

ANSYS

Your True Partner for CAE&CFD

IDAJ CAE
Solution
Conference

ICSC  2015

比想象中更强大!

IDAJ-China联合ANSYS-China
共同开创中国CAE&CFD新未来



ANSYS产品咨询:

Email: ansys@idaj.cn

Tel: 010-65881497; 021-50588290

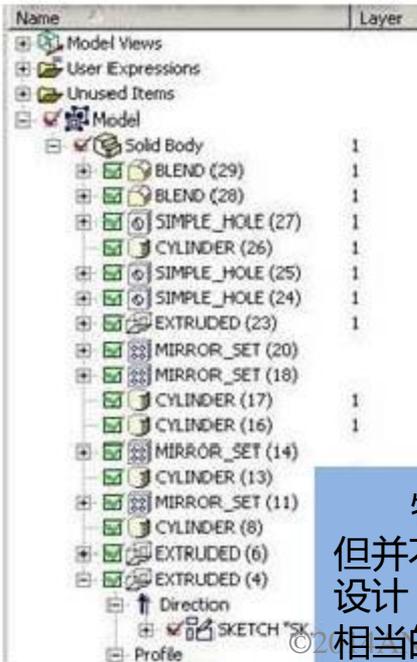
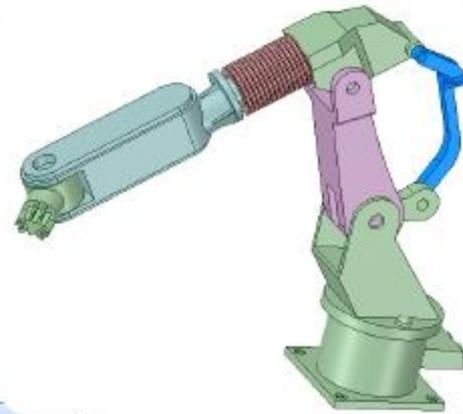
ANSYS SpaceClaim
直接建模技术

ANSYS CHINA

郭臻

什么是直接建模？

- ◆以几何拓扑为直接编辑对象的建模方式
- ◆建模过程不受模型建立过程和复杂的参数关联所约束
- ◆不管模型是否有特征（从其它CAD系统导入模型），用户都可以直接进行后续模型的创建，都无需关注模型的建立过程



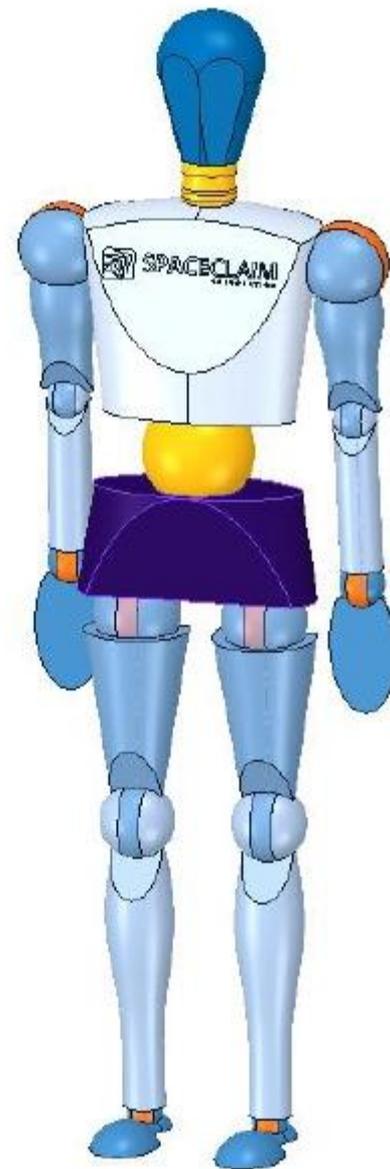
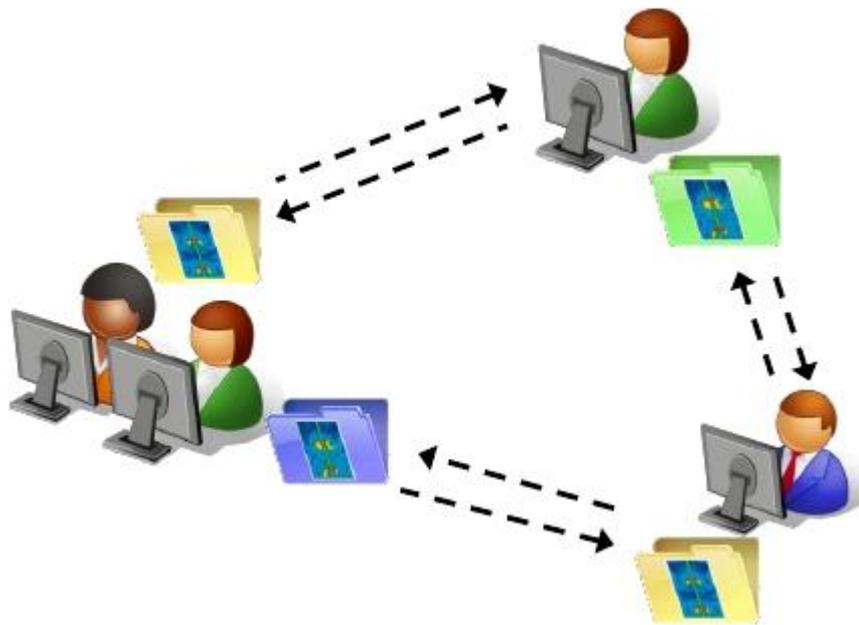
特征建模在详细设计中虽强大，但并不适合概念设计、仿真驱动型设计，对于模型的修改和重利用也相当的困难。



