ANSYS Mechanical介绍及其在工程机械行业的应用

周 乐
上海琨钦

Ansys

全球领先多学科CAE综合服务供应商

专注仿真

仿真是我们的专业。
由全球研发与技术团队支持的领先多物理场仿真产品技术。

广泛信任

全球
40,000
多
客户

ISO
CERTIFIED
9001

市场验证

知名企业

Nasdaq

STANDARD &POOR'S

250亿美元以上市值

全球部署

全球4,800多名员工

40个 国家设有
75个 办事处

规模最大

规模是实力最接近的竞争对手的2倍（收入）

稳定独立

长期财务稳定

CAD/PLM/IoT无关
开放式架构

兑现承诺

全球总体客户满意度达到

83%

推动创新

帮助客户解决新工程挑战：数字化、辅助驾驶、电气化、增材和数字孪生

Ansys 全产品家族：全域覆盖、多学科集成、面向未来

AV Simulation
- AVxcelerate Autonomy
- AVxcelerate Sensor
- AVxcelerate Headlamp

Safety Analysis
- Digital Safety Manager
- medini analyze
- medini analyze for Cybersecurity
- medini analyze for Semiconductors

Embedded Software
- Scade One
- SCADE Suite
- SCADE Display
- SCADE Architect
- SCADE Test

Connect
- Granta MI
- Granta Selector
- Granta MDS
- Granta EduPack
- optiSLang
- Minerva
- ModelCenter

Digital Twin
- Twin Builder

Cloud
- Ansys Cloud Direct
- Ansys Gateway powered by AWS

Electronics
- HFSS
- Maxwell
- Slwave
- Icepak
- ConceptEV
- Motor-CAD
- Q3D Extractor
- EMC Plus
- Charge Plus
- Nuhertz

Mission Engineering
- STK
- ODTK
- DME Component Libraries

Semiconductor
- RedHawk-SC
- RedHawk-SC Electrothermal
- Totem / Totem-SC
- PathFinder-SC
- PowerArtist
- RaptorX / H / Qu
- Exalto
- VeloceRF
- ParagonX
- PrimeX
- RedHawk-SC Security
- Clock FX

3D Design
- Discovery
- SpaceClaim
- DesignModeler

Optics
- Lumerical FDTD
- Lumerical Multiphysics
- Lumerical INTERCONNECT + CML compiler
- Speos
- Speos – NX
- Zemax OpticStudio
- HPC/Accelerators

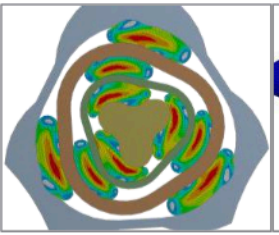
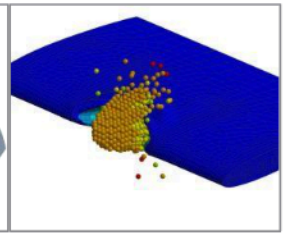
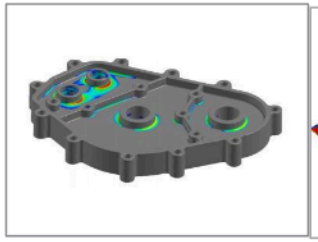
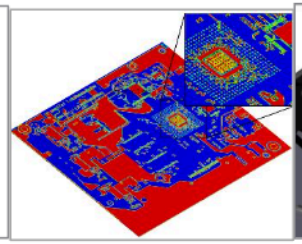
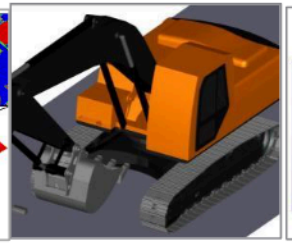
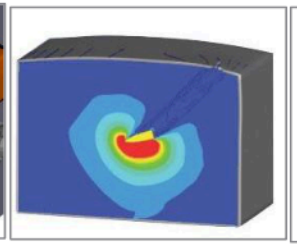
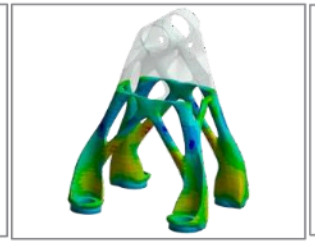
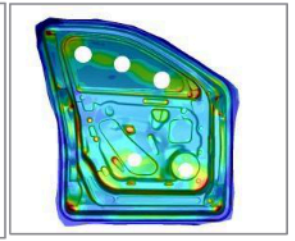
Acoustics Simulation
- Sound

Structures
- Mechanical
- LS-DYNA
- Forming
- Motion
- Sherlock
- Additive Solutions
- nCode DesignLife
- Autodyn
- Sound

Fluids
- Fluent
- CFX
- Chemkin-Pro
- Forte
- FENSAP-ICE
- EnSight
- Turbo Tools
- Polyflow
- Rocky

Ansys结构产品序列

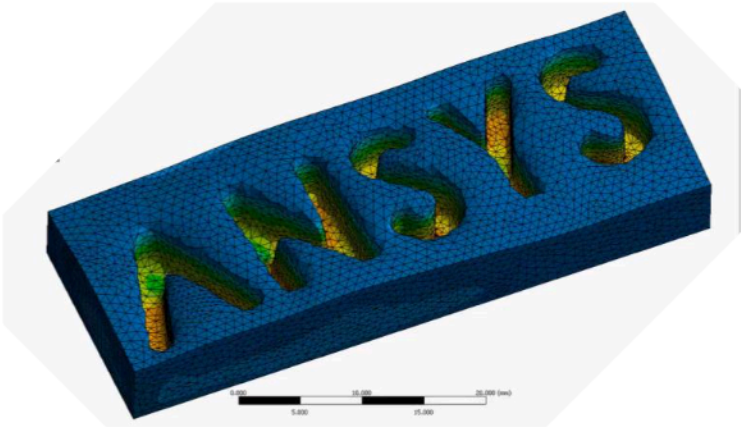
Ansys 结构分析软件使您能够解决复杂的结构工程问题，并做出更好、更快的设计决策。 借助软件中提供的有限元分析 (FEA) 求解器，您可以为您的结构力学问题定制和自动化解决方案，并将其参数化以分析多个设计场景。

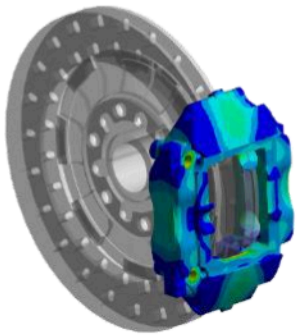
Mechanical	LS-DYNA	nCode DesignLife	Sherlock	Motion	Autodyn	Additive	Formin	Sound
								
一套有限元分析 (FEA) 解决方案，可对结构和耦合场行为进行深入分析，以满足广泛的结构分析需求。	集成到 Mechanical 中的强大显式仿真工具。 大量的功能和材料模型使复杂的仿真模型具有极大的可扩展性。	疲劳预测仿真解决方案，它使用 Ansys Mechanical 和 Ansys LS-DYNA 的有限元分析 (FEA) 来累积重复加载造成的损坏，以确定产品的预测寿命。	电子设计软件，可在早期设计阶段为组件、电路板和系统级别的电子硬件提供快速准确的寿命预测。	用于分析刚体和柔体的多体动力学求解器，能够通过对整个系统的分析来准确评估物理事件。	计算材料对短时间、严重冲击或爆炸载荷的响应。	一个全面且可扩展的软件解决方案，可最大限度地降低 AM 流程的风险，以确保高质量、可认证的零件。包括准备、及打印。	通过端到端工作流程模拟所有金属冲压任务，允许用户在单个平台中执行整个模具过程。	使您能够根据真实录音或声学模拟结果收听、分析和设计声源。

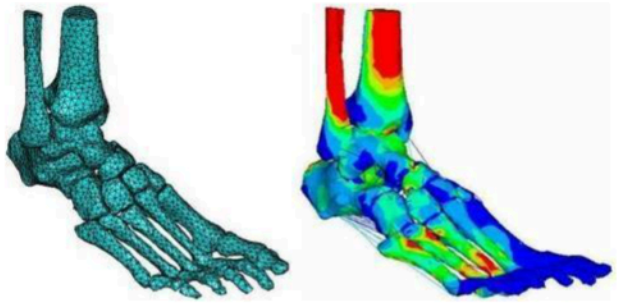
核心产品

Ansys Mechanical 是用于结构、热学、声学仿真的旗舰有限元分析（FEA）工具。

- 从线性静力学到涉及非线性材料的复杂瞬态仿真的应用
- 内置拓扑优化功能，可找到有效满足目标的最佳几何形状
- 与电磁、流体和系统领域的Ansys工具实现强大的双向连接





