem
The Joint Probe cannot Evaluate Results
The License Manager Server Is Down
Linux Platform - Localized Operating System
The Low/High Boundaries of Cyclic Symmetry
The Remote Boundary Condition object is defined on the Cyclic Axis of Symmetry
The Solution Combination Folder
The Solver Engine was Unable to Converge
The Solver Has Found Conflicting DOF Constraints
Problem with RSM-Mechanical Connection
Unable to Find Requested Modes
You Must Specify Joint Conditions to all Three Rotational DOFs

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Fracture Meshing Problems

Lustre Parallel File Systems on Linux

A Linearized Stress Result Cannot Be Solved.

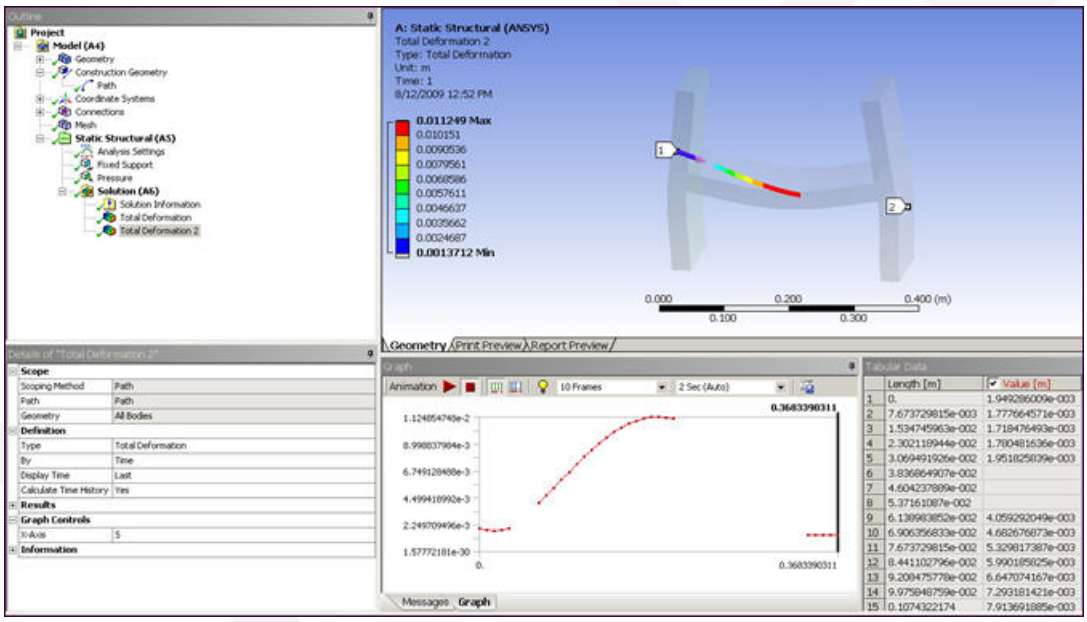
...The path is not entirely contained within the finite element mesh.

To solve a Linearized Stress (p. 2041) result, a necessary condition is that the associated path be totally contained within the model.

If the start/endpoints of the path are not within the model (likely to occur when the mesh is coarse and when using the XYZ **Coordinate** option for picking), you can use the **Snap to mesh nodes** (p. 883) feature to adjust the endpoints to be coincident with the nearest nodes in the mesh.

Occasionally however, other internal "knots" of the path are not inside the model due to a hole or other missing material in the model. These situations can prevent the solving of a Linearized Stress result and cause this error message to appear, even after using the **Snap to mesh nodes** feature.

To verify that a discontinuity is the cause of the error, apply a result other than a Linearized Stress result to that path, and solve it. By doing so you will take advantage of the fact that other results do not require that the full path be inside the model. The results are displayed and discontinuities are indicated by any gaps or missing fields shown in the **Graph** and **Tabular Data** windows. The following example illustrates a Total Deformation result where gaps in the **Graph** window and empty fields in the **Tabular Data** window provide evidence of discontinuities.



A Load Transfer Error Has Occurred.

... A load could not be applied to small or defeatured entity. See the Troubleshooting section of the Help System for more information.

At least one load is not able to be applied. This may be due to mesh-based defeaturing of the geometry. You can modify the mesh defeaturing settings to restore the nodes and elements where the loads need to be applied. Another possible reason could be that the Details View property **Send to Solver** was incorrectly set to **No** for the node-based Named Selection used for some Direct FE boundary condition scoping. Additionally, in a Thermal environment, the load applied to a whole body might lose its scope if all pertinent faces have other surface loads applied. See the Resolving Thermal Boundary Condition Conflicts (p. 1790) section of the Help for a description of precedence.

A Master Node is Missing from the Condensed Part

A master node is missing from the condensed part named...

This error indicates that a remote point on the interface of the condensed part cannot be found at the location advertised in its definition. This typically occurs if the solver automatically suppresses one or more remote points whose geometry selections overlap during **Condensed Part Generation**. See the Remote Points with Overlapping Geometry Selections are not Recommended within a Condensed Part (p. 2373)Troubleshooting section.

Although the Exported File Was Saved to Disk

... the Microsoft Office application failed to load. See the Troubleshooting section for details.

This message is displayed when you have chosen to export a file to Microsoft Excel, but the Microsoft application is either not supported or not installed correctly. The Microsoft Excel file is still exported and can be opened provided the application is resident. To prevent this error message from appearing again, you can either install Microsoft Excel or set **Automatically Open Excel** to **No** in the **Export** (p. 222) preferences, available in the **Options** dialog. Open the **Options** dialog from the **File** tab.

Although the Solution Failed to Solve Completely at all Time Points.

... partial results at some points have been able to be solved. Refer to Troubleshooting in the Help System for more details.

This message displays if for some reason (such as non convergence or the user choosing the **Stop** button) the simulation does not run to completion, but the solution does produce at least some results that can be post processed.

If such a condition occurs, any applicable results in the tree that you request will be calculated (provided that they are defined at a Time/Frequency/Set/Mode that is less than or equal to the final Time/Frequency/Set/Mode in the result file). These results will be assigned a green check state (up to date) but the solution itself will still be in an obsolete state because it is not fully complete.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Note that there are exceptions to this scenario. If the result's specified Time/Frequency/Set/Mode property is equal to the maximum time/frequency/set/mode in the result file, the application:

- Assigns the result with a red check state.
- Indicates the action as a user error. The state icon on all of the results will be red until the result with the out-of-bounds time/frequency/set/mode is suppressed or corrected and the solution is recomputed.

Use the **Evaluate Results** right mouse button option on a Solution object or a result object in order to additionally postprocess the partial solution.

See Reviewing Unconverged Results (p. 1995) for further details.

An Error Occurred Inside the SOLVER Module: Invalid Material Properties

... See the Troubleshooting section of the Help system for possible causes.

Check the following:

Material Definition

Check the Details view for each part to see that you selected the correct material for each part. Go to Engineering Data to edit and check your material files and data and to verify the material definitions (including numbers and units). Note that, depending on the type of result, you will have a minimum of properties to be set.

Structural, Vibration, Harmonic, and Shape Results:

- Need to define the Modulus of Elasticity
- If you don't define the Poisson's Ratio it will default to 0.0. Also note that the Solver engine will not accept values of Poisson's Ratio smaller than 0.1 or larger than 0.4 for Shape Results.
- For bodies whose Stiffness Behavior (p. 851) is set to Stiff Beam (p. 856), make sure that Isotropic Modulus of Elasticity is defined. Note that other types of elasticity, such as orthotropic or hyperelasticity, are not allowed.
- For Vibration and Harmonic results, include the Mass Density of your material.
- For Thermal-stress results, you will need the Coefficient of Thermal expansion.

Thermal Results:

Thermal conductivity is required. Can be constant or temperature-dependent.

Specific Heat is required in a thermal transient analysis. Can be constant or temperature-dependent.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Check Thermal Data

For thermal analysis, go to the Engineering Data to edit and check thermal conductivity in the material files and to check thermal convection in the convection files. Verify the 'smoothness' of the temperature-dependent conductivity data and convection data. Non-smooth curves will lead to Solve failures.

Electromagnetic Materials - Minimum Requirements

For a **Conductor** scoped to a body, the associated material must have either **Resistivity** or **Orthotropic Resistivity** specified in order for the simulation to continue on to a solve.

For all materials in an electromagnetic simulation, one of the following four conditions must be met. These conditions are mutually exclusive of each other so only one condition can exist at a time for a material.

- **Linear "Soft" Magnetic Material** properties specified: Either **Relative Permeability** or **Linear Orthotropic Permeability** are set.
- **Linear "Hard" Magnetic Material** properties specified. Only **Linear "Hard" Magnetic Material** property is set.
- **Nonlinear "Soft" Magnetic Material** properties specified: Either only **BH Curve** or **BH Curve** and **Nonlinear Orthotropic Permeability** are set.
- **Nonlinear "Hard" Magnetic Material** properties specified: Only **Demagnetization BH Curve** is set.

An Error Occurred While Solving Due To Insufficient Disk Space

... See the Troubleshooting section of the Help system for more information.

Possible reasons that this message appears:

- You may be running out of disk space during the Mechanical APDL solution due to the writing of large solution files. Verify that there is sufficient free disk space on the drive where the solver directory exists.
- You do not have write permissions to the solution directory.
- Files from a previous Workbench or Mechanical APDL session already reside in the solution directory.

An Error Occurred While Starting the Solver Module

To get further information on what the issue may be, insert a Solution Information object under Solution in the tree, and view the contents.

Possible reasons that the solver may fail are:

- Insufficient memory - You may not have enough virtual memory assigned to your system. To increase the allocation of virtual memory (total paging file size), go to **Settings> Control Panel> System**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

(on your Windows Start Menu). Click the **Advanced** tab and then click **Performance Options**. Increase the size of your virtual memory.

- Insufficient disk space - You may not have enough disk space to support the increase in virtual memory and the temporary files that are created in the analysis. Be sure you have enough disk space or move to an area where you have enough.
- Corrupt product installation
- License request rejected
- The startup directory for **cmd.exe** has been overridden by the AUTORUN option and as a result causes the solver to be unable to locate the solver input files.

Solving and UNC Paths

If a Workbench database resides on a UNC path (for example, \\pghxpuser\Shares) for which you have write permissions, the Ansys input file will be written successfully but will fail to start the solver executable. To solve, map a drive to the location and then reopen the project. If you did not have write permissions, Workbench will instead write the Ansys input file to your temp directory (%tmp%) and perform a solution from that directory.

An Internal Solution Magnitude Limit Was Exceeded.

… Check your Environment for inappropriate load values or insufficient supports. See the Troubleshooting section of the Help System for more information.

In most cases this message will occur if your model is improperly constrained or extremely large load magnitudes are applied relative to the model size. First check that the applied boundary conditions are correct. In some cases, loads that are self-equilibrating with no support may be desired. To help in these cases, if this message occurs, consider adjusting the weak spring stiffness (p. 1433) or turning on inertia relief (p. 1435).

Thermal Solutions

For Thermal Solutions, verify that your set up has at least one heat sink and one heat source as well as closed contact regions.

An Iterative Solver Was Used for this Analysis

...However, a direct solver may enhance performance. Consider specifying the use of a direct solver.

An iterative solver was used to obtain the solution; however, a large number of iterations were needed in order to get a converged answer.

By default, the program will either choose a direct or iterative solver based on analysis type and geometric properties. (In general, thin models perform better with a direct solver while bulky models perform better with an iterative solver.) However, sometimes the iterative solver is chosen when the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

direct solver would have performed better. In such cases, you may want to force the use of the direct solver. You may specify the solver type in the *Details view* of the **Analysis Settings** folder.

At Least One Body Has Been Found to Have Only 1 Element

...in at least 2 directions along with reduced integration. This situation can lead to invalid results. Consider changing to full integration element control or meshing with more elements. Refer to Troubleshooting in the Help System for more details.

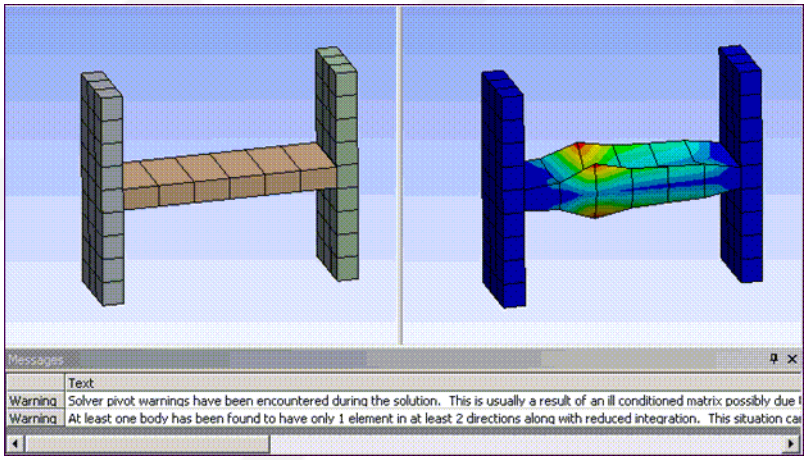
This scenario is based on the following conditions:

- Structural solid model.
- Brick meshes that have only one element in less than two directions.
- Reduced element integration (p. 857) is assigned. (This can happen by default if **Element Control** in the Geometry object is set to **Program Controlled**.)

If the above conditions are met, there is a strong likelihood that your analysis will excite hourglass modes. In such cases solver pivot warnings will be reported and nonphysical deformations will result (see examples below). If this occurs, first determine which bodies have one element through the thickness (Right-click in **Geometry** window, choose **Go To> Bodies With One Element Through the Thickness**, and observe selected body objects in the tree). The offending bodies can then be corrected by doing one of the following:

- Modify the mesh to have more than 1 element in at least 2 directions. This will remove the hourglass modes in most cases. In rare cases you may need to modify the mesh such that more than 1 element exists in all 3 directions.
- Use **Full** integration (p. 857) on the offending bodies.
- Consider using lower order elements.

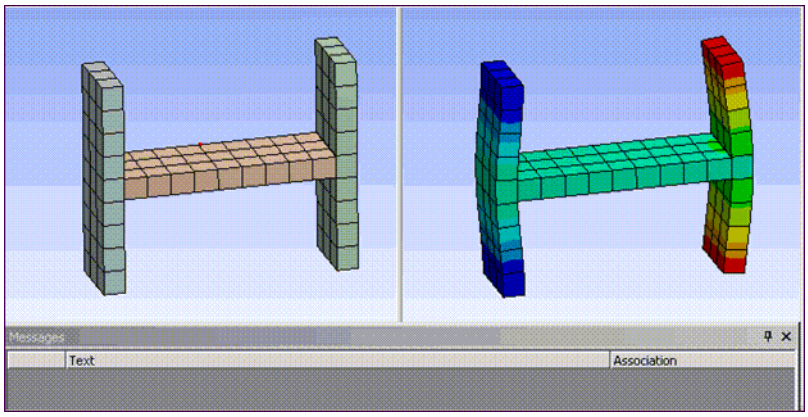
Example of a "bad" mesh for reduced integration:



Example of a "good" mesh for reduced integration:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



At Least One Spring Exists with Incorrectly Defined Nonlinear Stiffness

The force-deflection curve is incorrectly defined using tabular input for nonlinear stiffness for one or more spring objects existing in the model, see the details in COMBIN39 element description for more information.

Note:

Support Requirements

- **Tabular Data** requires at least two rows of data.
- The properties **Longitudinal Damping** and **Preload** are not applicable for Springs with nonlinear stiffness.

Animation Does not Export Correctly

When exporting an AVI file, make sure that you keep the Workbench module window in front of other windows until the exporting is complete. Opening other windows in front of the module window before the exporting is complete may cause those windows to be included in the AVI file capture.

Application Not Closing as Expected

During shutdown, if Mechanical is responding slowly, that is, taking two or more minutes to close, you may want to review your Filesystem Settings.

Assemblies Missing Parts

When reading assemblies from CATIA V5, all part files that are referenced by assemblies must be accessible in order for the importing to occur.

Cannot Undo Node Move

...The node is no longer at the location that it was moved to.

Or...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

...The node no longer exists in the mesh.

Node movements listed in the **Worksheet** can become obsolete if you implement mesh changes such as **Mesh Numbering**, **Fracture**, **Mesh Connections**, etc. Undo actions are not available once a node movement has become obsolete.

To return to a base mesh, you need to remove your mesh changes in the reverse order of their implementation. If this is not possible, you may need to clean and re-mesh your model.

CATIA V5 and IGES Surface Bodies

CATIA V5 and IGES surface bodies consisting of closed faces are transferred as solid bodies.

Constraint Equations Were Not Properly Matched

... for all node pairs across the low and high sector boundaries in the cyclic symmetry. See the Troubleshooting section of the Help System for more information

This message may occur if the solver does not succeed to reproduce the exact pairing of nodes between the low and high sector. An approximate technique was used to group like nodes and distribute the loads, but this can reduce solution accuracy.

Element *n* Located in *Body* (and maybe other elements) Has Become Highly Distorted

The application detected an element distortion error because one or more elements failed to meet certain solver criteria (p. 1838) during the solution process.

If the application detects more than one instance of the error during the solver's iterative process, then a message for the last instance is reported. The application obtains errors from `file.err` in the solver files directory.

Use the **Identify Element Violations** (p. 1838) property on the **Solution Information** object to generate Named Selections for the offending element(s).

Error Inertia tensor is too large

This message is shown by the LS-DYNA solver if your model includes rigid bodies with large dimensions, for example a few meters in length. Such rigid geometries cause the inertia tensor limit of the solver to be exceeded.

You can attempt to resolve this issue by running the *double precision* LS-DYNA solver, which has a much larger inertia tensor limit. The double precision solver executable can be accessed with the -dp command line option as follows `LSDYNA120.exe -dp`.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Equivalent Creep Strain Ratio has Exceeded the Specified Limit Value

... Since the time increment has reached the minimum value, the analysis is stopped.

The application detected one or more elements failed to meet certain solver criteria (p. 1838) because the creep strain increment is too large.

If the application detects more than one instance of the error during the solver's iterative process, then a message for the last instance is reported. The application obtains errors from `file.err` in the solver files directory.

Use the **Identify Element Violations** (p. 1838) property on the **Solution Information** object to generate Named Selections for the offending element(s).

Failed to Load Microsoft Office Application

... See the Troubleshooting section for details.

This message is displayed when you have chosen a feature that is dependent on a Microsoft Office application, such as exporting a file to Microsoft Excel, and the related Microsoft Office application is not installed correctly.

Illogical Reaction Results

Cause

Loads, supports, or contact items are applied to the same or shared topology.

Reason

It is unclear or ambiguous as to which reaction should be attributed to which support, load, or contact item. Refer to this Note (p. 2115) for details.

Large Deformation Effects are Active

... Which may have invalidated some of your applied supports such as displacement, cylindrical, frictionless, or compression only supports. Refer to Troubleshooting in the Help System for more details.

In a large deformation analysis, the program updates the nodal coordinates as the solution progresses towards the final configuration. As a result, supports that fix only some of the degrees of freedom of a node but not all (for example fix only UX=0), may become invalid as the model's nodal coordinates and thus nodal rotation angles are updated. The imposed DOF displacement directions do not change even though rotation angles change. This may or may not be a desirable situation. A classic example is a simple torsion of a rod. Initially the nodes at zero degrees have a circumferential direction of UY but after a twist of 90 degrees, have a circumferential direction of UX.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

The user is responsible for determining if any nodal rotation at the support is significant enough to cause undesired results.

The following is a list of supports which only fix the movement of a node partially and thus are susceptible to large deformation effects:

- Displacement (p. 1666)
- Cylindrical support (p. 1688)
- Frictionless (p. 1682)

In addition a Compression Only Support (p. 1685) may be susceptible to large deformation effects because if large sliding occurs, the face can literally "slide off" the compression only support.

Missing fonts for the Docking Pane Buttons (Linux Platform Only)

If you are missing fonts for the docking pane buttons Menu, Auto Hide, Maximize, and Close and you are displaying remotely, you need to install the fonts on your XWindows display server.

Copy the font files found under $INSTALL_DIR/aisol/CommonFiles/Fonts/linx64 locally to your XWindows display server.

Use the command "xset fp+ <local-dir>" on your XWindows display server so your XWindows server can find the font files.

MPC equations were not built for one or more contact regions or remote boundary conditions

...Due to potential conflicts with the cyclic symmetry constraints. This may reduce solution accuracy. Refer to the Troubleshooting section.

Cyclic symmetry is enforced with the help of constraint equations between pairs of nodes on the low and high sector boundaries respectively. When such nodes also participate in MPC contact, which requires constraint equations of its own, conflicts may arise. Review results carefully, since the MPC contact will be compromised at these locations.

One or More Contact Regions May Not Be In Initial Contact

… Check results carefully. Refer to Troubleshooting in the Help System for more details.

During the solution it was found that one or more of the contact pairs was not initially in contact. You may check the solution output located in the **Worksheet** of a **Solution Information** object (p. 1833) to determine exactly which contact pairs are initially open, and take the appropriate action.

- This message is expected if a contact pair is meant to be initially open and may become closed after the load application.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- If initial contact was desired and the contact pair has a significant geometric gap, setting the **Pinball Radius** (p. 1253) manually to a sufficiently large value may be required.
- If symmetric contact is active, it is possible that one pair may be initially open and its symmetric pair be initially in contact. Check the solution output to confirm this.

One or more MPC or Lagrange Multiplier formulation based contact may have conflicts

...With other applied boundary conditions or other contact or symmetry regions. This may reduce solution accuracy. For MPC formulation based remote boundary conditions, you can set Relaxation Method property to Yes to eliminate over constraints. Tip: You can graphically display FE Connections from the Solution Information Object. Refer to Troubleshooting in the Help System for more details.

During solution it was found that one or more contact pairs using MPC (multi point constraint) or Lagrange Multiplier based contact formulation (p. 1240) overlaps with another contact region or boundary condition. The same is true for remote boundary conditions (p. 1735) overlapping with another contact region or boundary condition.

Due to the fact that MPC formulation can cause over constraint if applied to the same nodes more than once, the program may have not been able to completely bond the desired entities together. You may check the solution output located in the **Worksheet** of a **Solution Information** object (p. 1833) to determine which pairs and nodes are affected by this condition. Specifically this can happen when:

Due to the fact that MPC formulation can cause over constraint if applied to the same nodes more than once, the application may have not been able to completely bond the desired entities together. For MPC formulation-based remote boundary conditions (**Formulation** property set to **MPC**), you can set **Relaxation Method** property to **Yes** to eliminate over constraints. If this does not resolve the over-constraint issue, you may check the solution output located in the **Worksheet** of a **Solution Information** (p. 1833) object to determine which pairs and nodes are affected by this condition. Specifically this can happen when:

- A contact pair entity (either an edge or face) also has a Dirichlet (prescribed displacement/temperature) boundary condition applied to it. In this case, the MPC constraints will not be created at nodes that have prescribed conditions and may possibly causing parts to lose contact. Sometimes this warning may be disregarded in cases such as a large face with a fixed support at one edge and a contact pair on another. If it is determined that overlap does indeed exist, consider relocating the applied support or using a formulation other than MPC.
- Two MPC contact pairs share topology (such as a face or an edge). Again it is possible for one or both of these pairs to lose contact. This message may especially occur when edge/face contact is automatically generated by the program because often 2 complementary contact pairs (that is, edge part 1/face part 2 and edge part 2/face part 1) are created. Often in this case the message can be ignored after verifying result correctness and if necessary, deleting/suppressing one of the inverse pairs. This condition may also occur when 1 part (typically a surface body), is being contacted by 2 or more parts in the same spatial region. In this case it is possible for one or more of the parts to lose contact. Consider reducing the **Pinball Radius** (p. 1253) to avoid overlap or changing one or more of the regions in question to use a contact formulation other than MPC.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- When MPC contact is used to connect rigid bodies and joints, the overconstraint situation can sometimes occur.
- When Remote Boundary conditions overlap the Low/High Regions of the Periodic/Cyclic symmetry regions, you may experience an overconstraint situation. In a severe situation, the application may terminate.

Note:

For bonded contacts involving shell bodies, the **Program Controlled** setting for the **Formulation** property internally selects the MPC formulation. If these contacts have overlapping geometry or mesh scoping with other connections or boundary conditions it can lead to overconstraint. You can either flip the contact/target scoping or use any formulation other than MPC to resolve this issue.

One or More Parts May Be Underconstrained

...and experiencing rigid body motion.

This message may occur for one of several reasons: If the program detects that the model may be underconstrained, weak springs will be added to the finite element model to help obtain a solution. In addition, the program will automatically add weak springs if unstable contact (frictionless, no separation, rough) or compression only supports are active in order to make the problem more numerically stable. Since the weak springs have a low stiffness relative to the model stiffness, they will not have an effect on a properly constrained model. If you are confident that weak springs are not needed for a solution and the program adds them anyway, you may disable them by setting the **Weak Springs** option to **Off** in the Details view (p. 119) of the **Analysis Settings** object.

One or More Remote Boundary Conditions is Scoped to a Large Number of Elements

...which can adversely affect solver performance. Consider using the Pinball setting to reduce the number of elements included in the solver.

Remote boundary conditions scoped to a large number of elements can cause the solver to consume excessive amounts of memory. Point masses in an analysis where a mass matrix is required and analyses that contain remote displacements are the most sensitive to this phenomenon. If this situation occurs, consider modifying the **Pinball** setting to reduce the number of elements included in the solver. Forcing the use of an iterative solver may help as well.

The reason for the excessive memory consumption is that the remote boundary conditions generate internal constraint equations to distribute the remote mass, displacement, or loads from one node of the model to all other selected nodes. As described in Chapter 15.14. Constraint Equations, in the *Mechanical APDL Theory Reference*, constraint equations could change a sparse matrix (for example, a stiffness matrix, mass matrix, or damping matrix) to a dense matrix. An increase in the number of constraint equations used increases the density of the final matrix, which in turn places a higher demand for more memory (or longer CPU time) in the solution of a problem.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Normally, if the maximum number of remote nodes selected is about 3000, then the increased memory usage or CPU time is not significant. Caution should be taken to not use too many remote nodes in these applications. Other techniques are available to distribute loads or masses. For example, to distribute a point mass to the entire model, you might consider specifying density directly instead of using the point mass approach.

Problems Unique to Background (Asynchronous) Solutions

Consider the following hints when troubleshooting background (asynchronous) solution problems:

- For security reasons, RSM will not allow any job to be run by the "root" user on Linux, including primary and alternate accounts.
- It may sometimes be necessary for you to enter the full path to the solver executable file in the **Solve Process Settings**.
- It may sometimes be necessary for you to enter the full path to the Linux working directory in the Linux **Working Folder** field of the **Solve Process Settings**.
- The LSF administrator should configure the Workbench job server to disallow multiple, simultaneous jobs. Two solves running on the same server will interfere with each other, preventing successful completion of each.
- To help in debugging solver startup problems on the remote machine, it is sometimes useful for you to use the **Solution Information** object (p. 1833) under the **Solution** object in the tree. The **Solution Information** object will show the contents of the `solve.out` file that the remote solver produced, if the application was able to start.
- When using the **Stop Solution** option to stop a solve running on a Linux machine, it is possible that the solver will continue to run on that machine even though the Mechanical application thinks it has stopped. If this happens and you don't want the solve job to continue on the Linux machine it will be necessary for you to kill the process manually. The ability to solve to two different Linux machines simultaneously is not allowed.
- The solve command may have failed to execute on the remote Linux server. Verify the command's spelling and/or path. Solve commands are issued to the remote server using the *rexec* interface. Failures may occur if the resulting path (`$path`) is insufficient. `$path` can be verified by issuing `rexec` on the command prompt on the local machine. For example:

  rexec *machinename* -l *username* echo $path > *diagnosticsfile*

  (where "l" is the letter "el)"

  The *machinename* and *username* match the entries in the **Solve Process Settings**, and *diagnosticsfile* corresponds to the recipient on the local machine for the command output.