无关特征，无关建立过程

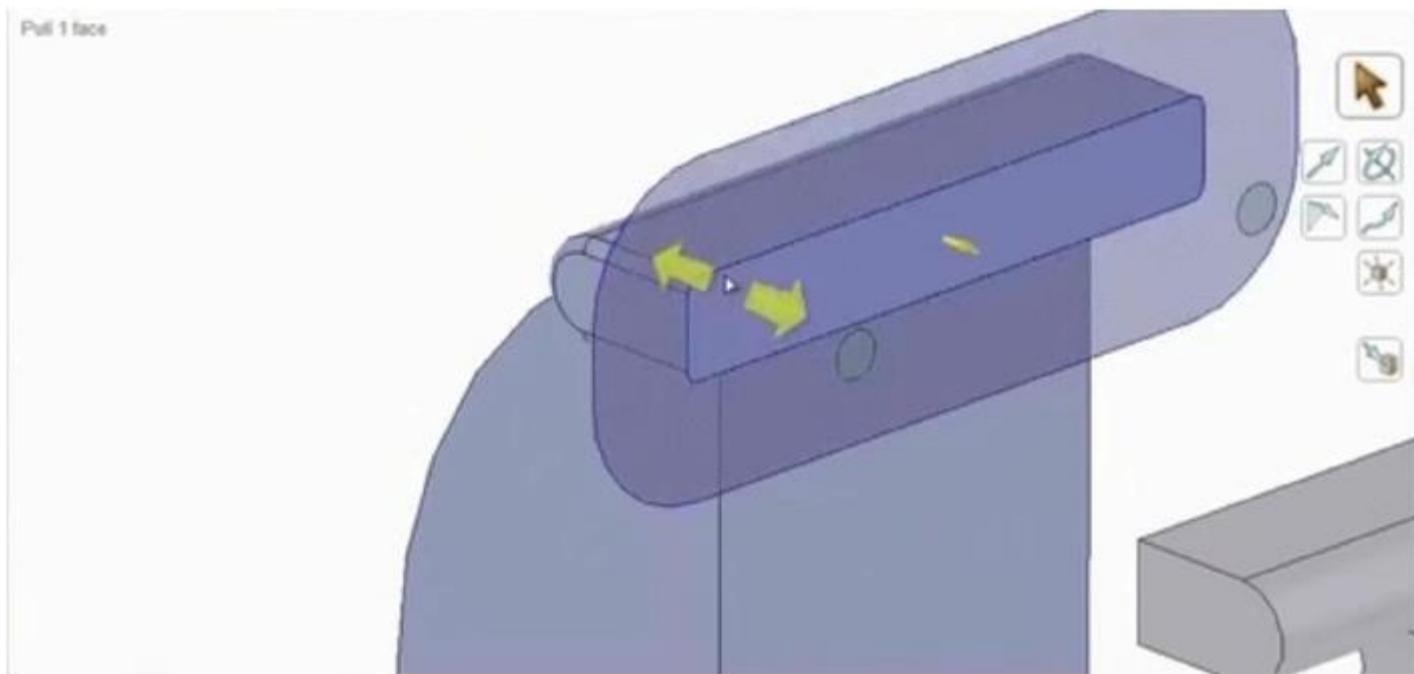
直接建模有什么优势？

- ◆建模方式最直观，更能充分表达设计者的创意
- ◆设计方案变更响应更加快速，缩短产品上市周期
- ◆灵活处理多源CAD数据，提高模型的再生成功率和重用率
- ◆增强不同部门间的协作能力



ANSYS SpaceClaim是怎样的工具？

- ◆ ANSYS软件体系中几何建模工具的重要组成部分
- ◆ 融合了直接建模思想的建模及模型修改的高效工具
- ◆ 专门面向CAE仿真模型的处理利器
- ◆ 三维设计和有限元仿真之间的重要桥梁





◆直接几何建模

1. 灵活高效的直接建模
2. 模型快速修改
3. 全面兼容的数据接口
4. 独特的钣金设计模块
5. 易用的零件装配
6. 丰富多样的实用功能

◆适用CAE仿真的模型处理

1. 模型错误检查
2. 破损模型修补
3. 模型特征简化
4. 特定仿真模型的建立
5. 优化网格质量的工具
6. 完美集成于ANSYS Workbench



直接几何建模

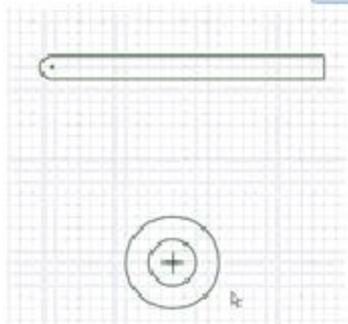
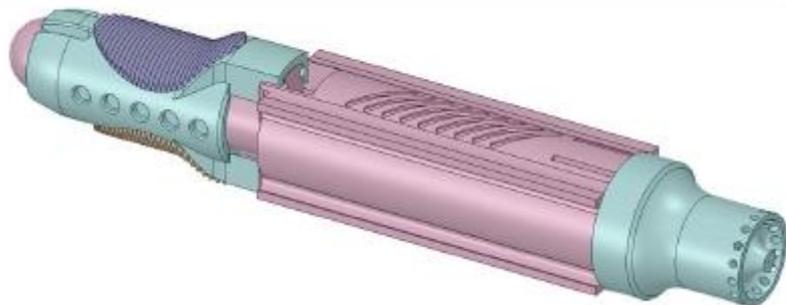
灵活高效的直接建模