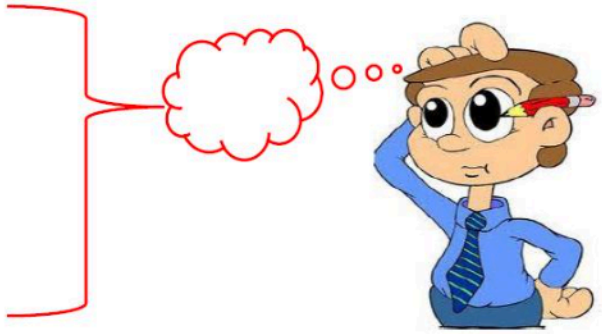


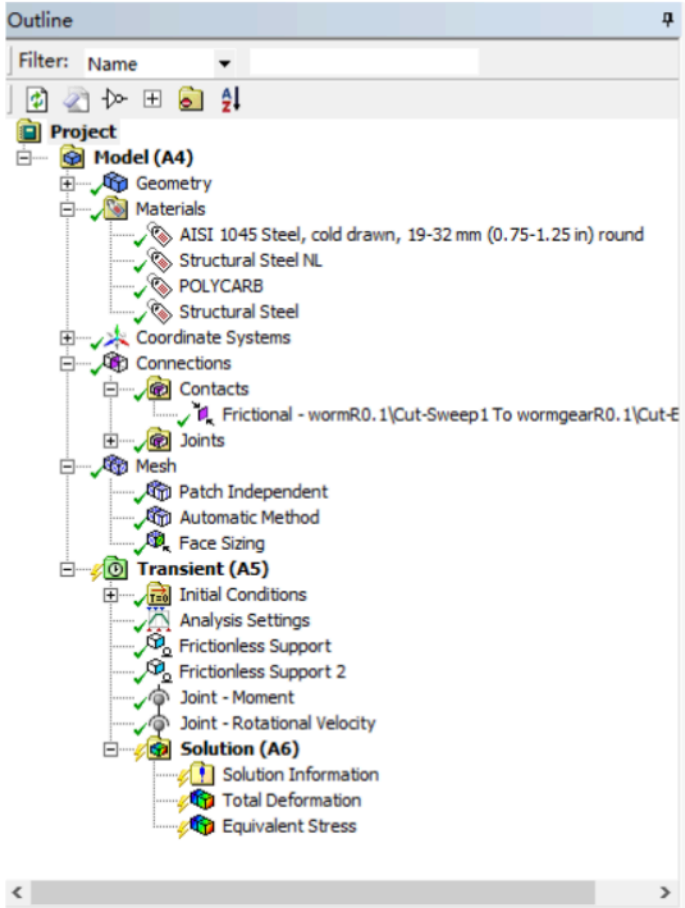
ANSYS Mechanical 基本分析流程

Mechanical 分析步骤

一般有限元分析过程主要包括以下四个步骤：

- 初始分析确定
- 有限元前处理
- 解决方案或者解决流程
- 有限元分析后处理

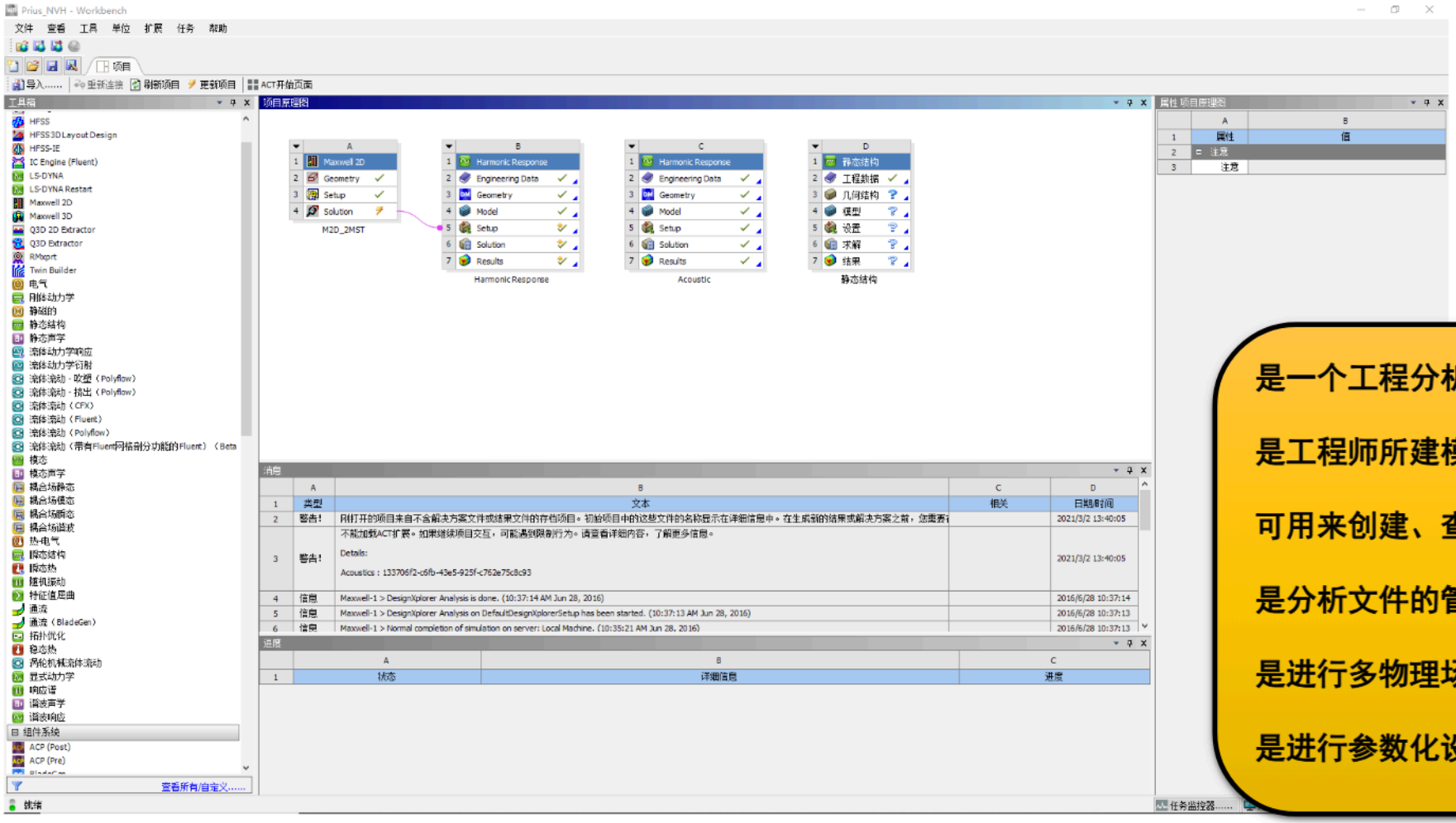




PS. 在Mechanical界面当中，基本的分析流程通常反应在分析树中，通常也称为 Outline Tree。

Mechanical 模块

ANSYS Workbench GUI 多物理场用户界面（具备官方中文版本）



是一个工程分析项目管理工具；

是工程师所建模型与分析的交互接口；

可用来创建、查看您的分析项目；

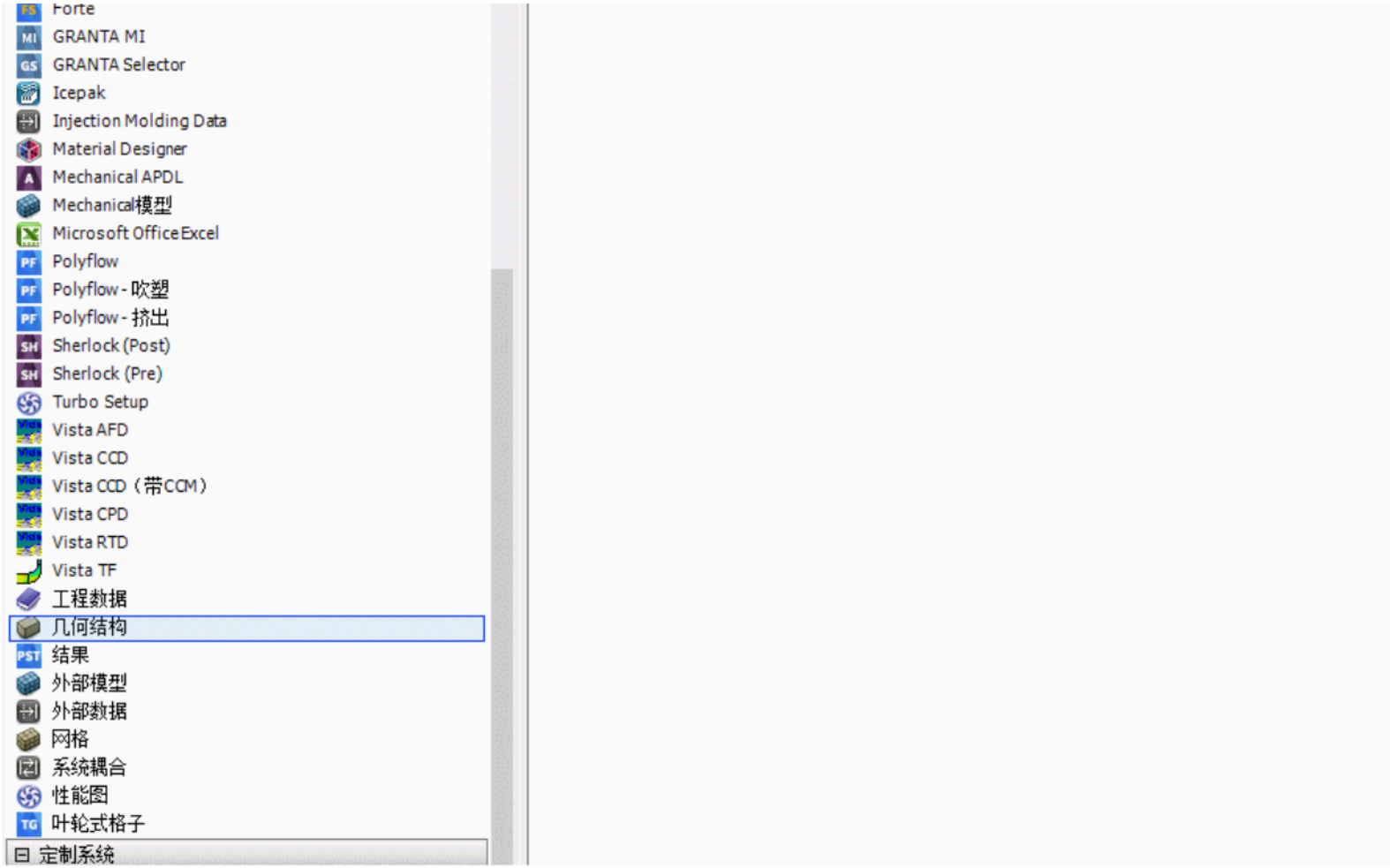
是分析文件的管理器；

是进行多物理场耦合的工具

是进行参数化设计和优化设计的工具。

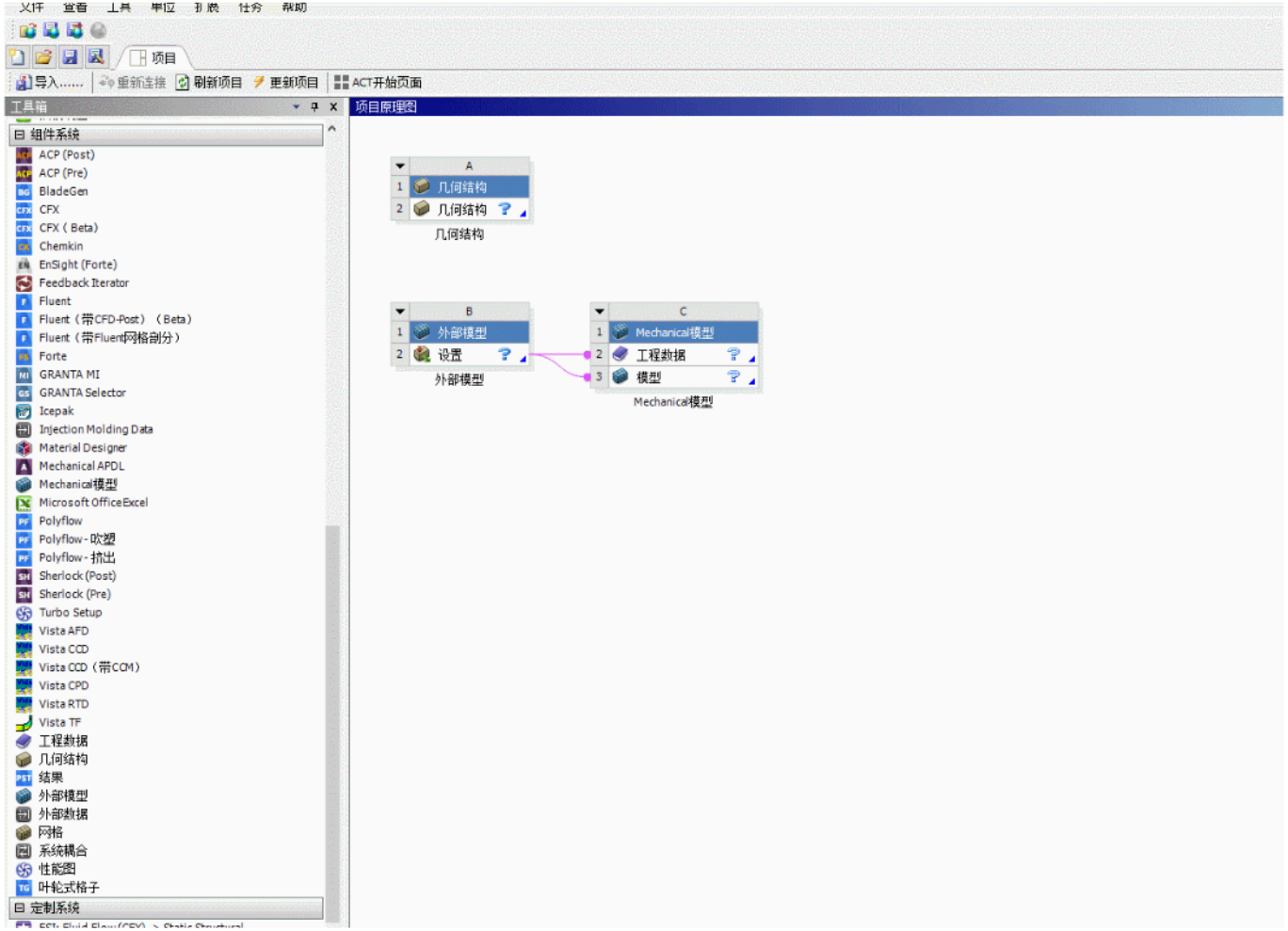
Mechanical 模块

ANSYS Geometry 快速几何模型文件导入



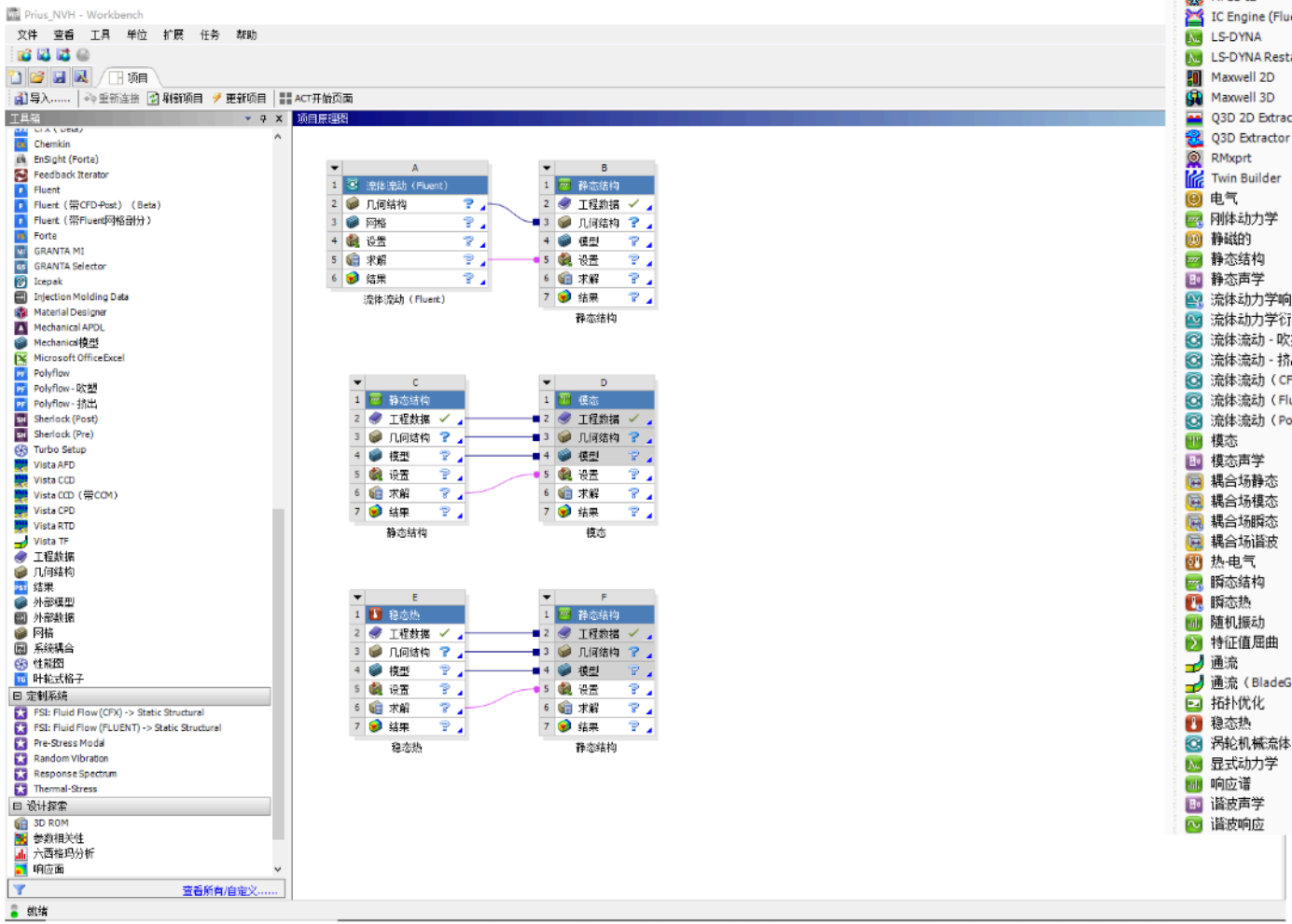
