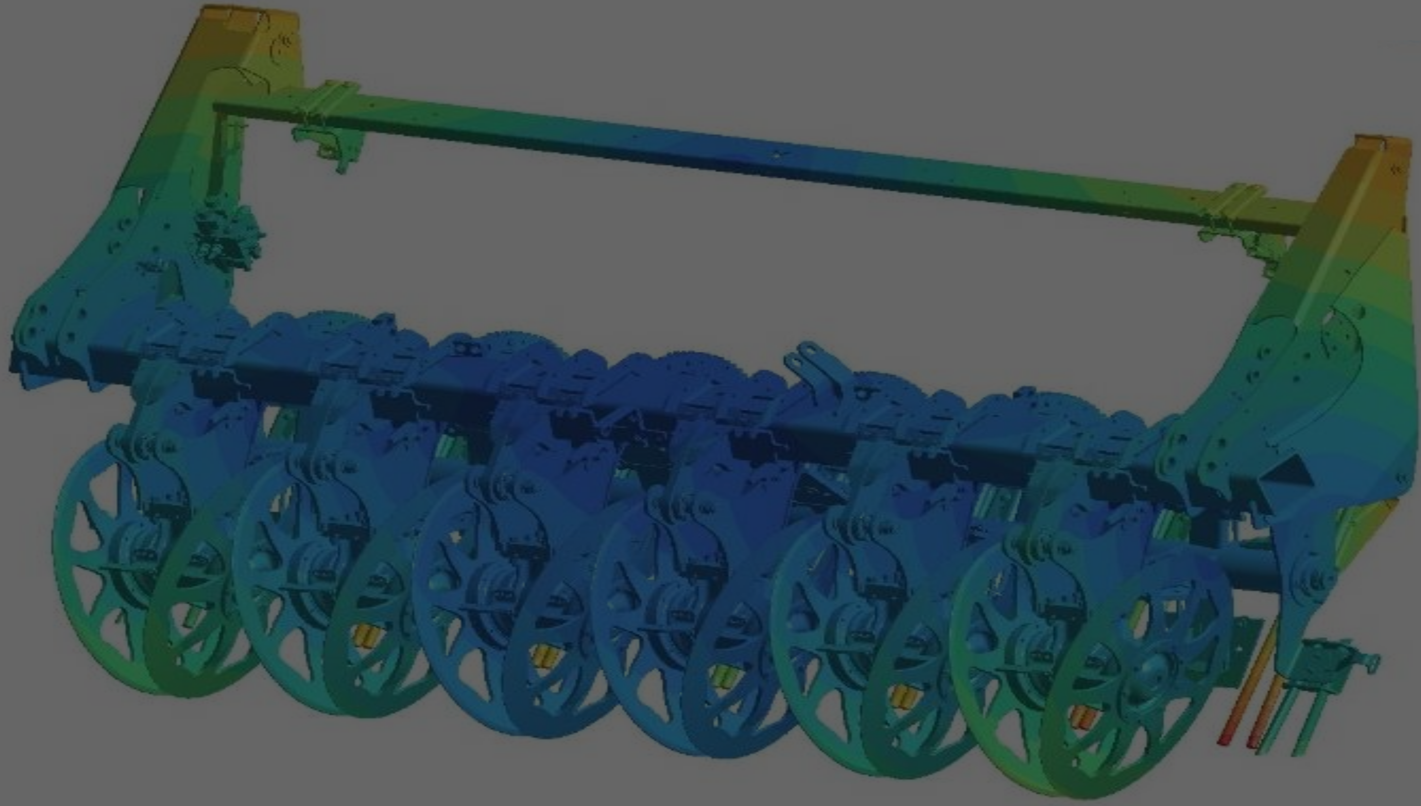


Altair SimSolid 复杂装配体无网格结构仿真介绍

汤凯利 Altair SimSolid中国区技术负责人
kaili@altair.com

SimSolid技术革新

求解速度快	探索更多	快速校验，缩短研发周期

建模速度快
几分钟内完成分析

大型装配体
复杂几何模型

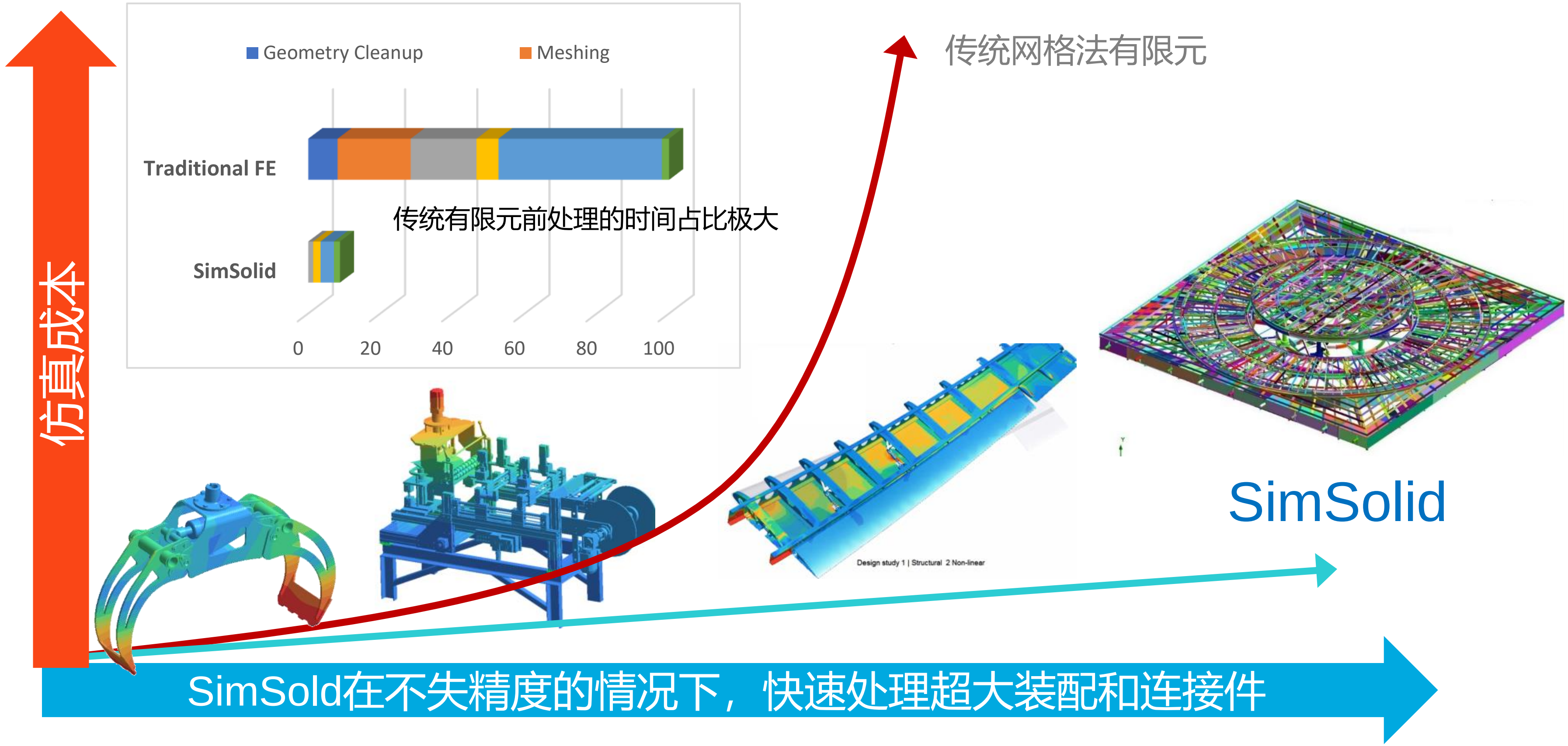
计算精度高
分析流程简单