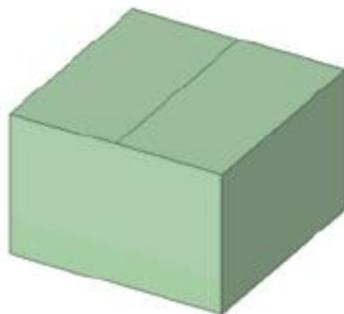
◆融合了直接建模思想的草图、剖面和三维建模方式

◆拖放式设计，大大地提高了用户与图形的交互性，所见即所得

◆菜单简洁清晰，操作简便



草图模式



剖面模式



三维模式

模型快速修改

- ◆ 无论是SpaceClaim建立的模型，或是从外部导入CAD数据，SpaceClaim都可以快速地进行修改
- ◆ 对于外部导入的无参数中性CAD数据，SpaceClaim都可以对其进行参数化



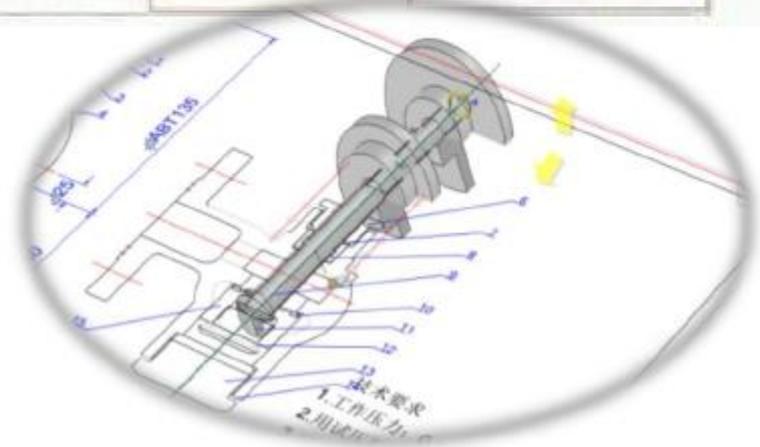
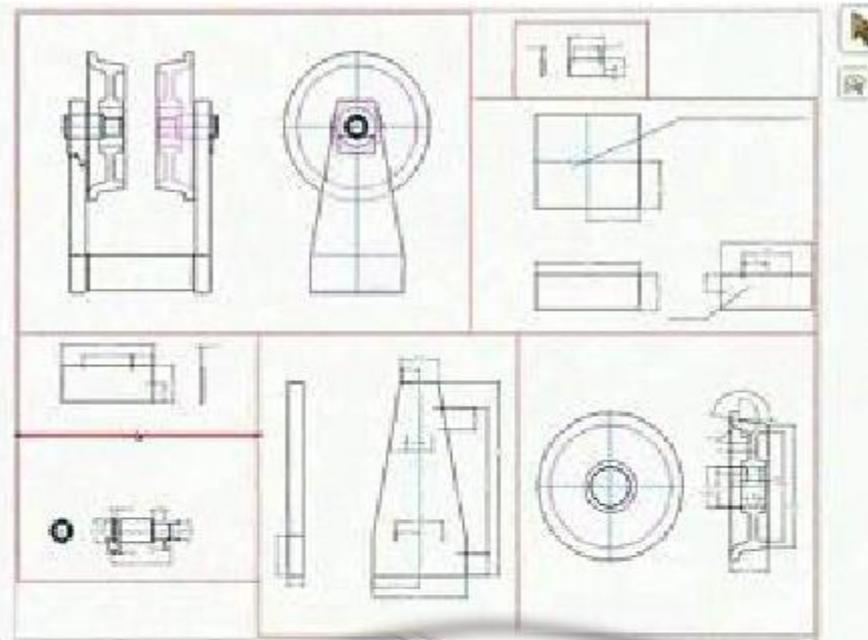
全面兼容的数据接口

- ◆模型导入和导出的格式十分丰富
- ◆基于二维图档快速建立三维模型

- Acis
- Step
- Iges
- Parasolids
- DWG、DXF
- Pro/Engineer
- Inventor
- Catia
- Solid Works
- Unigraphic
- STL
- VDA
- ECAD Files (ldf, emn)
-