Mechanical 模块

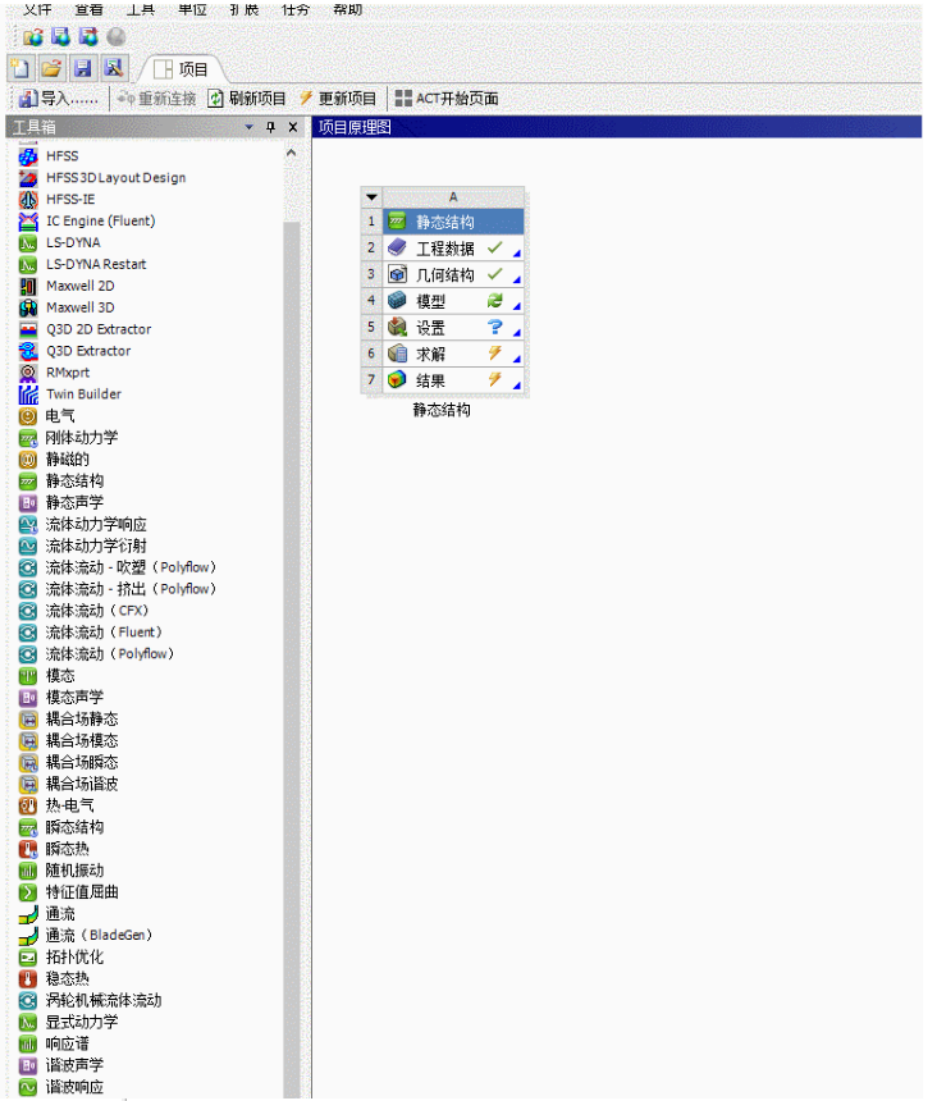
ANSYS External Data 高兼容性外部模型文件导入



Mechanical 模块

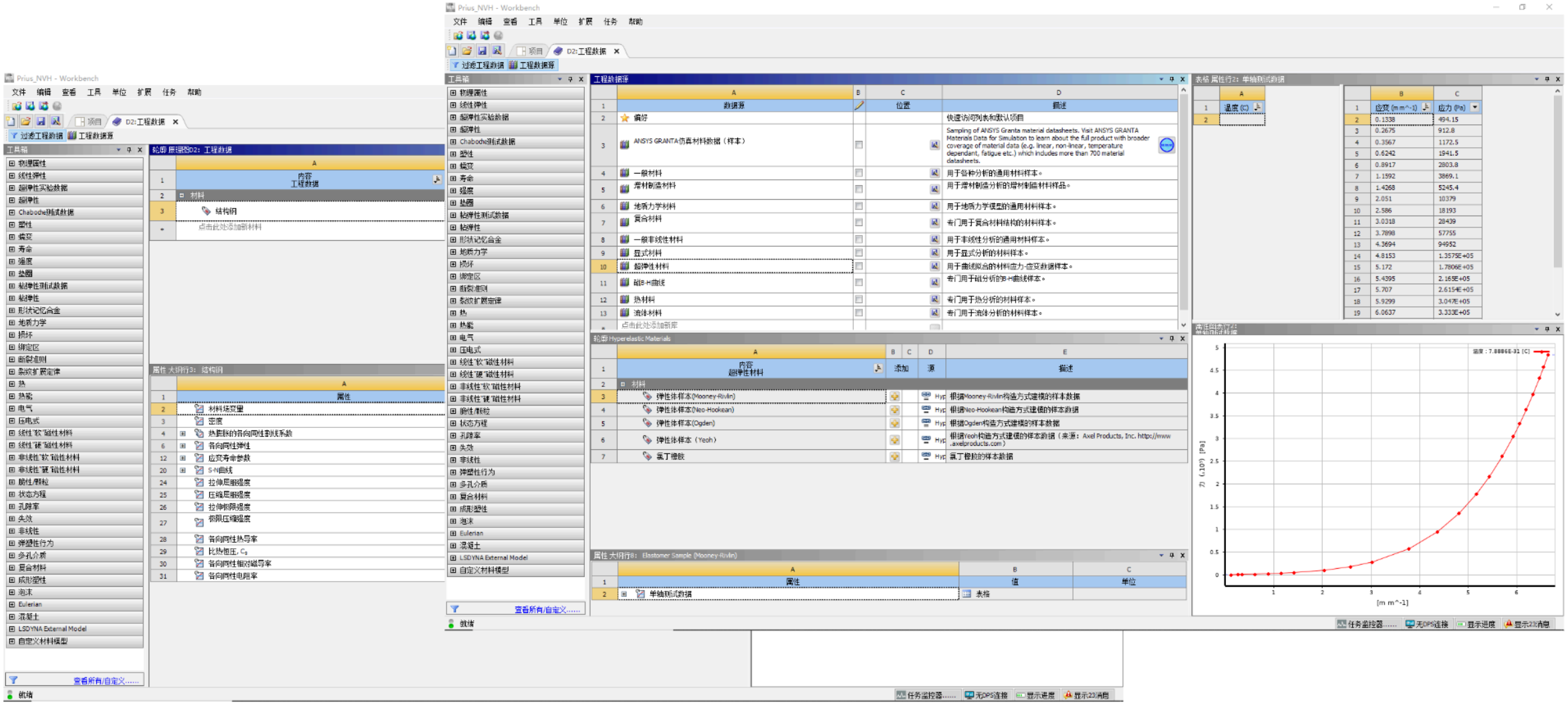
ANSYS Workbench 一键式快速多物理场耦合





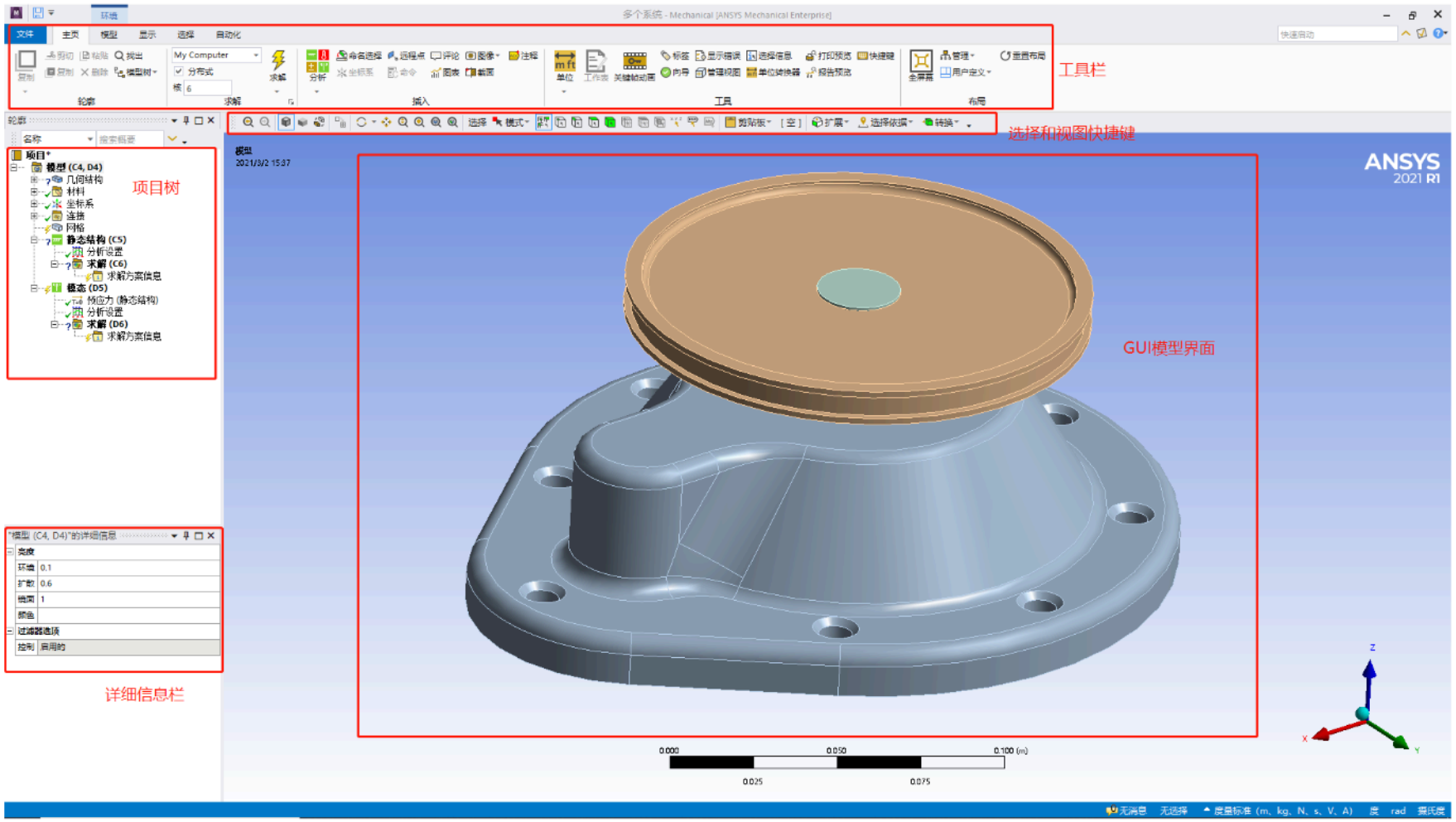
Mechanical 模块

ANSYS Engineering_Data 丰富的工程数据文件库



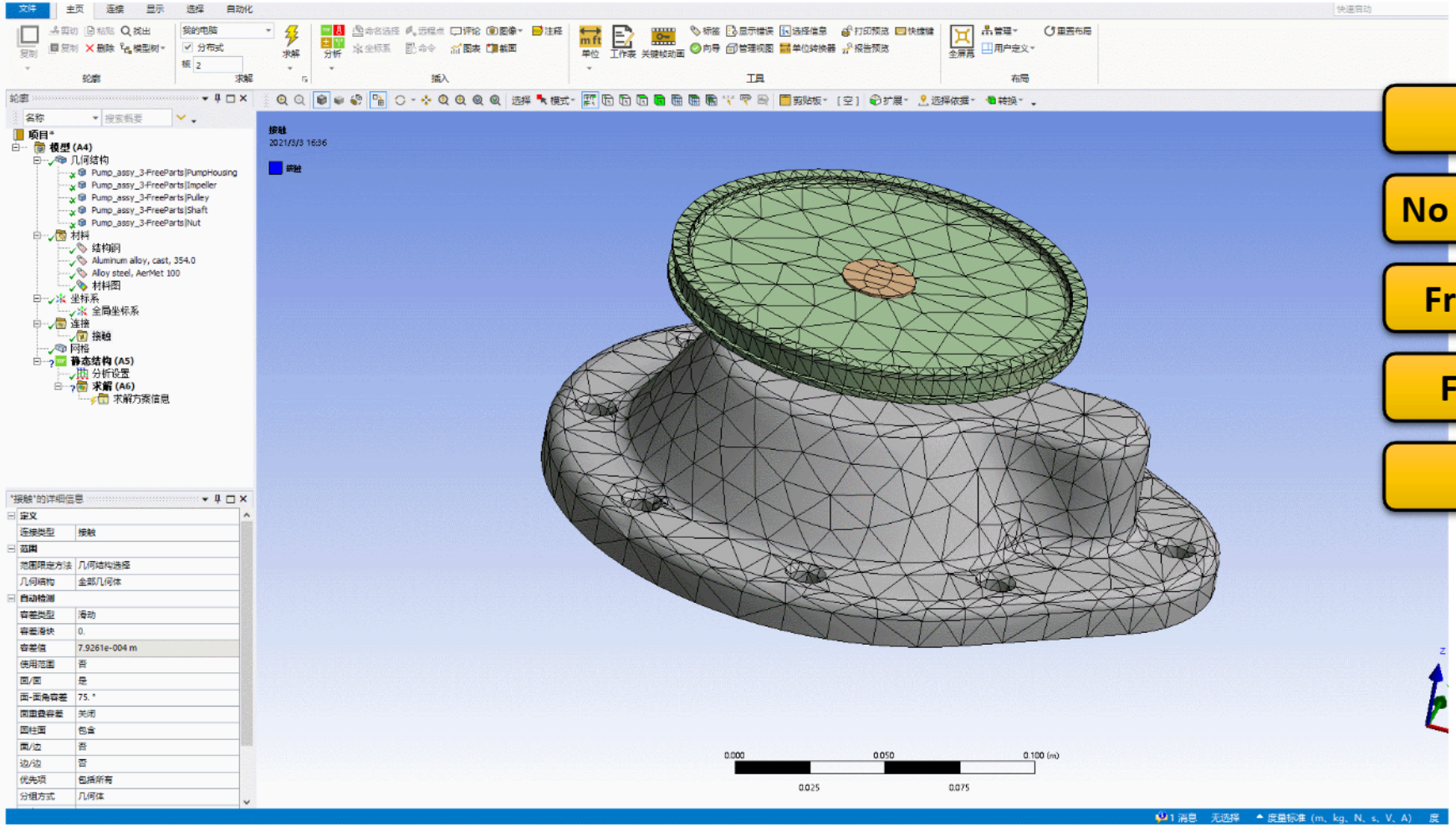
Mechanical 模块

ANSYS Mechanical GUI 可视化图形用户界面（具备官方中文版本）



Mechanical 模块

ANSYS Mechanical ——Connections（自动化接触功能）

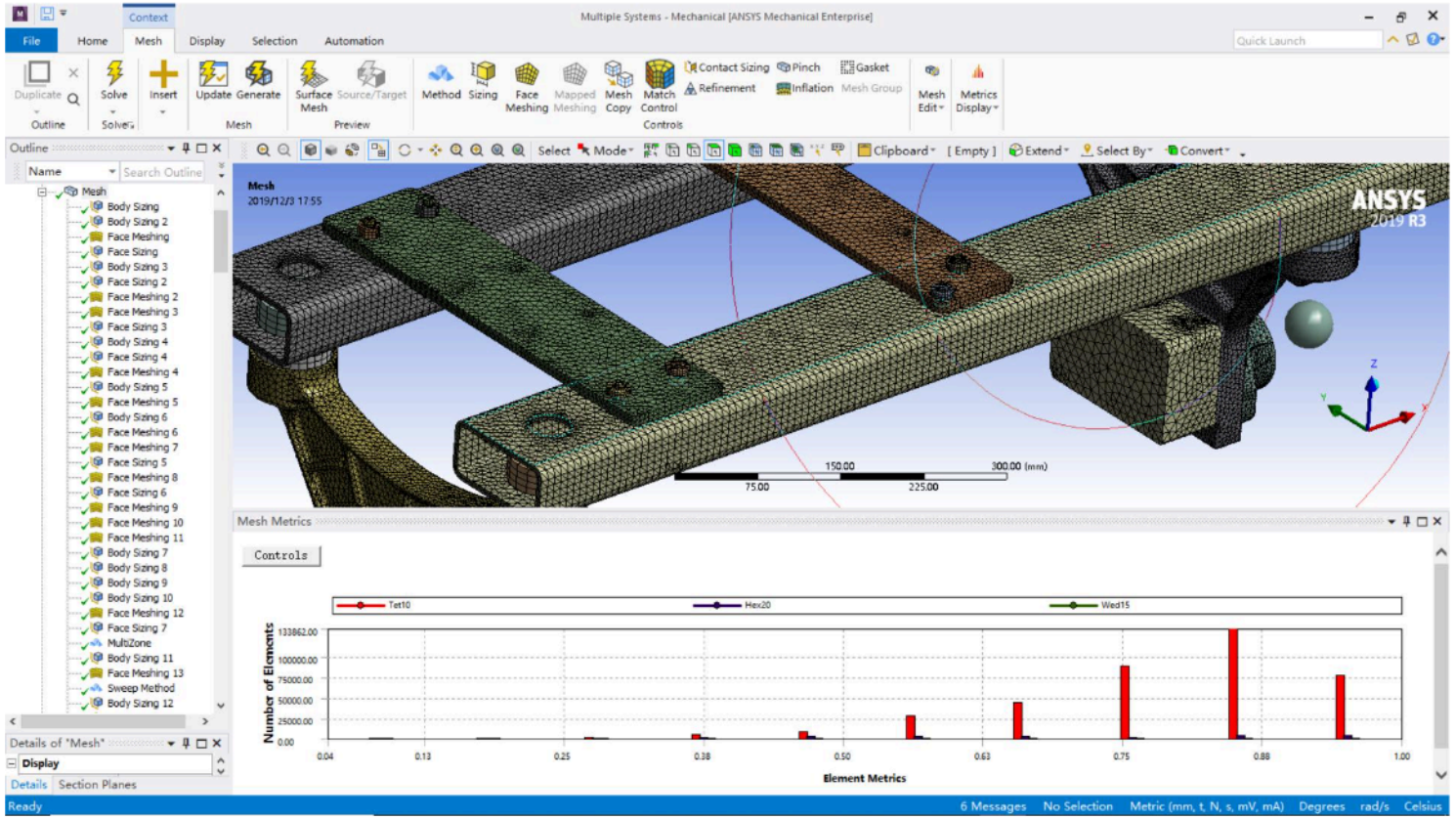


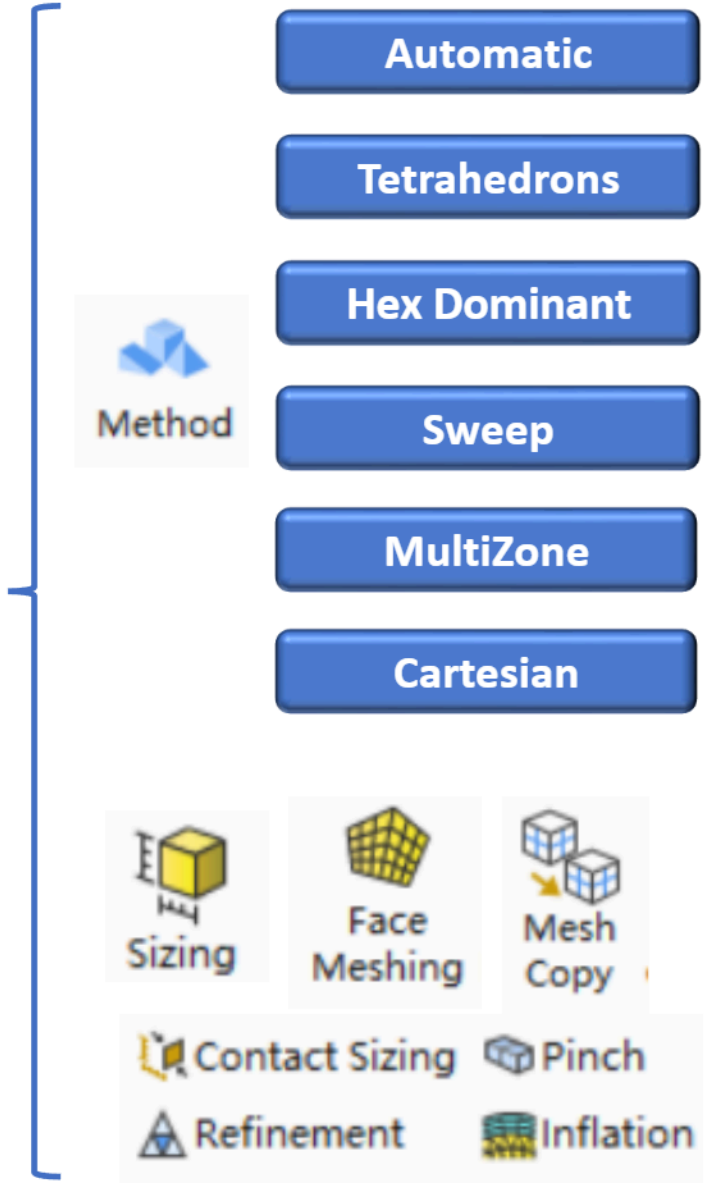
- Bonded（绑定）
- No separation（不分离）
- Frictionless（无摩擦）
- Frictional（有摩擦）
- Rough（粗糙）