  Note:

  After issuing `rexec`, if you receive the following message, *rexec* isn't enabled on the remote Linux server. This feature must be enabled on the remote Linux server in order for the solution to proceed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
> rexec:connect:Connection refused

rexec: can't establish connection
```

If the path to the solve command is unavailable on the remote server, it can be added to user or system-wide files that initialize the startup shell (for example, `.cshrc` or `/etc/csh.login` on C-shells). Consult the Linux server's *rexec* interface and appropriate shell manual pages for details.

- If you cannot make ASCII transfers to a Linux server, changes need to be made on the server. Background solutions on a remote Linux server use file transfer protocol (ftp). Therefore, the system administrator must install ftp and enable it. Ftp uses ASCII transfer mode to convert PC text to Linux text. If ASCII mode is disabled, it is not obvious because error messages do not imply this. On some ftp servers (vsftpd, for example), by default, the server will pretend to allow ASCII mode, but in fact, will ignore the request. You will need to ensure that the ASCII upload and download options are enabled to have the server actually do ASCII mangling on files when in ASCII mode. To enable these options, the system administrator should consult the operating system documentation. The following `vsftp.conf` modification procedure is Linux platform specific and is provided as an example only.

  1. In `/etc/vsftpd/vsftpd.conf`, uncomment the following lines (that is, remove the # at the beginning of these lines):

     ```
     ascii_upload_enable=YES

     ascii_download_enable=YES
     ```

  2. Restart the server.

Problems Using Solution

If Solution fails to complete, try the following suggestions.

Verify the Environment

Verify that the loads and supports in the Environment meet the requirements for Stress, Thermal, Thermal-Stress, Shape or Vibration.

You can verify the environment quickly by looking at the icons adjacent to each environment item in the Tree Outline. A green check indicates that the requirements are met. A indicates that the requirements were not met.

Check System Requirements

Verify that your system meets the minimum requirements at the time you start Solution. Disk space and memory may fluctuate depending on how the system is used. See also General Solver Error.

For Thin-Walled or Finely Detailed Parts

If your parts contain features whose size or thickness is extremely small in comparison to the principal dimensions of the assembly, try adjusting the variables used in modeling geometry.

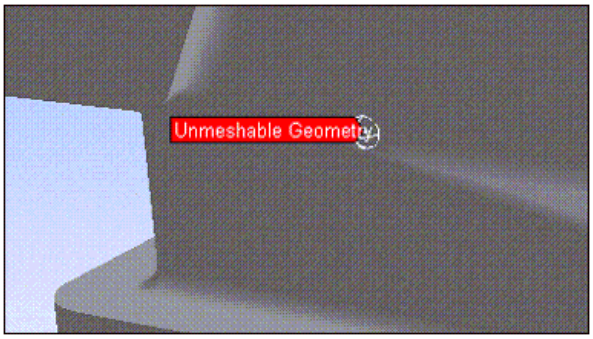
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Set the variable DSMESH DEFEATUREPERCENT to 1e-5. To set variables, select the **Variable Manager** option from the **File** tab.
- If that fails, change the setting to 1e-6.

Invalid or Poorly Defined Models

At the end of the Solution procedure, the region of a part that caused the problem is usually labeled.



.

If the geometry that is notated looks valid, but is small compared to the rest of the model, adjusting the Sizing Control may correct the problem.

Proxy Server Environment Variable

If you set the environment variable HTTPS_PROXY for a proxy server, make certain that the value of the variable starts with "https://" and not "http://."

If using the "https://" protocol for HTTPS_PROXY is not possible, there is an alternative. Set the environment variable **`no_grpc_proxy`** with internal URLs to ignore the proxy settings for internal communications. Depending on the required system configuration, this environment variable may contain one or more of the following as a comma-separated list:

- 127.0.0.1
- localhost
- localhost.localdomain

The setting "no_grpc_proxy=localhost" is generally sufficient, however, "no_grpc_proxy=127.0.0.1,localhost,localhost.localdomain" is also valid.

Remote Points with Overlapping Geometry Selections are not Recommended within a Condensed Part

Remote points with overlapping geometry selections are not recommended within a condensed part...

This warning indicates that two or more remote points on the interface of a condensed part have been found to share a face, edge or vertex. These remote points may have been defined in the Tree View, or may have been created internally by the use of a joint, spring or other remote conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This situation may lead the solver to suppress one or more remote points from the interface during Condensed Part generation, rendering them unavailable for the use pass.

To eliminate the overlap, it is recommended to edit the pinballs so they select disjoint portions of the resulting numerical model. It may also be possible to edit the geometry to separate the modeling features where they are applied.

Running Norton AntiVirus™ Causes the Mechanical Application to Crash

If the Norton AntiVirus™ product is running and you choose **Allow the entire script once** to resolve a script error, the Mechanical application crashes. Choose **Authorize this script** to allow the Mechanical application to function normally.

The Correctly Licensed Product Will Not Run

If you have installed a license file for a valid Mechanical product, but the product continues to run in read-only mode or, in the case of an upgrade to a higher product, continues to run the lower product, make sure you have specified the correct product in the launcher.

This situation can occur if you install the Mechanical application before creating your license file. In this case, the Mechanical application will run only in read-only mode. When you create your license file later, you must choose a license under **Mechanical APDL Product Launcher** in the **Start** menu. Once there, select the product that you have licensed to reset the default to the correct product. Otherwise, the Mechanical application will continue to run in read-only mode.

This situation can also occur if you upgrade your license to a higher Mechanical product. Again, you must choose a license under **Mechanical APDL Product Launcher** in the **Start** menu. Then reset to the appropriate product. Otherwise, the Mechanical application will continue to run as the lower, previously-licensed product.

The Deformation is Large Compared to the Model Bounding Box

... Verify boundary conditions or consider turning large deflection on.

This message will be displayed any time the software detects nodal deformations exceeding 10% of the model diagonal. Exceeding 10% of this length suggests model mechanics that depart from linearity in response to the applied boundary conditions. Load magnitudes, surface body thicknesses, and contact options (p. 1207), if applicable, should be verified. If these are intended, a nonlinear analysis is advised. To request a nonlinear analysis, set **Large Deflection** to **On** in the Details view of the **Analysis Settings** folder.

The Initial Time Increment May Be Too Large for This Problem

... Check results carefully. Refer to Troubleshooting in the Help System for more details.

This message will appear if the program determines that the initial time increment used in the thermal transient analysis may be too large based on the "Fourier modulus" (Fo). This dimensionless quantity

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

can be used as a guideline to define a conservative time step based on thermal material properties and element sizes. It is defined as:

$Fo = k (\Delta t) / \rho c (length_e^2)$

where:

$length_e$ = Average element length

Δt = Time step

k = Thermal Conductivity

c = Specific Heat

ρ = Density

Specifically this warning will be issued if the program finds that the Fourier modulus is greater than 100, that is, Fo > 100. Stated in terms of the initial time step (ITS), this warning appears when the ITS is 100 times greater than the time step suggested by the Fourier modulus in the form expressed below:

$\Delta t = length_e^2 / (k / (c \rho))$

This check is done on a per body basis and the results are echoed in the Mechanical APDL output listing. For example:

```
********* Initial Time Increment Check And Fourier Modulus *********
  Specified Initial Time Increment: .75
  Estimated Increment Needed, le*le/alpha, Body 1: 0.255118
  Estimated Increment Needed, le*le/alpha, Body 2: 1.30416
  Estimated Increment Needed, le*le/alpha, Body 3: 0.158196
  Estimated Increment Needed, le*le/alpha, Body 4: 0.364406
```

If this warning is issued make sure that the specified time step sizes are sufficiently fine to accurately capture the transient phenomenon. The proper use of this guideline depends on the type of problem being solved and on accuracy expectations.

The Joint Probe cannot Evaluate Results

...A possible cause is that the joint is a fixed body-body joint on a rigid body.

This message displays because fixed body-body joints on rigid bodies do not report a reaction. See the Probes (p. 1946) section of the help for more information.

The License Manager Server Is Down

Unless a connection is reestablished, the Mechanical application will exit in *nn* minutes.

Cause

This message occurs in a one-server license environment if your license manager has quit running. In a three-license server environment, the Ansys license manager must be running on at least two of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

the three license server machines at all times. If two of the license server machines go down, or two of the machines are not running the license manager, this error message will appear in the program output or in a message box. The program will continue to run for *nn* minutes to allow the license manager to be restarted or to be started on a second machine if using redundant servers. When the message first displays, *nn* = 60. The message then reappears every five minutes with *nn* displaying the elapsed time at each 5 minute increment (55, 50, 45, etc.) until the connection is established.

Resolution

When this error message appears, start the license manager on the other machines designated as license servers. If you get this message and determine that the license manager is still running, and you are running in a one-server environment, then the IP address of the license server machine was changed while the application was running (this is usually caused by connecting to or disconnecting from an Internet Service Provider (ISP) that dynamically allocates IP addresses). To correct this situation, you must return the IP address to the same address that the license server had when the application was started. If the IP address changes after you start the application (either because you connected to or disconnected from your ISP), you can correct the error by restarting the application. You should not need to restart the license manager.

You can avoid this problem by remaining connected to or disconnected from the ISP the entire time you are running the application.

Linux Platform - Localized Operating System

Specific to the Linux platform: if you are using a localized operating system (such as French or German), or set your preferences to use regional settings for numbers and dates (comma delimiter versus period), there is a discrepancy between applications: The Workbench application will honor the setting and display the numbers with comma delimiter. However, some of the components (e.g. Geometry, Meshing, Mechanical, etc.) can only recognize periods; numbers will be displayed and entered with periods. As a result, you may have to use commas when working in Workbench, and periods when working within those components. If this causes any inconvenience or confusion, define the "LANG" environment variable and set to "en-us" (e.g. "setenv LANG en-us" for csh shell) to force ALL applications (including Workbench) to use the period delimiter consistently throughout. Note that setting LANG to en-us may also cause some strings to be displayed in English, even if your language preference was set to a non-English language. Within Mechanical, analysis settings for Explicit Dynamics and Rigid Dynamics, as well as Imported Load mapping settings are not localized.

If you are using a localized operating system (such as French or German), you must set the following VisualMainWin control on any machines running these applications in order for these applications to recognize the correct numerical format. Ansys Workbench must already be installed before setting this control.

1. cd to:

   ```
   <wb_install directory>/v140/aisol
   ```

2. Issue the following command:

   ```
   ./.workbench -cmd mwcontrol
   ```

3. On the MainWin Control Panel, select Regional Settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

4. Select the Regional Settings tab.

5. Change the language in the drop-down to match the language you want to use.

The Low/High Boundaries of Cyclic Symmetry

... Have been found to include one or more nodes along the axis of symmetry. This may reduce solution accuracy. Refer to the Troubleshooting section.

Cyclic symmetry does not support the presence of nodes along the axis of symmetry. There, the node pair on the high and low sector boundary degenerates to a single node. Consider removing the axial nodes, fixing the nodes, or providing a much finer mesh in the vicinity.

The Remote Boundary Condition object is defined on the Cyclic Axis of Symmetry

... This may reduce solution accuracy. Refer to the Troubleshooting section in the Help System.

This message is displayed when the software detects that a Remote Boundary Condition object is defined on the Cyclic Axis of Symmetry. Make sure that the location of **Remote Points** does not coincide with the cyclic symmetry axis, as this could generate incorrect results. Two exceptions exist: if either a **Moment** or a **Displacement** is applied about the cyclic symmetry axis and then to all sectors, it must:

- be scoped to a **Remote Point** that has its **Behavior** property set to **Deformable**,
- have all DOFs fixed except the DOF about the cyclic symmetry axis, and
- the **Remote Point** must lie on the cyclic axis of symmetry.

The Solution Combination Folder

...is underdefined due to invalid input environments.

When the Solution Combination Folder is underdefined, verify that:

- The selected environments are supported. Supported environments include:
  - Static Structural
  - Structural Transient
  - Harmonic Response
  - Response Spectrum
  - Random Vibration
- The selected environments do not contain convergence.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- During a symmetry analysis, for the **Solution** folder within one or more selected environments, the properties **Number of Sectors** and **Starting at Section**, of the **Cyclic Solution Display** category, are set to **Program Controlled**.

For more information, see Solution Combination (p. 2288).

The Solver Engine was Unable to Converge

Cause

The solver engine was unable to converge on a solution of a nonlinear problem.

Recommendations

- **When Advanced Contact is NOT Present in the Model ...**

  1. Check for sufficient supports to prevent rigid body motion (structural) or check for thermal material curves or convection curves which rise and/or fall sharply over the temperature range (thermal).

  2. If you encounter a convergence error during a thermal analysis that is using contact, consider modifying the **Thermal Conductance** (p. 1253) property.

- **When Advanced Contact IS Present in the Model ...**

  1. Check for sufficient supports to prevent rigid body motion or that contact with other parts will prevent rigid motion.

  2. Check that the loading is of a reasonable nature. Unlike linear problems whose results will scale linearly with the loading, advanced contact is nonlinear and convergence problems may arise if the loading is too big or small in a real world setting.

  3. If the contact type is frictionless (p. 1235), try setting the type (p. 1234) to rough (p. 1235). This may help some problems to converge if any possible sliding is not constrained.

  4. Check that the mesh is sufficiently fine on faces that may be in contact. Too coarse a mesh may cause inaccurate answers and convergence difficulties.

  5. Consider softening the normal contact stiffness KN to a value of .1. The default value is 1 and may be changed by setting the **Normal Stiffness** (p. 1249). Smaller KN multipliers will allow more contact penetration which may cause inaccuracies but may allow problems to converge that would not otherwise.

  6. If symmetric contact is being used (by default the contact is symmetric), consider using asymmetric contact pairs (p. 1237). This may help problems that experience oscillating convergence patterns due to contact chattering. The program can be directed to automatically use asymmetric contact in the **Details** view of the Contact Folder.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

The Solver Has Found Conflicting DOF Constraints

...at one or more nodes. Refer to the Troubleshooting section in the Help System.

A variety of boundary conditions in Workbench direct the solver to apply a specific value of displacement or rotation to one or more nodes. Among these are fixed supports, simple supports, rotational supports, frictionless supports, cylindrical supports, symmetry planes and displacements. Workbench also allows you to rotate nodes using the **Nodal Orientation** (p. 1719) boundary condition. Although Workbench attempts to negotiate these constraints, along with the nodal rotations applied, there may be instances in which a node is directed to take on different and incompatible values of displacement or rotation by two or more of these boundary conditions. For such situations, Workbench will report a conflict.

One example could be to apply non-zero displacements to two faces of a model that meet at an edge, especially when the displacements do not act in perpendicular directions. Nodes along the edge may find conflicting instructions as they are instructed to move different amounts along the same direction in space. If this is the case, consider modifying the non-zero displacements so they act in perpendicular directions.

Another example could be when one or more nodal orientations are added in Workbench with other boundary conditions which are applied to same section of geometry (for example by selecting the same "Scope", or one "Scope" being a part of the other). Each Nodal Orientation prescribes a Nodal Coordinate System to a subset of nodes. Only one Nodal Coordinate System can be prescribed to a given node. Whenever this condition is not met, Workbench creates an error that "The solver has found conflicting DOF constraints with Direct FE loading at one or more nodes".

Direct FE (p. 1719) boundary conditions cannot be applied to nodes that are already scoped with geometry-based constraints which may modify Nodal Coordinate system.

Problem with RSM-Mechanical Connection

If Mechanical appears to hang up as a result of a job processing in the RSM, select the right-mouse option **Disconnect Job from RSM** from the Solution folder to disconnect Mechanical from the current RSM job.

Unable to Find Requested Modes

If this message occurs during a modal analysis (p. 515), most likely a frequency search range was specified but no natural frequencies were found in the specified range. Either increase search range or specify that the first N frequencies be found.

If this message occurs during an Eigenvalue Buckling analysis (p. 487), verify that the loading is in the correct direction (that is, compressive) and that the structure is well constrained so that no *rigid body motion* can occur. If the applied boundary conditions appear to be correct, it is likely that a buckling failure will not occur.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

You Must Specify Joint Conditions to all Three Rotational DOFs

...for one or more joints in the model. Refer to the Troubleshooting section in the Ansys Workbench Manual

Rotations are not independent in 3D. You must define all three rotations for a Joint Condition before proceeding to a solve. The problem is mathematically different on the velocities, as the 3 components are perfectly independent, thus you can define any of the components.

Fracture Meshing Problems

If you encounter fracture meshing failures, first refer to the Fracture Meshing (p. 415) section of the Help for detailed information about the requirements, characteristics, and limitations of fracture meshing. Otherwise, review the troubleshooting options described below.

Handling Cracks Close to Face Boundaries

When a crack is defined close to a common edge shared by two faces and that common edge has been defeatured so that the mesh topology does not conform well to the topology of the geometry, the fracture mesh generation might fail. This is because there may be elements which span across the common edge.

To correct this problem, be careful to ensure the edge of the geometry is captured in the mesh.

See Topology Checking for more information.

Handling Shallow Crack Failures

A crack failure can be caused if the Major/Minor Radius ratio is greater than 3:1 and there are a high number of crack divisions. To correct this problem, reduce the Crack Front Divisions when defining the crack. The recommended range is from 9 to 21 divisions.

A **Large Contour Radius** relative to the size of the crack might also cause a Fracture Mesh generation failure. In this case, reduce the contour radius.

For more information, see the Defining a Semi-Elliptical Crack (p. 433) topic in the Fracture Meshing (p. 415) section of the Help.

Handling Crack Errors for Multiple Cracks

If you define multiple cracks it might take a long time before the mesh fails. This is because multiple cracks are generated sequentially. When more than one crack is defined, a crack with an undesired parameter definition can only be detected after the cracks in front of it are generated.

To correct this problem, either increase the buffer zones so that they overlap each other or reduce them so that they are farther apart from each other.

Handling Buffer Zone Issues

When two Buffer Zones are very close, but not close enough to be combined as one zone during crack mesh generation, crack mesh generation might fail. To prevent this, re-size the buffer zones such that they either overlap or are farther apart from each other.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

If long and skinny tetrahedral elements are generated inside the Buffer Zone, increase the size of the zone using the scale factors. Also, refine the Base Mesh relative to the crack size.

Handling Base Mesh Issues

If the tetrahedra near the crack template boundary are long and skinny, the cause might be that the base mesh is coarse compared to the crack size. To fix the tetrahedra, either increase the buffer zone or refine the base mesh around the cracks.

If the crack mesh generation fails, it might be because the base mesh is either very coarse or very fine compared to the crack definition. To correct this problem, regenerate the base mesh with a mesh size relative to the crack definition. You should also scale the buffer zone to the crack definition.

If the crack mesh generation fails due to base mesh size, regenerate the base mesh with a size relative to the crack size.

Handling Fracture Affected Zone Definition Failures

If an error occurs when the crack template is inserted, it can be caused by one or more of the following:

- The **Fracture Affected Zone Height** is big compared to the contour radius. If the height is too large, layers outside of the contour will not look good.
- The Contour Radius is too big compared to the crack definition
- There are a large number of mesh contours
- There are a large number of Crack Front definitions

To correct these problems, you can:

- Reduce the **Fracture Affected Zone Height** manually or use the **Program-Controlled** option.
- Reduce the contour radius. The radius should be small enough to allow room for one layer of an element outside the contour.
- Reduce the number of mesh contours.
- Reduce the Crack Front divisions. Make sure, however, that there are at least nine Crack Front divisions.

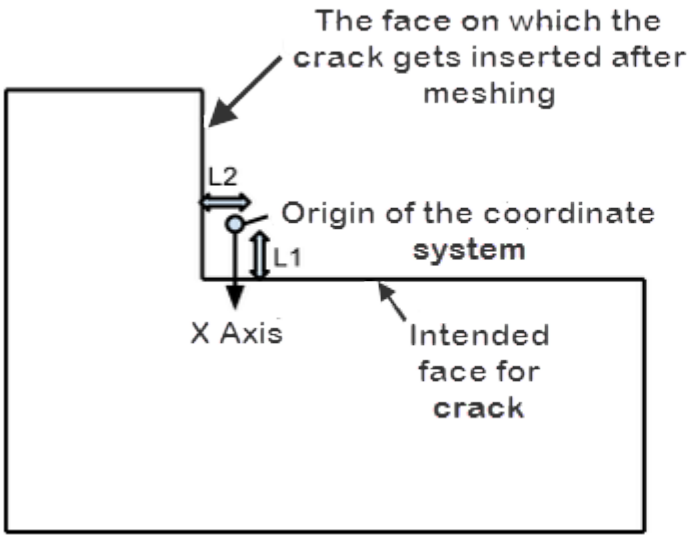
Handling X-Axis Rotation and Alignment

If the X-Axis of the coordinate system which is used to define the crack is not aligned along normal of the surface on which the crack is inserted, the mesher issues a warning message and rotates the X-Axis to be consistent with the surface normal.

Handling Projection to the Surface

If the origin point of the coordinate system is not on the surface of the scoped body, the crack mesher automatically inserts a crack on a face of the scoped body that is nearest to the origin location. Note that the chosen model face may be different than the intended model face. The mesher picks the face nearest to the location of the origin.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Since, L2 < L1 the crack will be inserted by mesher on the vertical face instead of the intended horizontal face.

To ensure the mesher chooses the intended model face, use the **Create Coordinate System Aligned with Hit Point Normal** option:

1. Click the **Hit Point Coordinate** button on the Graphics Toolbar (p. 97).
2. Select the point on the face where you want the origin to be located.
3. Right-click the point and choose **Coordinate System Aligned with Hit Point Normal**.

Improving Mesh Quality

Note that aggressive shape checking is available for Base Mesh generation, but is not available for Fracture meshing.

High aspect ratio elements might be generated:

- Around the crack tip if a smaller number of crack front divisions are used.
- Near the buffer zone boundary if a small crack is defined in a coarse base mesh.

To improve mesh quality:

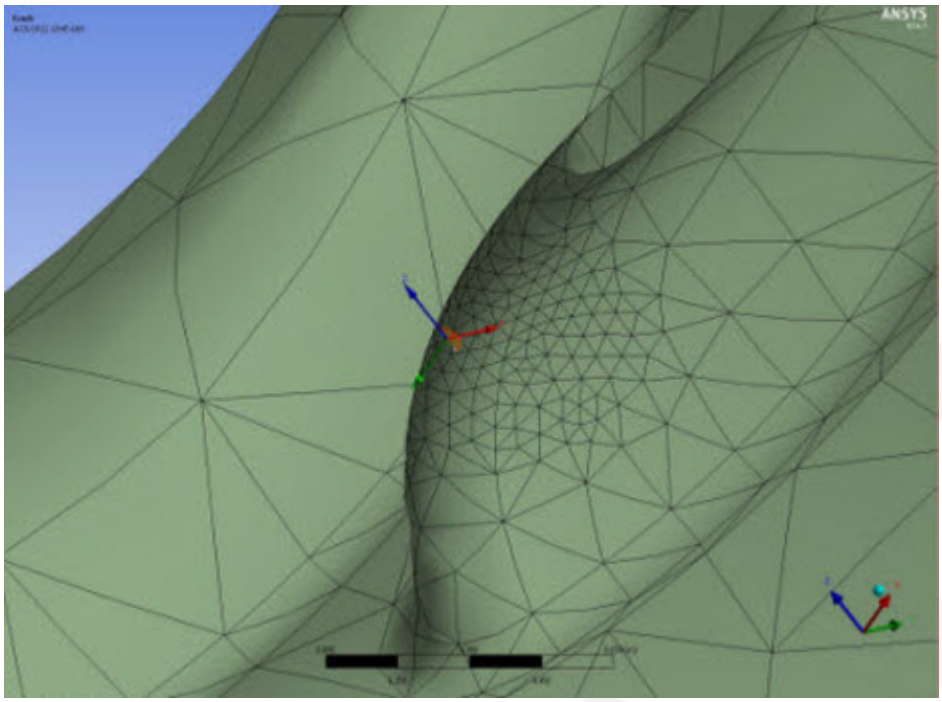
- Increase the Crack Front divisions within a range of 9-21 divisions.
- Increase the Buffer Zone or regenerate the Base Mesh with respect to the crack size by using **Local Sizing Control** with **Sphere of Influence**.

Handling Cracks on High Curvature Surfaces

When you define a crack on a highly curved surface, such as the one shown below, the generation of the crack may fail.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



To correct this, use the information from feedback messages to redefine the crack definition and/or modify the mesh region on which the crack is inserted.

Lustre Parallel File Systems on Linux

Projects created prior to Release 2024 R1 will not be usable on Lustre until migrated as follows:

1. Load the project into 2024 R1 software on a system that does not use a Lustre parallel file system.
2. Perform an operation that changes each model in the Mechanical system (for example, hide and then show a part). If systems share the same model, the change needs to be done for only one of the systems.
3. Save the project.

Lustre Parallel File Systems Requirement on Linux

In order to enable file locking on the filesystem, Mechanical requires that you specify a LUSTRE mounting option. Supported options include **`flock`** (preferred) and **`localflock`**.

Recommendations

Microsoft ClearType edge smoothing option may cause font display problem

If you use Microsoft ClearType edge smoothing method with Large size DPI setting, you may see distorted dimension text in DesignModeler and legend text in the Mechanical application. The problem occurs when the user minimizes or maximizes the Workbench window. In DesignModeler the display can be corrected on some machines by nudging the graphics window pane a pixel or two. This will cause a

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

resize event in the graphics browser which will redraw the dimension text properly. Nudging the graphics window pane does not correct the problem in the Mechanical application, however. Alternatively, if the edge smoothing method is set to Standard instead of ClearType, then the text display appears correctly in both applets. Note though, this is machine dependent, so the suggestions may not work on all machines. To ensure the text appears properly, you should turn off edge smoothing entirely.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Appendix A. Glossary of General Terms

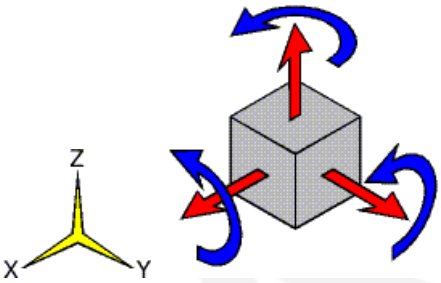
API	Application Program Interface: This is a defined interface of functions that can be called by the scripts. This interface will remain reasonably constant and no functions will be removed without deprecation and warning.
Callout	A message that appears as a result of an action initiated within the wizard. Callouts usually point to a toolbar button, a row in the Details Pane (p. 119), or object in the Outline (p. 105). The message contains descriptive and instructive text.
Context Menu	Provides a short list of options applicable to a specific object or window. To view a context menu, click the right mouse button on an object or in a window.
Context Toolbar	A toolbar containing options appropriate for the current level in the Outline (p. 105).
Deprecate	When a function in the API is removed it will be deprecated and undocumented. This means that it will still be available for the next release, but will be removed in the future. A warning will be provided with a suggested alternative method of achieving the same function.
Details View	Provides information on the highlighted object in the Outline (p. 105).
Displacement	A vector quantity used to measure the movement of a point from one location to another. The basic unit for displacement is (Length).
Double	Data type that can be assigned to real (decimal) numbers, e.g. 2.3462
Drag	Moving an on-screen object in the Outline (p. 105) from one location to another using the mouse cursor while holding down the left button. The drag is interpreted as "move" if the object is dragged from the outline and "copy" if the object is dragged from the outline while holding down the **Ctrl** key
Edge	A selectable entity on a part that occurs at the intersection of two surfaces. In a surface model, an edge can also exist on the edge of one surface.
Elastic Strain	Normal elastic strain is a measure of the elongation or contraction of a hypothetical line segment inside a body per unit length. Normal elastic strain is dimensionless, however in practice it is common to assign normal elastic strain the basic unit of (Length / Length). Shear elastic strain is a measure of the change in angle that occurs between

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

	two initially perpendicular hypothetical line segment inside a body. The basic unit for shear elastic strain is radians.
Environment Temperature	This property of the analysis environment object, defines the temperature of the body unless this temperature is specified by a particular load such as a thermal condition or an imported temperature. This will also be the material reference temperature unless overridden by the Body. Not a valid property for thermal analyses.
Face	A selectable area on a part bordered on all sides by edges. Periodic, non-boundary edged faces (like spheres) may occasionally appear.
Factor of Safety	Factor of safety is defined as the ratio of the limit strength of a material to the maximum stress predicted for the design. This definition of factor of safety assumes that the applied load is linearly related to stress (an assumption implicit in all calculations performed in the application). A factor of safety of less than one generally predicts failure of the design; in practice a factor of safety of one or greater is required to help avoid the potential for failure.
FEA	Finite Element Analysis. A robust and mature technique for approximating the physical behavior of a complex system by representing the system as a large number of simple interrelated building blocks called elements.
Fundamental Frequencies	The fundamental frequencies are the frequencies at which a structure under free vibration will vibrate into its fundamental mode shapes. The fundamental frequencies are measured in Hertz (cycles per second).
Heat Flux	A measure of heat flow per unit area. The basic unit for heat flux is (Heat / Length*Length).
Int	Data type that can be assigned to integer (whole) numbers, e.g.2
Margin of Safety	Margin of safety is always equal to the factor of safety minus one.
Multiple Select	Select more than one surface, edge or vertex by holding the **Ctrl** key.
Object	A set of information displayed visually as an icon (usually in the Outline (p. 105)).
Python	This is a non-proprietary scriptable programming language that is commonly used throughout the world. Full details can be found at www.python.org. A number of debuggers are available to enable a script to be stepped through.
Reference Temperature	The reference temperature defines the temperature at which strain in the design does not result from thermal expansion or contraction. For many situations, reference temperature is adequately defined as room temperature. The reference temperature is defined for each body in a model. A coefficient of thermal expansion curve will be

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

	adjusted for the body's reference temperature if the reference temperature of the coefficient of thermal expansion is different.
Right-Hand Rule	The right-hand rule is a convenient method for determining the sense of a rotation defined by a vector: close your right hand and extend your thumb in the direction of the vector defining the rotation. Your fingers will indicate the sense or direction of the rotation. The direction in which your fingers curl is the positive direction.
Rigid Body Motion	Might occur when the part is free to translate or rotate in one or more directions. For example, a body floating in space is free to move in the X-, Y-, and Z-directions and to rotate about the X-, Y-, and Z-directions. 
Stress	A measure of the internal forces inside a body. The basic unit for stress is (Force / Length*Length).
String	Data type that can be assigned to one or more characters of text, e.g. Hello World
Temperature	A scalar quantity used to measure the relative hotness or coldness of a point from one location to another. The basic units for temperature are degrees Fahrenheit or Celsius.
Vertex	A selectable entity on a part that occurs at the intersection of two or more edges.
World Coordinate System	The fixed global Cartesian (X, Y, Z) coordinate system defined for a part by the CAD system.
XML	eXtensible Markup Language: This is a standard layout of text based files in a metalanguage that enables users to define their own customized markup languages.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Appendix B. Data Transfer Mapping and Validation

This appendix examines the properties that are available when you map node and element data from external systems to Mechanical systems. It defines the requirements for these properties and also describes additional properties that become available depending upon how you have configured your up and downstream systems. In addition, this appendix discusses support properties that include graphical controls and Named Selection generation options as well as the Mapping Validation feature that, if available for your analysis configuration, performs an accuracy evaluation of the mapping process.

Data Transfer Mesh Mapping

Mapping Validation

Data Transfer Mesh Mapping

To transfer data across a dissimilar mesh interface, the nodes of one mesh must be mapped to the local coordinates of a node/element in the other mesh. This section describes the settings that are available in Mechanical when data is mapped across two different meshes.

You can add the exported mesh and loads as external data in the project schematic and couple a new Mechanical analysis system with this external data. The Mapping Settings described below are available within Mechanical for Thermal-Stress (p. 836) coupling with dissimilar mesh, Submodeling (p. 807), when temperatures or displacements are transferred from Mechanical to Ansoft, or when the source data comes from an External Data system.

Mapping Settings

The **Settings** category provides the following properties.

Mapping Control

By default, when **Program Controlled** is selected, the software will determine the appropriate algorithm and settings based on the source and target mesh data, as well as the data type being transferred. See Program Controlled Mapping (p. 2408) topic below for additional information. You may choose to modify the advanced features by setting this to **Manual**.

Mapping

This read-only property displays the mapping algorithms the application selects. Options include:

- **Profile Preserving**: Using this mapping option, the application simply takes the profile of the variable (for example, temperature) on one mesh, and matches or maps it to the other mesh as best as it can.
- **Conservative**: Using this mapping option, the application makes sure that the profile is interpolated in such a way as to ensure that a total quantity passing across the interface is conserved, that is the same total passes out of one mesh and into the other. For example,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

a conservative interpolation of force ensures that the total force on one side of the boundary exactly matches the total force received by the other side of the boundary, even if the mesh resolution is poor. Conservative interpolation does not make sense for a variable such as temperature where there is no corresponding physical quantity to conserve.

Note:

Conservative algorithms are only available for Imported Force loads. Conservative algorithms are not available for 2D to 3D data transfers. If conservative algorithms are not available, it is a read-only field displaying that a "Profile Preserving" algorithm is being used.

Weighting

Choose which type of weighting should be performed. This option can be changed only if **Mapping Control** is set to **Manual**.

- **Direct Assignment** applies the source value directly on the nodes/elements identified by Node IDs/Element IDs in the External Data specification.

  Note:

  This mapping only supports loads applied to nodes or elements. The following loads applied to element faces are not supported:

  – Pressure (on element faces)

  – Convection Coefficient

  – Heat Flux

- **Triangulation** creates temporary elements from the *n* closest source nodes to find the closest points that will contribute portions of their data values. For 3D, 4-node tetrahedrons are created, and for 2D, 3-node triangles are created by iterating over all possible combinations of the source points (maximum number controlled by the **Limit** property), starting with the closest points. If the target point is found within the element, weights are calculated based on the target's location inside the element.
- **Distance Based Average** uses the distance from the target node to the specified number of closest source node(s) to calculate a weighting value.
- **Quaternion** (**Imported Element Orientations** only) performs the interpolation in the quaternion space rather than directly on the Euler angles. Weighting values are calculated as in the **Distance Based Average** case. All options available for the **Distance Based Average** also apply to this Weighting type. This is the preferred approach for mapping orientations because:

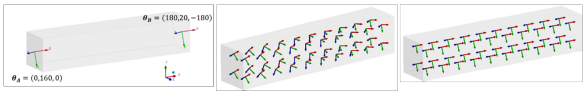
  Several Euler angles parametrize the same orientation. Interpolating in the quaternion space enables the application to account for equivalence. See Figure 34.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

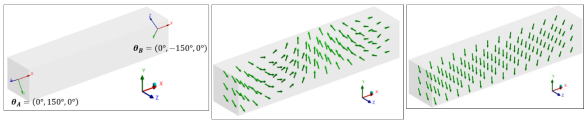
Quaternion interpolation provides the shortest path between two orientations as compared to interpolating the Euler angles directly, which can lead to counter-intuitive results. See Figure 35.

Figure 34: Interpolating Between Different (but equivalent) Euler Angles



Consider interpolating between same orientations A and B expressed with different (but equivalent) Euler angles (left). Euler angles interpolation (center) leads to counter-intuitive results. The quaternion interpolation (right) delivers the expected results (constant orientations over the part).

Figure 35: Quaternion versus Euler Angle Interpolation

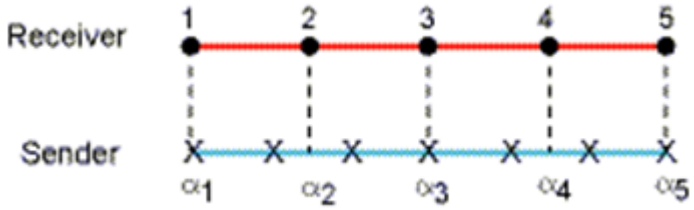


Consider interpolating between the orientations A and B shown on the left. Quaternion interpolation (right) provides the shortest path between the two orientations, while Euler angles interpolation (center) takes the longest one.

- **Weighting**:
  - **Shape Function**: Two mapping methods are available for a load transfer: "Profile Preserving" and "Conservative".

    In a Profile Preserving mapping, each node on the target (receiver) side maps onto an element on the source (sender) side (α_1). The transfer variable is then interpolated at α_1. The transfer value is $\mathbf{T}_1 = \phi\ (\alpha_1)$. Thus, all nodes on the target side query the source side.

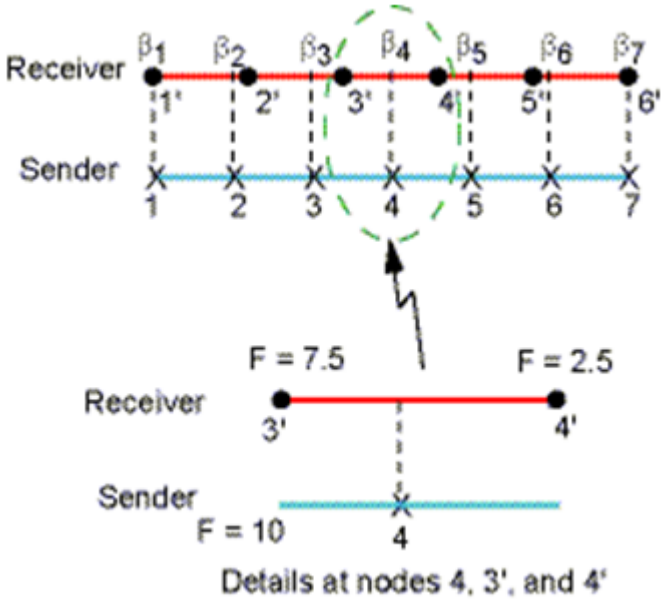
    Figure 36: Profile Preserving Mapping



    In a conservative mapping, each node X on the source (sender) maps onto an element on the target (receiver) side. Thus, the transfer variable on the source is split into two quantities that are added to the target nodes. As shown in the following figure, the force at node 4 splits into forces at nodes 3' and 4'.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 37: Conservative Mapping



Thus profile preserving (conservative) version of **Shape Function** algorithm loops over the target (source) nodes and tries to locate a source (target) element that each target (source) node can be mapped to. Weights for each of the source nodes are then assigned based on the location of the target (source) node and the shape function of the element. For each target (source) node, the search efficiency can be improved by restricting the search to a subset of the source (target) elements. The search algorithm works by:

→ Distributing all source (target) elements into Cartesian boxes or buckets. The number of buckets is controlled by the **Scale** property.

→ Locating each of the target nodes in a box

→ Finding an element that each target (source) node can be mapped to by restricting the search with each target's (source's) box

---

Note:

→ When there is a significant distance between target (source) node and the closest element, e.g. Shell-Solid submodeling, the node and the element may not be found in the same box. In order to improve mapping accuracy in such cases, the **Pinball** control may be used. See **Pinball** in the Advanced (p. 2397) section for more details.

→ For conservative mapping, the value on a source node is distributed only on the nodes of the target element it is mapped to. Therefore, it is possible, especially if you are mapping from coarse to fine meshes, that some of the mapped target nodes get a zero value. This is because none of the adjacent elements are mapped to one or more source nodes.

→ For the Linux Platform, this mesh mapping option is limited to the use of only one processor core.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

– **Kriging** is a regression-based interpolation technique that assigns weights to surrounding source points according to their spatial covariance values. The algorithm combines the kriging model with a polynomial model to capture local and global deviations. The kriging model interpolates the source points based on their localized deviations, while the polynomial model globally approximates the source space. See Kriging in the *DesignXplorer User's Guide* for more information.

Note:

By default, the Kriging technique uses an adaptive algorithm and ensures that the interpolated values do not exceed specific limits. The adaptive algorithm starts by using the higher-order **Cross Quadratic** polynomial to interpolate data. If the interpolated value of each target point is outside the extrapolation limit you specified, the algorithm re-interpolates data by reducing the polynomial order and the number of source points. Target nodes whose values are outside the limits when the lowest polynomial type is used are not assigned a value.

The Kriging algorithm, when used with the higher-order **Cross Quadratic** or **Pure Quadratic** polynomial, may fail to correctly interpolate data for a target point if multiple source points are spaced close to one another or if the target point is outside the region enclosed by the source points that are selected for interpolation. This may introduce gross errors in the estimation of the target value and manifests itself mostly when mapping data on surface or edge geometries. In such cases, you should change the **Polynomial Type** to **Constant** or **Linear** and, if necessary, reduce the number of source points to be included for the interpolation.

– **UV Mapping** can be used to transfer data from one surface to another. Unlike other algorithms, UV mapping does not require the surfaces to be coincident. This allows for mapping between deformed and un-deformed geometries, as well as transfers between dissimilar geometry. Element data is required from both the source and the target mesh. If the source is an Mechanical APDL `.cdb` file containing volumetric element data, a nodal component must also be specified which will be used to define the surface from which the data transfer will occur.

Polyhedral Surface Creation and Conversion to UV

To map a mesh in UV space, the application first creates polyhedral surfaces from the given mesh data. If the source mesh is volumetric data, an associated node-based component must be selected such that the nodes consist of the surface area where the mapping takes place. Once the application creates the source and target surfaces, they are then 'unfolded' and converted into UV coordinates. The application defines the UV space as a parametric space where the axis data equals 0.0 to 1.0. Alignment points anchor the node locations to the corners of the 1x1 box.

Interpolation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Once the source and target data is converted to UV space, the target nodal UV locations are used to locate the source element that would contain the target node. The value for the target is then calculated based on the values provided from the source elements nodes.

Note:

Available weighting options depend upon the data available from source and target and on the **Mapping** setting. Some of the weighting options may not be available for certain mesh data or **Mapping** settings. For example, when **Mapping** is set to *Profile Preserving*, **Shape Function** and **UV** are only available when the source provides element information. For *Conservative* **Mapping**, only Shape Function for Weighting and Surface for Transfer Type is available.

Interpolation

Interpolation (Imported Trace only): This property species how the application calculates effective conductivity on a per element basis when you are performing an analysis that involves trace mapping of a circuit board. Options include **Nondirectional** (default) and **Directional**. The **Nondirectional** option calculates effective conductivity by averaging the trace data in each element. Using the **Directional** option, the application calculates the effective orthotropic conductivity for each element using the position and values of the trace data within each element.

Transfer Type

Enables you to choose the dimension of the transfer (for 3D profile preserving transfers only). This option is available only for **Triangulation**, **Shape Function**, and for adaptive **Kriging**. For best results, use the **Surface** option when mapping data across surfaces and the **Volumetric** option when mapping data across volumes.

When used with **Triangulation**:

- The **Surface** option tries to map each target point by searching triangles that are created from the set of closest source points. The target point will be projected onto the plane relative to the triangle surface. If the point is found inside the triangle, the weights are calculated based on the target's projected location inside the triangle.
- The **Volumetric** option tries to map each target point by searching tetrahedrons that are created from the set of closest source points.