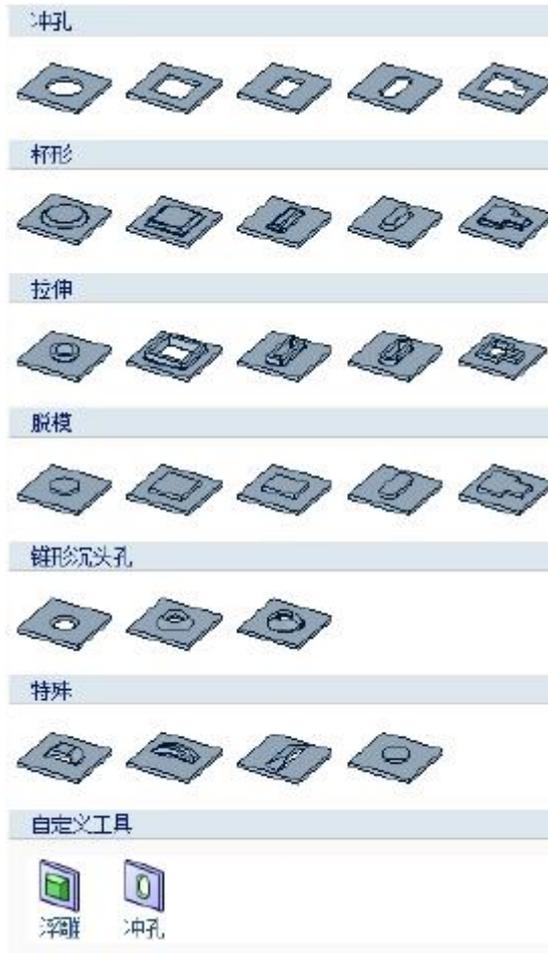
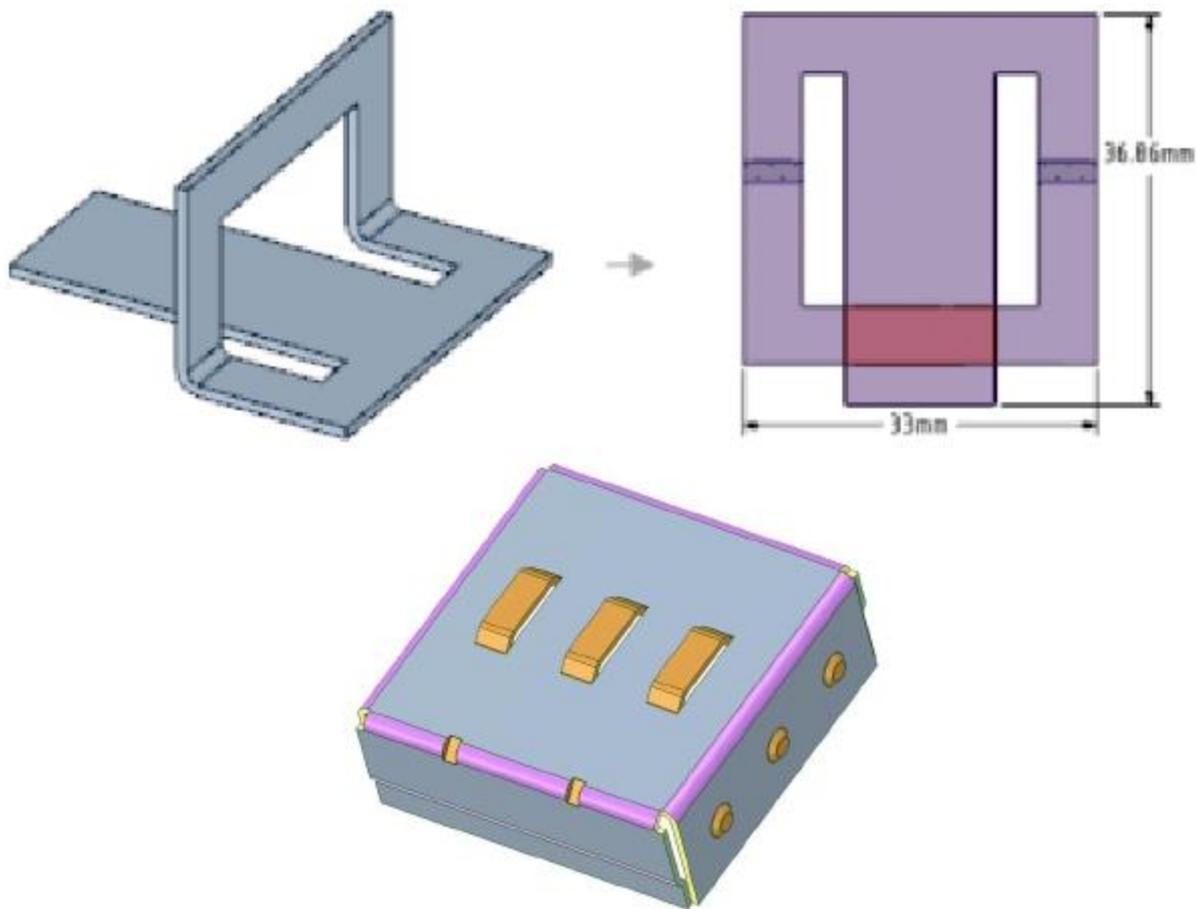


- Acis
- Step
- Iges
- Parasolids
- DWG、DXF
- STL
- VRML
- XAML
-



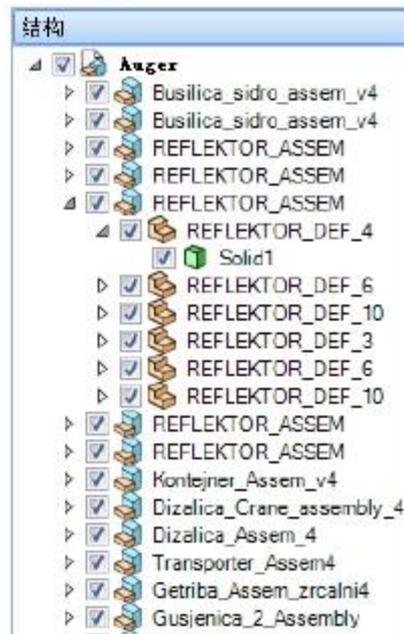
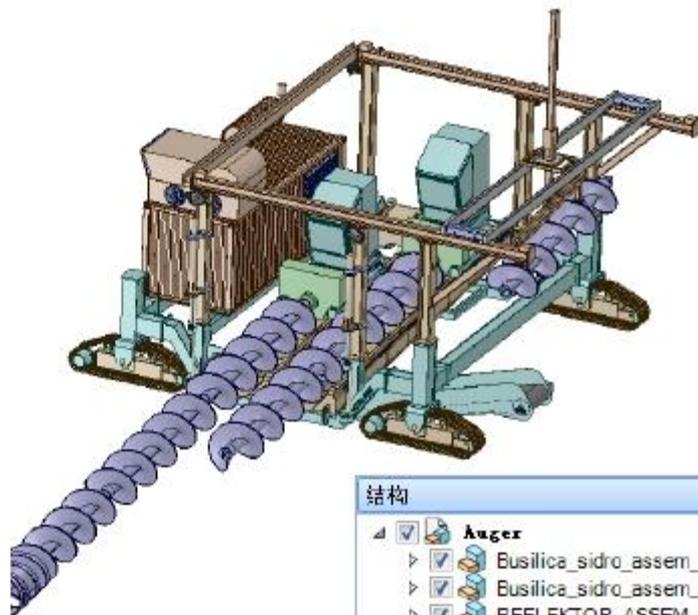
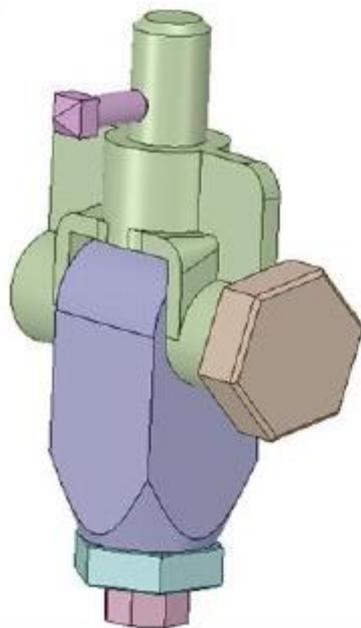
独特的钣金设计模块