Mechanical 模块

ANSYS Mechanical FEA Mesh

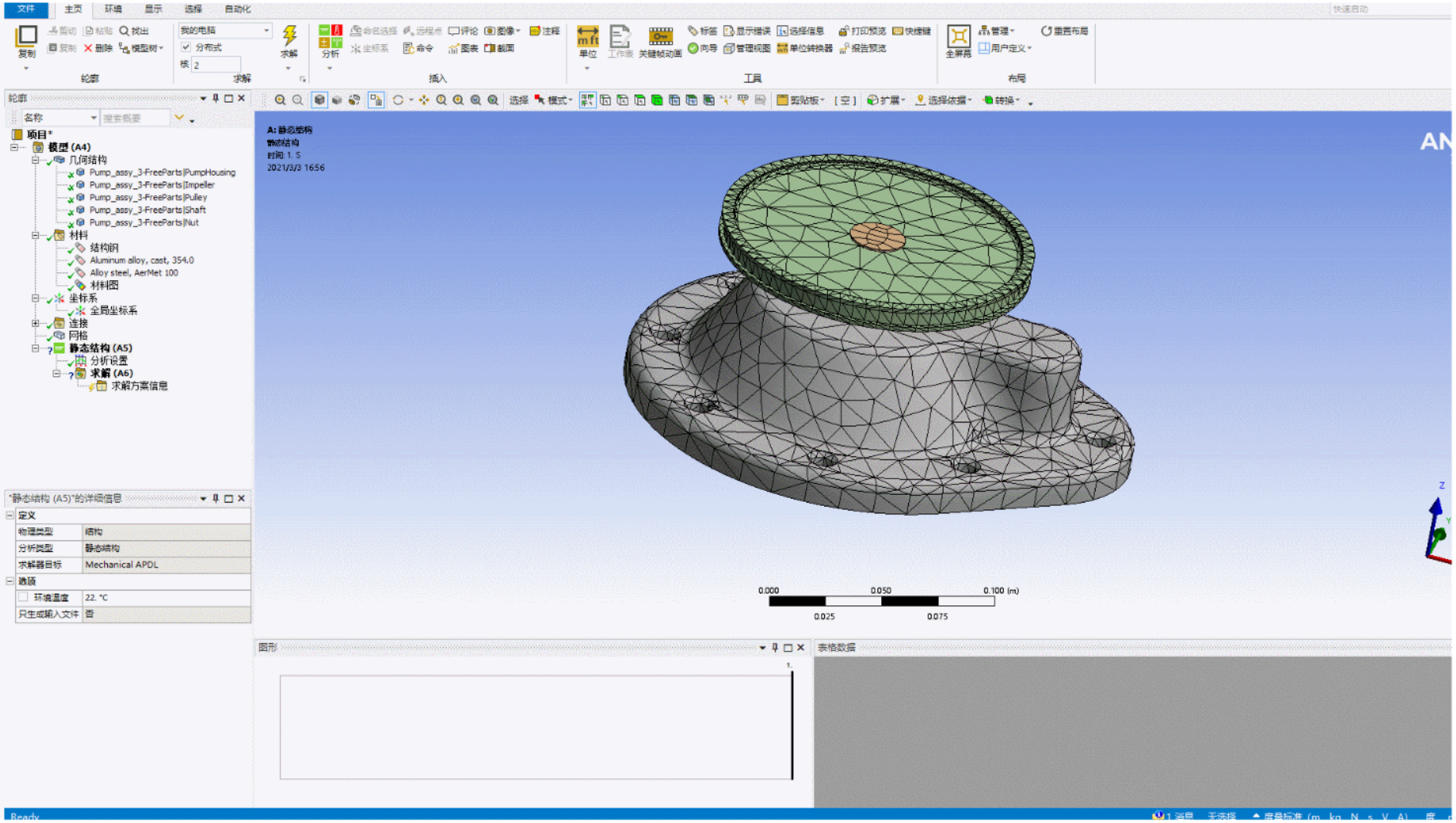
丰富的自动化网格划分功能





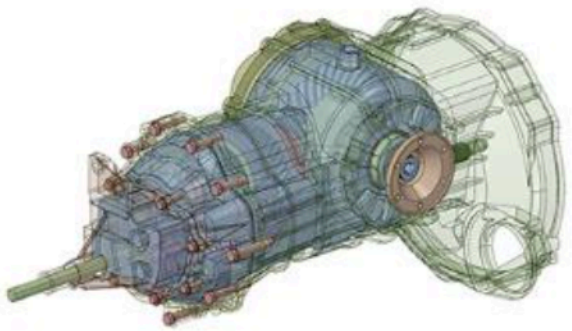
Mechanical 模块

ANSYS Mechanical ——多种边界条件及载荷功能

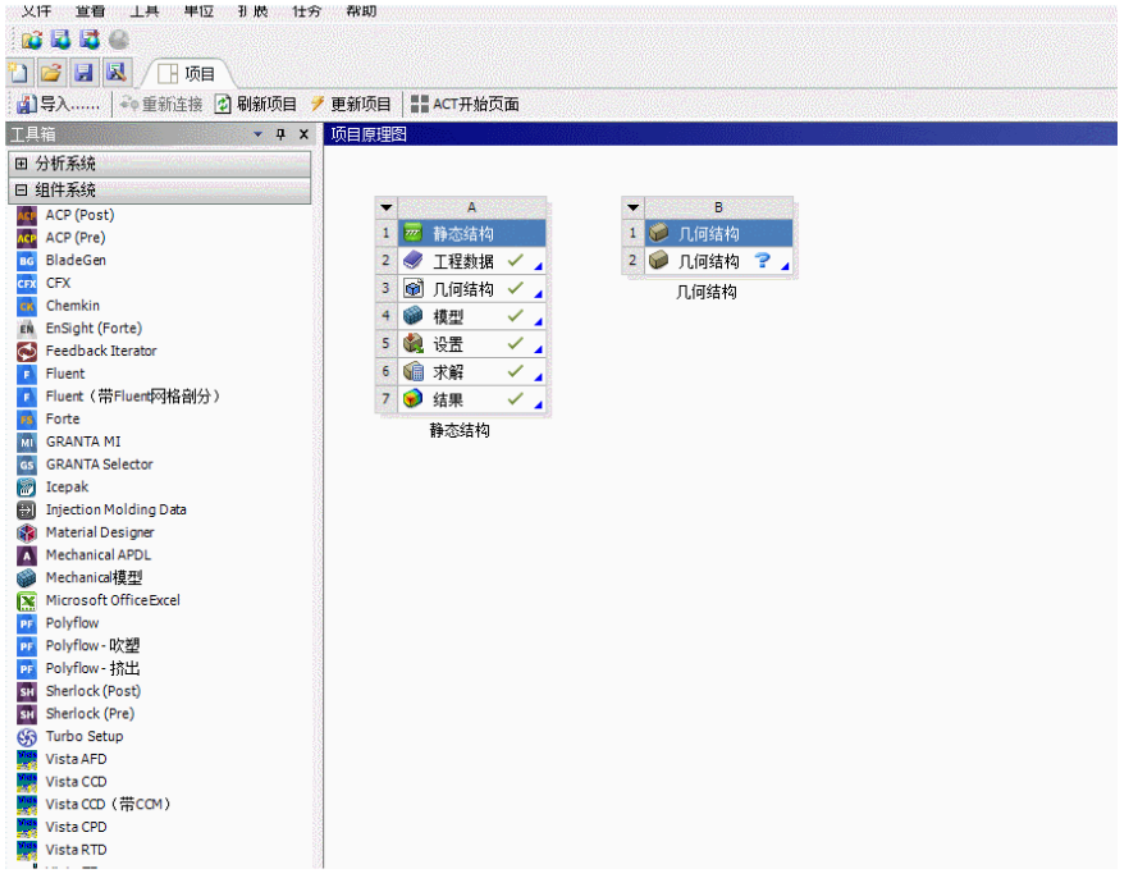


Mechanical 建模与预处理

ANSYS SpaceClaim Direct Modeler



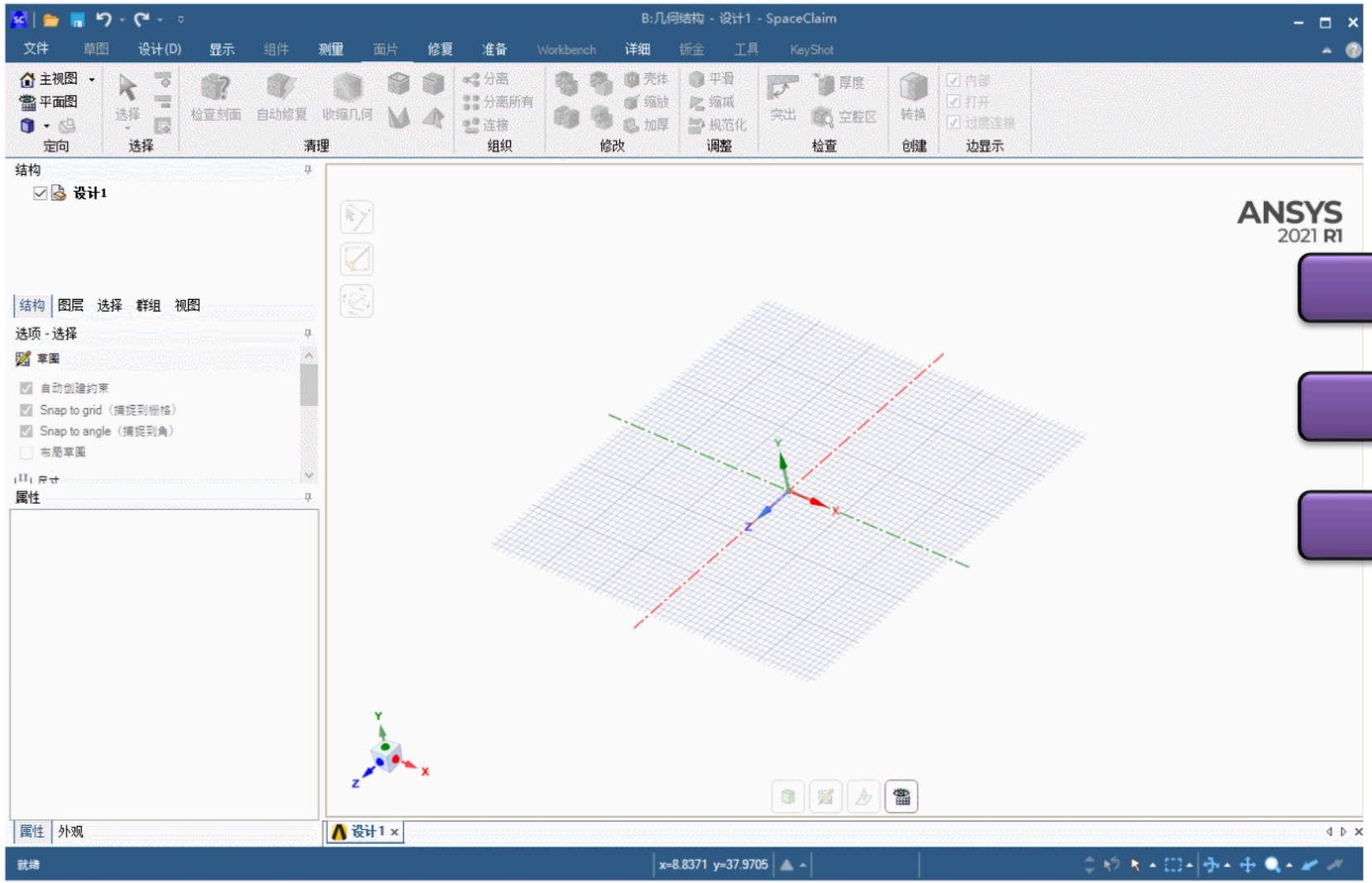
SCDM 2019 R3



Ansys Discovery SpaceClaim 是一款多功能 3D 建模应用程序，能够为常见的建模任务提供高效的解决方案。Discovery SpaceClaim 基 于直接建模技术，能够解决与各项 3D CAD 操作相关的几何问题，例如设计或概念建模、已转化 CAD 文件的修复 、一般模型简化以及完整的模型编辑。Discovery SpaceClaim 简洁易用，能够有效地处理各种大小的建模问题