When used with the **Shape Function**:

- The **Surface** option uses the bucket surface search algorithm to locate a source element that each target node can be mapped to. This option supports only triangle and quadrilateral source elements; do not use it if your source elements are other shapes because the algorithm does not account for these shapes.
- The **Volumetric** option uses the bucket volume search algorithm to locate a source element that each target node can be mapped to. This option supports triangle, quadrilateral, tetrahedron, hexahedron, and wedge source elements.

When used with adaptive **Kriging**, the **Surface** option uses fewer surrounding source points to interpolate data than the **Volumetric** option does.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

2D Projection

Available only for 2D to 3D data transfers from an External Data system connected to Mechanical. The default option is **Normal To Plane**. You will be able to choose between the default as well as all application and user input coordinate systems.

Important:

When you use a cylindrical coordinate system , the application projects the points on the mapped surface around the Z axis and into the ZX Plane on the positive X axis side.

Rigid Transformation

The properties of the **Rigid Transformation** category enable you to apply coordinate system transformations to the source points using the options of the **Mesh Alignment** property listed below.

Important:

- For source data imported from an upstream **External Data** system, any transformation in Mechanical is added onto the Rigid Transformation performed in upstream **External Data** system.
- Rigid Transformations are not supported when the **Weighting** (p. 2390) property is set to **Direct Assignment**.

The **Mesh Alignment** property includes two options:

Use Origin and Euler Angles

You change the source point coordinate locations by specifying distance and rotational degree entries in the Origin and Theta properties. For example, an entry of 0.1 meters in the Origin X property would offset the X location of all the source points by 0.1 meters. In addition, it is important to note that rotational changes are **made first** and in the following order:

1. Theta ZX
2. Theta YZ
3. Theta XY

Once complete, translational changes are made.

Use Coordinate Systems

When you select this option, the **Source Coordinate System** and the **Target Coordinate System** properties display. Each property provides a drop-down list of available coordinate systems to select from. Based on your selections, the transformations are calculated so that the **Source Coordinate System** automatically aligns with the **Target Coordinate System**. For example, when the source mesh is defined in the XY plane, whereas the target geometry is defined in a plane obtained by applying the Euler rotations RXY, RYZ and RZX to the XY plane. Then choosing Global Coordinate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

System as Source Coordinate System and the coordinate system created by applying the transformations RXY, RYZ and RZX to the Global Coordinate System as the Target Coordinate Systems, the source mesh is transformed such that it is aligned with the target geometry. This option is useful if the source points are defined with respect to a coordinate system that is not aligned with the target geometry system.

Note:

When your upstream system is an **External Data** system, the default option for both properties is **External Data Coordinate System**. This internal coordinate system is not visible in the Outline. It represents the coordinate system that transformed the source points in the **External Data** system. If there are no Rigid Transformations defined in the upstream **External Data** system, the **External Data Coordinate System** is the same as the **Global Coordinate System**.

Note:

For an **Imported Load** or **Imported Thickness** object, set the **Display Source Points** property to **On** to display the alignment of the source and target points.

Graphics Controls

The **Graphics Controls** category provides the following properties.

- **Display Source Points**: Toggle display of source point data. This can be helpful in visualizing where the source point data is in reference to the target mesh.
- **Display Source Point Ids**: Toggle display of source point identifiers. This can be helpful in conjunction with validation objects when trying to identify nodes with undefined values. Note that if a column is not defined with the Node ID **Data Type**, the source point ids will correspond to the row from which they come in the file. For formatted and delimited files, ids will start after skipped lines.
- **Display Interior Points**: Available when **Display Source Points** or **Display Source Point Ids** is set to **On**. Toggle allowing source point data to be displayed through the model so that interior points can be seen.
- **Display Projection Plane**: Toggle display of project plane (available only for 2D to 3D mapping).

Legend Controls

The **Legend Controls** category provides the following properties.

- **Legend Range**: **Program Controlled** (default) or **Manual** control of the legend minimum and maximum values. When **Program Controlled** is selected, the target data's minimum and maximum values will be used in the legend. When **Manual** is selected, control of the **Maximum** and **Minimum** values can input and the graphics will be drawn based on these values.
- **Minimum**: When **Legend Range** is set to **Manual**, this option is available for inputting the minimum legend value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Maximum**: When **Legend Range** is set to **Manual**, this option is available for inputting the maximum legend value.
- **Source Minimum**: Read only field providing the source data minimum value.
- **Source Maximum**: Read only field providing the source data maximum value.

Named Selection Creation

The **Named Selections** category provides the following properties.

- **Unmapped Nodes**: Activating this property creates a named selection containing all of the points that cannot be mapped. The default setting is **Off**.

  In addition, when you activate this property, an associated **Name** property displays. This property displays the name of the Named Selection. You can edit this field. By default, the application assigns name "Unmapped Nodes."

- **Mapped Nodes**: Activating this property creates a named selection that contains all mapped points. The default setting is **Off**.

  In addition, when you activate this property, an associated **Name** property displays. This property displays the name of the Named Selection. You can edit this field. By default, the application assigns name "Mapped Nodes."

- **Outside Nodes**: Activating this property create a named selection containing all the points that cannot be found within tetrahedrons/triangles when **Triangulation** is used. The default setting is **Off**.

  In addition, when you activate this property, an associated **Name** property displays. This property displays the name of the Named Selection. You can edit this field. By default, the application assigns name "Outside Nodes."

These settings are only available for profile preserving mapping.

Advanced

The application filters the properties of the Advanced category based on the settings made in the Mapping Control and Weighting properties in the Mapping Settings (p. 2389) category. Properties include:

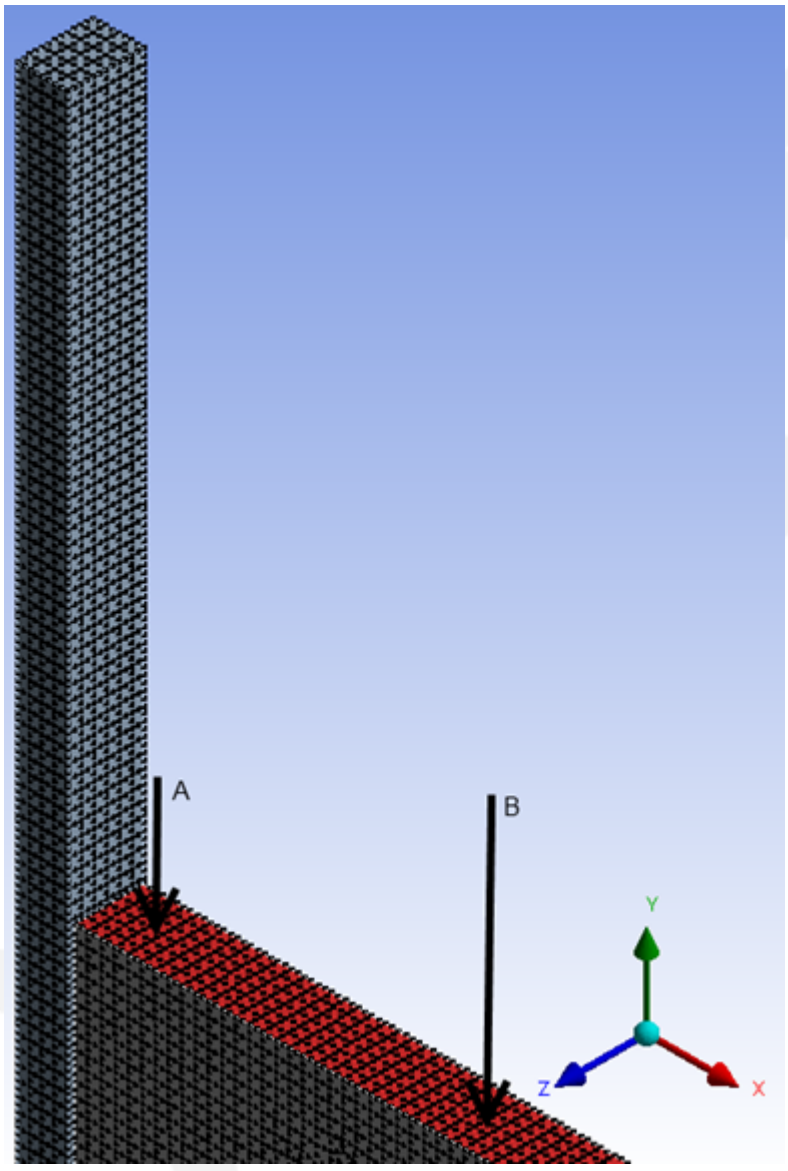
- **Pinball**: The **Pinball** property enables you to specify a region of interest around a target point. Only the source points/elements inside the pinball region are considered for mapping and any point/element outside of the pinball will not be used. Specific behavior of the **Pinball** control is dependent on the **Weighting** type selected as discussed below:
  - **When used with Triangulation or Distance Based Average**, a bounding box is created around the target point based on the value of the pinball to find the closest source points. Any point outside of the bounding box will not be used. By default, the **Program Controlled** value is 0.0, which calculates the distance based on .05% of the source region's bounding box size. The bounding box will automatically resize if the mapping is unable to find the minimum number of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

points required to calculate weighting factors. (Note that resizing occurs only for **Program Controlled**.)

Note:

In certain cases when **Pinball** is set to **Program Controlled**, the process of searching for source nodes around a target node can take a long time. In the image below, the target nodes are located on the red face. The target nodes (A) closest to the vertical body will quickly find nodes in the +Y axis direction. Target nodes (B) further down the X axis will take longer to find.



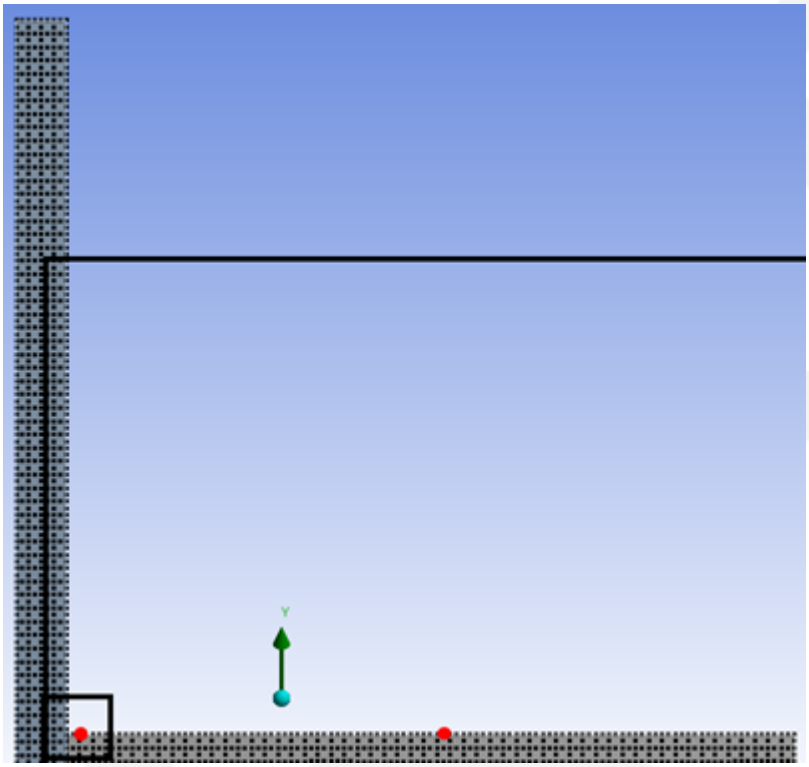
As an example, consider the case shown in the image below. The two red dots indicate target nodes in regions A and B. For each target node, the triangulation algorithm will begin its search for source nodes within the perimeter of a psuedo cube (bounding box) centered at its location. For the first pass, the edge length of the cube is set to be 0.05% of the maximum bounding box length of the source region. The algorithm looks to find 'n' source points (set by the limits property) in the positive and negative X, Y, and Z axes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

of the cube. If 'n' source points cannot be found in any of the six directions (±X, ±Y, and ±Z), the size of the search region is doubled and the process repeated. The search process continues until the required number of source points are found in all directions or until the search region extends beyond the limits of the source bounding box.

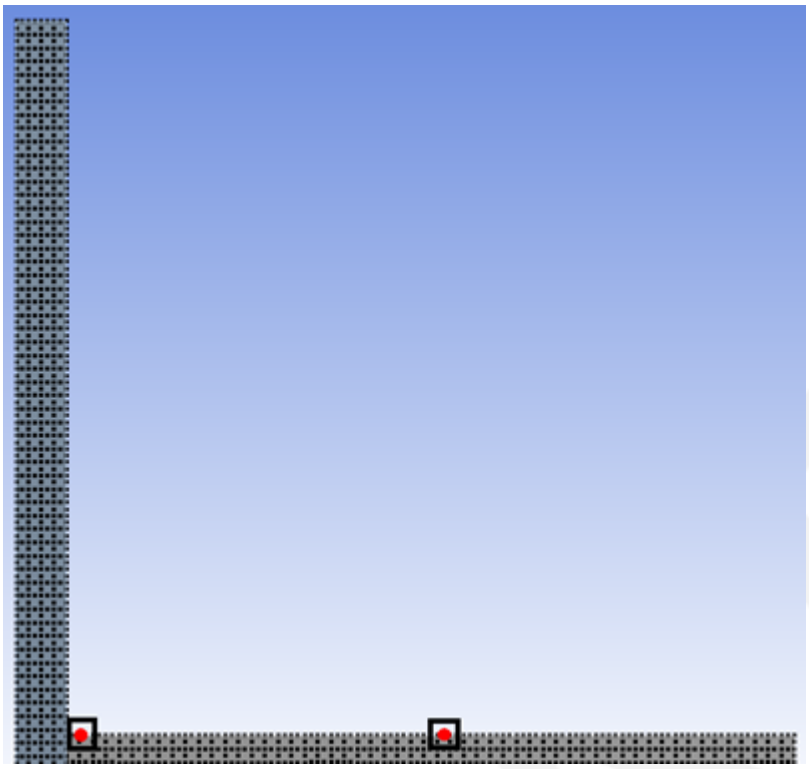
During the first pass, for the target node in region A, the algorithm is able to find the required number of source nodes. However, for the target node in region B, sufficient nodes cannot be found in the +Y direction and the size of the search area is increased. As illustrated in the figure below, for the target node in region B, the algorithm runs through several iterations before it is able to find the required number of source nodes. This results in an increase in time as well as the possible inclusion of source nodes that are significantly further away from the target node.



Please note that for each target node the pinball is reset to its initial size (0.05% of the maximum bounding box length) before the search begins.

For such cases it is recommended that you specify a pinball value so that the search box can be controlled to only find nodes within a certain region. This allows for triangulation to quickly search for source nodes, as well as to ignore source nodes that are sufficiently far away from the target node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



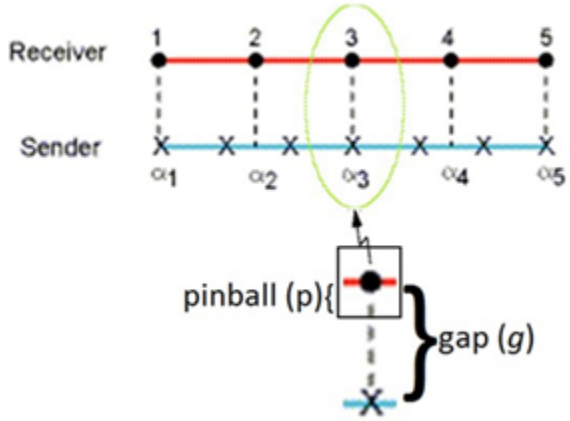
– **When used with Shape Function**, the **Pinball** control can be used to:

→ Exclude from mapping to elements far away from the target point. When **Transfer Type** is **Surface**, the target point is projected onto the source elements to find the matching element. Due to projection, the gap (the distance between target point and its projection on the matching element) between the target point and the matching element may be large. Such elements are excluded from mapping if the gap is larger than the **Pinball Value** specified.

→ Expand the search region to find matching elements. Shape function algorithm works by distributing the source elements into regions called buckets, and then for each target point, finding the appropriate bucket and searching for the matching element in the bucket. When there is a significant distance between a target node and the closest element, e.g. Shell-Solid submodeling, the node and the element may not be found in the same bucket. In order to improve mapping accuracy in such cases, the **Pinball** control may be used to include additional buckets for mapping. When a **Pinball Value** greater that 0 is specified, then a bounding region is created around the target node using the **Pinball Value** and all the buckets associated with the region are used to find the appropriate element. To improve the mapping efficiency, the search is restricted only to the elements within the bounding region.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23



α_3 is excluded when pinball (p) < gap (g), and included when pinball (p) $\geq$gap (g).

The **Pinball** option is not available when **Weighting** is set to **Kriging**.

- **Limit**: Number of nearby points considered for interpolation. Defaults to 20. Lower values will reduce processing time, however, some distorted or irregular meshes will require a higher **Limit** value to successfully encounter nodes for triangulation.

  When **Weighting** is set to **Kriging**, the minimum value that can be used is based on the selected **Polynomial** type.

Weighting	Minimum Limit	Maximum Limit
Triangulation	5	20
Kriging (Constant)	3 (3D), 2 (2D)	Number of source points
Kriging (Linear)	4 (3D), 3 (2D)	Number of source points
Kriging (Pure Quadratic)	7 (3D), 5 (2D)	Number of source points
Kriging (Cross Quadratic)	10 (3D), 6 (2D)	Number of source points

- **Outside Option**: Enables you to ignore or choose a different weighting algorithm for target points that cannot be found within the source mesh/points. Different options are available, based on the **Weighting** option chosen:
  - **When used with Triangulation**. For target points that cannot be found within tetrahedrons/triangles created for **Triangulation**.
    - → **Distance Based Average**: The mapping will use a weighted average based on distances to the closest **Number of Points**. **Distance Based Average** is the default option.
    - → **Ignore**: Target points will be ignored and no value will be applied.
    - → **Projection**: Triangles will be created from the closest **Number of Points** and the target point will be projected onto the plane relative to the triangle surface. If the point is found inside the triangle, the weights are calculated based on the target's projected location inside the triangle. This option is available only for 3D transfers when the **Transfer Type** is set to **Volumetric**.
  - **When used with Shape Function**. For target points that cannot be found within source elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

→ **Nearest Node**: The mapping will use the data from the nearest source node.

→ **Ignore**: Target points will be ignored and no value will be applied.

Note:

– For the **Conservative Shape Function** algorithm, the source mesh is mapped onto the target mesh (as opposed to profile preserving version, which maps target mesh onto source), and outside options control the contribution from source nodes which fall outside the target mesh.

– **Nearest Node** is the default option for the **Profile Preserving Shape Function** algorithm, while the **Ignore** option is the default for the conservative algorithm.

- **Number of Points**: When **Weighting** is set to **Distance Based Average**, or when **Outside Option** is set to **Distance Based Average** or **Projection**, this option is available to specify how many closest source points should be used when calculating weights. Valid range is from 1 to 8 for **Distance Based Average** and 3 to 20 for **Projection**. Defaults to 3.
- **Outside Distance Checking**: When **Weighting** is set to **Triangulation** and **Outside Option** is set to **Distance Based Average** or **Projection**, this option enables you to specify a **Maximum Distance** cutoff beyond which source points will be ignored. Defaults to **Off**. The maximum number of source points is limited to the value specified by the **Number of Points** setting.

  – If the **Outside Option** is set to **Distance Based Average**, only source points that lie on or within a sphere (centered at the targets location and radius defined by the Maximum Distance value) will provide contributions.

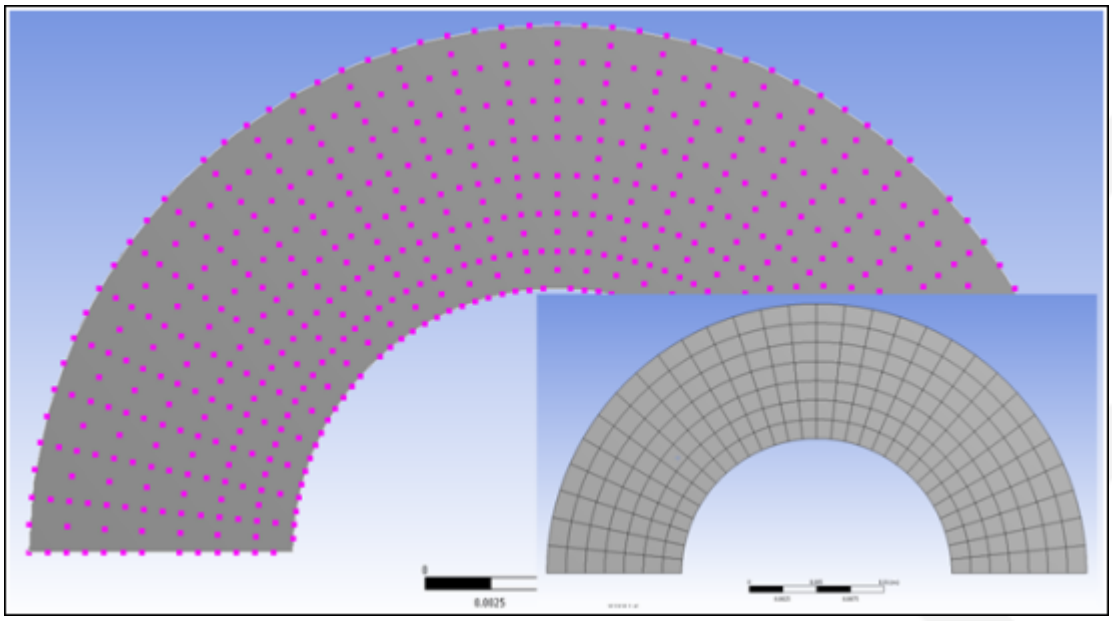
  – If the **Outside Option** is set to **Projection**, the algorithm only uses triangles with centroids that lie on or inside a sphere (centered at the targets location and radius defined by the **Maximum Distance** value).

  In Figure 38: Outside Nodes (Pink) with Mesh Overlay (p. 2403), all the pink nodes on the surface are found "Outside" the source points and will use the **Outside Distance Checking** based on the **Maximum Distance** specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

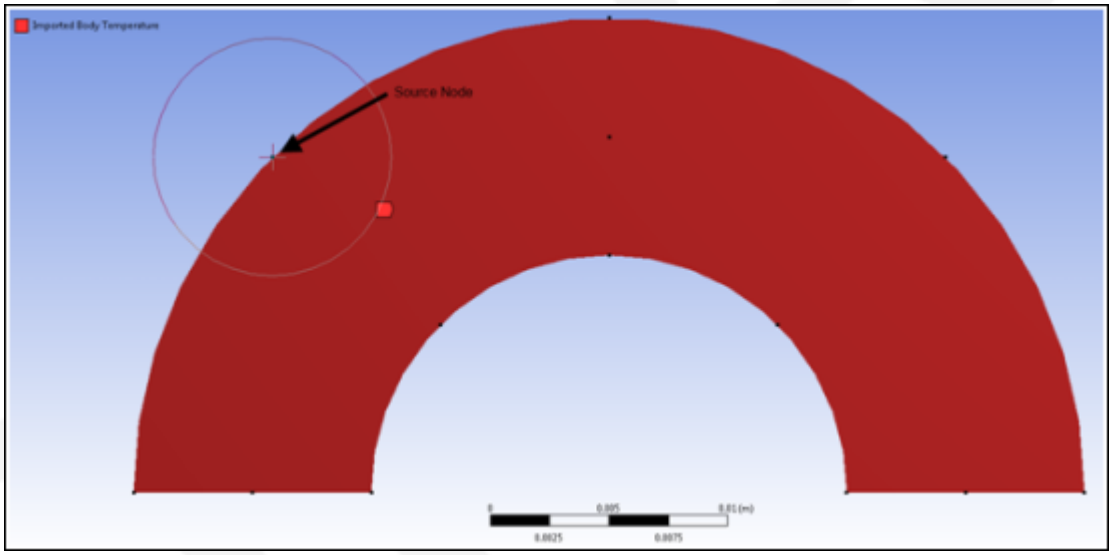
Published: 2023-10-23

Figure 38: Outside Nodes (Pink) with Mesh Overlay



In Figure 39: Maximum Distance set to 0.005 (m) (p. 2403), the circle is at the mouse location with radius set to 0.005 (m). Nodes within this radius will be mapped. The source nodes are drawn as black dots and come from an extremely coarse mesh.

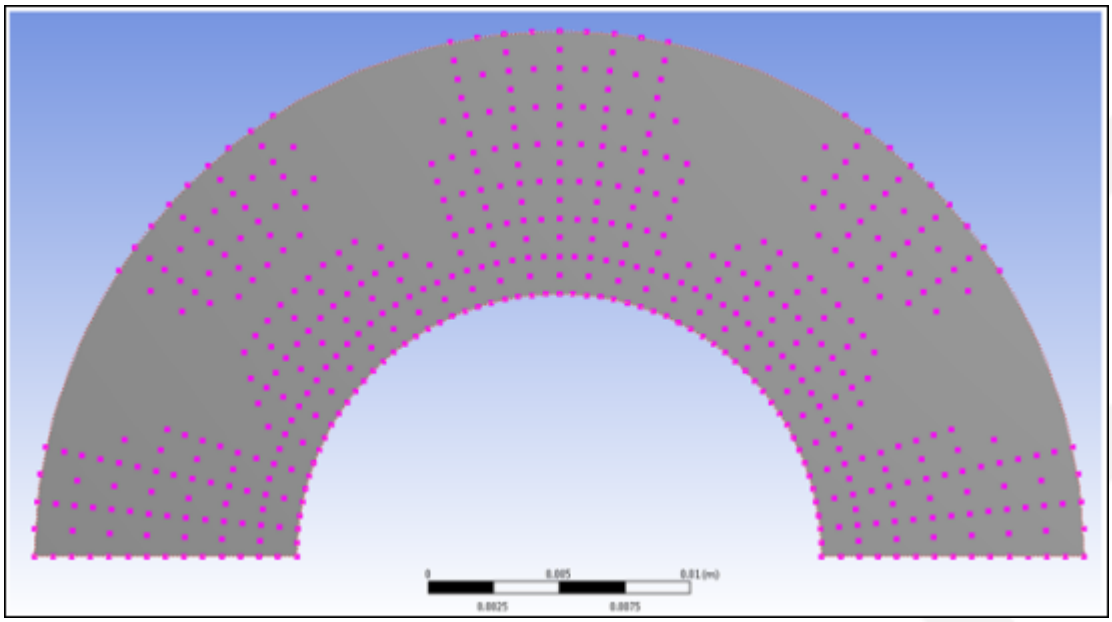
Figure 39: Maximum Distance set to 0.005 (m)



In Figure 40: Mapped Nodes (p. 2404), the "Outside" nodes get mapped because they are located within the Maximum Distance.

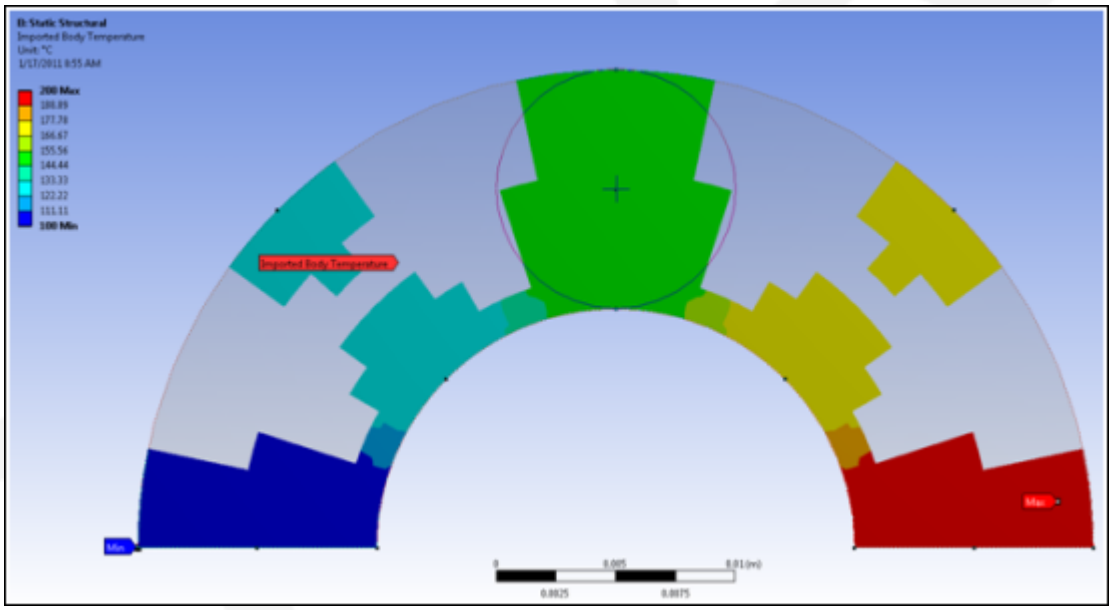
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 40: Mapped Nodes



The result of the import is shown in Figure 41: Imported Data using Maximum Distance for Outside Nodes (p. 2404). Transparent areas show target nodes that do not get mapped because there are no source nodes within the **Maximum Distance**.

Figure 41: Imported Data using Maximum Distance for Outside Nodes



When **Weighting** is set to **Kriging**, this option allows you to ignore target points that lie outside the source bounding box. Defaults to **Off**. When this option is set to **On**, the **Bounding Box Tolerance** property enables you to include target points that lie outside the source bounding box by specifying a tolerance value. The algorithm adds this tolerance value to the source bounding box when it checks to see if a target point should be ignored or not.

- **Scale**: When weighting is set to **Shape Function**, the scaling factor (%) determines the number of buckets used to distribute the source elements. Defaults to 50% (2 buckets).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- **Edge Tolerance**: Dimensionless mapping tolerance (default = 0.05).

  – Shape Function for Surface/Edge topology.

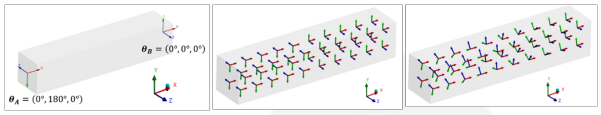
- **Correlation Function**: When weighting is set to **Kriging**, this property enables you to change the mathematical function that is used to model the spatial correlation between the sample points. Defaults to **Gaussian**.

- **Polynomial**: When weighting is set to **Kriging**, this property enables you to change the mathematical function that is used to globally approximate the sample. Defaults to **Adaptive**.

- **Extrapolation Tolerance**: You can use this option with adaptive **Kriging** to ensure that the interpolated value for each target point lies within specific limits. The tolerance is applied to the source range (based on the source points used for each target point) to determine if the interpolated value is satisfactory or if the data needs to be re-interpolated by reducing the polynomial order and the number of source points. For example, consider a target point having source values between 99 and 100. The default tolerance value of 10% will ensure that the mapped value is between 98.9 and 100.1. Target points whose values are outside the limits when the lowest polynomial type is used are not assigned a value.

- **Orientation Realignment**: This option is available when the **Weighting** property is set to **Quaternion**. When selected, the interpolation calculation treats orientations that have been flipped as equivalent. That is, given an element orientation identified by the axes (X,Y,Z), the algorithm treats as equivalent the orientations (X,Y,Z), (-X, -Y, Z), (X, -Y, -Z), and (-X, Y, -Z). See Figure 45.

  Figure 42: Interpolating Flipped Orientations

  Consider interpolating between the orientations A and B shown on the left. Note that A can be obtained by flipping the Z axis in the reference coordinate system. Mapped orientations obtained with and without **Orientation Realignment** are shown in the center and right, respectively.

Advanced Shell-Solid

Advanced shell-solid settings are filtered based on the **Mapping Control** and **Weighting** type selected in Mapping Settings (p. 2389). They are only available for **Shell-Solid** submodeling. In the case of imported cut boundary conditions, **Shape Function** is the only available **Weighting** type.

Pinball Factor: This value is used to calculate the **Pinball Value** for shell-solid submodeling. The **Pinball Value** is calculated by scaling the maximum shell thickness with the **Pinball Factor**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Figure 43: Shell-Solid Submodeling with Pinball Factor = 1.0

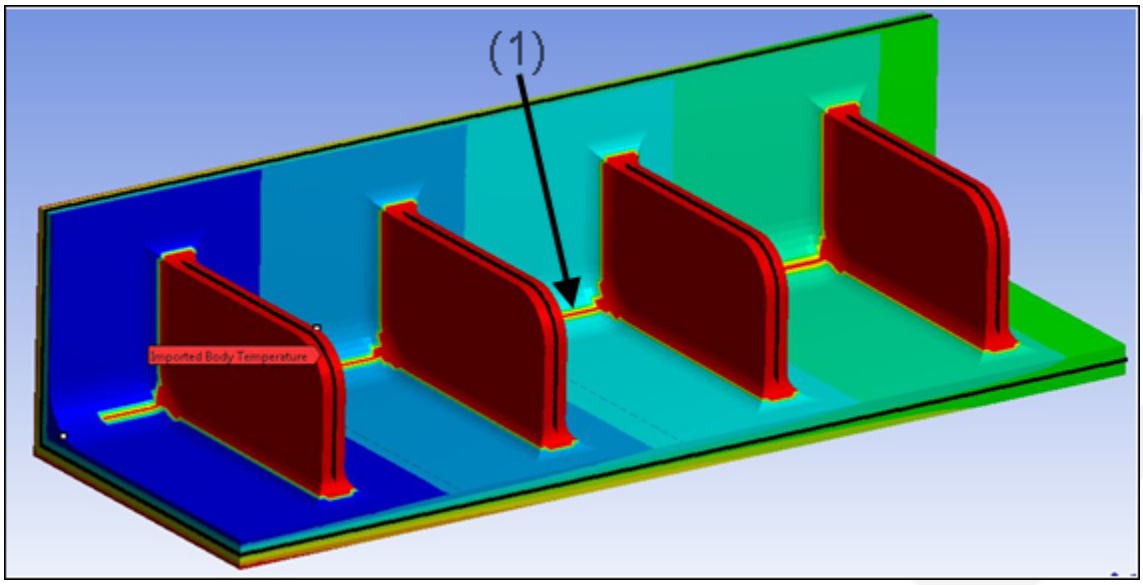
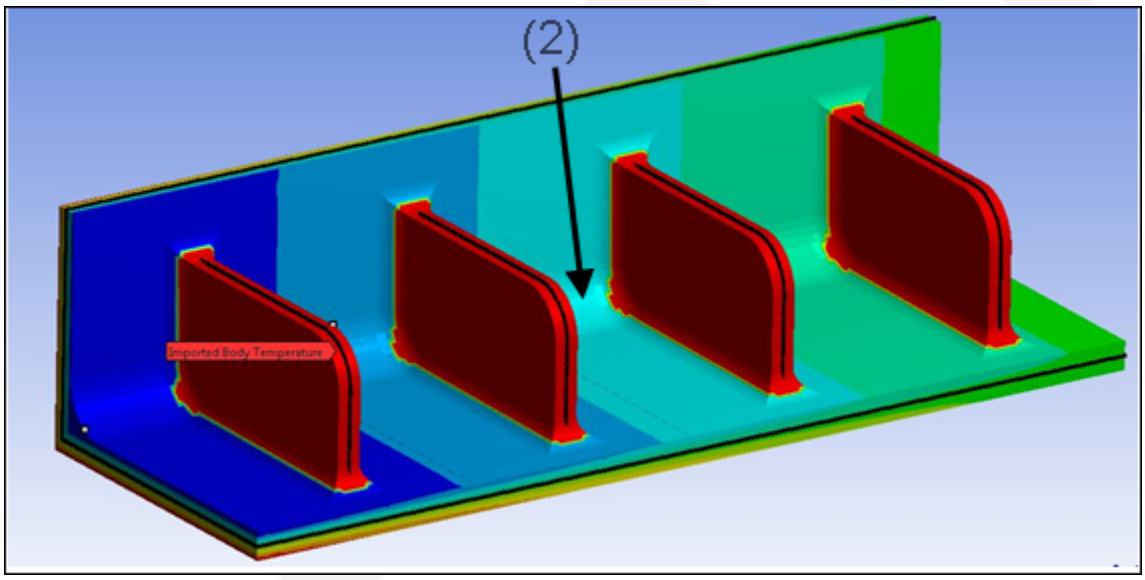


Figure 44: Shell-Solid Submodeling with Pinball Factor = 1.2



As shown in Figure 43: Shell-Solid Submodeling with Pinball Factor = 1.0 (p. 2406) and Figure 44: Shell-Solid Submodeling with Pinball Factor = 1.2 (p. 2406), the gap between the nodes in the filleted region is greater than the maximum shell thickness for the model. Hence using a **Pinball Factor** equal to 1 results in nodes in the fillet not finding appropriate matching elements.(1) When **Pinball Factor** of 1.2 is used, then additional buckets are included in the search resulting in better mapping results.(2)

Note:

Increasing the **Pinball Factor** increases the number of buckets searched to find the matching element hence, may decrease the efficiency of the mapping. An appropriate value should be chosen so that the resulting bounding region includes the matching element but not too big so as to negatively affect the efficiency of the search.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Shell Thickness Factor: For shell models with variable thickness, the gap between the target node, and matching element may be large. **Shell Thickness Factor** is used to exclude any matching element which has a gap greater than **Thickness* Shell Thickness Factor**. **Thickness** is the average element thickness of the matching element.

Figure 45: Shell-Solid Submodeling with Shell Thickness Factor = 0.6

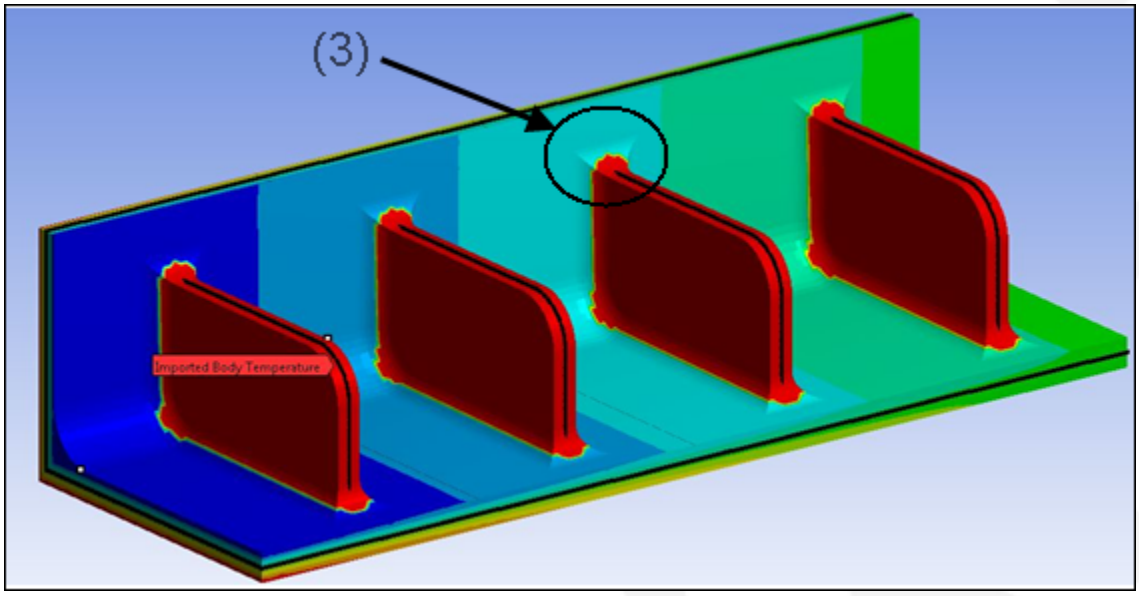
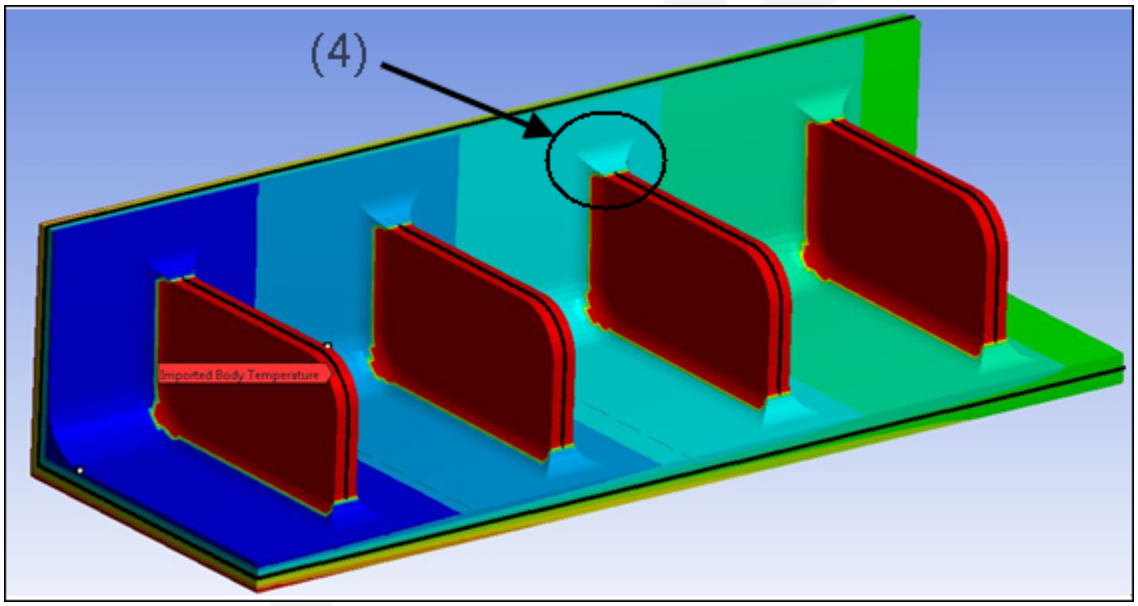


Figure 46: Shell-Solid Submodeling with Shell Thickness Factor = 1.2



Note:

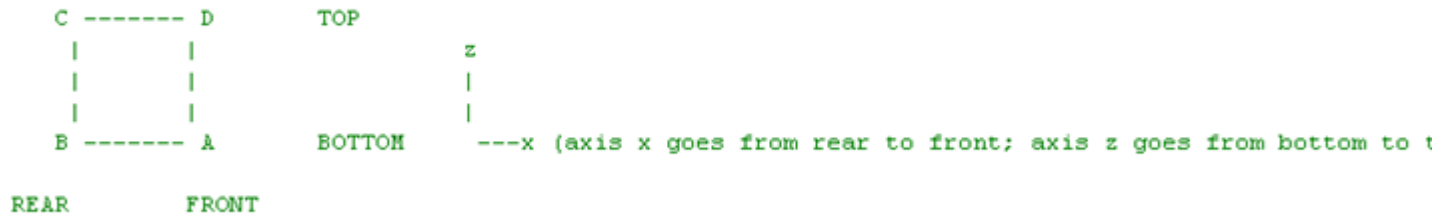
Increasing the **Shell Thickness Factor** to allow submodel nodes to be "found" can produce poor submodel results as shown in Figure 45: Shell-Solid Submodeling with Shell Thickness Factor = 0.6 (p. 2407) and Figure 46: Shell-Solid Submodeling with Shell Thickness Factor = 1.2 (p. 2407). where large Shell Thickness Factor causes the target nodes on the web region

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

to be matched with the base (3), whereas the target nodes are more appropriately matched for a smaller Shell Thickness Factor (4).

UV Source Controls/UV Target Controls

- **Alignment: Program Controlled** (default) or **Manual** control of selecting the four alignment points needed for UV Mapping. The process of UV mapping involves aligning both the source and target nodal data from XYZ coordinates into the equivalent UV space. To do this, the mapper needs to have access to four alignment locations as reference points for unfolding and flattening the nodal information. These four locations are referred to as "Front Bottom", "Rear Bottom", "Rear Top", and "Front Top". When the **Program Controlled** alignment option is selected, the associated coordinate systems ZX plane is used in relation to the associated mesh nodal locations.



  - **Coordinate System**: Available when **Alignment** is set to **Program Controlled**. One of the available coordinate systems must be selected as a reference point for **Program Controlled** alignment. The mesh nodal data is transformed related to the ZX plane of the selected coordinate system. A mean Z value is determined so that the nodes can be split into 2 groups, an upper and lower section. The nodes in each section are then sorted based on their X position. If there are nodes at the same X position, these points are then sorted based on their Z location. For the "Rear Bottom" and "Front Bottom" points, the minimum sorted Z point will be used, and for the "Rear Top" and "Front Top", the maximum Z point will be used.

  - **Nodes**: Available when **Alignment** is set to **Manual** for UV Source Controls. The user must list the 4 node locations in the text entry separated by commas. The order must be input as Front Bottom, Rear Bottom, Rear Top, Front Top.

  - **Target Front-Bottom**, **Target Rear-Bottom**, **Target Front-Top**, **Target Front-Top**: Available when **Alignment** is set to **Manual** for UV Target Controls. The user must select geometric vertices for each alignment point.

Program Controlled Mapping

Refer to the following table for the appropriate settings for when you set the **Mapping Control** property to **Program Controlled** in order to determine which type of mapping algorithm to use. The application determines default settings based on the properties described above.

When **Mapping** is **Profile Preserving**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Source mesh can provide:	Target mesh can provide:	Weighting that will be used:
Node IDs Only	Nodes	Uses **Direct Assignment** to assign values to target nodes.
Element IDs Only	Elements	Uses **Direct Assignment** to assign values to target elements.
Nodes Only	Nodes Only	Uses **Triangulation** to calculate mapping data.
Nodes and Elements	Nodes Only	Uses **Shape Function** to calculate mapping data.

When **Mapping** is **Conservative**

Source mesh can provide:	Target mesh can provide:	Weighting that will be used:
Nodes Only	Nodes and Elements	Uses **Shape Function** to calculate mapping data.

Manual Mapping

When manual mode is selected, you will be able to control advanced settings for the mapper. Based on the mapping chosen (conservative or profile preserving) and mesh data provided from the source and target, you will be able to choose the type of weighting algorithm.

For Profile Preserving Mapping:

If the source mesh contains only points, you will be able to select from the following:

- Direct Assignment (requires source Node or Element IDs)
- Triangulation
- Distance Based Average
- Kriging

If the source mesh also contains element data, you will have the items listed above as well as:

- Shape Function

For Conservative Mapping:

If the source mesh contains only points, you will be able to select from the following:

- Shape Function

Element shapes supported during mapping when **Shape Function** is selected:

Element Shape	Supported
3 Node Triangle	X (2D) (3D)
6 Node Triangle	X (2D) (3D)
4 Node Quadrilateral	X (2D) (3D)

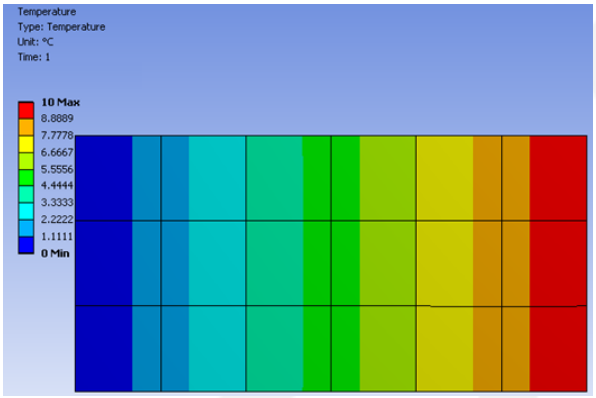
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Element Shape	Supported
8 Node Quadrilateral	X (2D) (3D)
4 Node Tetrahedron	X (3D)
10 Node Tetrahedron	X (3D)
8 Node Hexahedron	X (3D)
20 Node Hexahedron	X (3D)
6 Node Wedge	X (3D)
15 Node Wedge	X (3D)

2D to 3D Mapping

Mapping point data from 2D to 3D analyses is possible using the External Data system connected to a downstream Mechanical system. This mapping is performed by collapsing the 3D mesh data into a 2D plane and calculating target point weighting factors from the source point data.

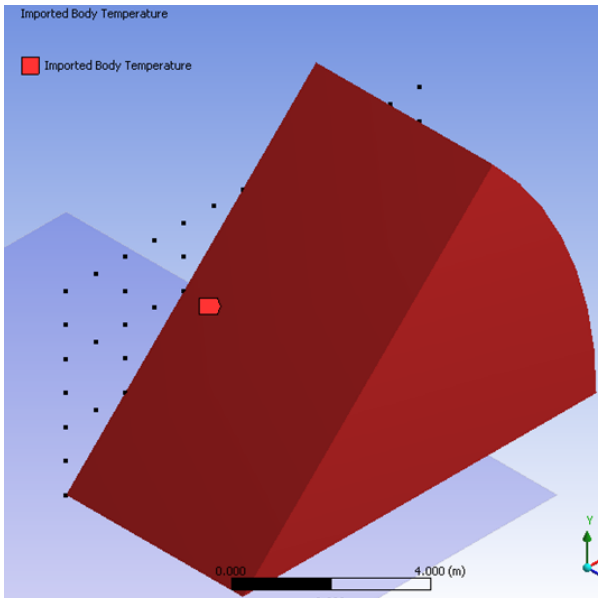
2D results in the XY Plane:



You will be able to select the 2D project plane to use based on the available coordinate systems as well as an option to select normal to the 2D source point data (**Normal To Plane**). Using the **Graphics Controls** described above, you will be able to turn on and off visualization of the source point data and the 2D projection plane.

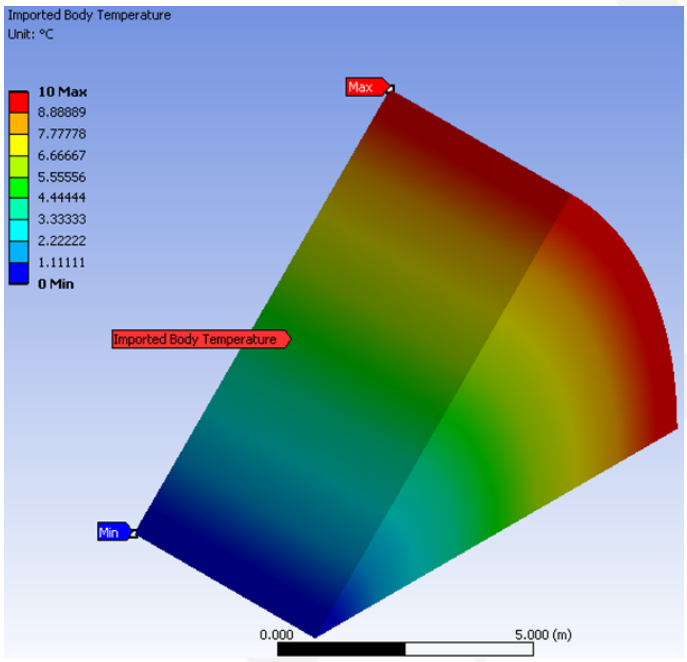
Source point and 2D projection plane displayed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



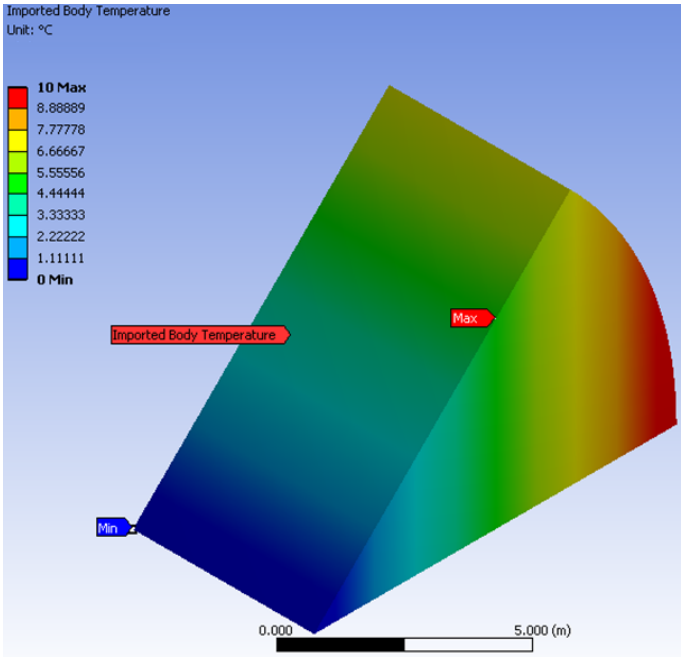
When selecting Cartesian coordinate systems, the projection will be done on the XY Plane. If the co-ordinate system is cylindrical, the projection will be rotated about the Z axis into the ZX Plane. **Normal To Plane** will project the target points into the source point plane.

3D mapped data using cylindrical coordinate system projection:



3D mapped data using Normal To Plane:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23



Note:

Conservative mapping is not available for 2D to 3D transfers.

Notes

When mapping point cloud data, the mapping utility does not know where body boundaries are. If you have a model with contact between two bodies, the mapping may pick up points from both bodies causing undesired results.

Mapping Validation

Mapping Validation objects can be inserted under imported data objects* to allow for an evaluation of how the mapping operation performed, by either right-clicking and selecting **Insert** > **Validation** from the context menu, or by clicking the **Validation** option. To perform a validation, right-click the **Validation** object and select **Analyze**. The following sections describe different methods to help analyze and determine if the mapping and interpolation that was performed produced an accurate representation of the mapped value data transferred from the source mesh onto the target mesh.

*Mapping Validation is supported on the following:

- External Data Import
- Submodeling (p. 807) (Not supported for Shell-Solid Submodeling)
- Thermal-Stress Analysis (p. 836)
- One-way Acoustic Coupling Analysis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Definition

The variable to display the validation information can be identified using the following properties:

1. **File Identifier***: A list of variables obtained from the parent object will be listed in the File Identifier drop-down. The validation information will be displayed based on the selected item.
2. **Layer**: This property is only available when validating Imported Trace from External Data system. Specify the layer from the PCB layout.
3. **Row**: The row of the parent worksheet.
4. **Data**: The data type for the imported load.
5. **Component**: The vector component (X, Y, Z).
6. **Complex Component**: The real/imaginary component for complex loads.
7. **Shell Face**: Top/Bottom for loads applied to shells.

Note:

- **File Identifier*** property is only available for data imported through the External Data system.
- The source mesh referenced by the File Identifier property must provide node locations in order for the application to be able to perform a validation.
- **Properties 2-6 are not available for** data imported through the External Data system. Instead the validation information is displayed for the variable identified using the **File Identifier** property.

Settings

Within the **Settings** category, the **Type** of validation must be specified by selecting **Reverse Validation**, **Distance Based Average Comparison**, or **Source Value**:

- **Reverse Validation**. **Reverse Validation** takes the results of the imported data (based on the File Identifier) and maps these values back onto the source points. These newly mapped values are compared to the source variables original values.
- **Distance Based Average Comparison**. **Distance Based Average Comparison** compares the results from the parent (based on the **File Identifier**) to mapped results obtained by using the distance-based average algorithm. Distance-based mapping will be done using the **Number of Points** specified. The output graphics will be displayed at the nodal locations of the target mesh.
- **Source Value**. **Source Value** displays the selected **File Identifier** data values. With the **Display In Parent** turned **On** and the parent of the validation tree node item selected, the interpolated values calculated on the target mesh can easily be compared to the original source point values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- **Undefined Points**. **Undefined Points** displays the nodes which do not have an associated value based on the selected **File Identifier**.

Note:

The **Reverse Validation** and **Distance Based Average Comparison** options are not available when validating **Imported Trace** and **Imported Heat Generation**.

The **Output Type** can be set to Absolute or Relative Difference (default). For **Relative Difference**, the percent error is calculated and any values that are above 0.01% will be displayed in the graphics window. For **Absolute Difference**, any non-zero difference will be displayed. The **Minimum** and **Maximum** values will be displayed in the **Statistics** category of the details view. Subsets of the full set for either relative or absolute differences can be shown by adjusting the **Display Minimum** and **Display Maximum** fields. These fields must be within the **Maximum** and **Minimum** range defined within the **Statistics** category.

Graphics Controls

There are multiple display options available: **Scaled Spheres**, **Colored Spheres**, **Colored Diamonds**, **Colored Points**, **Contours**, and **Isolines**. **Colored Spheres** and **Scaled Spheres** consume more memory and take longer to display on the screen due to the number of sides being drawn for each sphere. **Colored Diamonds** consume less memory and time, and **Colored Points** use the least amount. **Contours** and **Isolines** option will only be available when source mesh element connectivity is provided. Use External Data with an Mechanical APDL `.cdb` formatted file containing elements. All displays will be based on the range entered in the **Display Minimum/Display Maximum** fields. Display items that are colored will have a discrete legend displayed based on the **Display Minimum** and **Display Maximum**, divided equally into ranges. **Scaled Spheres**, **Colored Spheres**, and **Colored Diamonds** can be scaled based on the **Scale** field value.

If the **Display** option is set to **Isolines**, a **Line Thickness** option will be available to control how the isolines are drawn. This setting will be respected when drawing isolines on the parent object when **Display In Parent** is **On**.

If the **Display In Parent** property is set to **On**, the validation data will also be displayed when the parent object is selected. The validation data that is displayed in the parent object respects the **Active Row** and, if available, the **Data**/**Component** option selected in the details pane of the Imported Load object.

- If the **Component** property in the details pane of the Imported Load object is set to **All** or **Total**, the displayed data represents the vector magnitude of the validation results corresponding to the source identifiers defined in the worksheet of the active row.

- If the **Component** property is set to **X**, **Y** or **Z** component for vectors, the displayed data represents the validation results in the global **X**, **Y** or **Z** directions for the source identifiers defined in the worksheet of the active row.

- If the **Component** property is set to XX/YY/ZZ/XY/YZ/ZX component for tensors, the displayed data represents the validation results in the **global coordinate system** for the source identifiers defined in the worksheet of the active row.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- If the **Data** property is set to **Temperature** or **Convection Coefficient**, the displayed data represents the validation result for the corresponding source identifier selected in the worksheet of the active row.

Legend Divisions control how many contour colors to use and must be within the range from 3 to 14.

Statistics

The **Maximum** and **Minimum** read-only fields show the full range of available results from the validation. **Number Of Items** shows how many items are currently being displayed in the graphics window. This number is based on the **Display Minimum** and **Display Maximum** values.

Once a validation has been performed, the data can be exported to a file by simply right-clicking the **Validation** object and selecting **Export**.

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Appendix C. Workbench Mechanical Wizard Advanced Programming Topics

This appendix examines programming techniques and provides a reference for customizing the Mechanical Wizard.

Topics

Overview (p. 2417)
URI Address and Path Considerations (p. 2418)
Using Strings and Languages (p. 2419)
Guidelines for Editing XML Files (p. 2420)
About the TaskML Merge Process (p. 2420)
Using the Integrated Wizard Development Kit (WDK) (p. 2421)
Using IFRAME Elements (p. 2422)
TaskML Reference (p. 2423)
Standard Object Groups Reference (p. 2455)
Tutorials (p. 2458)
Wizard Development Kit (WDK) Groups (p. 2469)

Overview

From a programming perspective, the Mechanical Wizard system is best described as a "task browser." As a "web browser" used to view and navigate pages on the Internet, a task browser is used to view and navigate tasks in an engineering system. A web browser accesses HTML files and resources on a network; a task browser accesses `TaskML` files and resources on a network.

TaskML is an XML vocabulary that defines the rules and data necessary to display and process pages of tasks in the Mechanical application. Like HTML, TaskML allows for general scripting and for inserting arbitrary HTML content and user interface controls. Basic wizard customization using TaskML is similar to working with HTML and requires only a text editor.

The Mechanical Wizard runs as a web application (specifically, a dynamic HTML page) inside of a web browser control (Microsoft Internet Explorer). The web browser control is hosted by the Mechanical application. Consequently, the Mechanical Wizard system has full access to the capabilities of the web browser and the Mechanical application. Development of the Mechanical Wizard involves use of the HTML, CSS, XML, JScript web standards, and, for access to and automation of the application, use of the Mechanical application object model. The Mechanical Wizard displays tasks organized into groups. A task displays a caption and a status or descriptive icon. Activating a task (by clicking) typically involves automatic navigation to a particular context and selection in the user interface and display of a "callout"

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

with a text message pointing to a specific control. Custom tasks may perform any operation via TaskML elements or scripting.

The Mechanical Wizard responds to events that occur in the Mechanical application. Adding a load is an example of an event. When such an event occurs, each task is given the opportunity to determine its status or take an action.

The user may open a TaskML file inside the Mechanical Wizard from their local disk or from a network location. Therefore, saving TaskML to a network server makes custom wizard definitions available to any user with access to the server. Additionally, the Mechanical Wizard system itself may be run by any number of clients from a network location.

TaskML, along with HTML and scripting, offers an efficient and powerful means of extending the Mechanical application user interface.

URI Address and Path Considerations

The Merge (p. 2424), Script (p. 2425), task (p. 2431), set-icon (p. 2453), open-url (p. 2447), display-help-topic (p. 2443) and iframe (p. 2434) TaskML elements use URIs to link together files to form a complete wizard definition. TaskML supports the following URI formats.

Note:

Standard network security conditions apply to these URIs. As a general rule, if a user cannot open a linked file in their web browser, the file cannot be accessed by the Mechanical Wizard.

Local Machine and LAN