- ◆结合直接建模和钣金工艺特征的钣金设计
- ◆对不符合钣金生产标准的部件进行检查



易用的零件装配

- ◆ 搭配强大的移动功能，装配更加易用
- ◆ 装配好的部件不受相互约束，避免模型出错
- ◆ 支持多层部件关系，不怕零件图丢失

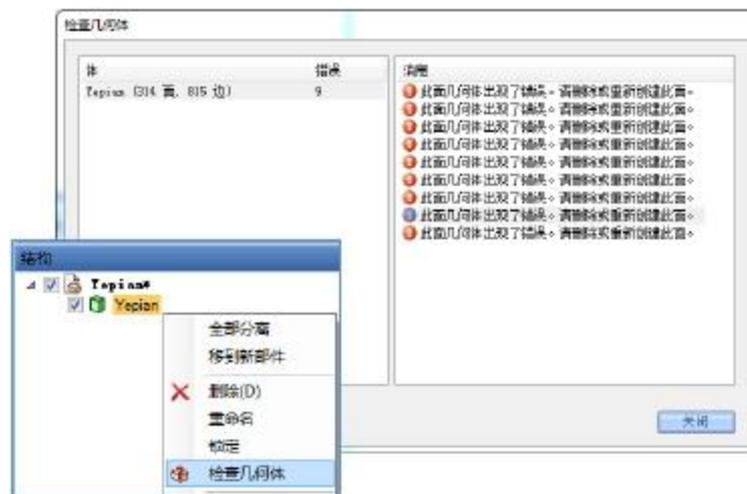
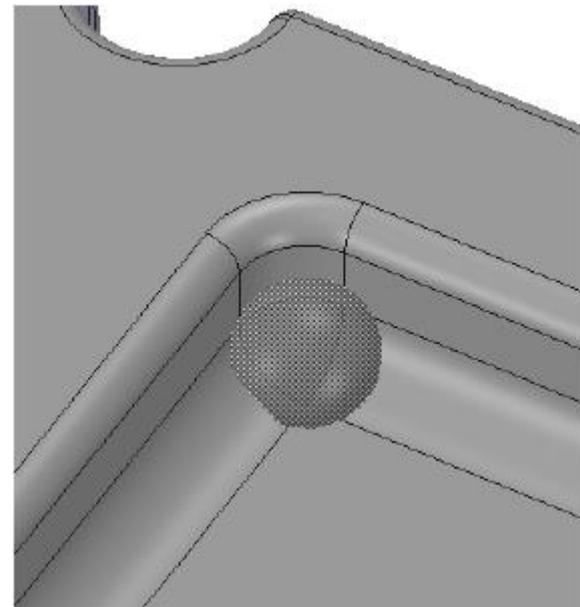
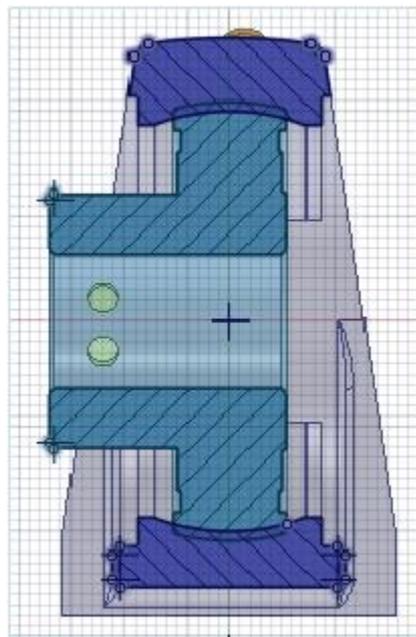
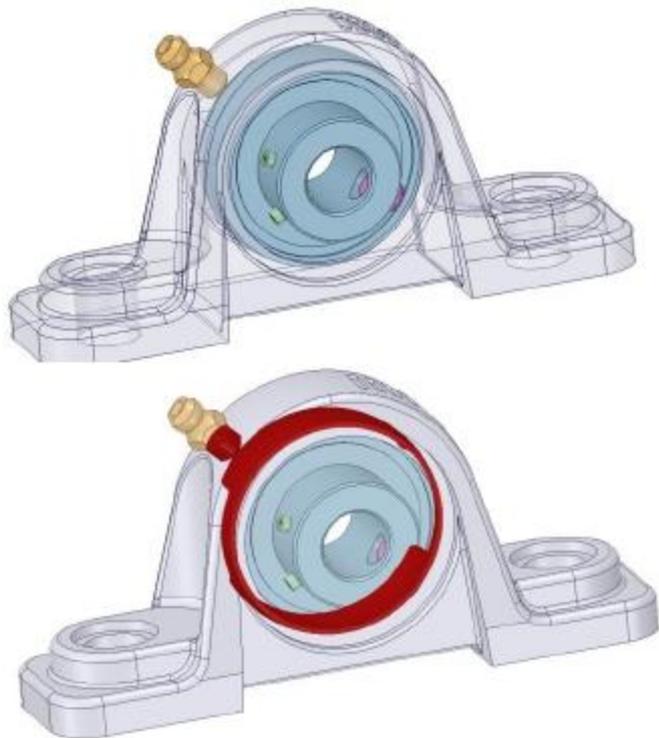