Mechanical 建模与预处理

ANSYS SpaceClaim Direct Modeler



官方的中文界面

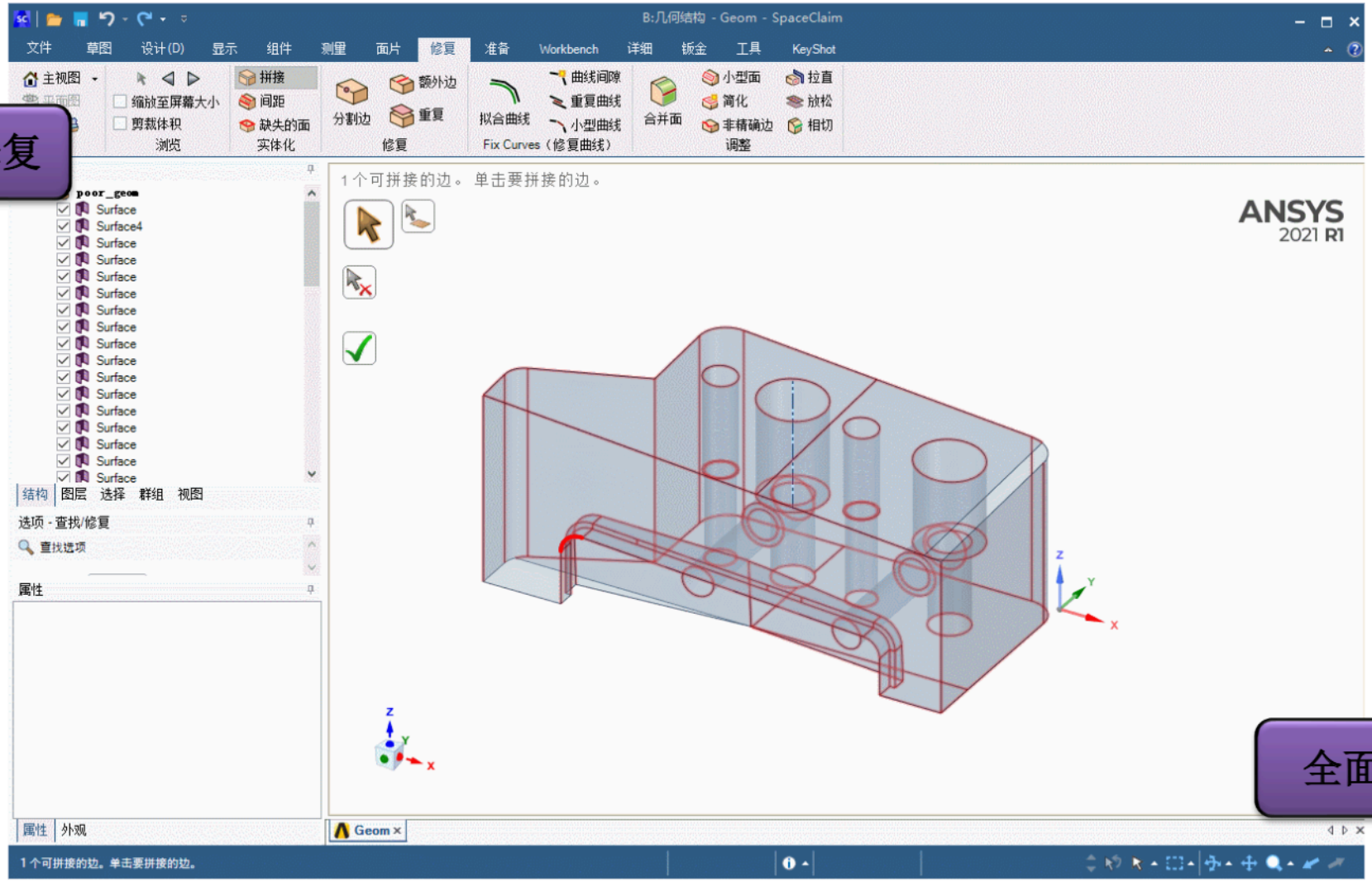
全面的直接建模功能

丰富的几何模型接口

Mechanical 建模与预处理

ANSYS SpaceClaim Direct Modeler

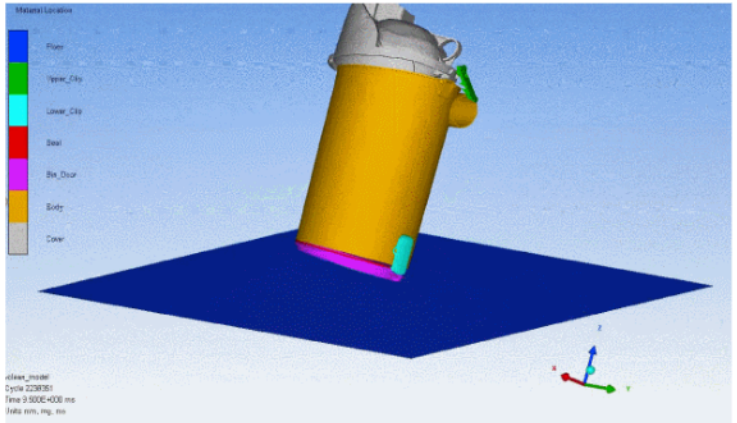
一键式几何缺陷修复

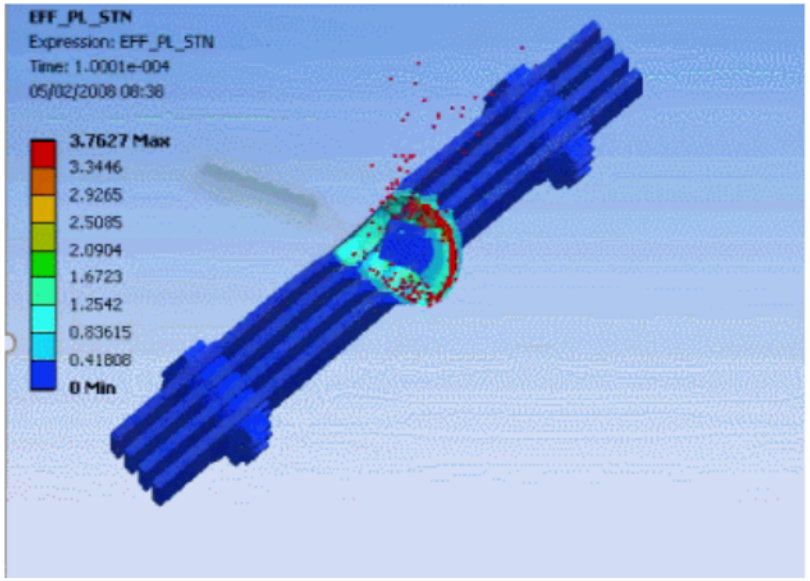


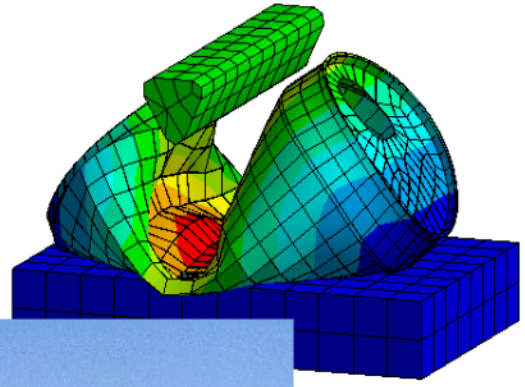
全面的直接建模功能

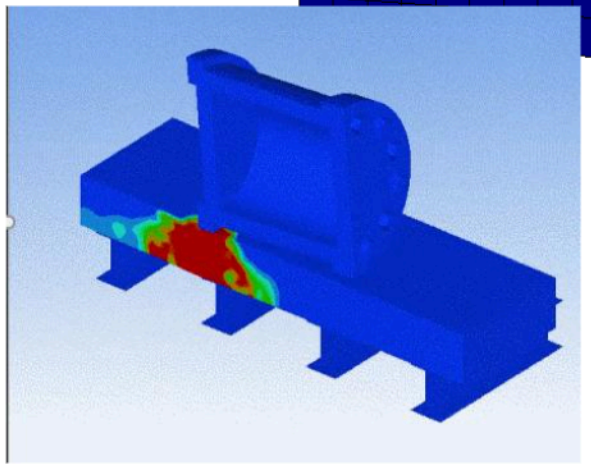
Mechanical 显示动力学/跌落/碰撞

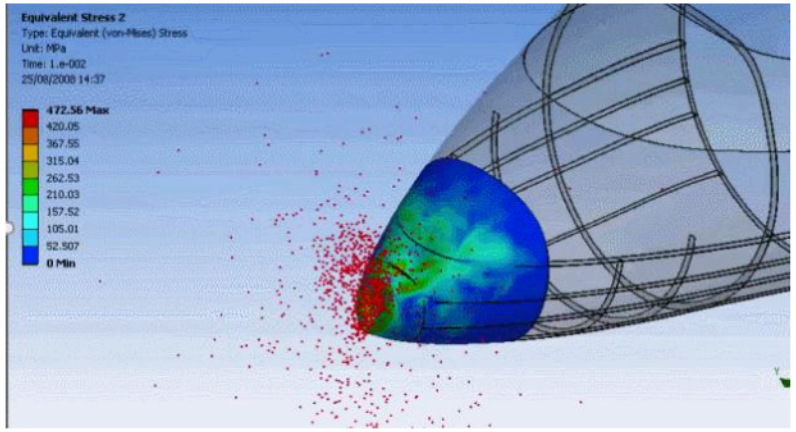
ANSYS Explicit Dynamics & Autodyn









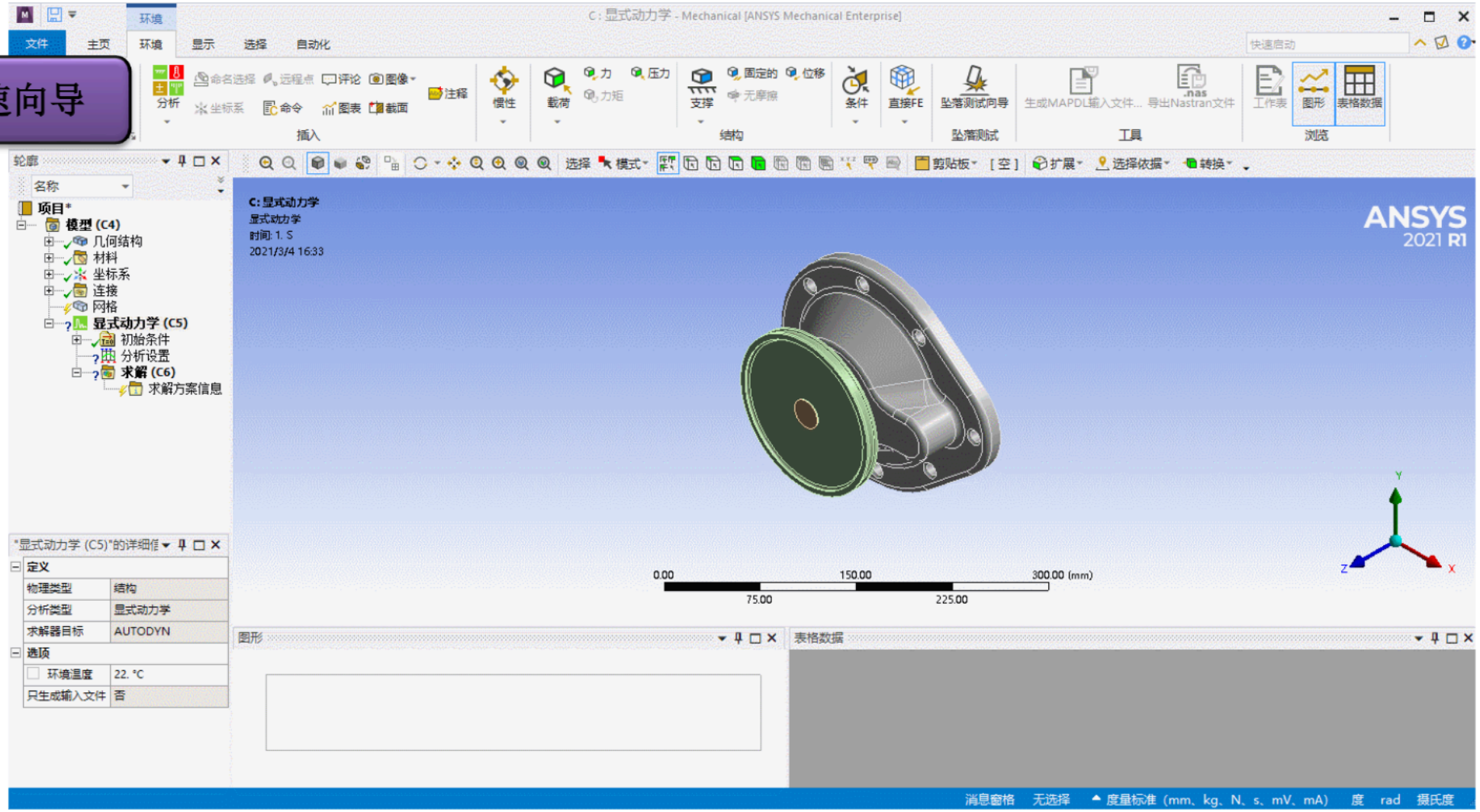


Explicit Dynamics，可对两个或更多个几何体之间的冲击进行建模。这些程序可以计算两个或更多相撞实体之间的力，以及所导致的变形或损坏。Explicit Dynamics通常用于碰撞、跌落等结构大变形或者离散分析。

Mechanical 显示动力学/跌落/碰撞

ANSYS Explicit Dynamics & Autodyn

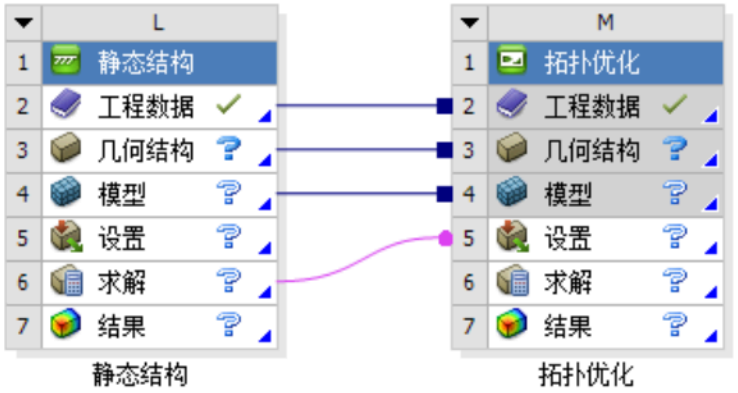
Drop test 快速向导

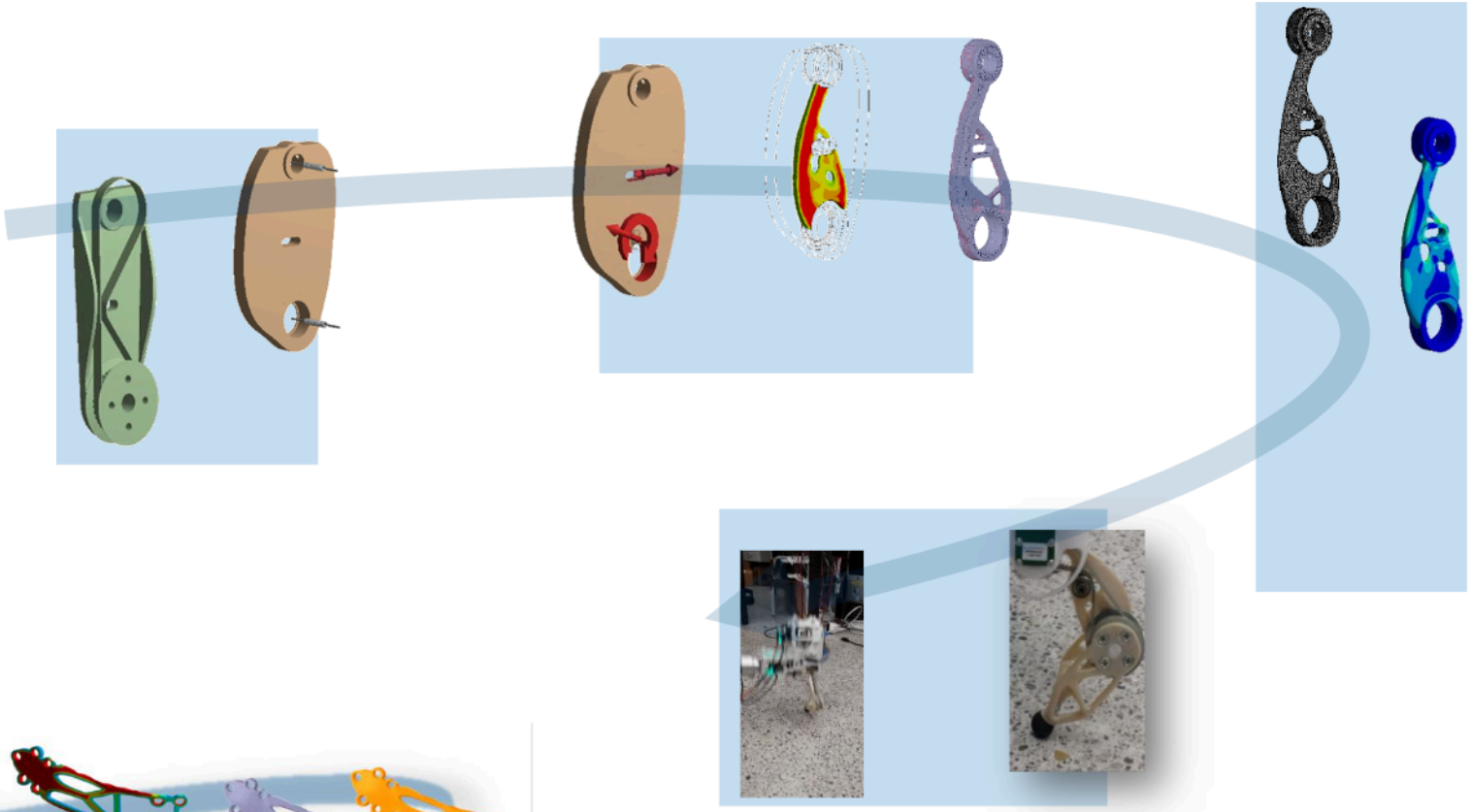


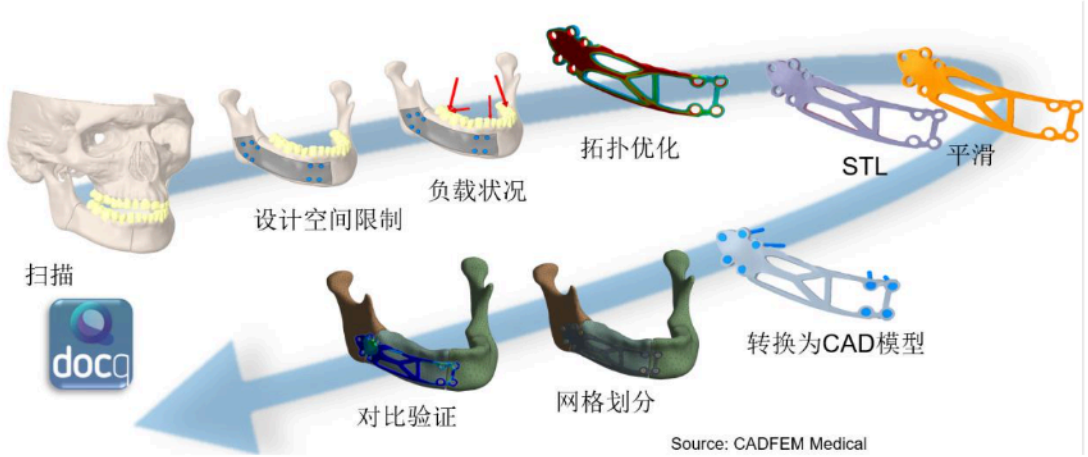
Mechanical 模块

Topology Optimization

拓扑优化



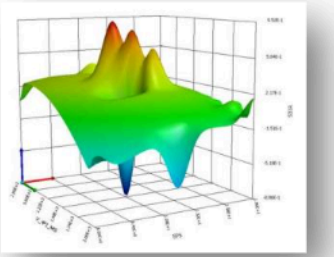




Mechanical 模块

Design Exploration

参数优化



	E
1	Response Surface Optimization
2	Design of Experiments ✓
3	Response Surface ✓
4	Optimization ✓