Altair **SimSolid** 是一个结构仿真工具，直接用**原始的未经简化的几何装配体**建模，并且**不需要划分网格..**

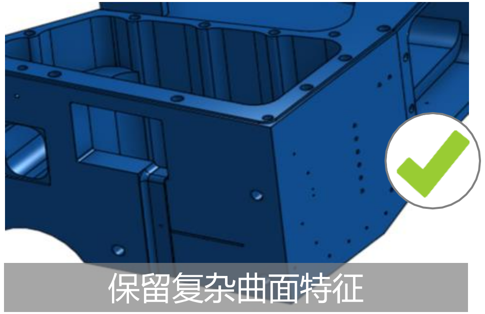
桌面级配置

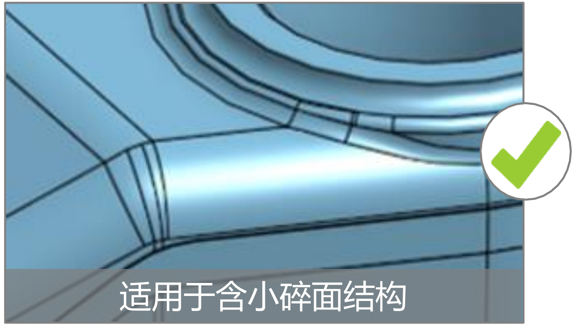
..在**几秒至几分钟**内提供结果

有限元技术的革新—SimSolid无网格分析方法

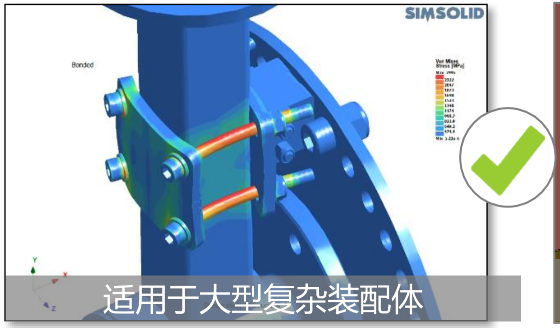


无几何简化，自动识别连接

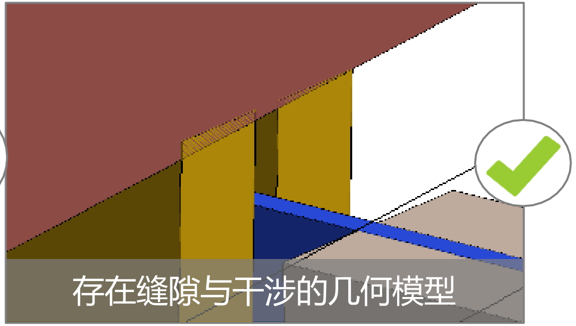