适用CAE仿真的模型处理

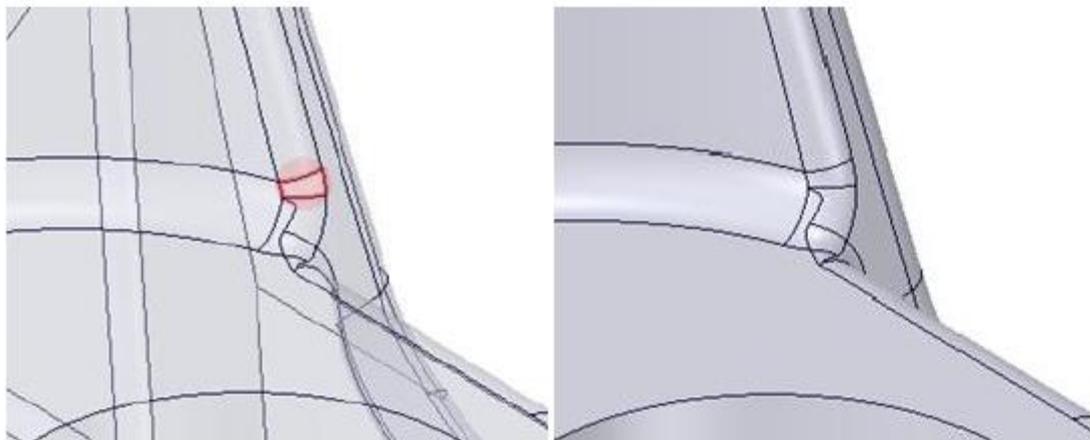
模型错误检查

- ◆ 检查和修复因建模错误而导致的模型干涉等问题
- ◆ 因数据转移而造成的破面等拓扑问题

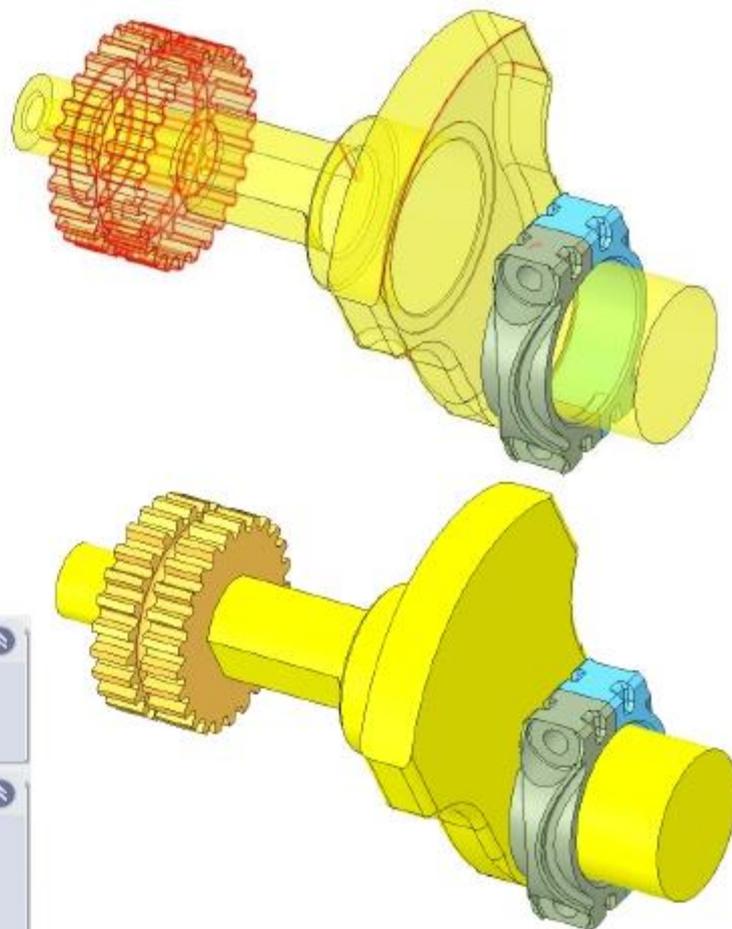


破损模型修复

◆ 修补模型的缺面、面间隙等问题



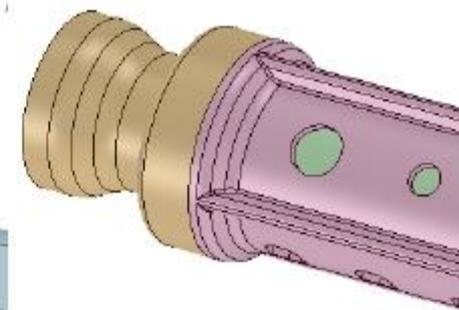
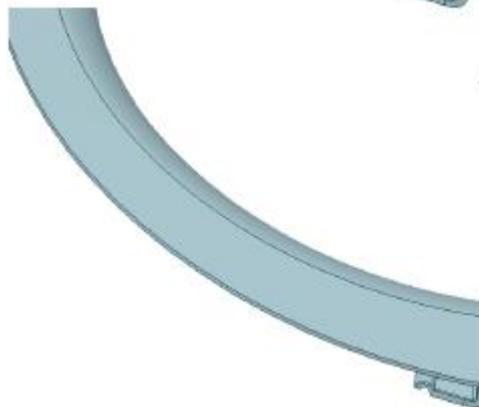
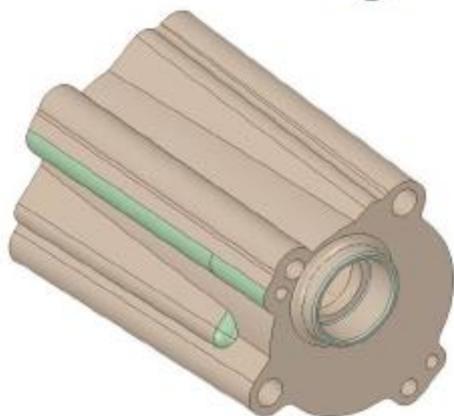
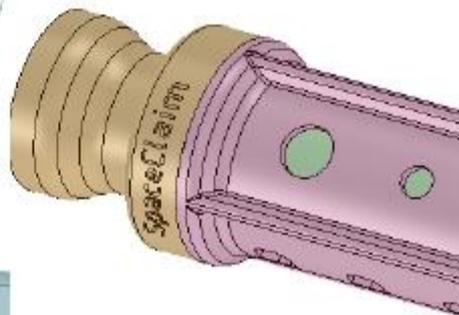