Minimize Mass

	F
1	Direct Optimization
2	Optimization ✓

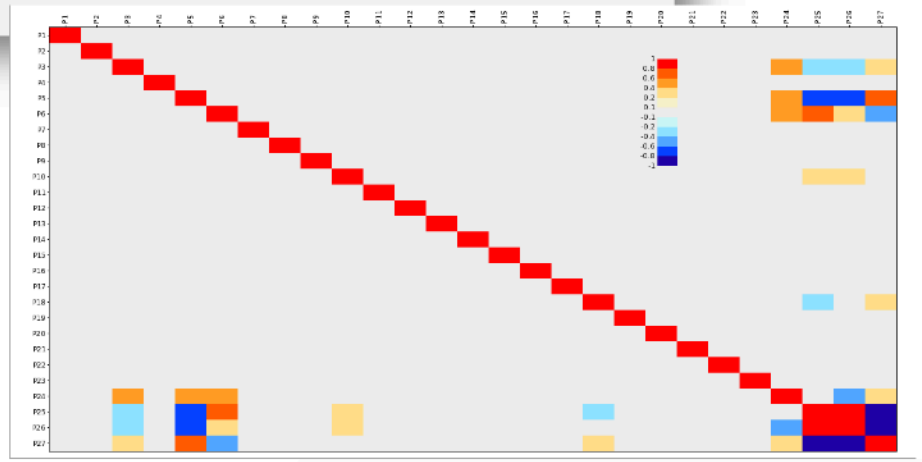
Minimize Stress

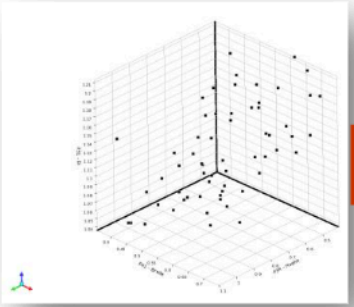
Reconnect | Refresh Project | Update Project | Resume | Update All Design Points | Return to Project | Compact Mode

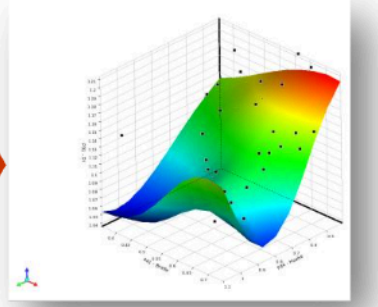
Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - velocity-1	P2 - Face Sizing Element Size	P4 - PipeLength	P3 - Solid Volume	P5 - PressureDrop	Retain
2		m s^-1	m		m^3	Pa	
3	Current	1	0.001	1	3.0844	1.1146E+05	
4	DP 1	2	0.001	1			☐
5	DP 2	1	0.002	2			☑
6	DP 3			2			☑
*							☐

- Copy
- Paste
- Set Update Order by Row
- Show Update Order
- Optimize Update Order
- Delete Design Point
- Copy inputs to Current
- Duplicate Design Point
- Update Selected Design Points
- Export Data (Beta)



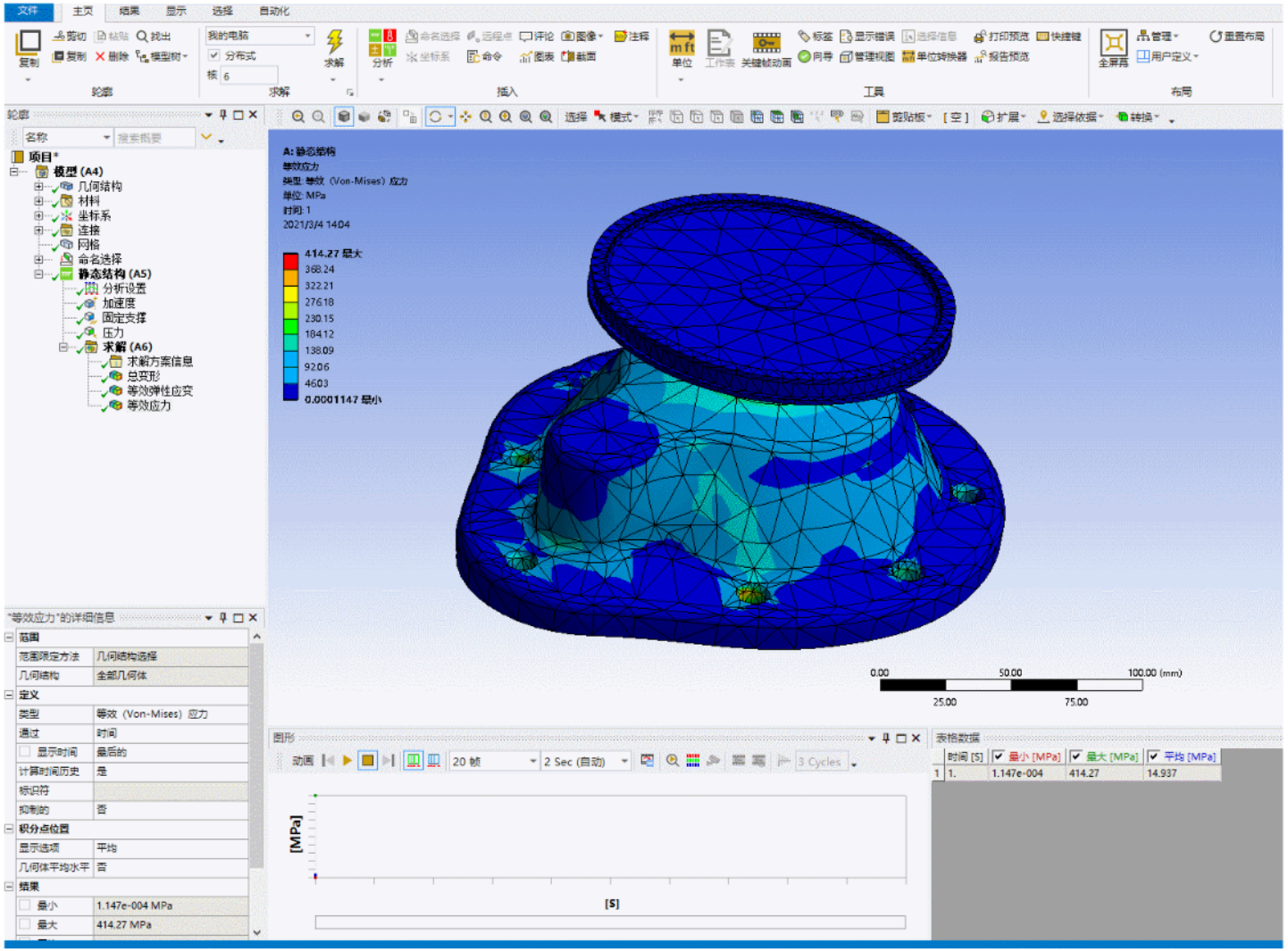




Reference	Name	P8 - WB_DIS (mm)	
		Parameter Value	Variation from Reference
◉	---- Initial Design ----	✕ 79.999	0.00 %
○	---- Initial Design ---- (verified) (DP 0)	✕ 80	0.00 %
○	Candidate Point 1	★★★ 41.449	-48.19 %
○	Candidate Point 1 (verified)	★★★ 41.632	-47.96 %
○	With Rounded Values	★★★ 41.186	-48.52 %
○	With Rounded Values (verified)	★★★ 41.373	-48.28 %
	New Custom Candidate Point		

Mechanical 模块

Post 后处理



- Contact tool（接触工具）
- Fatigue tool（疲劳工具）
- Bolt tool（螺栓工具）
- Probe （探针）
- Linearization tool（线性化工具）

Mechanical 后处理理论

1,第一强度理论：最大拉应力理论，认为当最大拉应力σ1≥ σb 时，材料发生断裂。

2,第二强度理论：最大伸长线应变理论，认为当最大伸长线应变是引起断裂的主要原因，即材料断裂条件为σ1-μ(σ2+03)≥σb(μ 为泊松比)。

3,第三强度理论：最大切应力理论，认为当最大切应力 tmax=(σ1- σ3)/2≥σs/2 时，材料发生屈服，即塑形材料不失效条件为σ1- σ3≤σs。

4,第四强度理论：畸变能密度理论(冯米斯Von-Mises 等效应力理论),认为畸变能密度是引起屈服的主要原因。即塑形材料不失效条件为σVon≤σs。

理论	第一理论	第二理论	第三理论	第四理论
材料力学中名称	最大拉应力理论	最大伸长线应变理论	最大切应力理论	畸变能密度理论
求取对象在Ansys中名称	Maximum Principal	无	1. Maximum Shear 2. Intensity	Equivalent (von-Mises)
适用材料性质	脆性材料	脆性材料(Ansys中无)	塑形材料	塑性材料
在Ansys工程应用中使用频率	脆性常用	无	一般	塑形最常用

结构仿真在工程机械行业的应用

Ansys

挖掘机强度分析

Goal

- Optimize lifting bucket designed by Caterpillar® for increased load capacity

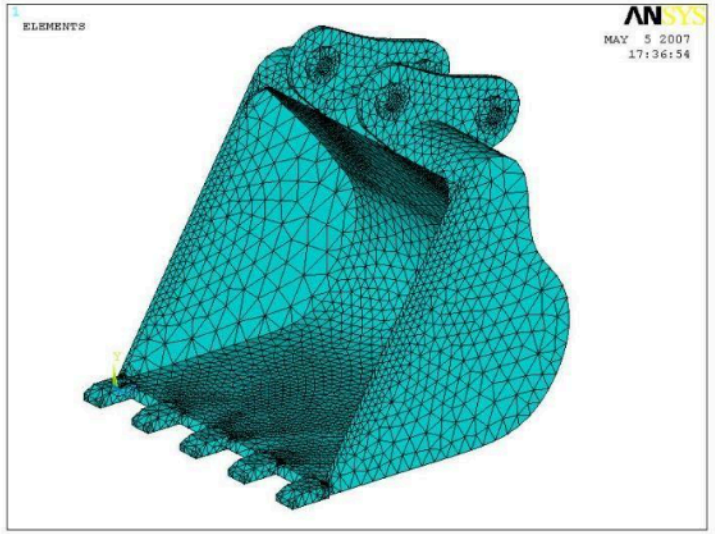
Challenge

Evaluate design for different field loading conditions

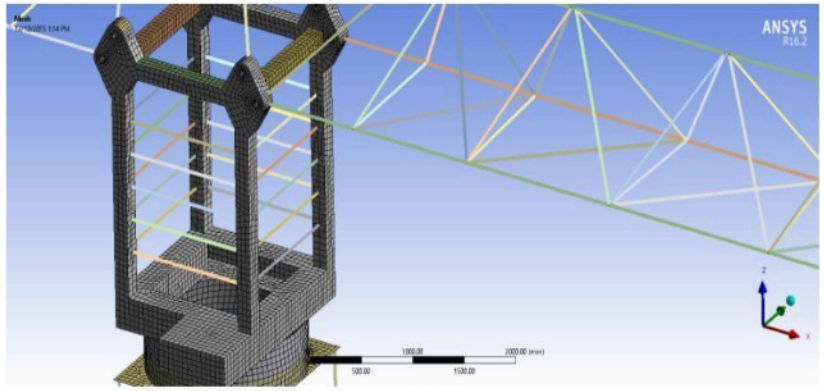
- Maintain a safety factor
- Maintain bucket volume

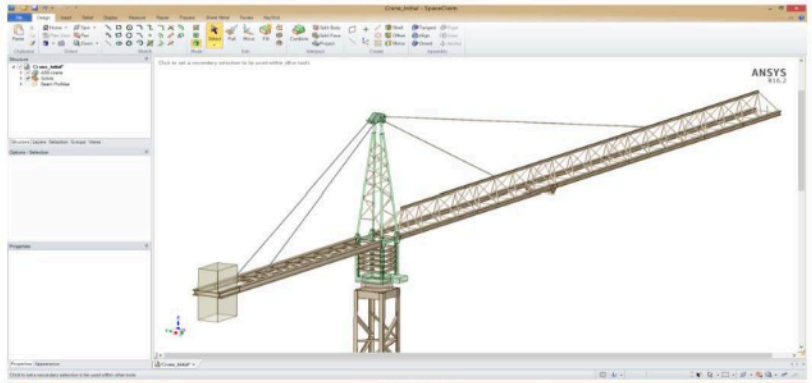
Solution

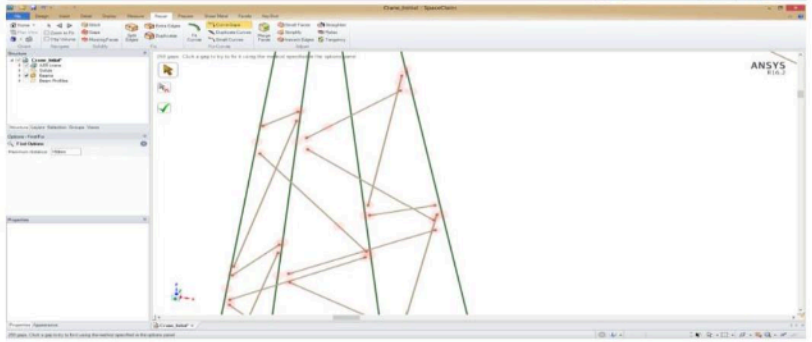
- Model a bucket used on the hydraulic excavators
- Analyse stresses under normal loading conditions