```
C:\folder\Wizard.xml

M:\folder\Wizard.xml

\\server\share\Wizard.xml
```

Standard Protocols

```
http://webserver/share/Wizard.xml

ftp://ftp.webserver.com/pub/Wizard.xml

file:///C:/folder/Wizard.xml
```

SIMWIZ Protocol

The SIMWIZ protocol supports paths relative to the location of the Mechanical Wizard (specifically, relative to the location of the file Default.htm in the Mechanical Wizard folder). The SIMWIZ protocol allows custom TaskML files published to any arbitrary location to reuse standard TaskML files and other components of the system.

```
simwiz://Tasks/StandardTasks.xml
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Relative Paths

All relative paths are relative to the location of the file containing the link. Note that this behavior is different from version 6.0, in which relative links were relative to the location of the Mechanical Wizard.

```
folder/Wizard.xml

./folder/Wizard.xml

../folder/Wizard.xml

/rootfolder/Wizard.xml
```

Using Strings and Languages

The Mechanical Wizard obtains all strings from TaskML. The language-related section of the TaskML uses the following structure:

```
<strings>
    <language xml:lang="language-code">
        <string id="String_ID">Sample Text</string>
    </language>
</strings>
```

The Mechanical Wizard determines which strings to use by matching the Language setting in the Wizard page of the Control Panel to the **xml:lang** attribute of a **language** element. If no **language** element with a matching **xml:lang** attribute exists, or if no **string** element with the necessary ID exists, the Mechanical Wizard takes the string from the **language** element with the **xml:lang** attribute set to "en-us" (English, United States). If the default English string doesn't exist, the Mechanical Wizard takes the first string with a matching ID or displays the string ID in place of the text.

Recommended Localization Process

This process describes how to localize all strings in a TaskML file:

1. Open the TaskML file in a text editor.
2. Copy the section of the file from:

   ```
   <language xml:lang="en-us">
   ```

   to

   ```
   </language>
   ```

3. Paste the copy into the<**string**> element below the last <**language**> close tag.
4. Change the language code from **en-us** to the code appropriate for the localization.
5. Localize each <**string**> element within the new <**language**> element. String IDs must remain unchanged.
6. Test the new language by entering the language code in the Language setting in the Wizard page of the Control Panel.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

English Customization Process

This process describes how to customize individual English strings with specific information or terminology:

1. Create a new <**language xml:lang="x-foo"**> element at the bottom of the <**string**> element below the last <**/language**> close tag. Set the **xml:lang** attribute to an arbitrary "x-code" descriptive of the customization (no spaces).
2. Copy individual <**string**> elements to customize from the < **language xml: lang="en-us"**> element to the new <**language xml: lang="x-foo"**> element. Strings omitted from the new <**language**> element will be obtained from the <**language xml: lang="en-us"**> element.
3. Customize the strings. String IDs must remain unchanged.
4. Test the customized strings by entering the x-code in the Language setting in the Wizard page of the Control Panel.

Guidelines for Editing XML Files

TaskML is an XML vocabulary. As such, TaskML consists of Unicode (wide character) text files that must follow the standard XML rules for well-formedness. When editing a TaskML file, use caution to ensure that the XML remains well-formed. For example, omitting a close tag will cause an error and may prevent the wizard from loading. To test for well-formedness, open the file in Internet Explorer 5 or later.

Note:

- XML is case-sensitive. All TaskML tags are lower-case.
- Attribute values must be in quotes.
- Use only the five predefined XML entity references for special characters if needed: & (**`&`**), < (**`<`**) > (**`>`**) " (**`"`**) ' (**`&apo;`**).
- White space (new lines, tabs, etc) is generally discarded. However, within a string element extra white space may result in multiple spaces between words. At this release there is no way to insert a line break within a string element.
- string elements contain only text; string (p. 2429) elements may not contain any XML or HTML elements.
- XML comments are allowed.

About the TaskML Merge Process

The merge process facilitates reuse of wizard components from local or network locations.

The merge process is the first step in loading TaskML into the Mechanical Wizard. The process involves selectively copying information from a merged TaskML document into a parent TaskML document. The parent document includes a Merge (p. 2424) element linking to the merged file. The merge process

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

generates a composite TaskML document in memory; neither the parent or merged TaskML files are modified.

The merge process consists of the following steps:

1. If the merged TaskML document contains Merge (p. 2424) elements, this process is called recursively. That is, a TaskML document may merge a file that merges a file, and so on.
2. Script (p. 2425) elements are copied to the parent only if the **src** attribute is unique.
3. object-group (p. 2426) elements are copied to the parent only if the merged **object-group** has a unique **name** attribute.
4. status (p. 2427) elements are copied to the parent only if the merged **status** has a unique **id**.
5. language (p. 2429) collections (and contained string elements) are copied only if the **language** has a unique **xml:lang** attribute.
6. string (p. 2429) elements are copied only if the merged **string** has a unique **id**.
7. task (p. 2431) elements are copied only if the merged **task** has a unique **id**.
8. If both the parent and the merged TaskML documents contain a group (p. 2433) with the same **id**:
   - Attributes defined for the merged **group** but omitted in the parent **group** are copied to the parent **group**.
   - All children of the merged **group** are appended to the parent **group**.

For diagnostic purposes the merge process automatically adds a **merged-from** attribute to elements added to the parent TaskML file. The **merged-from** attribute contains the url of the TaskML file from which the element was obtained.

Using the Integrated Wizard Development Kit (WDK)

The Mechanical Wizard system includes an integrated toolkit to assist in customizing wizards.

The following topics describe the tools:

- WDK: Tools Group (p. 2469)
- WDK: Commands Group (p. 2470)
- WDK Tests: Actions (p. 2470)
- WDK Tests: Flags (Conditions) (p. 2471)

To enable the toolkit:

- In the Mechanical application, select **Tools>Options**.
- Select **Wizard** and set **Enable WDK Tools** to **yes**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Enabling the WDK toolkit adds four groups to the bottom of every panel displayed in the Mechanical Wizard. The WDK toolkit does not change the behavior of other groups in the panel.

Using IFRAME Elements

An IFRAME (inline frame) functions as an HTML document within a Mechanical Wizard group. An IFRAME may contain any content, from static text to detailed user interface controls. IFRAMEs have full script access to the Mechanical Wizard, and therefore full access to the Mechanical application.

The **Options** group in the **Insert Geometry** panel demonstrates a simple user interface extension using an IFRAME. Other examples of IFRAME usage in the Mechanical application include the WDK: Tools (p. 2469) group and "Tip of the Day."

IFRAMEs in the Mechanical Wizard provide a way to customize the Mechanical application without modifying the main user interface. IFRAMEs may be published on a network, enabling customized user interfaces for multiple users without requiring changes to each installation.

Working with IFRAMEs requires familiarity with HTML and JScript coding. See also Tutorial: Adding a Web Search IFRAME (p. 2463).

Security Restrictions

Due to the cross-frame scripting security model enforced by the web browser control, custom IFRAME HTML pages should reside in the same location as the Mechanical Wizard. IFRAME pages from a different domain as the parent page cannot access the parent via script.

IFRAME Toolkit

The WDK includes the following resources for developing IFRAMEs:

- The file `MechanicalWizard\WDK\Info_IFRAME.htm` contains a template HTML document for an IFRAME. View the source for descriptions of recommended HTML elements and JScript functions.
- The file `MechanicalWizard\System\IFrame.js` implements generic functions for use in IFRAMEs.

The following files demonstrate use of IFRAMEs:

- `MechanicalWizard\WDK\Tools_IFrame.htm` contains implementation for the WDK: Tools (p. 2469) IFRAME. See `MechanicalWizard\WDK\Tools_Merge.xml` for corresponding TaskML.
- `MechanicalWizard\Panels\InsertGeometry_IFrame.htm` contains implementation for the Insert Geometry panel Options group. See `MechanicalWizard\Panels\InsertGeometry.xml` for corresponding TaskML.
- `MechanicalWizard\TipoftheDay\IFrame.htm` contains implementation for Tip of the Day. See `MechanicalWizard\Panels\Startup.xml` for corresponding TaskML.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

TaskML Reference

This reference describes each element defined in TaskML. See XML Notes (p. 2420) for general usage guidelines.

The Overview Map (p. 2423) contains a diagram showing the basic structure of TaskML.

- Document Element (p. 2424)
- External References (p. 2424)
- Object Grouping (p. 2425)
- Status Definitions (p. 2427)
- Language and Text (p. 2428)
- Tasks and Events (p. 2430)
- Wizard Content (p. 2432)
- Rules (p. 2435)
- Scripting (p. 2454)

Overview Map of TaskML

The following illustrates the basic hierarchical structure of TaskML.

- simulation-wizard (p. 2424) document element
  - Merge (p. 2424) elements
  - Script (p. 2425) elements
  - object-groups (p. 2426) collection
  - statuses (p. 2428) collection
  - strings (p. 2430) collection
  - tasks (p. 2432) collection
    - → task (p. 2431) elements
      - update-event (p. 2432) element
        - Rules (p. 2435) sequence
      - activate-event (p. 2430) element
        - Rules (p. 2435) sequence
  - body (p. 2432) element

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

→ group (p. 2433) elements

- taskref (p. 2434) elements
- iframe (p. 2434) elements
- eval (p. 2454) statements

→ eval (p. 2454) statements

Document Element

- simulation-wizard (p. 2424)

simulation-wizard

Identifies the start of a TaskML file.

```
<simulation-wizard
    version="1.0">
```

Attributes

version

Specifies the version of the TaskML vocabulary. The current version is "1.0."

Element Information

Parents	None. This is the document element (root) of the XML structure.
Children	Merge (p. 2424), Script (p. 2425), object-groups (p. 2426), statuses (p. 2428), strings (p. 2430), tasks (p. 2432), body (p. 2432)
End Tag	Required

External References

- Merge (p. 2424)
- Script (p. 2425)

Merge

Merges an external TaskML file.

```
<merge
    src="url" />
```

Attributes

src

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifies the URL of the TaskML file to merge.

Table 95: Element Information

Parents	simulation-wizard (p. 2424)
Children	None
End Tag	No - close element with "/>"

See Also

About the TaskML Merge Process (p. 2420) and URI Address and Path Considerations (p. 2418).

Script

Specifies an external JScript file to load into the Mechanical Wizard.

```
<merge
    src="url" />
```

Attributes

src

Specifies the URL of the JScript file to load.

Remarks

- JScript files use the `.js` file extension.
- Code in the JScript file outside of any function is evaluated immediately upon loading.
- The eval element may directly call functions defined in the JScript file.

Table 96: Element Information

Parents	simulation-wizard (p. 2424)
Children	None
End Tag	No - close element with "/>"

See Also

URI Address and Path Considerations (p. 2418).

Object Grouping

- object-group (p. 2426)
- object-groups (p. 2426)
- object-type (p. 2426)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

object-group

Organizes objects by placing them in an assigned group.

```
<object-group
    name="group_name">
```

Attributes

name

Specifies the name of the group.

Element Information

Parents	object-groups (p. 2426)
Children	object-type (p. 2426)
End Tag	Required

See Also

object (p. 2440), select-first-object (p. 2449), select-all-objects (p. 2447), Standard Object Groups Reference (p. 2455).

object-groups

Contains an unordered collection of object group definitions.

```
<object-groups>
```

Element Information

Parents	simulation-wizard (p. 2424)
Children	object-group (p. 2426)
End Tag	Required

See Also

Standard Object Groups Reference (p. 2455).

object-type

Specifies an Outline object by its internal identifiers.

```
<object-type
    class="id_Constant"
    type="id_Constant" />
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Attributes

class

Identifies the class ID constant.

type

Identifies the type ID constant. Applies only for a class of "id_Load" or "id_Result."

Remarks

ID constants are defined in the script file **DSConstants.js**.

The class attribute corresponds to the "Class" property of the Mechanical application objects.

The type attribute corresponds the "loadType" or "ResultType" property of specific the Mechanical application objects.

Element Information

Parents	object-group (p. 2426)
Children	None
End Tag	No - close element with "/>"

See Also

Standard Object Groups Reference (p. 2455).

Status Definitions

- status (p. 2427)
- statuses (p. 2428)

status

Defines a task status.