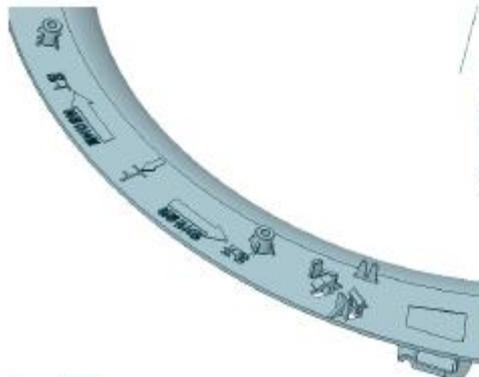
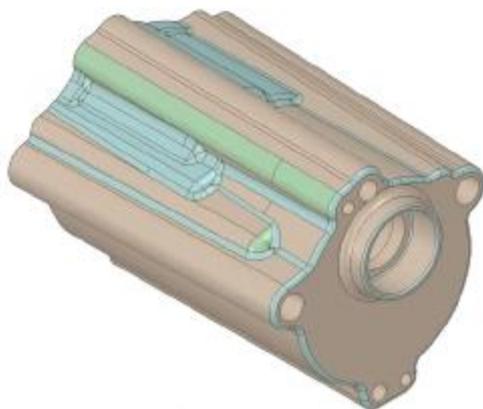
根据设定的数值，自动探测和修补



模型特征简化

◆圆角、倒角、小孔、凸台、刻字等工艺性特征的简化

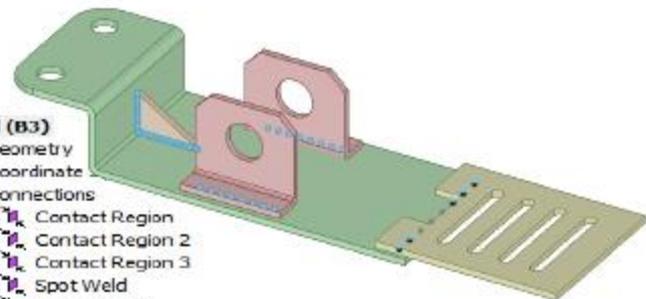
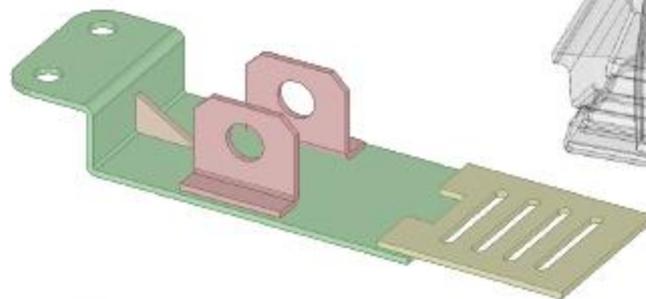
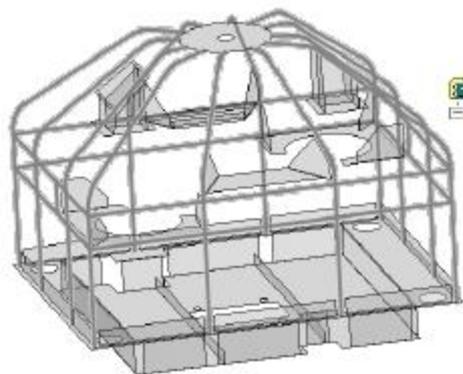
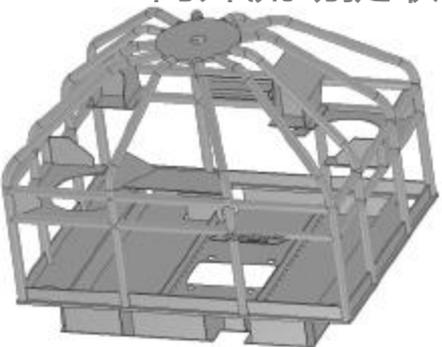
配合智能的选择引擎，大大提高了模型简化的效率