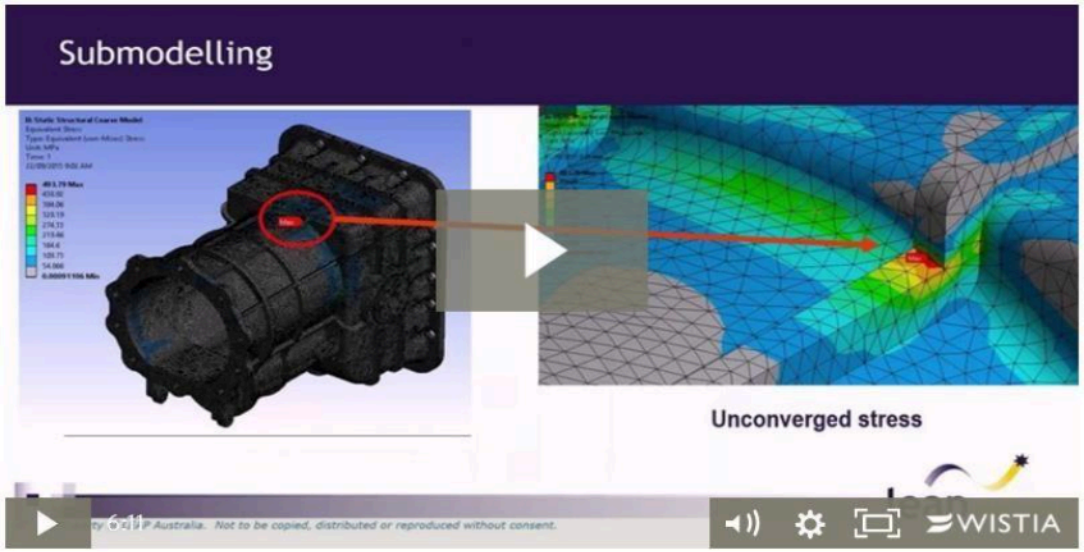


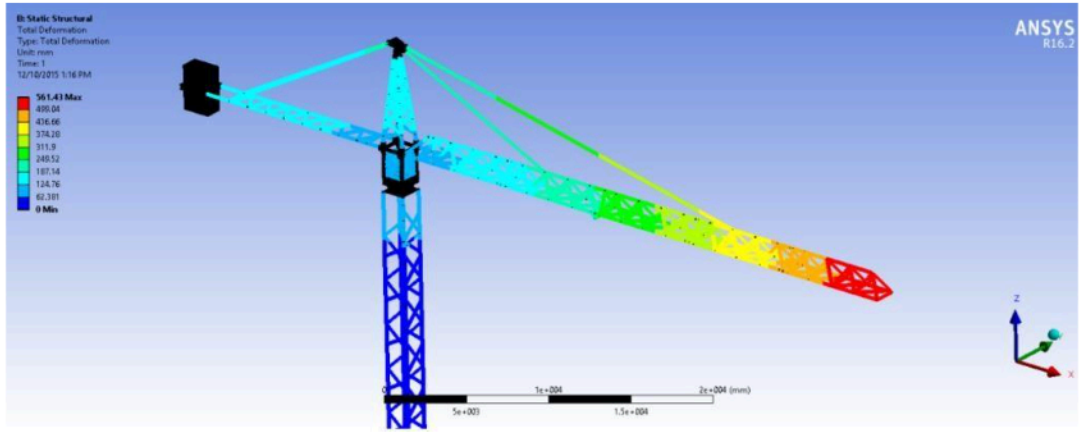
起重机强度分析


ANSYS
R16.2


ANSYS
R16.2

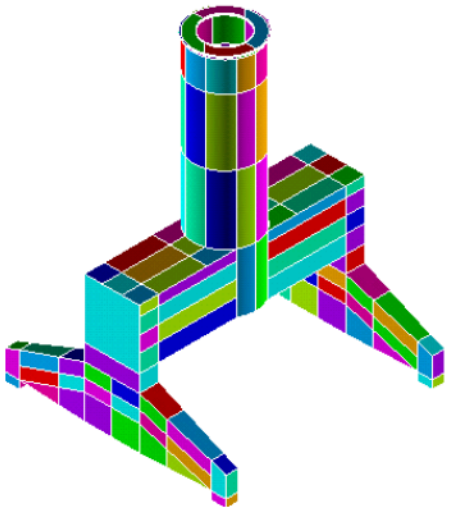

ANSYS

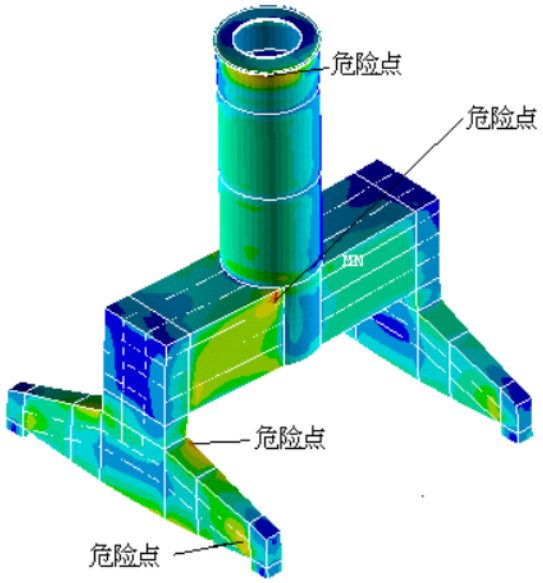

Submodelling
Unconverged stress
WISTIA

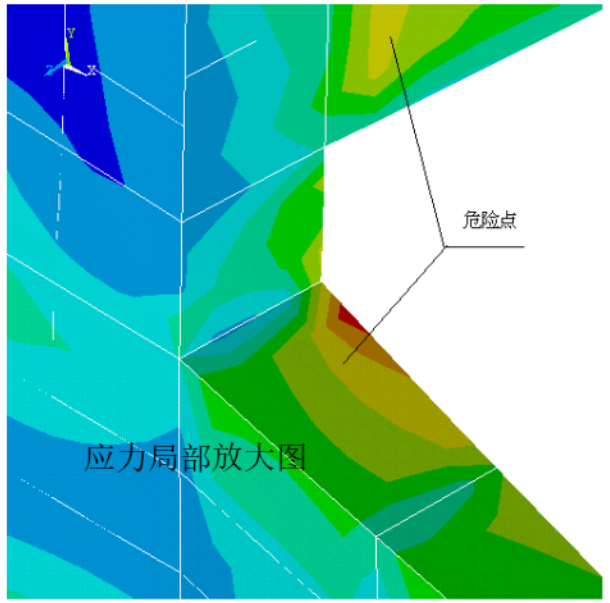

B: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
12/10/2015 1:16 PM
561.43 Max
499.04
436.66
374.28
311.9
249.52
187.14
124.76
62.381
0 Min
ANSYS
R16.2

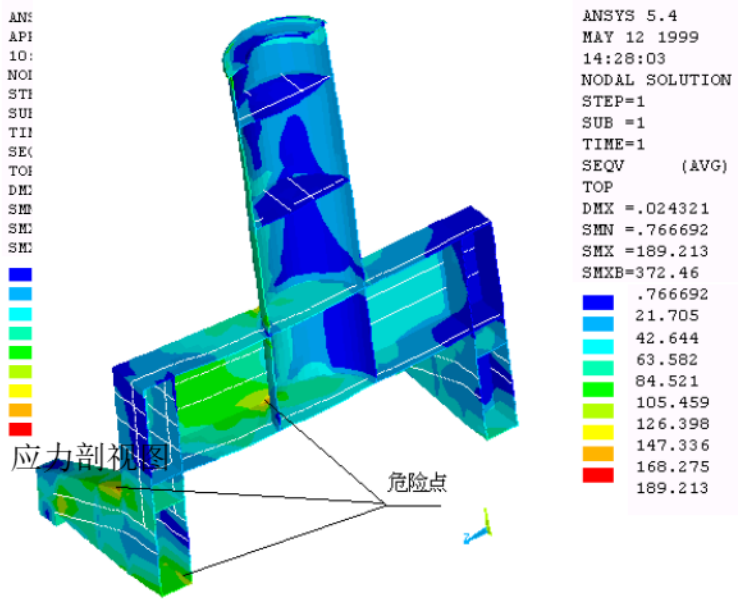
天津港16t门机门架结构

使用过程中产生裂缝
找出各构件的应力分布规律及裂纹敏感区







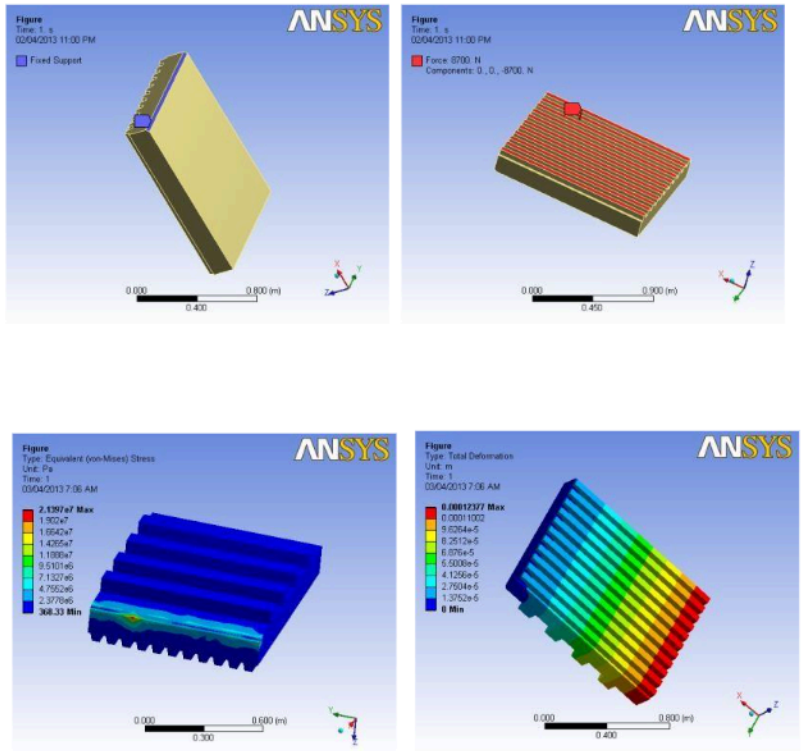


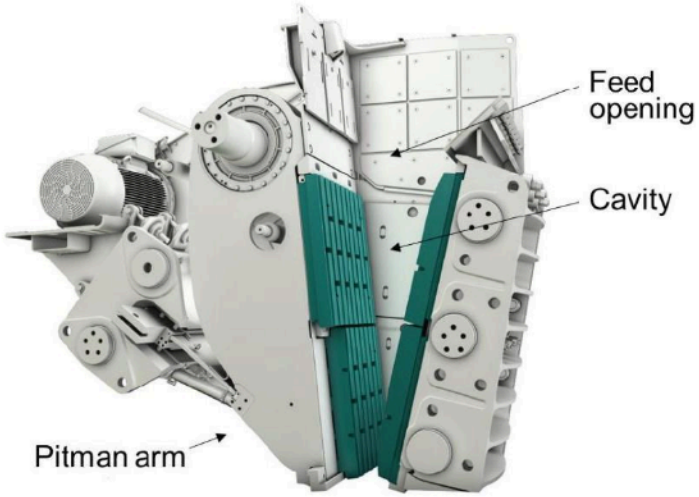
门架结构最大等效应力均在屈服应力230Mpa范围之内，但几个关键部位已超过许用应力[σ]=170Mpa，其位置在端梁上与横梁交界处。

钻机强度分析

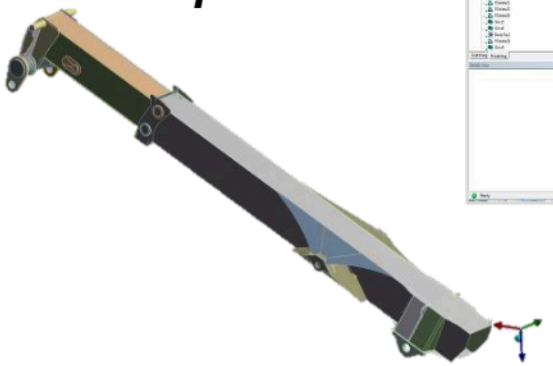
设计新型C130颚式钻机

- 不增加机器重量的情况下提供更大的工作能力
- ANSYS SpaceClaim创建了多种设计方案
- ANSYS Mechanical来分析这些设计
- 新设计——每小时270公吨，（原始设计245公吨）
- ANSYS HPC将每次计算从21小时缩短到1小时。