```
<status
    id="statusID"
    css-class="status-class"
    tooltip="statusID_Tooltip" />
```

Attributes

id

Unique identifier for the status.

css-class

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifies the class in the skin (cascading style sheet) to apply to the task. The style class defines the visual appearance of task status.

tooltip

Optional. Specifies the string ID of text to display in a tooltip when the cursor hovers over the task. Defaults to "statusID_*Tooltip*."

Element Information

Parents	statuses (p. 2428)
Children	None
End Tag	No - close element with "/>"

See Also

set-status (p. 2453).

statuses

Contains an unordered collection of status definitions.

```
<statuses>
```

Element Information

Parents	simulation-wizard (p. 2424)
Children	status (p. 2427)
End Tag	Required

See Also

set-status (p. 2453).

Language and Text

- data (p. 2428)
- language (p. 2429)
- string (p. 2429)
- strings (p. 2430)

data

Data placeholder within a string.

```
<string id="stringID">string text<data />string text</string>
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Remarks

Used only with the Lookup method on a Strings object as defined in **StringLookupObject.js**. Allows JScript functions to retrieve a localized string containing arbitrary data.

Element Information

Parents	string (p. 2429)
Children	None
End Tag	No - close element with "/>"

language

Contains an unordered collection of strings in a specified language.

```
<language
    [xml:lang="en us"]>
```

Attributes

xml:lang

Specifies the language code. Defaults to "en-us" (English, United States).

Remarks

The language code corresponds to the Language setting in the Wizard page of the Control Panel.

Element Information

Parents	strings (p. 2430)
Children	string (p. 2429)
End Tag	Required

string

Specifies the text for a given string ID.

```
<string
    id="stringID">string text</string>
```

Attributes

id

Unique identifier assigned to the string.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Element Information

Parents	language (p. 2429)
Children	data (p. 2428)
End Tag	Required

strings

Contains an unordered collection of languages.

```
<strings>
```

Element Information

Parents	simulation-wizard (p. 2424)
Children	language (p. 2429)
End Tag	Required

Tasks and Events

- activate-event (p. 2430)
- task (p. 2431)
- tasks (p. 2432)
- update-event (p. 2432)

activate-event

Contains a sequence of rules to process when the user clicks on a task.

```
<activate-event
    tab="{design | print | report | help | any}">
```

Attributes

tab

Optional. Selects a specific tab before processing the activate event rules.

design	Selects the Design View tab. Default behavior if attribute omitted.
print	Selects the Print Preview tab.
report	Selects the Report Preview tab.
help	Selects the Quick Help tab.
any	Does not change tab selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Information

Parents	task (p. 2431)
Children	if (p. 2436), set-icon (p. 2453), set-caption (p. 2452), set-status (p. 2453), select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), click-button (p. 2442), display-task-callout (p. 2446), display-outline-callout (p. 2444), display-details-callout (p. 2443), display-toolbar-callout (p. 2446), display-tab-callout (p. 2445), display-status-callout (p. 2445), open-url (p. 2447), display-help-topic (p. 2443), send-mail (p. 2452), eval (p. 2454), update (p. 2438), debug (p. 2436)
End Tag	Required

task

Defines a task.

```
<task
id="uniqueID"
caption="uniqueID_Caption"
tooltip="uniqueID_Tooltip"
disable-if-missing="group_name"
hide-if-missing="group_name"
check-ambiguity="{model | environment | solution}"
icon="url"
deemphasize="{yes | no}">
```

Table 97: Attributes

Attribute	Description
id	Arbitrary unique identifier assigned to the task.
caption	Optional. Specifies the string ID of the text to display in the task caption. Defaults to "uniqueID_Caption" if not specified.
tooltip	Optional. Specifies the string ID of the text to display in the task tooltip. Defaults to "uniqueID_Toolip" if not specified.
disable-if-missing	Optional. Disables the task if an object matching the group name does not exist.
hide-if-missing	Optional. Hides the task if an object matching the group name does not exist.
check-ambiguity	Optional. Automatically tests for ambiguity of an outline level prior to processing event rules.
icon	Optional. Specifies the URI of an image to use as the task icon. See URI Address and Path Considerations.
deemphasize	Optional. Causes a task inside an emphasized group to render with a deemphasized style.

Table 98: Element Information

Parents	tasks (p. 2432)
Children	update-event (p. 2432), activate-event (p. 2430)
End Tag	Required

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Also See: taskref

tasks

Contains an unordered collection of task definitions.

```
<tasks>
```

Element Information

Parents	simulation-wizard (p. 2424)
Children	task (p. 2431)
End Tag	Required

update-event

Contains a sequence of rules to process when the user navigates or modifies information in the Mechanical application.

```
<update-event>
```

Element Information

Parents	task (p. 2431)
Children	if (p. 2436), set-icon (p. 2453), set-caption (p. 2452), set-status (p. 2453), select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), click-button (p. 2442), display-task-callout (p. 2446), display-outline-callout (p. 2444), display-details-callout (p. 2443), display-toolbar-callout (p. 2446), display-tab-callout (p. 2445), display-status-callout (p. 2445), open-url (p. 2447), display-help-topic (p. 2443), send-mail (p. 2452), eval (p. 2454), debug (p. 2436)
End Tag	Required

Wizard Content

- body (p. 2432)
- group (p. 2433)
- iframe (p. 2434)
- taskref (p. 2434)

body

Specifies content to display inside the Mechanical Wizard.

```
<body title="stringID">
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Attribute

title

Optional. Specifies the string ID of text to display in the title of the panel containing the Mechanical Wizard. Defaults to the text "Mechanical Wizard."

Element Information

Parents	simulation-wizard (p. 2424)
Children	group (p. 2433), eval (p. 2454)
End Tag	Required

group

Defines a collapsible group of tasks or iframes.

```
<group
    id="uniqueID"
    caption="uniqueID_Caption"
    description="uniqueID_Description"
    emphasize="{yes | no}"
    collapsed="{yes | no}"
    onupdate="foo()">
```

Attributes

id

Arbitrary unique identifier assigned to the group.

caption

Optional. Specifies the string ID of the text to display in the group caption. Defaults to "*uniqueID*_Caption" if not specified.

description

Optional. Specifies the string ID for a brief paragraph to display at the top of the group. Defaults to "*uniqueID*_Description" if not specified. If the string ID is undefined the group contains no description.

emphasize

Optional. Emphasizes the group via different visual styles. Defaults to "no."

collapsed

Optional. Initially displays the group collapsed. After first use the collapsed status of each group is persisted. Defaults to "no."

onupdate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional. JScript expression to evaluate on the Update event prior to processing the update-event (p. 2432) rules for tasks the group contains.

Element Information

Parents	body (p. 2432)
Children	taskref (p. 2434), iframe (p. 2434), eval (p. 2454)
End Tag	Required

iframe

Inserts an HTML IFRAME element within a group. The IFRAME may contain any arbitrary web page and may communicate with the Mechanical Wizard via script.

```
<iframe
src="uri" />
```

Attributes

src

Specifies the URI of the web page to load into the IFRAME. See the topic on IFRAME Elements for notes on security restrictions.

Table 99: Element Information

Parents	group
Children	None
End Tag	No - close element with "/>"

See Also

Using IFRAME Elements (p. 2422).

taskref

Inserts a task into a group.

```
<taskref
    task="uniqueID" />
```

Attributes

task

Specifies the ID of a task defined elsewhere in the merged TaskML file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Information

Parents	group (p. 2433)
Children	None
End Tag	No - close element with "/>"

See Also

task (p. 2431).

Rules

- Statements (p. 2435)
- Conditions (p. 2438)
- Actions (p. 2442)

Statements

- and (p. 2435)
- debug (p. 2436)
- if then else stop (p. 2436)
- not (p. 2437)
- or (p. 2437)
- update (p. 2438)

and

Performs a logical conjunction on two conditions. Equivalent to the JScript **&&** operator.

```
condition1 <and> condition2 </and>
```

Element Information

Parents	if (p. 2436)
Children	Conditions: level (p. 2439), object (p. 2440), changeable-length-unit (p. 2439), assembly-geometry (p. 2438), geometry-includes-sheets (p. 2439), zero-thickness-sheet (p. 2441) Actions: select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), eval (p. 2454)
End Tag	Required

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

debug

Attempts to launch a script debugger to debug the JScript code corresponding to the rules in the current event. Equivalent to the JScript **debugger** keyword.

```
<debug />
```

Element Information

Parents	update-event (p. 2432), activate-event (p. 2430), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"

if then else stop

Conditionally processes a sequence of rules, depending on the value of a condition.

```
<if> condition
    <then>
        rules
        <stop/>
    </then>
    <else>
        rules
        <stop/>
    </else>
</if>
```

Remarks

eval (p. 2454) statement. The not (p. 2437) operator negates the value of a condition. The and (p. 2435) and or (p. 2437) operators perform logical operations on two conditions within an **if** statement.

The **then** statement contains a sequence of rules to process when the resolved value of the condition is **true**. An **if** statement must contain one **then** statement.

The **else** statement contains a sequence of rules to process when the resolved value of the condition is **false**. The **else** statement is optional. If used it must follow the close of the **then** statement.

The **if...then...else** structure is equivalent to the **if...else** statement in JScript:

```
if( condition )
    { statements }
    else { statements }
```

The **stop** statement ends processing of an event at a specific point. If a **stop** statement is not included within a **then** or **else** statement, rules following the **if** statement are processed. The **stop** statement is equivalent to the JScript **return** statement.

Element Information for <if>

Parents	update-event (p. 2432) and activate-event (p. 2430)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Children	Operators: and (p. 2435), or (p. 2437), not (p. 2437) Conditions: level (p. 2439), object (p. 2440), changeable-length-unit (p. 2439), assembly-geometry (p. 2438), geometry-includes-sheets (p. 2439), zero-thickness-sheet (p. 2441) Actions: select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), eval (p. 2454)

Element Information for <then> and <else>

Parents	if (p. 2436)
Children	set-icon (p. 2453), set-caption (p. 2452), status (p. 2427), select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), click-button (p. 2442), display-task-callout (p. 2446), display-outline-callout (p. 2444), display-details-callout (p. 2443), display-toolbar-callout (p. 2446), display-tab-callout (p. 2445), display-status-callout (p. 2445), open-url (p. 2447), display-help-topic (p. 2443), send-mail (p. 2452), eval (p. 2454), update (p. 2438), debug (p. 2436)
End Tag	Required

not

Performs logical negation on a condition. Equivalent to the JScript **!** operator.

```
<not> condition </not>
```

Element Information

Parents	if (p. 2436)
Children	Conditions: level (p. 2439), object (p. 2440), changeable-length-unit (p. 2439), assembly-geometry (p. 2438), geometry-includes-sheets (p. 2439), zero-thickness-sheet (p. 2441) Actions: select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448), select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), eval (p. 2454)
End Tag	Required

or

Performs a logical disjunction on two conditions. Equivalent to the JScript **||** operator.

```
condition1 <or> condition2 </or>
```

Element Information

Parents	if (p. 2436)
Children	Conditions: level (p. 2439), object (p. 2440), changeable-length-unit (p. 2439), assembly-geometry (p. 2438), geometry-includes-sheets (p. 2439), zero-thickness-sheet (p. 2441) Actions: select-first-object (p. 2449), select-all-objects (p. 2447), select-field (p. 2448),

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	select-first-undefined-field (p. 2451), select-first-parameter-field (p. 2450), select-zero-thickness-sheets (p. 2451), eval (p. 2454)
End Tag	Required

update

Forces an Update event to fire. In general, this statement is necessary only if preceding rules in the event cause the status of other tasks to become out of sync.

```
<update />
```

Element Information

Parents	activate-event (p. 2430), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"

Conditions

- assembly-geometry (p. 2438)
- changeable-length-unit (p. 2439)
- geometry-includes-sheets (p. 2439)
- level (p. 2439)
- object (p. 2440)
- zero-thickness-sheet (p. 2441)

assembly-geometry

Tests if the geometry in context of the current selection contains an assembly or a single part.

```
<assembly-geometry />
```

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	True if the geometry contains an assembly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

changeable-length-unit

Tests if the geometry in context of the current selection does not explicitly specify a length unit (e.g. for ACIS geometry types). Useful in prompting the user to verify a correct length unit setting.

```
<changeable-length-unit />
```

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	True if the length unit is not read-only.

geometry-includes-sheets

Tests if the geometry in context of the current selection contains sheet parts.

```
<geometry-includes-sheets />
```

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	True if the geometry contains one or more sheets.

level

Tests the level of the current selection in the Outline.

```
<level
    type="{project | model | environment | solution}"
    condition="{is-ambiguous | is-not-ambiguous | is-selected | is-not-selected}" />
```

Attributes

type

Identifies the level. A level consists of a container (for example, the Environment) and all children excluding other containers.

condition

Specifies a condition to test.

is-ambiguous	Returns true if a specific container cannot be resolved given the current Outline selection.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is-not-ambiguous	Returns true if a specific container is identified given the current Outline selection.
is-selected	Returns true if any object at the given level is currently selected.
is-not-selected	Returns true if no object at the given level is currently selected.

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	As defined by the condition attribute.

object

Tests the **Outline** tree for an object matching the given criteria. Searches only non-ambiguous objects given the current selection.

```
<object
    type="group_name"
    state="{any | stateless | fully-defined | under-defined | suppressed |
       not-updated | updated | obsolete | error | bad-license}"
    name-regexp="regular_expression"
    condition="{exists | does-not-exist | is-selected | is-not-selected}" />
```

Note:

It was necessary to "word wrap" the long line of code in the above example.

Attributes

type

Optional. Identifies an object group name or an object type constant as a search criteria. If omitted, the object type is not considered.

Object groups are defined by using the object-group (p. 2426) element. Refer to the Standard Object Groups Reference (p. 2455).

Type constants for specific objects (prefixed by "id_") are defined in the script file **DSConstants.js**.

state

Optional. Specifies an object state as a search criteria. If omitted, the default of "any" is used, meaning that object state is not considered.

States are defined in the script file **DSConstants.js**.

name-regexp

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Optional. Specifies a regular expression of an object's name as a search criteria. For example, "part" matches any object that includes "part" in its name (e.g. "part 2"). If omitted, object names are not considered. See the Microsoft regular expressions reference .

condition

Specifies a condition to test.

exists	Returns true if an object matching the criteria exists.
does-not-exist	Returns true if no object matches the criteria.
is-selected	Returns true if an object matching the criteria is currently selected.
is-not-selected	Returns true if no object matching the criteria is currently selected.

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	True if an object matching the criteria meets the condition.

zero-thickness-sheet

Tests if the geometry in context of the current selection contains any sheet with zero thickness specified. Useful in prompting the user to enter valid information for sheet thickness.

```
<zero-thickness-sheet />
```

Element Information

Parents	if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437)
Children	None
End Tag	No - close element with "/>"
Return Value	True if any sheet has a zero thickness value.

valid-emag-geometry

Tests if the geometry in context of the current selection meets the requirements for performing an electromagnetic simulation.

```
<valid-emag-geometry />
```

enclosure-exists

Tests if the geometry in context of the current selection contains an enclosure body for electromagnetic simulation.

```
<enclosure-exists />
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Actions

- click-button (p. 2442)
- display-details-callout (p. 2443)
- display-help-topic (p. 2443)
- display-outline-callout (p. 2444)
- display-status-callout (p. 2445)
- display-tab-callout (p. 2445)
- display-task-callout (p. 2446)
- display-toolbar-callout (p. 2446)
- open-url (p. 2447)
- select-all-objects (p. 2447)
- select-field (p. 2448)
- select-first-object (p. 2449)
- select-first-parameter-field (p. 2450)
- select-first-undefined-field (p. 2451)
- select-zero-thickness-sheets (p. 2451)
- send-mail (p. 2452)
- set-caption (p. 2452)
- set-icon (p. 2453)
- set-status (p. 2453)

click-button

Simulates a toolbar button click.

```
<click-button
    toolbar="key"
    button="key" />
```

Attributes

Use the WDK command View Current Toolbar Keys (p. 2469) to determine values for the attributes below.

toolbar

Specifies the key for the toolbar.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

button

Specifies the key for the button.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

See Also

display-toolbar-callout (p. 2446).

display-details-callout

Displays a callout pointing to the currently selected Details field.

```
<display-details-callout
    message="stringID" />
```

Attributes

message

Specifies the string ID (p. 2429) of the text to display in the callout.

Remarks

Before using this action:

Use select-first-object (p. 2449) or select-all-objects (p. 2447) to select one or more Outline objects prior to accessing the Details control.

Use select-field (p. 2448) to select a Details field.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

display-help-topic

Displays a topic from a Windows HTML Help file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<display-help-topic
    href="uri"
    topic="path" />
```

Attributes

href

Optional. Defines the URI of the CHM file. Defaults to `simw-iz://../HHelp/DesignSpace.chm`, the location of the Ansys Workbench Help system relative to the standard Mechanical Wizard location. **NOTE:** The default value no longer exists - Specify a valid help system path. See URI Address and Path Considerations (p. 2418).

topic

Optional. Specifies an internal path to a desired topic.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	None

display-outline-callout

Displays a callout pointing to the currently selected Outline object.

```
<display-outline-callout
    message="stringID" />
```

Attributes

message

Specifies the string ID (p. 2429) of the text to display in the callout.

Remarks

Use select-first-object (p. 2449) or select-all-objects (p. 2447) to select one or more Outline objects prior to displaying the callout.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Return Value	True if successful.

display-status-callout

Displays a callout pointing to a status bar panel.

```
<display-status-callout
    panel="index"
    message="stringID" />
```

Attributes

panel

Specifies the index of the status bar panel. The index of the leftmost panel is 1.

message

Specifies the string ID (p. 2429) of the text to display in the callout.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

display-tab-callout

Displays a callout pointing to a tab.

```
<display-tab-callout
    tab="{design | print | report | help}"
    message="stringID" />
```

Attributes

tab

One of the following keywords:

design	Design View tab.
print	Print Preview tab.
report	Report Preview tab.
help	Quick Help tab.

message

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifies the string ID (p. 2429) of the text to display in the callout.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

display-task-callout

Displays a callout pointing to the task itself.

```
<display-task-callout
    message="stringID" />
```

Attributes

message

Specifies the string ID (p. 2429) of the text to display in the callout.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

display-toolbar-callout

Displays a callout pointing to a toolbar button.

```
<display-toolbar-callout
    toolbar="key"
    button="key"
    message="stringID" />
```

Attributes

Use the WDK command View Current Toolbar Keys (p. 2469) to determine values for the **toolbar** and **button** attributes below.

toolbar

Specifies the key for the toolbar.

button

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Specifies the key for the button.

message

Specifies the string ID (p. 2429) of the text to display in the callout.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

See Also

click-button (p. 2442).

open-url

Opens a new web browser window and navigates to a given URI (URL).

```
<open-url
    href="uri" />
```

Attributes

href

Any valid URI. See URI Address and Path Considerations (p. 2418).

Element Information

Parents	activate-event (p. 2430), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	None

select-all-objects

Selects a set of objects based on given criteria. Searches only non-ambiguous objects given the initial selection.

```
<select-all-objects
    type="group_name"
    state="{any | stateless | fully-defined | under-defined | suppressed |
```

```
       not-updated | updated | obsolete | error | bad-license}"
    name-regexp="regular_expression" />
```

Note:

It was necessary to "word wrap" the long line of code in the above example.

Attributes

type

Optional. Identifies an object group name or an object type constant as a search criteria. If omitted, the object type is not considered.

Object groups are defined by using the object-group (p. 2426) element. Refer to the Standard Object Groups Reference (p. 2455).

Type constants for specific objects (prefixed by "id_") are defined in the script file `DSConstants.js`.

state

Optional. Specifies an object state as a search criteria. If omitted, the default of "any" is used, meaning that object state is not considered.

States are defined in the script file `DSConstants.js`.

name-regexp

Optional. Specifies a regular expression of an object's name as a search criteria. For example, "part" matches any object that includes "part" in its name (e.g. "part 2"). If omitted, object names are not considered. See the Microsoft regular expressions reference .

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one or more objects meeting the criteria were selected.

See Also

select-first-object (p. 2449).

select-field

Selects a field in the Details control by name.

```
<select-field
    name="stringID" />
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Attributes

name

Specifies the string ID for name of the field. Use the Details Field String ID (p. 2469) section in the WDK Tools group to determine the string ID of a field.

Remarks

Use select-first-object (p. 2449) or select-all-objects (p. 2447) to select one or more Outline objects prior to accessing the Details control.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one Details meeting the criteria was selected.

See Also

select-first-parameter-field (p. 2450), select-first-undefined-field (p. 2451).

select-first-object

Selects the first object matching given criteria. Searches only non-ambiguous objects given the initial selection.

```
<select-first-object
    type="group_name"
    state="{any | stateless | fully-defined | under-defined | suppressed |
       not-updated | updated | obsolete | error | bad-license
}"
    name-regexp="regular_expression" />
```

Note:

It was necessary to "word wrap" the long line of code in the above example.

Attributes

type

Optional. Identifies an object group name or an object type constant as a search criterion. If omitted, the object type is not considered.

Object groups are defined by using the object-group (p. 2426) element. Refer to the Standard Object Groups Reference (p. 2455).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Type constants for specific objects (prefixed by "id_") are defined in the script file `DSConstants.js`.

state

Optional. Specifies an object state as a search criteria. If omitted, the default of "any" is used, meaning that object state is not considered.

States are defined in the script file `DSConstants.js`.

name-regexp

Optional. Specifies a regular expression of an object's name as a search criterion. For example, "part" matches any object that includes "part" in its name (for example, "part 2"). If omitted, object names are not considered. See the Microsoft regular expressions reference .

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one object meeting the criteria was selected.

See Also

select-all-objects (p. 2447).

select-first-parameter-field

Selects the first parameter field in the Details control.

```
<select-first-parameter-field />
```

Remarks

Parameter fields contain a check box to the left of the name. If checked, the parameter field is exposed for use in the Parameter Workspace.

Use select-first-object (p. 2449) or select-all-objects (p. 2447) to select one or more Outline objects prior to accessing the Details control.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one Details meeting the criteria was selected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

See Also

select-field (p. 2448), select-first-undefined-field (p. 2451).

select-first-undefined-field

Selects the first undefined Details field.

```
<select-first-undefined-field />
```

Remarks

The Details control highlights undefined fields in yellow.

Use select-first-object (p. 2449) or select-all-objects (p. 2447) to select one or more Outline objects prior to accessing the Details control.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one Details meeting the criteria was selected.

See Also

select-field (p. 2448), select-first-parameter-field (p. 2450).

select-zero-thickness-sheets

Selects all parts containing zero-thickness sheet geometry.

```
<select-zero-thickness-sheets />
```

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if one or more objects meeting the criteria were selected.

select-enclosures

Selects any enclosure bodies in the current geometry.

```
<select-enclosures />
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

send-mail

Opens a new email and fills in envelope information and default text. Does not send the email.

```
<send-mail
    to="addr;addr"
    cc="addr;addr"
    bcc="addr;addr"
    subject="stringID"
    body="stringID" />
```

Attributes

to

Semicolon-delimited list of email addresses.

cc

Optional. Semicolon-delimited list of email addresses.

bcc

Optional. Semicolon-delimited list of email addresses.

subject

Optional. Default subject line.

body

Optional. Default body text.

Element Information

Parents	activate-event (p. 2430), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	None

set-caption

Sets the caption of the task.

```
<set-caption
    caption="stringID" />
```

Attributes

caption

Specifies the string ID (p. 2429) of the text.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

See Also

task (p. 2431).

set-icon

Sets the task icon to an image at a given URL.

```
<set-icon
     src="url" />
```

Attributes

src

Specifies the URI of the icon. See URI Address and Path Considerations (p. 2418).

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

See Also

task (p. 2431).

set-status

Sets the status of the task.

```
<set-status
    status="{non-status | incomplete | complete | information | undefined |
       indeterminate | solve | obsolete | ambiguous | caution |
       warning | disabled | hidden}" />
```

Note:

It was necessary to "word wrap" the long line of code in the above example.

Attributes

status

A status keyword. Status keywords are defined by using the status (p. 2427) element.

Remarks

The element definition shown above lists the standard statuses. The TaskML file `MechanicalWizard\Data\Statuses.xml` defines the standard statuses and is merged automatically while loading any wizard.

Element Information

Parents	activate-event (p. 2430), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436)
Children	None
End Tag	No - close element with "/>"
Return Value	True if successful.

Scripting

- eval (p. 2454)

eval

Evaluates a JScript expression.