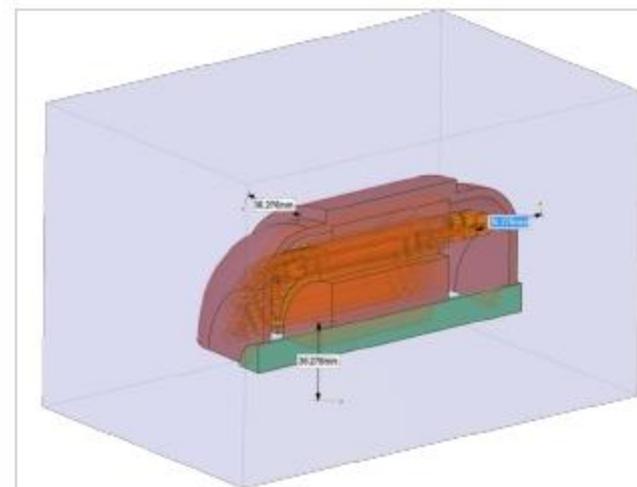
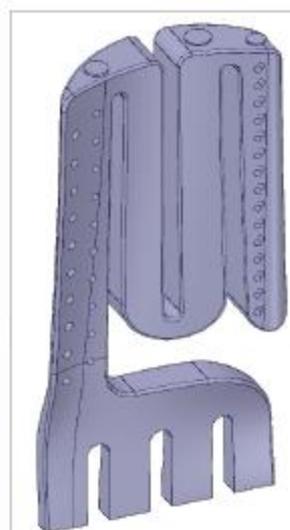
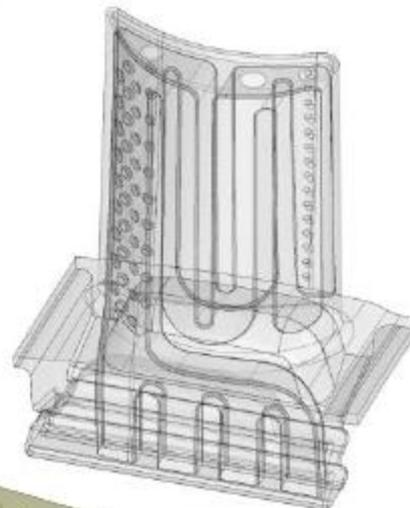
©2014 ANSYS Inc.

特定仿真模型的建立

- ◆ 截面梁定义
- ◆ 中间面抽取
- ◆ 焊点设定
- ◆ 内外流场提取等

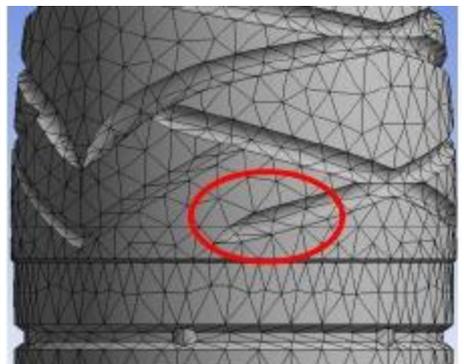
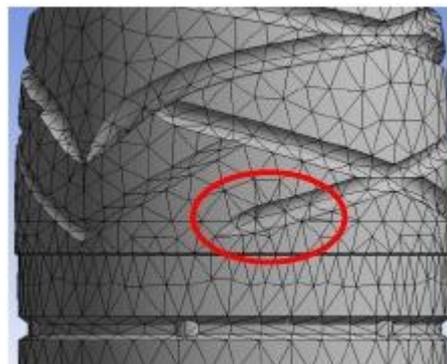
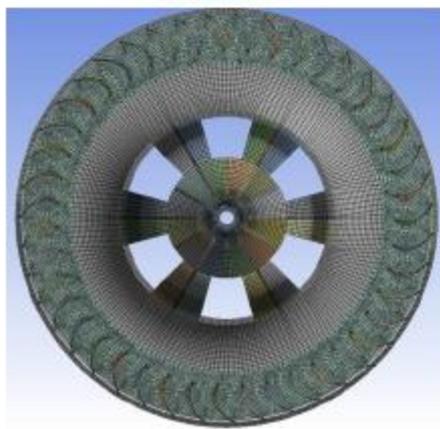
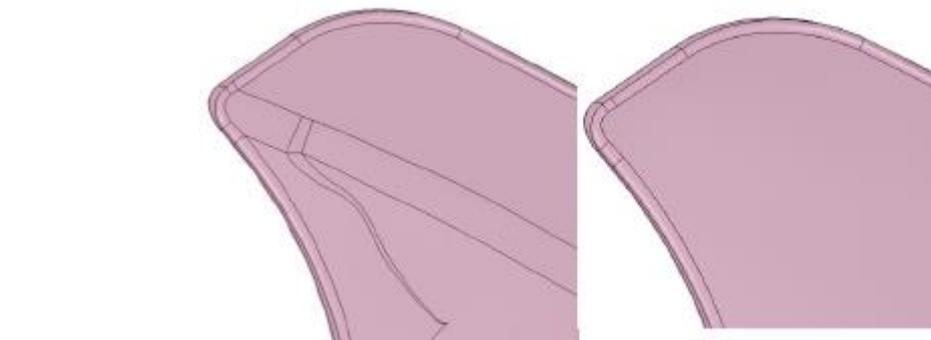


- Project
- Model (B3)
 - Geometry
 - Coordinate
 - Connections
 - Contact Region
 - Contact Region 2
 - Contact Region 3
 - Spot Weld
 - Spot Weld 2
 - Spot Weld 3
 - Spot Weld 4
 - Spot Weld 5
 - Spot Weld 6
 - Spot Weld 7
 - Spot Weld 8
 - Spot Weld 9



优化网格质量的工具

- ◆ 实体拆分、表面切割
- ◆ 额外点的删除
- ◆ 狭长面的合并



完美集成于ANSYS Workbench

◆ 结合几何参数化，可在ANSYS Workbench中快速对不同的方案进行多物理场仿真及优化



The screenshot shows the ANSYS Workbench Project Schematic interface. It features a 'Toolbox' on the left with various simulation types. The main workspace contains six components (A-F) connected by lines, representing a multi-physics simulation. Component A is 'Geometry', B is 'Static Structural', C is 'Modal', D is 'Explicit Dynamics (LS-DYNA Export)', E is 'Fluid Flow (Fluent)', and F is 'System Coupling'. Each component has a list of steps (e.g., Geometry, Engineering Data, Model, Setup, Solution, Results, Parameters). A 'Parameter Set' is defined at the bottom, which is detailed in the table below.

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Geometry (A1)			
4	P1	XX直径	10	
5	P2	XX壁厚	30	
6	New input parameter	New name	New expression	
7	Output Parameters			
8	New output parameter		New expression	
9	Charts			



THANK YOU