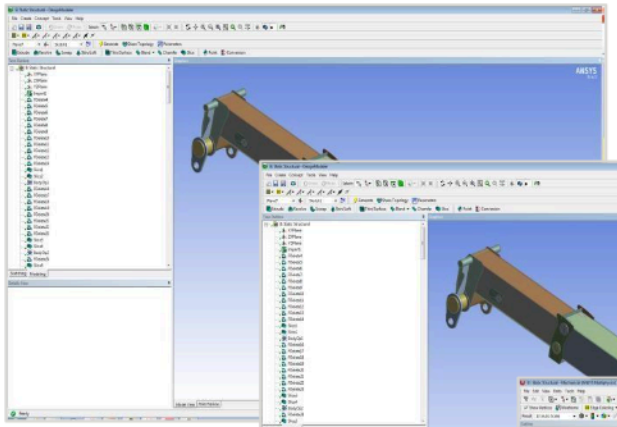


起重臂强度分析

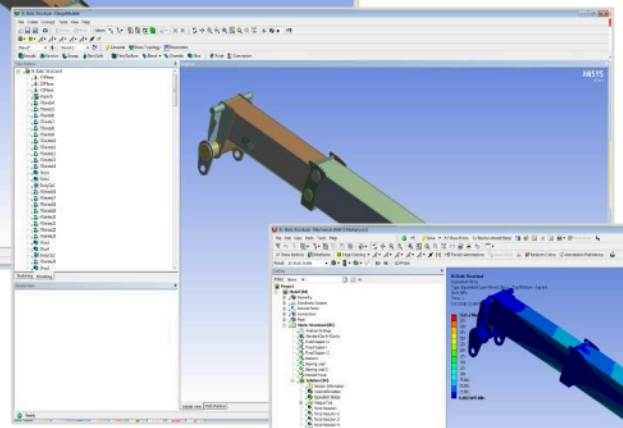


CAD Import

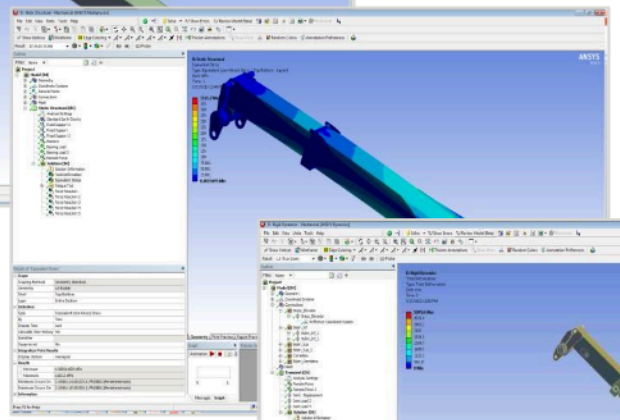
Model Courtesy of UPLIFTING



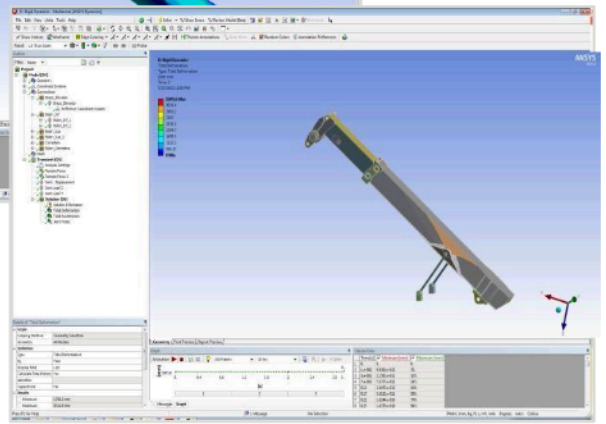
Geometry Preprocessing



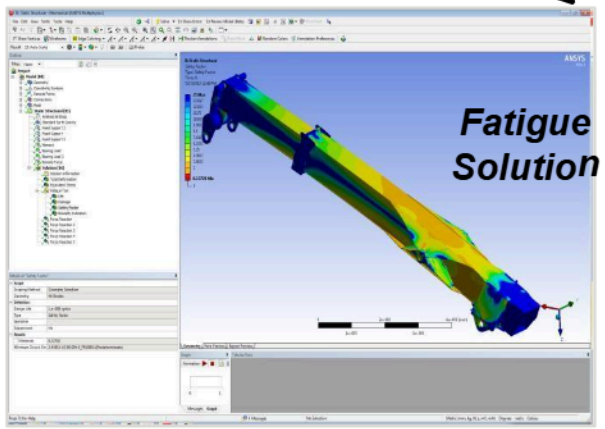
Meshing



Mechanical Solution

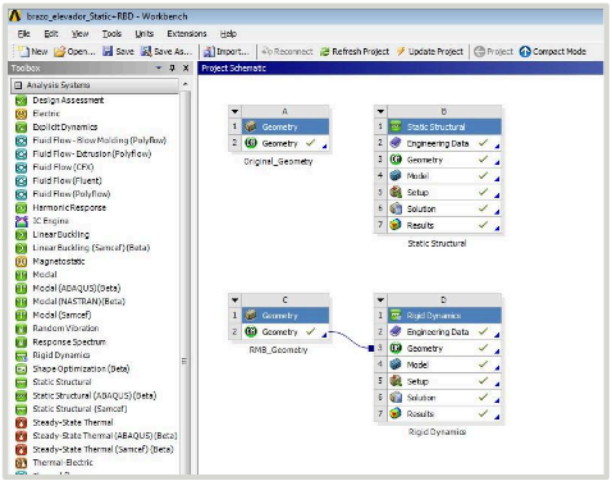


Kinematic Solution

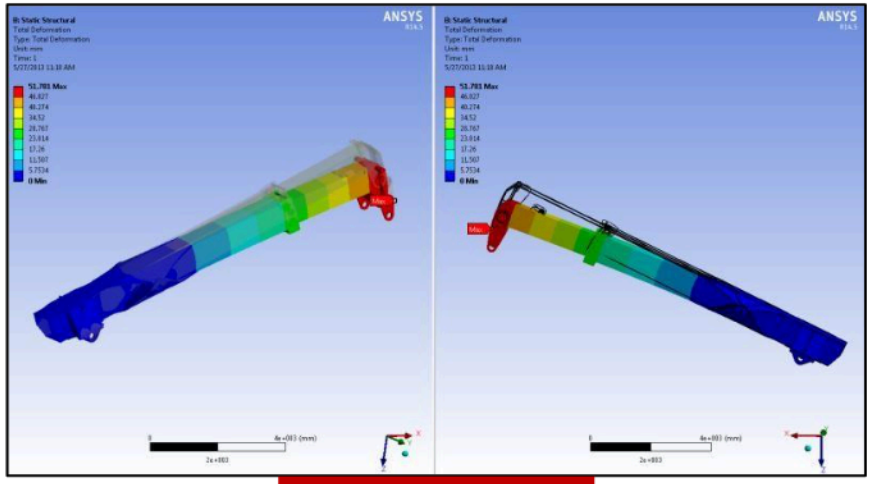


Fatigue Solution

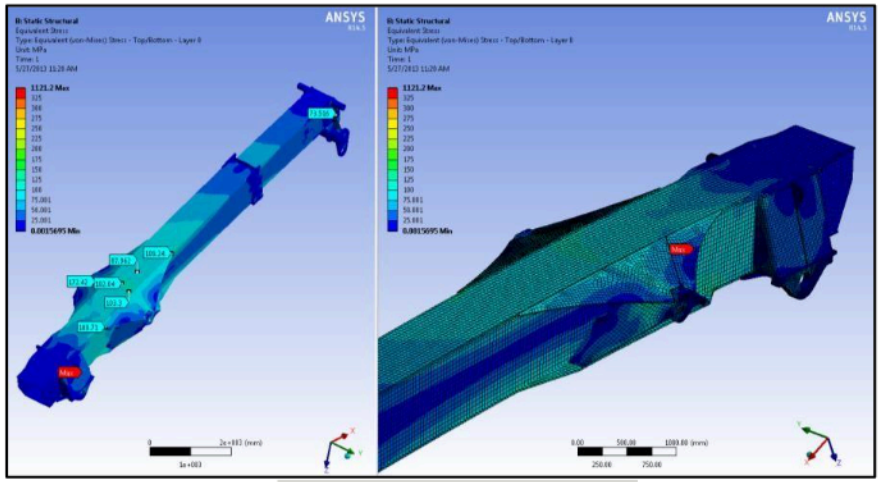
Workbench



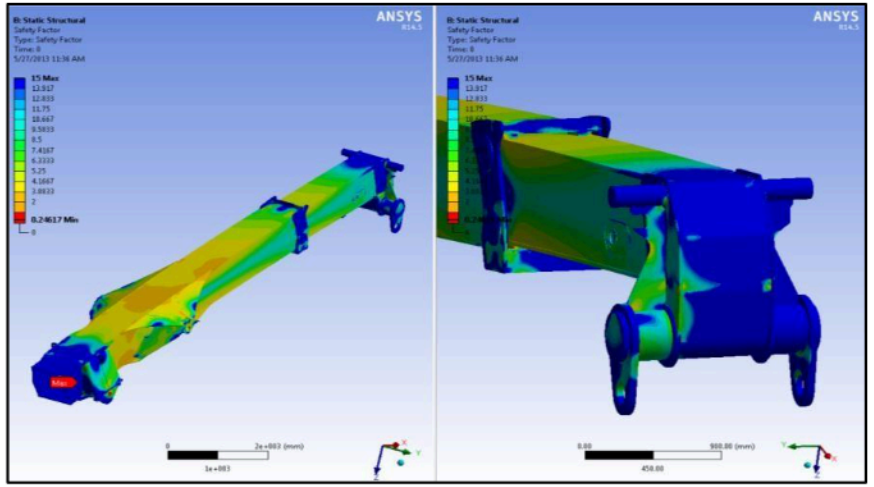
工况1: 结构静力学分析



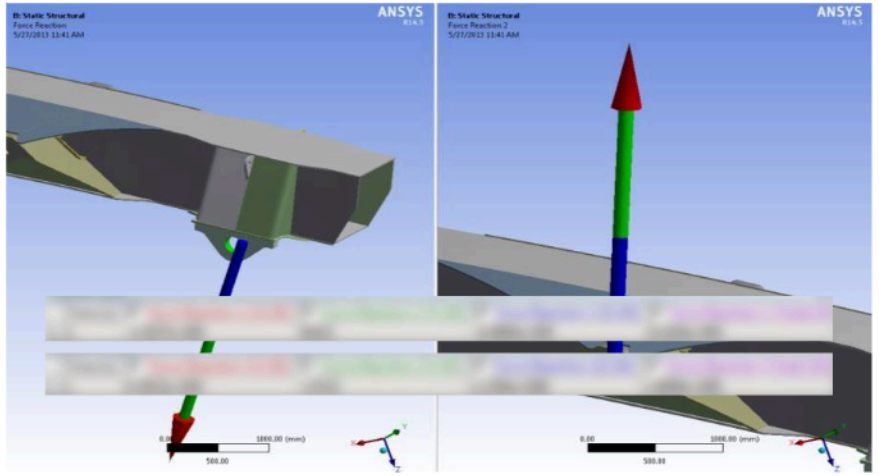
Displacements



Von Mises Stress



Safety Factor

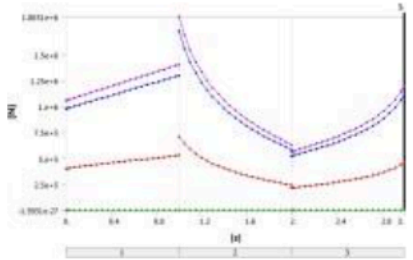


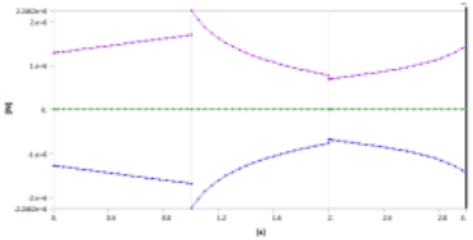
Reaction Evolution

工况2: 多体动力学分析

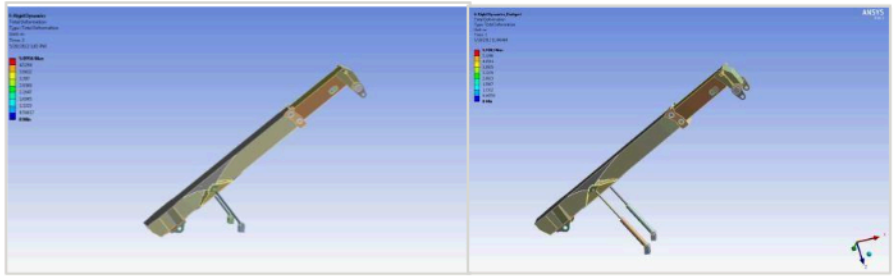
- 快速求解 (* mins)
- 过程动画
- 运动学结果:
  - Displacements
  - Velocities
  - Accelerations
- 支撑及接头处的反力变化

Time [s]	Joint Probe 2 (Total Force X) [N]	Joint Probe 2 (Total Force Y) [N]	Joint Probe 2 (Total Force Z) [N]	Joint Probe 2 (Total Force Tot
0.	-5.2387e-010	-4.8782e-010	-1.2781e+006	1.2781e+006
1.e-002	-8.7311e-010	-4.8939e-010	-1.2823e+006	1.2823e+006
3.e-002	5.2387e-010	-4.9253e-010	-1.2905e+006	1.2905e+006
7.e-002	-4.0745e-010	-4.9883e-010	-1.307e+006	1.307e+006
0.12	-5.8208e-011	-5.0669e-010	-1.3276e+006	1.3276e+006
0.17	-5.8208e-011	-5.1455e-010	-1.3482e+006	1.3482e+006
0.22	-5.8208e-011	-5.2242e-010	-1.3688e+006	1.3688e+006
0.27	5.8206e-011	-5.3028e-010	-1.3894e+006	1.3894e+006
0.32	-5.8208e-011	-5.3814e-010	-1.41e+006	1.41e+006
0.37	-1.7462e-010	-5.4601e-010	-1.4306e+006	1.4306e+006
0.42	-8.1491e-010	-5.5387e-010	-1.4512e+006	1.4512e+006



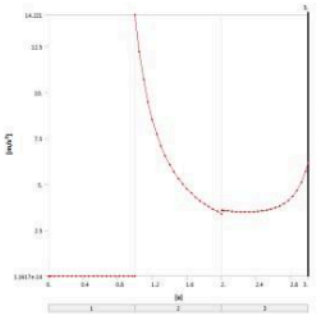


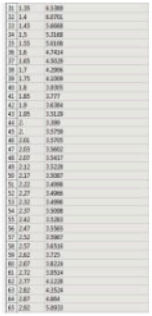
Displacements



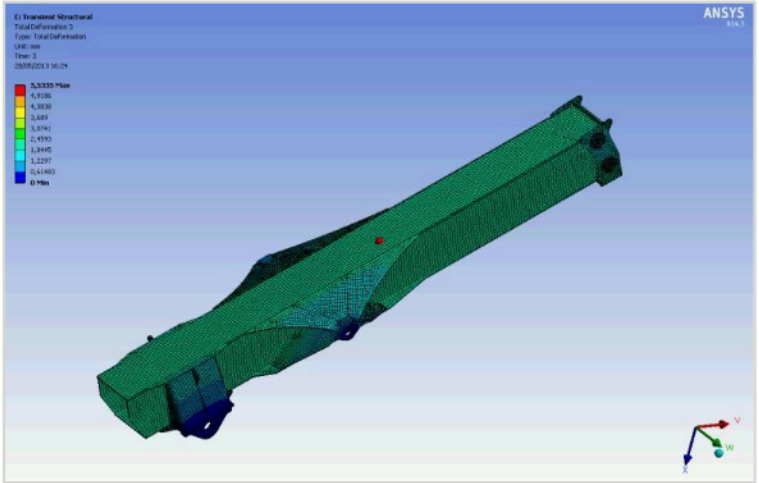
Reaction Evolution at Joints

Maximum Value Over Time	
Total	14.221 m/s²
Minimum Value Over Time	
Total	1.1617e-014 m/s²

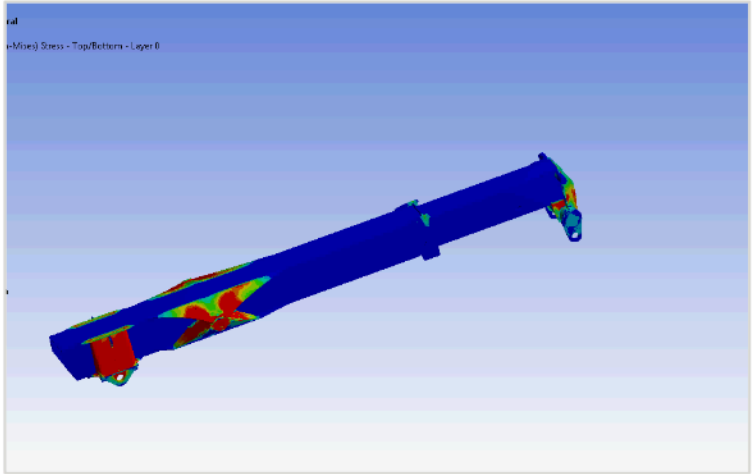




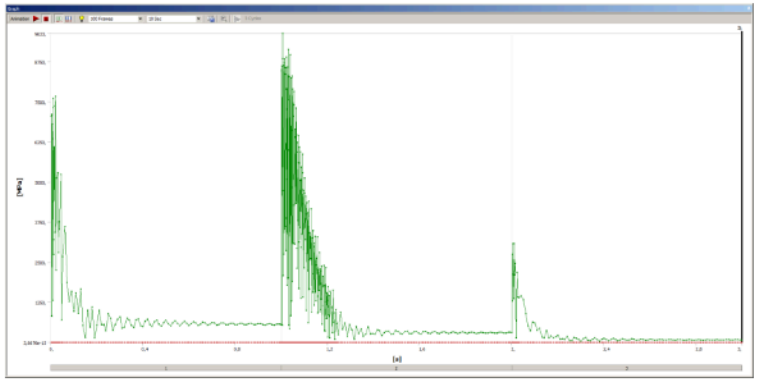
工况3: 瞬态分析

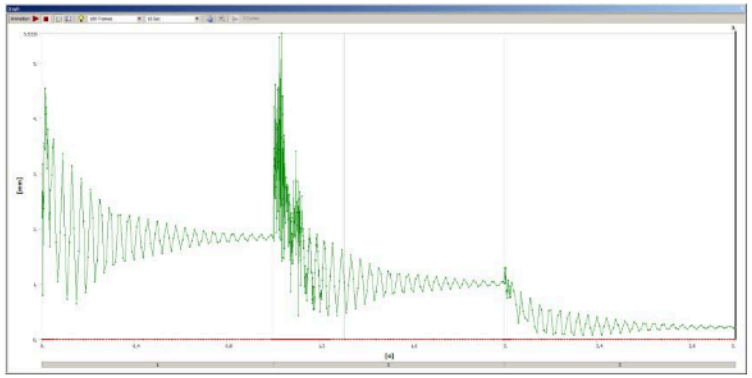


Displacements



Von Mises Stress



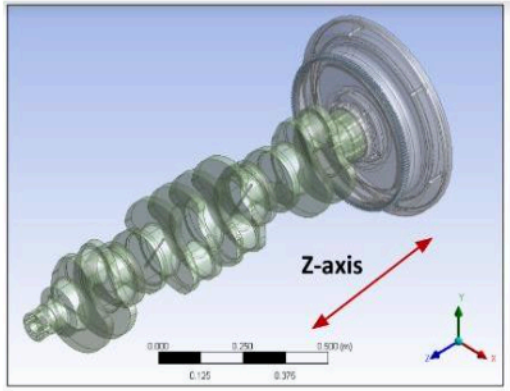


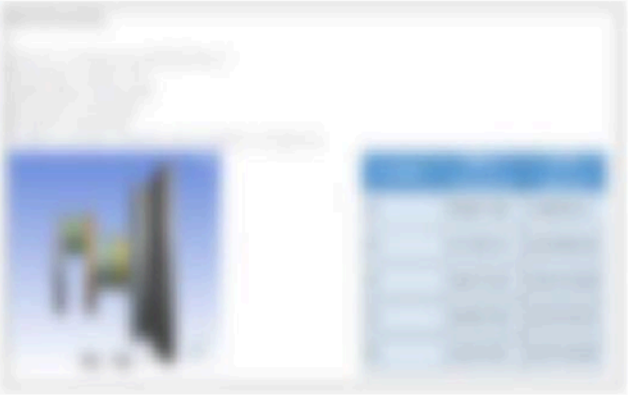
Damping and Reaction Evolution

结构仿真流程自动化

- 曲轴仿真定制化模块
- 概念设计→详细设计重复仿真
- 流程自动化，提升效率，减少错误

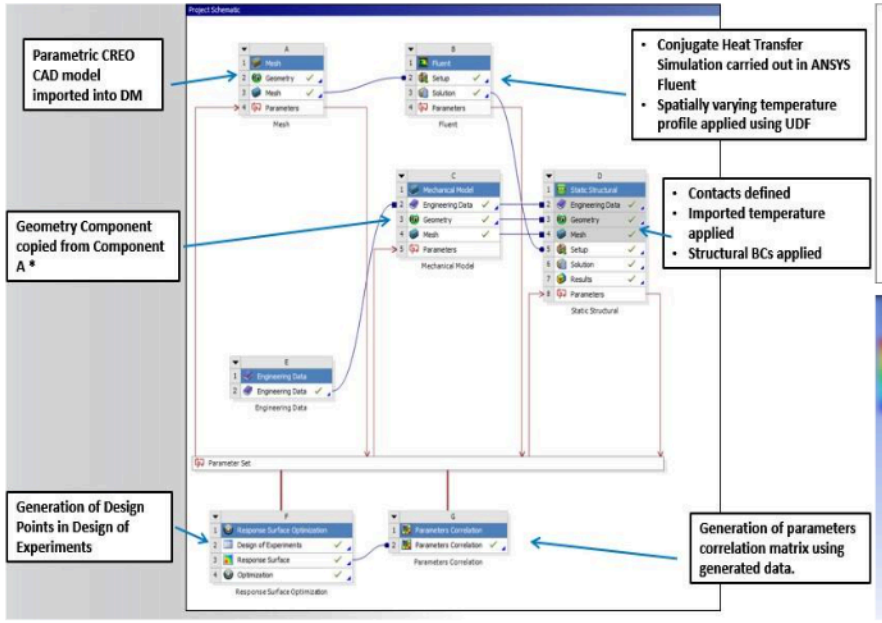
- 通过ACT Wizards， 整合Design Explorer, ANSYS Mechanical仿真流程

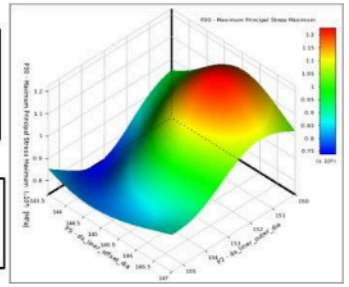


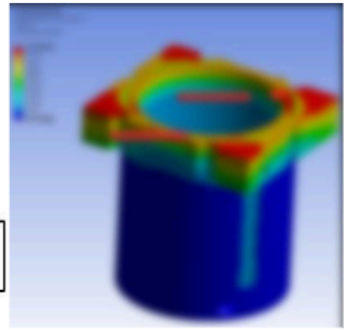


- 基于模型的分析定义(Mechanical GUI)
- 概念设计→详细设计重复仿真
- 易用工具集

- 通过ACT Wizards， 整合Design Explorer, ANSYS Mechanical仿真流程，平台化







Ansys