```
<eval
    code="expression" />
```

Attributes

code

A string of valid JScript code. For example, "foo()" evaluates the global function **foo**.

Remarks

Use the Script (p. 2425) element to make custom JScript available for use with the **eval** statement.

If the **eval** statement is a task rule, the expression is evaluated when the rule is processed as part of an event. Using **eval** in this context allows:

- custom code to determine the status of a task
- the task to perform any arbitrary operation

The file `MechanicalWizard\WDK\Tools_Merge.xml` demonstrates use of the **eval** statement to:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- Execute global functions defined in a script file referenced by a Script (p. 2425) element.
- Access the DOM to manipulate the DHTML page containing the wizard.
- Call methods on global objects to automate the Mechanical Wizard.

  If the **eval** statement exists inside of a **body** or **group** element, the expression evaluates at that point in the generation of the wizard DHTML. Using **eval** in this context allows for programmatically generating wizard content. See `Startup.xml`, `New.xml` and `InsertGeometry.xml` in the `MechanicalWizard\Panels` folder for examples. These examples call global functions defined in the script file `MechanicalWizard\System\PanelFunctions.js`.

  Complete coverage of scripting is beyond the present scope of this documentation. You may use the source code as a reference and a script debugger for exploring variables and object models. The following globally-available JScript objects are particularly useful:

- **g_Wizard** - the global Wizard object that controls the Mechanical Wizard. Defined in `MechanicalWizard\System\WizardObject.js`.
- **g_Wizard.App** - provides access to the key objects in the Mechanical application and Ansys Workbench. Defined in `MechanicalWizard\System\AppObject.js`.
- **g_Wizard.App.Scripting** - reference to the script block inside the Mechanical application.
- **g_Wizard.GlobalStrings** - a Strings object (`StringLookupObject.js`) containing generic strings defined in `MechanicalWizard\Data\GlobalStrings.xml`.
- **g_Wizard.Strings** - a Strings object containing strings from the loaded TaskML document.

Element Information

Parents	As an action or condition: activate-event (p. 2430), update-event (p. 2432), if (p. 2436), and (p. 2435), or (p. 2437), not (p. 2437), then (p. 2436), else (p. 2436) For evaluation as the wizard loads: body (p. 2432), group (p. 2433)
Children	None
End Tag	No - close element with "/>"
Return Value	Return value of the expression or **null**.

Standard Object Groups Reference

The following table lists standard object-group (p. 2426) names and the object-type (p. 2426) elements they contain. The corresponding TaskML file is `MechanicalWizard\Data\ObjectGroups.xml`, and is merged automatically while loading any wizard.

The elements object (p. 2440), select-first-object (p. 2449), and select-all-objects (p. 2447) use object groups. TaskML files may include an object-groups (p. 2426) section to define custom object-group (p. 2426) elements (for example, to identify a specific object such as pressure). See Tutorial: Creating a Custom Task (p. 2460) for an example.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Class and Type correspond to constants defined in the script file `DSConstants.js`. Type corresponds to the "loadType" or "ResultType" property of specific Mechanical application objects.

Group Name	Class	Type
project	id_Project	
model	id_Model	
environment	id_Environment	
solution	id_AnswerSet	
geometry	id_PrototypeGroup	
part	id_Prototype	
contact	id_ContactGroup	
contact region	id_ContactRegion	
mesh	id_MeshControlGroup	
mesh control	id_MeshControl	
global load	id_Acceleration	
	id_Rotation	
load	id_Load	
structural load	id_Load	id_SurfacePressure
	id_Load	id_SurfaceForce
	id_Load	id_EdgeForce
	id_Load	id_VertexForce
	id_Load	id_CylinderBoltLoad
	id_Load	id_ForceAtAPoint
	id_Load	id_SurfaceMoment
	id_Load	id_SurfaceRotation
	id_Load	id_EdgeRotation
	id_Load	id_VertexRotation
	id_Load	id_EdgeMoment
	id_Load	id_VertexMoment
displacement load	id_Load	id_SurfaceDisplacement
	id_Load	id_EdgeDisplacement
	id_Load	id_VertexDisplacement
	id_Load	id_SurfaceSupport
	id_Load	id_FixedEdgeSupport
	id_Load	id_FixedVertexSupport
	id_Load	id_CylinderRadialSupport
	id_Load	id_CylinderRadialAndAxialSupport
	id_Load	id_SurfaceFrictionlessSupport

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Group Name	Class	Type
	id_Load	id_CylinderFixedSupport
	id_Load	id_CylinderPinnedSupport
	id_Load	id_SimpleEdgeSupport
	id_Load	id_SimpleVertexSupport
thermal load	id_Load	id_SurfaceHeatFlux
	id_Load	id_SurfaceTemperature
	id_Load	id_EdgeTemperature
	id_Load	id_VertexTemperature
	id_Load	id_SurfaceConvection
	id_Load	id_SurfaceInsulation
	id_Load	id_SurfaceHeat
	id_Load	id_EdgeHeat
	id_Load	id_VertexHeat
	id_Load	id_InternalPartHeat
stress tool	id_StressSafetyTool	
stress tool result	id_Result	id_StressSafetyMargin
	id_Result	id_StressSafetyFactor
	id_Result	id_StressRatio
result	id_Result	
structural result	id_Result	id_EquivalentStress
	id_Result	id_MaximumPrincipalStress
	id_Result	id_IntermediatePrincipalStress
	id_Result	id_MinimumPrincipalStress
	id_Result	id_MaximumShearStress
	id_Result	id_StressIntensity
	id_Result	id_XComponentStress
	id_Result	id_YComponentStress
	id_Result	id_ZComponentStress
	id_Result	id_XYShearStress
	id_Result	id_YZShearStress
	id_Result	id_XZShearStress
	id_Result	id_EquivalentStrain
	id_Result	id_MaximumPrincipalStrain
	id_Result	id_IntermediatePrincipalStrain
	id_Result	id_MinimumPrincipalStrain
	id_Result	id_MaximumShearStrain
	id_Result	id_StrainIntensity
	id_Result	id_XComponentStrain

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Group Name	Class	Type
	id_Result	id_YComponentStrain
	id_Result	id_ZComponentStrain
	id_Result	id_XYShearStrain
	id_Result	id_YZShearStrain
	id_Result	id_XZShearStrain
	id_Result	id_TotalDisplacement
	id_Result	id_XComponentDisplacement
	id_Result	id_YComponentDisplacement
	id_Result	id_ZComponentDisplacement
thermal result	id_Result	id_Temperature
	id_Result	id_TotalHeatFlux
	id_Result	id_XComponentHeatFlux
	id_Result	id_YComponentHeatFlux
	id_Result	id_ZComponentHeatFlux
	id_Result	id_MaximumPrincipalThermalStrain
	id_Result	id_IntermediatePrincipalThermalStrain
	id_Result	id_MinimumPrincipalThermalStrain
	id_Result	id_XComponentThermalStrain
	id_Result	id_YComponentThermalStrain
	id_Result	id_ZComponentThermalStrain
fatigue tool	id_FatigueTool	
fatigue result	id_Result	id_FatigueLife
	id_Result	id_FatigueSafetyFactor
	id_Result	id_FatigueDamage
	id_Result	id_FatigueBiaxialityIndication
	id_Result	id_FatigueRainflowMatrix
	id_Result	id_FatigueDamageMatrix
	id_Result	id_FatigueSensitivity
frequency	id_Result	id_Frequency

Tutorials

- Tutorial: Adding a Link (p. 2459)
- Tutorial: Creating a Custom Task (p. 2460)
- Tutorial: Creating a Custom Wizard (p. 2462)
- Tutorial: Adding a Web Search IFRAME (p. 2463)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Tutorial: Adding a Link

This tutorial covers the steps needed to add a custom link to the Links group. The Links group is available in any of the standard wizards.

View (p. 2464) the completed TaskML file for this tutorial.

Steps

To add a link to the web site MatWeb:

Open the TaskML file `MechanicalWizard\Tasks\Links.xml` in a text editor such as Notepad. All standard wizards Merge (p. 2424) the `Links.xml` file; changes made to this file automatically appear in all standard wizards.

Create a new task (p. 2431) definition by adding the following to the tasks (p. 2432) section:

```
<tasks>
    <task id="DesignSpaceHomePage" icon="simwiz://Icons/Link.gif">
        <activate-event>
            <open-url href="http://www.designspace.com" />
        </activate-event>
    </task>
    <task id="DesignSpaceResources" icon="simwiz://Icons/Link.gif">
        <activate-event>
            <open-url href="http://www.designspace.com/designspace/user_support/" />
        </activate-event>
    </task>
    <task id="MatWeb" icon="simwiz://Icons/Link.gif">
        <activate-event>
            <open-url href="http://www.matweb.com/" />
        </activate-event>
    </task>
</tasks>
```

The value for the **id** attribute is arbitrary.

Define a new string (p. 2429) by adding the following to the strings (p. 2430) section:

```
<strings>
    <language xml:lang="en-us">
        <string id="Standard_Links_Caption">
            Links
        </string>
        <string id="DesignSpaceHomePage_Caption">
            DesignSpace.com
        </string>
        <string id="DesignSpaceResources_Caption">
            DesignSpace Resources
        </string>
        <string id="MatWeb_Caption">
            MatWeb Materials
        </string>
    </language>
</strings>
```

The value for the string **id** uses the built-in naming convention of the task **id** and "**_Caption**" to simplify the **task** element by omitting the **caption** attribute. The new string applies to the default language code "**en-us**." To support other languages, define a new string inside each language (p. 2429) section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Insert the new task into the Links group (p. 2433) by modifying the body (p. 2432) section as follows:

```
<body>
    <group id="Standard_Links" collapsed="yes">
        <taskref task="DesignSpaceHomePage" />
        <taskref task="DesignSpaceResources" />
        <taskref task="MatWeb" />
    </group>
</body>
```

The task attribute matches the id of the task.

Save the file.

Open a wizard in the Mechanical application. The Links group will contain a new link to the MatWeb website.

Tutorial: Creating a Custom Task

This tutorial describes the steps needed to develop a custom task for inserting a 100 psi pressure load. The tutorial for Creating a Custom Wizard (p. 2462) uses the task created below.

View (p. 2465) the completed TaskML file for this tutorial.

Steps

Copy the file `MechanicalWizard\Tasks\InsertStructuralLoad.xml` to a file named `Insert100psi.xml` in a different folder. Generally, the easiest way to create a custom task is to modify a similar existing task instead of starting from scratch.

task (p. 2431) element as follows:

```
<task id="Insert100psi" disable-if-missing="geometry" check-ambiguity="environment">
```

The other attributes on the task element disable the task if the Outline contains no geometry and prompts the user to select a particular Environment if the current selection is ambiguous.

Create an object-groups (p. 2426) section at the top of the file:

```
<simulation-wizard version="1.0">
    <object-groups>
        <object-group name="pressure">
            <object-type class="id_Load" type="id_SurfacePressure" />
        </object-group>
    </object-groups>
    ...
</simulation-wizard>
```

This creates a custom object-group (p. 2426) named "pressure" that contains a single object-group (p. 2426) corresponding to the Pressure object type in the Outline. This object group is available in addition to the Standard Object Groups Reference (p. 2455) to wizards merging (p. 2424) this task.

Modify the strings (p. 2430) section as follows:

```
<strings>
    <language xml:lang="en-us">
        <string id="Insert100psi_Caption">
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
            Insert Pressure
        </string>
        <string id="Insert100psi_Message">
            Use the Structural button to insert a Pressure load.
            Enter 100 psi for Magnitude.
        </string>
    </language>
</strings>
```

The value for the first string **id** uses the built-in naming convention of the task **id** and "**_Caption**" to simplify the **task** element by omitting the **caption** attribute. The value for the second string **id** is arbitrary and referenced by the **display-details-callout** action defined below. The strings apply to the default language code "**en-us**." To support other languages, define new strings inside each language (p. 2429) section.

Modify the update-event (p. 2432) as shown:

```
<update-event>
    <if><object type="pressure" condition="does-not-exist"/>
        <then>
            <set-status status="incomplete"/>
            <stop/>
        </then>
    </if>
    <if><object type="pressure" condition="exists" state="under-defined"/>
        <then>
            <set-status status="undefined"/>
            <stop/>
        </then>
    </if>
    <set-status status="complete"/>
</update-event>
```

Modify the activate-event (p. 2430) as shown:

```
<activate-event>
    <if><object type="pressure" condition="exists" state="under-defined"/>
        <then>
            <select-first-object type="pressure" state="under-defined"/>
            <select-first-undefined-field/>
            <display-details-callout message="Insert100psi_Message" />
            <stop/>
        </then>
    </if>
    <if><level type="environment" condition="is-not-selected"/>
        <then>
            <select-first-object type="environment"/>
        </then>
    </if>
    <click-button toolbar="DS_graphics" button="Surface"/>
    <display-toolbar-callout toolbar="Context" button="Structural"
        message="Insert100psi_Message" />***
</activate-event>
```

Note:

***Note that it was necessary to "word wrap" the long line of code in the above example.

The first **if** statement checks for an under-defined pressure. The second **if** statement ensures that the Outline selection is at the Environment level so that the user can insert a Pressure. The **click-button** action ensures that the surface selection mode is active.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

Save the file.

Proceed to the tutorial Creating a Custom Wizard (p. 2462) to use this custom task.

Tutorial: Creating a Custom Wizard

This tutorial describes the steps needed to develop a custom wizard. Before proceeding, complete the tutorial Creating a Custom Task (p. 2460).

View (p. 2466) the completed TaskML file for this tutorial.

Steps

Copy the file `MechanicalWizard\StressWizard.xml` to a file named `CustomWizard.xml` in the same folder as the file `Insert100psi.xml` created in the previous tutorial.

Change "InsertStructuralLoad.xml" to "Insert100psi.xml" in the Merge (p. 2424) element:

```
<merge src="Insert100psi.xml" />
```

This **merge** makes the custom task definition available to this wizard. Note that the URI to the file containing the task is relative to the location of the file containing the wizard. See URI Address and Path Considerations (p. 2418).

Modify the strings (p. 2430) section as follows:

```
<strings>
    <language xml:lang="en-us">
        <string id="Title_Caption">
            Tutorial Wizard
        </string>
        <string id="Title_Description">
            Demonstrates a custom wizard with a task for inserting a 100 psi Pressure.
        </string>
    </language>
</strings>
```

Change "InsertStructuralLoad" to "Insert100psi" in the taskref (p. 2434) element:

```
<taskref task="Insert100psi"/>
```

This **taskref** adds the task (p. 2431) to the body (p. 2432) of the wizard by its **id**.

Save the file.

In the Mechanical application, click the **Choose Wizard** option from the top of a standard wizard.

Choose "browse for a custom wizard definition.".

Select the file **CustomWizard.xml**.

Test the Insert Pressure task. The task should behave in the same way as the standard Insert Loads task but with specific instructions for defining a 100 psi pressure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

Tutorial: Adding a Web Search IFRAME

This tutorial describes the steps needed to add an Internet search capability to a wizard as an IFRAME. This tutorial uses the wizard created in Tutorial: Creating a Custom Wizard (p. 2462). See Using IFRAME Elements (p. 2422) for a discussion on IFRAMEs.

View the file Search.htm (p. 2467) or the modified TaskML file CustomWizard.xml (p. 2466).

Steps

Create a new text file with the following contents:

```
<html>
<head>
<script src="System/IFrame.js"></script>
<link ID="Skin" REL="stylesheet">
<script>
function IFrame_onload() { Skin.href = g_Wizard.GetSkin() }
</script>
<style>
INPUT { width: 100%; margin-bottom: 4px; }
</style>
</head>
<body scroll="no">
<center>
<form method="GET" action="http://www.google.com/search" target="_blank">
<a HREF="http://www.google.com/" target="_blank">
<img SRC="http://www.google.com/logos/Logo_40wht.gif" border="0" ALT="Google"
   width="128" height="53"></a><br>
<input TYPE="text" name="q" size="25" maxlength="255" value><br>
<input type="submit" name="btnG" VALUE="Google Search">
</form>
</center>
</body>
</html>
```

Note:

It was necessary to "word wrap" the long line of code in the above example.

Note:

- The script file `MechanicalWizard/System/IFrame.js` contains generic functions for use with IFRAMEs.
- The **link** element initially lacks a **href** element. The script block implements the **IFRAME_onload** function (called by `IFrame.js`) and sets **href** to the url returned by the **GetSkin** method on the **g_Wizard** object. The file `MechanicalWizard\WDK\Info_IFRAME.htm` contains an inaccuracy in that the link is not automatically assigned.
- The **style** element provides some additional formatting rules.
- The **body** element has the scroll element set to "no" to preserve margins and prevent scrollbars from appearing. As long as a reference to `IFrame.js` appears in the IFRAME the Mechanical Wizard will autosize the height such that scrollbars are unnecessary.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

- The contents of the body are based on Google's custom search engine.
- Note use of the **target** attribute to prevent the linked pages from opening in place of the Mechanical Wizard.

Save the file as **Search.htm** in the **Mechanical Wizard** folder. The files must reside together for web browser security to permit cross-frame scripting.

Open the file **CustomWizard.xml** from the previous tutorial.

Add the following **group** at the bottom of the **body**:

```
<group id="Search" collapsed="yes">
    <iframe src="simwiz://Search.htm" />
</group>
```

Add the following **string** to the <strings><language xml:lang="en-us"> section:

```
<string id="Search_Caption">
    Search the Web
</string>
```

Note the use of the "*groupID*_Caption" shortcut for the string id.

Save the file and open the wizard in the Mechanical application.

Completed TaskML Files

The following sections examine examples of completed TaskML files.

Links.xml

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<?xml version="1.0"?>

<simulation-wizard version="1.0">
 <strings>
  <language xml:lang="en-us">
   <string id="Standard_Links_Caption">
    Links
   </string>
   <string id="DesignSpaceHomePage_Caption">
    DesignSpace.com
   </string>
   <string id="DesignSpaceResources_Caption">
    DesignSpace Resources
   </string>
   <string id="MatWeb_Caption">
    MatWeb Materials
   </string>
  </language>
 </strings>
 <tasks>
  <task id="DesignSpaceHomePage" icon="simwiz://Icons/Link.gif">
   <activate-event>
    <open-url href="http://www.designspace.com" />
   </activate-event>
  </task>
  <task id="DesignSpaceResources" icon="simwiz://Icons/Link.gif">
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
   <activate-event>
    <open-url href="http://www.designspace.com/designspace/user_support/" />
   </activate-event>
  </task>
  <task id="MatWeb" icon="simwiz://Icons/Link.gif">
   <activate-event>
    <open-url href="http://www.matweb.com/" />
   </activate-event>
  </task>
 </tasks>
 <body>
  <group id="Standard_Links" collapsed="yes">
   <taskref task="DesignSpaceHomePage" />
   <taskref task="DesignSpaceResources" />
   <taskref task="MatWeb" />
  </group>
 </body>
</simulation-wizard>
```

Insert100psi.xml

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<?xml version="1.0"?>
<simulation-wizard version="1.0">
 <object-groups>
  <object-group name="pressure">
   <object-type class="id_Load" type="id_SurfacePressure" />
  </object-group>
 </object-groups>
 <strings>
  <language xml:lang="en-us">
   <string id="Insert100psi_Caption">
    Insert Pressure
   </string>
   <string id="Insert100psi_Message">
    Use the Structural button to insert a Pressure load.
    Enter 100 psi for Magnitude.
   </string>
  </language>
 </strings>
 <tasks>
  <task id="Insert100psi" disable-if-missing="geometry" check-ambiguity="environment">
   <update-event>
    <if><object type="pressure" condition="does-not-exist"/>
     <then>
      <set-status status="incomplete"/>
      <stop/>
     </then>
    </if>
    <if><object type="pressure" condition="exists" state="under-defined"/>
     <then>
      <set-status status="undefined"/>
      <stop/>
     </then>
    </if>
    <set-status status="complete"/>
   </update-event>
   <activate-event>
    <if><object type="pressure" condition="exists" state="under-defined"/>
     <then>
      <select-first-object type="pressure" state="under-defined"/>
      <select-first-undefined-field/>
      <display-details-callout message="Insert100psi_Message" />
      <stop/>
     </then>
    </if>
    <if><level type="environment" condition="is-not-selected"/>
     <then>
      <select-first-object type="environment"/>
```

```
     </then>
    </if>
    <click-button toolbar="DS_graphics" button="Surface"/>
    <display-toolbar-callout toolbar="Context" button="Structural"
             message="Insert100psi_Message" />***
   </activate-event>
  </task>
 </tasks>
</simulation-wizard>
```

Note:

***Note that it was necessary to "word wrap" the long line of code in the above example.

CustomWizard.xml

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<?xml version="1.0"?>

<simulation-wizard version="1.0">
 <merge src="Tasks/InsertGeometry.xml" />
 <merge src="Tasks/VerifyLengthUnit.xml" />
 <merge src="Tasks/DefineSheetThickness.xml" />
 <merge src="Tasks/AssignMaterial.xml" />
 <merge src="Insert100psi.xml" />
 <merge src="Tasks/InsertDisplacementLoad.xml" />
 <merge src="Tasks/ThermalStressNote.xml" />
 <merge src="Tasks/InsertStructuralResults.xml" />
 <merge src="Tasks/StressStiffeningNote.xml" />
 <merge src="Tasks/Solve.xml" />
 <merge src="Tasks/ViewResults.xml" />
 <merge src="Tasks/ViewReport.xml" />

 <merge src="Tasks/StandardTasks.xml"/>
 <strings>
  <language xml:lang="en-us">
   <string id="Title_Caption">
    Tutorial Wizard
   </string>
   <string id="Title_Description">
    Demonstrates a custom wizard with a task for inserting a 100 psi Pressure.
   </string>
  </language>
 </strings>
 <body>
  <group id="Title">
   <taskref task="ChooseWizard"/>
  </group>
  <group id="RequiredSteps" emphasize="yes">
   <taskref task="InsertGeometry"/>
   <taskref task="VerifyLengthUnit"/>
   <taskref task="DefineSheetThickness"/>
   <taskref task="AssignMaterial"/>
   <taskref task="Insert100psi"/>
   <taskref task="InsertDisplacementLoad"/>
   <taskref task="ThermalStressNote"/>
   <taskref task="InsertStructuralResults"/>
   <taskref task="StressStiffeningNote"/>
   <taskref task="Solve"/>
   <taskref task="ViewResults"/>
   <taskref task="ViewReport"/>
  </group>
  <group id="Standard_OptionalTasks" />
  <group id="Standard_ParameterTasks" />
  <group id="Standard_GeneralTasks" />
  <group id="Standard_AdvancedTasks" />
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  <group id="Standard_Links" />
 </body>
</simulation-wizard>
```

Search.htm

```
<?xml version="1.0" encoding="ISO-8859-1"?>
<!doctype HTML public "-//W3C//DTD HTML 4.0 Frameset//EN">

<html>

<!--(===========================================================)-->
<!--(Document created with RoboEditor. )=========================-->
<!--(===========================================================)-->

<head>

<title>Search</title>

<!--(Meta)=================================================-->

<meta name=generator content="RoboHELP by eHelp Corporation - www.ehelp.com">
<meta name=generator-major-version content=0.1>
<meta name=generator-minor-version content=1>
<meta name=filetype content=kadov>
<meta name=filetype-version content=1>
<meta name=page-count content=1>
<meta name=layout-height content=427>
<meta name=layout-width content=640>
<meta name=date content="07 9, 2003 11:30:11 AM">


<!--(Links)=================================================-->

<link ID=Skin REL=stylesheet>



<!--(Style Sheet)===========================================-->

<style>
<!--

INPUT {
 width: 100%;
 margin-bottom: 4px;
}

-->
</style>

<!--(Scripts)===============================================-->

<script src="System/IFrame.js"></script>

<script>function IFrame_onload() { Skin.href = g_Wizard.GetSkin() }</script>

</head>

<!--(Body)==================================================-->


<body scroll=no>

<form method=GET
  action="http://www.google.com/search"
  target=_blank>
<p style="text-align: center;"
 align=center><a HREF="http://www.google.com/"
```

```
    target=_blank><img src="http://www.google.com/logos/Logo_40wht.gif"
          ALT=Google
          style="width: 128px;
          height: 53px;
          border-style: none;"
          width=128
          height=53
          border=0></a><br>
<input TYPE=text
  name=q
  size=25
  maxlength=255
  value><br>
<input type=submit
  name=btnG
  VALUE="Google Search"></p>
</form>

</body>

</html>
```

CustomWizardSearch.xml

```
<?xml version="1.0"?>

<simulation-wizard version="1.0">
 <merge src="simwiz://Tasks/InsertGeometry.xml" />
 <merge src="simwiz://Tasks/VerifyLengthUnit.xml" />
 <merge src="simwiz://Tasks/DefineSheetThickness.xml" />
 <merge src="simwiz://Tasks/AssignMaterial.xml" />
 <merge src="Insert100psi.xml" />
 <merge src="simwiz://Tasks/InsertDisplacementLoad.xml" />
 <merge src="simwiz://Tasks/ThermalStressNote.xml" />
 <merge src="simwiz://Tasks/InsertStructuralResults.xml" />
 <merge src="simwiz://Tasks/StressStiffeningNote.xml" />
 <merge src="simwiz://Tasks/Solve.xml" />
 <merge src="simwiz://Tasks/ViewResults.xml" />
 <merge src="simwiz://Tasks/ViewReport.xml" />

 <merge src="simwiz://Tasks/StandardTasks.xml"/>
 <strings>
  <language xml:lang="en-us">
   <string id="Title_Caption">
    Tutorial Wizard
   </string>
   <string id="Title_Description">
    Demonstrates a custom wizard with a task for inserting a 100 psi Pressure.
   </string>
   <string id="Search_Caption">
    Search the Web
   </string>
  </language>
 </strings>
 <body>
  <group id="Title">
   <taskref task="ChooseWizard"/>
  </group>
  <group id="RequiredSteps" emphasize="yes">
   <taskref task="InsertGeometry"/>
   <taskref task="VerifyLengthUnit"/>
   <taskref task="DefineSheetThickness"/>
   <taskref task="AssignMaterial"/>
   <taskref task="Insert100psi"/>
   <taskref task="InsertDisplacementLoad"/>
   <taskref task="ThermalStressNote"/>
   <taskref task="InsertStructuralResults"/>
   <taskref task="StressStiffeningNote"/>
   <taskref task="Solve"/>
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

```
   <taskref task="ViewResults"/>
   <taskref task="ViewReport"/>
  </group>
  <group id="Standard_OptionalTasks" />
  <group id="Standard_ParameterTasks" />
  <group id="Standard_GeneralTasks" />
  <group id="Standard_AdvancedTasks" />
  <group id="Standard_Links" />
  <group id="Search" collapsed="yes">
   <iframe src="simwiz://Search.htm" />
  </group>
 </body>
</simulation-wizard>
```

Wizard Development Kit (WDK) Groups

- WDK: Tools Group (p. 2469)
- WDK: Commands Group (p. 2470)
- WDK Tests: Actions (p. 2470)
- WDK Tests: Flags (Conditions) (p. 2471)

WDK: Tools Group

The **WDK: Tools** group provides interactive access to the functionality of several of the most important TaskML elements and exposes some key internal data. The group also demonstrates how IFRAMEs allow arbitrary customization of the user interface.

The **WDK: Tools** group updates automatically when the selection in the Outline changes.

Level Testing

The Outline Level section exercises the functionality of the level (p. 2439) element.

Object Testing and Selection

The second section exercises the functionality of the object (p. 2440), select-first-object (p. 2449) and select-all-objects (p. 2447) elements. Expert users may find this section useful for automating selection in the Outline. For example, typing "prt" under Name Regular Expression and clicking Select All Matching Objects selects all Outline objects with "prt" in their name.

Details Field String ID

The third section exposes the string ID of the currently selected Details field for use with the select-field (p. 2448) element.

Preview Event Code

Advanced. If checked, displays a message box containing virtual JScript event code prior to its evaluation. Used for low-level debugging of task rules.

Folder

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displays the folder from which the Mechanical Wizard is currently running. Corresponds to "Mechanical Wizard URL" in the Control Panel.

WDK: Commands Group

The Commands group exposes options for viewing internal data and for manipulating the system.

Reload Sim Wizard

Reloads the HTML page containing the Mechanical Wizard. The system is reset and the Startup panel displayed.

Open Wizard

Displays an Open dialog to choose a TaskML file to load. Same as selecting the "browse" option from "Choose Wizard" on the Startup panel or in wizards.

Fire Update Event

Forces an update-event (p. 2432) to occur in the Mechanical Wizard.

View Current Toolbar Keys

Displays a temporary XML file containing the toolbar and button keys for the current state of the user interface. Toolbar and button keys are used to define the click-button (p. 2442) and display-toolbar-callout (p. 2446) elements.

View Wizard XML

Displays a temporary XML file containing the internal merged TaskML.

Remove Merge Information

Removes merge tracking information from the internal TaskML.

Snapshot Wizard DHTML

Saves an HTML file snapshot of the current Mechanical Wizard. The HTML snapshot is useful for developing CSS skins.

Clear UserData

Clears the Mechanical Wizard UserData store. The UserData store consists of Tip of the Day, group expansion, and other non-critical data.

WDK Tests: Actions

The Actions group exercises actions used to define task rules.

Actions

- display-outline-callout (p. 2444)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

- display-details-callout (p. 2443)
- display-task-callout (p. 2446)
- display-toolbar-callout (p. 2446)
- display-tab-callout (p. 2445)
- display-status-callout (p. 2445)
- send-mail (p. 2452)
- open-url (p. 2447)
- display-help-topic (p. 2443)
- select-zero-thickness-sheets (p. 2451)
- select-first-undefined-field (p. 2451)
- select-first-parameter-field (p. 2450)
- select-enclosures (p. 2451)

WDK Tests: Flags (Conditions)

The Flags group exercises conditions used to define task rules.

Flags (Conditions)

- changeable-length-unit (p. 2439)
- assembly-geometry (p. 2438)
- geometry-includes-sheets (p. 2439)
- zero-thickness-sheet (p. 2441)
- valid-emag-geometry (p. 2441)
- enclosure-exists (p. 2441)

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Index

Symbols

2D analyses - description and characteristics, 878

A

acceleration
  description, 1501
add linearized stress, 2041
add offset no ramping contact region setting , 1257
add offset ramped effects contact region setting, 1257
adding beams, 1404
Additive Manufacturing Process Simulation, 685
adjust to touch contact region setting, 1257
adjustment result in bolt tool
  description, 2086
AM Process, 685
  application, 685
  workflow, 685
ambient temperature - in radiation load, 1609
analysis
  2D analyses - description and characteristics, 878
  apply loads and supports step, 354
  apply mesh controls step, 348
  apply preview mesh step, 348
  approach - overall steps, 323
  composite, 685
  create report step, 358
  define initial condition step, 349
  define substructures, 347
  ecad trace mapping in Mechanical, 693
  review results step, 356
  set connections options step, 348
  solve step, 355
  types - listed, 359
analysis settings
  fracture controls, 1445
  role of time, 1491
  steps and step controls overall topics, 1491
  topic listing, 1417
analysis topics - special, 685
analysis types
  coupled field transient, 395
  eigenvalue buckling, 487
  electric, 403
  explicit dynamics, 406
  harmonic, 496
  listing, 359
  magnetostatic, 546
  modal, 515
  random vibration, 524
  response spectrum, 533
  rigid dynamics, 550, 844
  static structural, 652
  steady-state thermal, 657
  thermal-electric, 661
  transient thermal, 680
angular periodicity, 1131
annotations
  basics, 306
  environment, 306
  highlight and selection graphics, 306
  message, 306
  multiple objects, 309
  positioning, 306
  rescaling, 306
  solution, 306
Ansys CFX- solving with fluid solid interface , 1648, 1651
Ansys Product Improvement Program, 31
Ansys Workbench
  safety tools, 2058
anti-periodic symmetry type, 1131
anti-symmetric electromagnetic symmetry type, 1131
anti-symmetric structural symmetry type, 1128
anti-symmetric thermal symmetry type, 1128
anti-virus causing crash - troubleshooting, 2374
APIP (Ansys Product Improvement Program see)
application not closing - troubleshooting, 2365
applying a graphical view, 292
applying loads and supports
  overall analysis step, 354
applying pre-stress effects, 352
assemblies missing parts - troubleshooting, 2365
assemblies of surface bodies, 861
associative and non-associative coordinate systems, 1192
asynchronous solutions
  description, 1803
  troubleshooting, 2371
Autodesk Inventor
  assigning parameters, 2340
automatic contact, 1266
automatic time stepping analysis settings, 1493
averaged vs. unaveraged contour results, 1981
axisymmetric behavior - 2D simulation, 878

B

background solutions
  troubleshooting, 2371
beam, 1404
beam end releases, 1409

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

beam probe result, 2125
beam results, 2089
bearing load
  description, 1561
bearings
  description, 1413
bin size - fatigue simulations, 2176
body
  hide, 857
  suppress, 857
bolt pretension
  description, 1566
bolt thread contact region setting, 1257
bolt tool result
  description, 2086
boundary condition scoping method
  description, 1497
boundary condition types - listed, 1663
boundary conditions
  description, 1497
boundary conditions - electromagnetic, 1635

C

CAD parameters, 2340
CAD systems
  general information, 2351
  isotropic material limitation, 2352
  multiple versions, 2352
callouts in details view, 119
cannot undo node move - troubleshooting, 2365
CFD load transfer
  convections, 763
  structural, 762
  surface temperatures, 763
chart and table, 1929
clean results data, 1981
  geometry change, 1981
  mesh change, 1981
clipboard, 104
clipboard toolbar, 104
color by parts, 858
colors - contact initial information table, 2079
commands object
  properties, 2307
commands objects
  LS-DYNA Solver, 2321
  Rigid Dynamics Solver, 2321
composite analysis, 685
composite failure results, 2072
composite failure tool, 2072
composite results, 1990
compression only support
  description, 1685
condensed part
  application, 786
  best practices, 805
  limitations, 801
  overview, 784
  worksheet, 788
condensed part missing master node - troubleshooting, 2360
condensed parts, 781
condensing parts, 347
conditions, 1697
  constraint equation, 1702
  coupling, 1698
  element birth and death, 1707
  pipe idealization, 1704
conductor
  current excitation for solid source conductors, 1641
  description, 1637
  solid body as conductor, 1638
  stranded source body as conductor, 1642
  stranded source conductor, 1643
  voltage excitation for solid source conductors, 1640
connection detection global setting, 1212
connection lines with remote point, 1098
connections
  global settings - listed and defined, 1212
  setting options - overall analysis step, 348
connections worksheet, 1210
constraint equation
  description, 1702
constraint equations were not properly matched-troubleshooting, 2366
construction geometry
  path, 883
  solid, 892
  STL file import, 894
  surface, 890
contact
  automatic, 1266
  best practices, 1281
  controlling transparency for regions - animated example, 1269
  ease of use features - listed, 1267
  formulation theory, 1222
  hiding bodies not scoped to contact region - procedure, 1271
  initial, 2079
  loading region settings - procedure, 1274

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

locating bodies without contact - application and procedure, 1275
locating parts without contact - application and procedure, 1275
manual, 1266
merging regions - procedure, 1274
overview, 1222
reactions, 2079
renaming regions - animated example, 1271
resetting regions to defaults - procedure, 1275
saving region settings - procedure, 1274
setting conditions manually - guidelines and procedure, 1266
settings, 1226
supported contact types, 1265
supported formulations, 1265
symmetry, 1265
tool, 2079
contact based reactions, 2107
contact best practices
contact behavior, 1281
contact tool, 1285
diagnostics, 1290
mesh quality, 1286
non-convergence, 1290
overlap conditions, 1289
solver preparation, 1286
contact bodies
colors, 1269
contact debonding
application, 480
contact geometry correction contact region setting, 1257
contact region
automatically generate objects, 1268
contact region(s) not in initial contact - troubleshooting , 2368
contact tool result
description, 2079
contours during solve, 1844
controlling transparency for contact regions - animated example, 1269
convection
description, 1602
convections
at CFD boundary, 763
convective heat transfer, 1602
convergence
plots, 1833
coordinate systems
applying local coordinate systems, 1198
create section plane, 1199
creating, 1191
creating construction surfaces, 891
importing, 1198
orientation, 1194
overall topics, 1191
principal axis, 1194
reference number, 1192
references, 1198
result, 1974
transferring to the Mechanical APDL application, 1201
transformations, 1195
use in specifying joint locations., 1199
using, 1191
coordinate systems results, 1935
copying graphical view to Mechanical APDL, 294
coupled field analysis types, 376
coupled field transient analysis type, 395
coupling
description, 1698
crack analysis, 406
cracks, 424
create a simulation template, 346
create coordinate system from nodes, 274
create named selection from element faces, 276
create named selection from elements, 276
create named selection from nodes, 275
create solid geometry in Mechanical, 347
creating a graphical view, 292
creep controls, 1445
creep strain - equivalent, 2034
Creo Parametric
assigning parameters, 2340
current
description, 1630
current density result
description, 2159
current excitation
stranded source conductor body, 1643
current excitation for solid source conductors
description, 1641
cursors - rotation, 257
cut boundary displacement method (submodeling see)
cyclic axis of symmetry - troubleshooting, 2377
cyclic controls, 1446
cyclic region, 1134
cyclic symmetry in a harmonic response analysis, 1151
boundary conditions, 1151
results, 1156
cyclic symmetry in a modal analysis, 1159

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

cyclic symmetry in a static structural analysis, 1147
  boundary conditions, 1147
  results, 1149
cyclic symmetry in a thermal analysis, 1166
cylindrical joints, 1298
cylindrical support
  description, 1688

D

damping controls - analysis settings, 1462
data transfer
  Polyflow to Mechanical , 773
debonding
  fracture analysis, 476
define initial condition
  overall analysis step, 349
deformation , 2018
degrees of freedom and joint types, 1298
delamination
  fracture analysis, 476
deleting a graphical view, 293
details view
  description and user interactions, 119
direct element selection, 279
direct fe
  nodal displacement, 1719
  nodal force, 1719
  nodal orientation, 1719
  nodal rotation, 1719
direct fe types - listed, 1719
direct node selection, 279
direction
  defaults, 283
  selecting, 283
directional acceleration result
  description, 2018
directional deformation result
  description, 2018
Directional Force electromagnetic result
  description, 2159
directional heat flux result
  description, 2153
directional magnetic field intensity result
  description, 2159
directional magnetic flux density result
  description, 2158
directional velocity result
  description, 2018
displacement
  description, 1666
  edge, 1666
  remote, 1666
  surfaces, 1666
  vertex, 1666
display points - in solution information, 1833
Distance Based Average Comparison option
  for Mapping Validation, 2412
distributed mass
  description, 902
dynamic legend, 2271

E

ease of use contact features - listed, 1267
eigenvalue buckling analysis type, 487
elastic strain intensity result
  description, 2029
elastic support
  description, 1695
electric analysis type, 403
Electric Machines NVH Analysis
  results, 720
electric potential result
  description, 2158
electric results, 2162
electro-mechanical transducer
  description, 1732
electromagnetic boundary conditions - listed, 1635
electromagnetic periodic symmetry, 1132
electromagnetic-thermal load import, 707
element birth and death
  description, 1707
element distortion error - troubleshooting, 2366
element name results, 2002
element orientation, 1008
  application, 1008
  specification requirements, 1008
element selection
  direct, 279
element through the thickness - troubleshooting, 2364
element type results, 2002
elemental coordinate systems results, 1936
em transducer
  description, 1732
emissivity - in radiation load, 1609
enclosure - in radiation load, 1609
environment
  annotations, 306
environment filtering of GUI, 216
equivalent creep strain ratio has exceeded the specified
limit - troubleshooting, 2367
equivalent creep strain result
  description, 2034

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

equivalent plastic strain result
description, 2032
equivalent stress result
description, 2027
equivalent total strain result
description, 2035
error - magnetic result
description, 2161
error result
structural, 2030
thermal, 2154
excitations - electromagnetic, 1635
explicit dynamics analysis type, 406
export
description, 249
file format, 249
exported file saved to disk but microsoft office failed to load - troubleshooting, 2360
exporting a saved graphical view list, 293
external model
importing deformation-based geometry, 1022

F

failed to load microsoft office application - troubleshooting, 2367
fatigue combination
description, 2188
fatigue results
frequency-based fatigue, 2169
stress life versus strain life, 2167
fatigue simulations
loading options, 2176
material properties, 2165
overview, 2165
strain-life, 2165
stress-life, 2165
user life units, 2176
figure
description, 313
file names - CAD limitation, 2352
filter for objects, 114
filter for tags, 114
filter the tree, 114
filtering GUI based on environment, 216
fixed joints, 1298
fixed rotation support
description, 1692
fixed support
Edge: description, 1664
surFace: description, 1664
Vertex: description, 1664
flexible bodies, 851
flexible parts in a Rigid Dynamics analysis , 347
flip reference and mobile for joints, 1365
flipping periodic low and periodic high settings, 1181
fluid solid interface
description, 1648, 1651
fluid-structure interaction
convections, 763
surface temperatures, 763
fluid-structure interaction - one-way using imported loads, 758
fluid-structure interaction - overall description, 756
fluid-structure interaction - two-way, 757
fluid-structure interface
face forces, 762
flux linkage result
description, 2160
force
description, 1543
foundation stiffness - in elastic support , 1695
fracture, 406
define arbitrary crack, 426
define pre-meshed crack, 465
define semi-elliptical crack, 433
fracture analysis, 406
multi-point constraint contact, 482
workflows, 407
fracture controls, 1445
fracture results
probes, 2127
fracture tool
probes, 2127
frequency-based fatigue, 2169
material properties, 2169
result formulation methods, 2173
frictional stress result in contact tool
description, 2079
frictionless support
description, 1682
FSI - one-way imported loads, 758
FSI - overall description, 756
FSI - two-way, 757
full integration scheme, 857

G

gasket
mesh control, 853
gasket bodies, 851, 853
specify, 853
gasket meshing, 853
gasket results, 853, 2131

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

gaskets
defining, 853
general axisymmetric, 1167
application, 1188
general joints, 1298
generalized plane strain
behavior in 2D analyses, 878
description, 1582
reactions, 2107
using, 881
generate connections on update global setting, 1212
generate objects from template object, 2343
generating reports
tables, 176
geometric modification contact region settings - listed and defined, 1257
geometry
analysis template, 878
full screen mode, 129
legend, 134
tab, 129
window, 129
geometry from deformation results, 1022
geometry view
saving, 292
geometry window
model movement - manipulation, 280
global connection settings - listed and defined , 1212
glossary, 2385
graphical views
applying, 292
copying to Mechanical APDL, 294
creating, 292
deleting, 293
exporting list, 293
importing list, 293
managing, 292
renaming, 293
replacing saved, 293
graphics
control, 257
tips, 257
graphics - topic listing, 257
group by global connection setting, 1212
group tree objects, 208

H

harmonic analysis, 496
harmonic analysis amplitude result calculation, 505
heat flow
description, 1614
heat flux
description, 1617
heat flux results, 2153
heat reaction result
description, 2154
hide all other bodies, 857
hide body, 857
hide faces, 857
hide items, 113
hide other bodies for joints, 1365
hiding bodies not scoped to contact region - procedure, 1271
home tab, 40
hotkeys, 314
hydrostatic pressure
description, 1537

I

Icepak
Mechanical data transfer, 764
transient, 765
illogical reaction results - troubleshooting, 2367
Import thickness, 867
imported body force density load, 1739
imported body temperature load, 1740
imported convection coefficient load, 1744
imported cut boundary constraint, 1744-1745
imported cut remote force, 1746
imported displacement load, 1746
imported force, 1747
imported heat flux load, 1748
imported heat generation load, 1748
imported initial stain load, 1749
imported initial stress load, 1750
imported load
electromagnetic-thermal, 707
imported pressure load, 1752
imported surface force density, 1755
imported temperature load, 1756
imported velocity load, 1756
importing
coordinate systems, 1198
importing a graphical view list, 293
importing mesh-based databases
coordinate systems, 934
element orientations, 939
flexible remote connections, 942
loads, 919
named selections, 945
nodal orientations, 945
point masses, 947

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

rigid remote connections, 950-951
shell thickness, 954
springs, 956
importing mesh-based databases via external model
bolt pretensions, 918
composites, 924
constraint equations, 928
contacts, 930
in process solutions, 1803
inductance result
description, 2159
inertia relief analysis setting, 1428
inertia tensor is too large - troubleshooting, 2366
inertial loads - listed, 1500
infinite life - fatigue simulations, 2176
initial contact, 2079
initial information object in contact tool
description, 2079
initial information result in contact tool
colors in table, 2079
description, 2079
initial time increment problems - troubleshooting , 2374
insufficient disk space - troubleshooting, 2362
integration scheme, 857
interface behavior based on license levels, 215
interface delamination
application, 477
interface treatment contact region setting, 1257
internal heat generation
description, 1620
interpolation
path results, 1937
probe results, 1937
surface results, 1937
invalid material properties - troubleshooting , 2361
invert suppressed body set, 113
isotropic materials - CAD limitation, 2352
iterative solver problem - troubleshooting, 2363

J

joint
create manually, 1357
description, 1592
legend, 134
properties, 1316
joint checker, 1365
joint legend, 1365
joint probe problems - troubleshooting, 2375
joint probes
results, 2118
joints
characteristics, 1293
detecting overconstrained conditions, 1369
ease of use features, 1365
example, 1380
generate automatically, 1359
topics, 1292
types, 1298

K

Keyframe animation, 255
known temperature load, 1598

L

labeling objects, 2349
large deflections analysis setting, 1428
large deformation effects are active - troubleshooting , 2367
large deformation problems - troubleshooting, 2374
layered sections, 868
license manager server problems - troubleshooting , 2375
licensed product issues - troubleshooting, 2374
life - fatigue user life units, 2176
lighting controls, 312
line bodies
general description, 872
line pressure
description, 1585
line pressure result, 2151
linear dynamic analysis types, 486
linear periodic structural symmetry type, 1128
linear periodic thermal symmetry type, 1128
linear perturbation, 476
Linearized stress, 2041
linearized stress error - troubleshooting, 2359
load transfer error - troubleshooting, 2360
load transfer mesh mapping, 2389
load types - listed, 1500
loading contact region settings - procedure, 1274
loading types - fatigue simulations, 2176
local coordinate system
applying, 1198
locating bodies without contact - application and procedure , 1275
locating parts without contact - application and procedure , 1275
low/high cyclic symmetry - troubleshooting, 2377
LS-DYNA Solver
using commands, 2321
lustre
troubleshooting, 2383

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

M

macros - usage and accessing, 248
magnetic error result, 2161
magnetic field intensity result
  directional, 2159
  total, 2158
magnetic flux boundary condition, 1636
magnetic flux density result
  directional, 2158
  total, 2158
magnetic flux parallel load
  description, 1636
magnetostatic analysis type, 546
magnetostatic results, 2157
manage view settings, 292
manual contact, 1266
mapping -CFD results, 763
Mapping Control
  Distance Based Average weighting, 2389
  Manual enables modification of Advance Features, 2389
  Program Controlled gives best accuracy, 2389
  Shape Function weighting, 2389
  Triangulation weighting, 2389
Mapping Validation objects, 2412
mass flow rate
  description, 1622
mass moment of inertia, 901
material assignment, 1049
  application, 1049
  specification requirements, 1049
material plot, 1051
  application, 1051
material properties - fatigue, 2165
material properties - frequency-based Fatigue, 2165
material property usage in postprocessing, 1980
material type results, 2002
maximum data points to plot - fatigue simulations , 2176
maximum equivalent stress safety tool result
  description, 2059
maximum principal stress/elastic strain result
  description, 2028
maximum shear stress safety tool result
  description, 2061
maximum shear stress/elastic strain result
  description, 2029
maximum tensile stress safety tool result
  description, 2064
mean stress theory - fatigue simulations, 2176
Mechanical APDL application memory options, 1804
memory options - the Mechanical APDL application setting for a solution, 1804
merging contact regions - procedure, 1274
mesh control tools
  applying - overall analysis step, 348
mesh nodes
  result contour, 273
mesh numbering, 1035
mesh selection
  element, 276
  element face, 276
mesh selection - elements
  element information, 278
mesh selection - nodes, 269
messages window, 200
middle principal stress/elastic strain result
  description, 2028
minimum principal stress/elastic strain result
  description, 2028
modal analysis
  troubleshooting, 2379
modal analysis type, 515
model
  pan, 280
  rotate, 280
  triad, 280
  zoom, 280
model movement - manipulation, 280
Mohr-Coulomb stress safety tool result
  description, 2062
moment
  description, 1574
moment of inertia, 901
motion load
  description, 1645
  solving with inertia relief, 1645
mpc equations were not built for one or more contact regions - troubleshooting , 2368
multi-point constraint contact for fracture, 482
multibody parts, 850
multiple versions of CAD systems, 2352

N

named selection
  exporting, 1085
named selections
  converting to Mechanical APDL application components, 1087
  creating, 1057
  creating though promotion, 1077
  criteria rules, 1061

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

defining, 1060
display, 1078
including in program controlled inflation, 1084
merging, 1086
overview, 1057
protected, 1084
scoping analysis objects to, 1083
sending to solver, 1084
worksheet, 1074
Named Selections
importing, 1085
managing, 1075
ribbon, 1075
new section plane, 294
Newton-Raphson residuals, 1833
nodal coordinate systems results, 1935
nodal displacement
description, 1728
nodal force
description, 1721
nodal orientation
description, 1719
nodal pressure
description, 1724
nodal rotation
description, 1730
node selection
direct, 279
nonlinear controls analysis settings
rigid dynamic, 1472
static and transient, 1468
transient thermal, 1472
nonlinear solution, 1833
normal elastic strain result
description, 2385
normal stress result
description, 2028
number of processors solution setting, 1804
NX
assigning parameters, 2340

O

object generator, 2343
objects
generating multiple from template object, 2343
tagging, 2349
offsets
surface bodies, 862
other selection scoping in periodic symmetry region, 1181
out of process solutions, 1803
overconstrained conditions
joints, 1369

P

pan, 280
parameters
CAD, 2340
overall description, 2337
parameterizing a variable, 128
restrictions, 2337
specifying, 2337
part transform, 1014
partial solution returned - troubleshooting, 2360
path result
scoping, 2002
path results, 1940
peak result, 1990
perfectly insulated load
description, 1614
periodic high scoping in periodic symmetry region, 1181
periodic low scoping in periodic symmetry region, 1181
periodic region
using, 1181
periodic symmetry type, 1131
periodicity - angular, 1131
phase angle
in current for solid source conductor, 1641
in current for stranded source conductor body, 1643
in voltage for solid source conductor, 1640
pipe idealization
description, 1704
pipe pressure
description, 1532
pipe temperature
description, 1534
planar joints, 1298
plane strain behavior - 2D simulation, 878
plane stress behavior - 2D simulation, 878
Plastic Heating
description, 1715
plastic strain - equivalent, 2032
point mass
description, 901
Polyflow to Mechanical
data transfer, 773
position probe, 2102
postprocessing features, 1844
pressure load
at CFD boundary, 762
preview mesh - overall analysis step, 348

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

priority global connection setting, 1212
probe
  description, 1946
probes
  volume, 2152
procedure
  overall steps in an analysis, 323
psd base excitation
  description, 1589

Q

quick rainflow counting - fatigue simulations , 2176

R

radiation
  description, 1609
radiosity controls, 1446
rainflow counting - fatigue simulations, 2176
random colors
  contact bodies, 1269
  named selections, 1078
random vibration analyses
  considerations for acceleration, 2018
  considerations for deformation, 2018
  considerations for velocity, 2018
random vibration analysis type, 524
reactions
  bolt load, 2107
  contact, 2079
  generalized plane strain, 2107
  overall list, 2107
reduced integration scheme, 857
reference number - coordinate system, 1192
remote boundary conditions, 1735
remote boundary conditions - troubleshooting, 2370
remote displacement support
  description, 1674
remote force
  description, 1552
remote point
  commands objects, 1098
  connection lines, 1098
  features, 1098
  geometry behavior, 1095
  overview, 1089
  promote, 1098
  support specifications, 1097
remote solving, 1803
rename based on definition
  results and result tools, 2283
rename tree objects, 105
renaming a graphical view, 293
renaming contact regions - animated example, 1271
renaming joints based on geometry, 1365
replacing a saved graphical view, 293
report
  creating - overall analysis step, 358
resetting contact regions to defaults - procedure, 1275
response psd tool, 2130
response spectrum analysis type, 533
restart analysis, 1437
restart controls, 1438
result
  coordinate systems, 1974
  legend, 134
result coordinate systems, 1974
result tracker
  thermal, 1852
results
  composite failure, 2072
  display issues, 1996
  electric, 2162
  fracture tool probes, 2127
  gasket, 2131
  geometry represented, 1927
  how to apply, 1927
  introduction and overview, 1927
  line pressure, 2151
  magnetostatic, 2157
  response psd tool, 2130
  reviewing - overall analysis step, 356
  thermal, 2153
  unaveraged contour, 1981
  unconverged results, 1995
  vector plots, 1968
  volume, 2151
results set listing, 1957
Reverse Validation option
  for Mapping Validation, 2412
revolute joints, 1298
ribbon
  overview, 37
rigid bodies, 851
rigid body motion - troubleshooting , 2370
Rigid Dynamics Analysis
  to static structural analysis, 806
rigid dynamics analysis type, 550
Rigid Dynamics analysis type
  commands, 561
Rigid Dynamics Solver
  using commands, 2321
rotate, 280

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

rotating force, 1655
rotation cursor, 280
rotation cursors, 257
rotational acceleration
description, 1516
rotational order of coordinate systems results, 1937
rotational velocity
description, 1511
rotordynamics analysis, 780

S

safety factor
for maximum equivalent stress safety tool result, 2059
for maximum shear stress safety tool result, 2061
for maximum tensile stress safety tool result, 2064
for Mohr-Coulomb stress safety tool result, 2062
safety factor result
[stress] description, 2058
safety margin
for maximum equivalent stress safety tool result, 2059
for maximum shear stress safety tool result, 2061
for maximum tensile stress safety tool result, 2064
for Mohr-Coulomb stress safety tool result, 2062
saving contact region settings - procedure , 1274
scenarios - solving, 1831
scope
graphics, 306
search across global connection setting, 1212
search object names, 119
search the tree, 119
section planes
creating, 294
display options, 303
working with results, 306
selecting
mesh nodes on result contour, 273
selecting direction, 283
selecting element faces on the mesh, 276
selecting elements on the mesh, 276
selecting nodes on the mesh, 269
setting contact conditions manually - guidelines and procedure, 1266
setting variables, 247
shear elastic strain result
description, 2385
shear moment diagram, 2090
shear stress result
description, 2026
shell element results, 2026
show all bodies, 857
show body, 857
show faces, 857
simply supported
Edge: description, 1690
Vertex: description, 1690
simply supported support
description, 1690
simulation wizard - features and types, 321
slot joints, 1298
smart crack growth
fracture analysis, 468
snap to mesh nodes, 883
solid bodies - using, 860
Solid Edge
assigning parameters, 2340
solid source conductor body, 1638
SOLIDWORKS
assigning parameters, 2340
solution
annotations, 306
solving overview, 1791
troubleshooting (convergence problems), 2378
troubleshooting (general), 2372
solution combination
description, 2288
troubleshooting, 2377
solution coordinate system, 1975
solution information
description, 1833
solution magnitude limit exceeded - troubleshooting , 2363
solve process settings, 1804
solver - conflicting DOF constraints
troubleshooting, 2379-2380
solver component results, 2002
solver failure - troubleshooting, 2362
solver pivot checking, 1428
solver type, 1428
solving
overall analysis step, 355
overview, 1791
units, 1874
solving scenarios, 1831
Source Value option
for Mapping Validation, 2412
special analysis topics, 685
specifying surface body layered sections, 868
specifying surface body thickness, 864
spot weld
using, 1407

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

spot welds
assumptions and restrictions, 1407
spring behavior
rigid dynamics analysis settings, 1397
springs
applying springs, 1396
behavior property, 1401
incompatibility, 1404
nonlinear spring stiffness, 1399
pinball region property, 1401
preload, 1400
results, 2123
scoping, 1400
using, 1395
stabilization energy result
description, 2036
standard earth gravity
description, 1509
static structural analysis type, 652
status of variables, 247
steady-state thermal analysis type, 657
steps
details of equilibrium iterations, 1492
details of steps, 1492
details of substeps, 1492
guidelines for integration step size, 1493
overall topics, 1491
stiffness behavior
defining, 851
flexible bodies, 852
gasket bodies, 853
rigid bodies, 852
stiff beam, 856
STL (Stereolithography)
import via construction geometry, 894
strain energy result
description, 2037
strain-life fatigue, 2165
stranded source conductor body, 1642
strength factor - fatigue simulations, 2176
stress intensity result
description, 2029
stress ratio
for maximum equivalent stress safety tool result, 2059
for maximum shear stress safety tool result, 2061
for maximum tensile stress safety tool result, 2064
for Mohr-Coulomb stress safety tool result, 2062
stress tools
how to add, 2058
listed, 2058
stress-life fatigue, 2165
stress/strain results - overall description, 2026
structural error result
description, 2030
submodeling, 807
beam, 807, 813
introduction, 807
structural, 808
thermal, 816
understanding, 807
substructures, 781
support loads - listed, 1663
suppress all other bodies, 857
suppress body, 857
suppress objects, 113
surface bodies
faces with multiple thicknesses and layers specified, 871
general description, 860
importing , 862
importing thickness, 862
offsets, 862
specifying layered sections, 868
specifying thickness, 864
thickness, 861
surface body results, 1993
surface coating
description, 903
surface coating results, 2008
surface results, 1959
scoping, 2002
surface temperatures
at CFD boundary, 763
surfaces transferred as solids - troubleshooting, 2366
symmetric electromechanical symmetry type, 1128
symmetric structural cyclic symmetry type, 1128
symmetric structural symmetry type, 1128
symmetric thermal symmetry type, 1128
symmetry
defining in DesignModeler, 1180
defining in Mechanical, 1181
general axisymmetric, 1167
regions, 1128
using, 1127
symmetry region
using, 1181
symmetry regions
types, 1128
synchronous solutions, 1803

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23

T

tagging objects, 2349
tags
  applying to objects, 2349
  creating, 2349
  deleting, 2349
  filtering, 114
  highlighting tree objects, 2350
  renaming, 2350
Tags window, 2349
temperature
  description, 1598
temperature result
  description, 2153
thermal capacitance, 905
thermal contact results, 2156
thermal fluid flow rate result
  thermal result, 2155
thermal fluid heat conduction rate result
  thermal result, 2155
thermal point mass
  description, 905
thermal results, 2153
  contact results, 2156
thermal steady-state analysis type, 657
thermal strain result, 2032
thermal-electric analysis type, 661
thermal-stress analyses, 836
thermal/structural loads
  importing, 707
thermal/structural results
  exporting, 718
time
  role in analysis settings, 1491
tolerance slider global connection setting, 1212
tolerance type global connection setting, 1212
tolerance value global connection setting, 1212
topics - special analysis, 685
total acceleration result
  description, 2018
total deformation result
  description, 2018
total force electromagnetic result
  description, 2159
total heat flux result
  description, 2153
total magnetic field intensity result
  description, 2158
total magnetic flux density result
  description, 2158
total strain - equivalent, 2035
total velocity result
  description, 2018
transfer
  volumetric temperature, 763
transferring coordinate systems to the Mechanical APDL application, 1201
transient thermal analysis type, 680
translational joints, 1298
transparency for joints, 1365
tree
  filtering, 114
  search, 119
tree objects
  generating from template object, 2343
  grouping, 208
  highlighting tagged objects, 2350
tree outline, 105
troubleshooting
  listing of overall problem situations, 2357
Twin Builder
  Pins, 844

U

unaveraged contour results , 1981
underconstrained parts - troubleshooting, 2370
units - fatigue user life, 2176
units - solving, 1874
universal joints, 1298
unsuppress all bodies, 857
unsuppress body, 857
unsuppress objects, 113
use range global connection setting, 1212
user defined result
  description, 2192
user interactions - details view, 119

V

variable
  parameterizing, 128
  setting, 247
  status, 247
vector heat flux result plots, 2153
vector plot result display, 1968
velocity support
  description, 1680
view results during solve, 1844
virtual topology
  in Mechanical, 349
Viscoelastic Heating
  description, 1717
visibility - analysis settings, 1491

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
Published: 2023-10-23

voltage
description, 1625, 1627, 1633
voltage excitation for solid source conductors
description, 1640
volume probe, 2152
volume result, 2151
von Mises stress result, 2027

W

weak springs analysis setting, 1428
window
geometry, 129
wizards
description and screen location, 320
simulation wizard - features and types, 321
workflows
fracture analysis, 407
working load result in bolt tool
description, 2086
worksheet
connections, 1210
writing and reading files, 1867
LS-DYNA, 1864
Mechanical APDL, 1861

Z

zoom, 280

DRAFT

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Published: 2023-10-23