保留复杂曲面特征

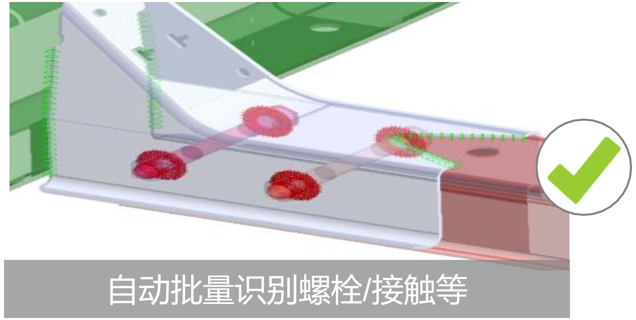

适用于含小碎面结构


非常复杂的几何模型



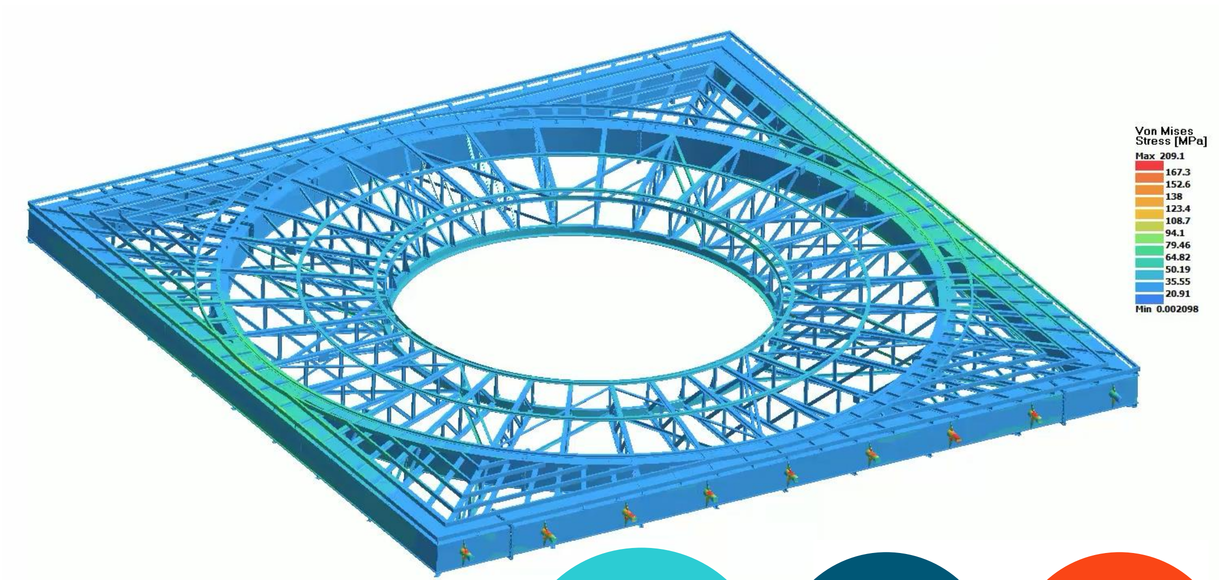
适用于大型复杂装配体

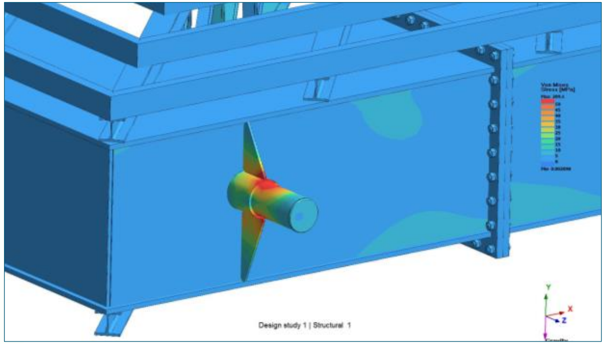

存在缝隙与干涉的几何模型

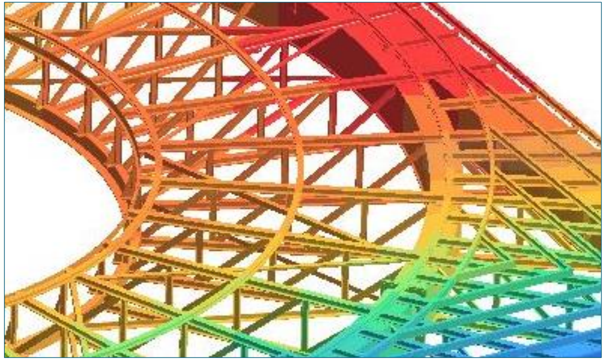

自动批量识别螺栓/接触等

获得同步设计的仿真速度

从CAD到分析只需要几分钟







桌面级配置

7700
部件

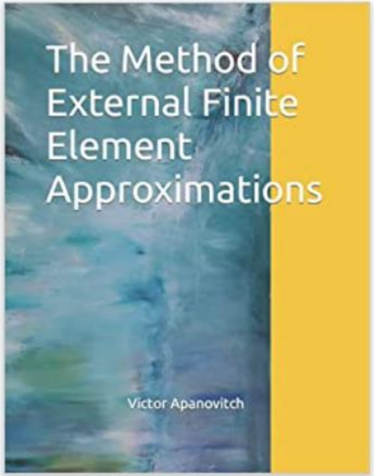
60
分钟
建模

30
分钟
CPU求解

ALTAIR

SimSolid 独特的理论基础

- 基于有限元算法的扩展算法，无需使用有限元算法的特定形状的单元网格
- 离散单元可以是单个部件
- 有限元形函数（插值函数）可以是任意阶数的多项式函数，并且可以随着计算过程自动调整阶数或者使用非多项式的特殊函数，以提高计算精度
- 求解精度控制基于局部能量密度改变和边界条件的误差，适用于局部与全局，适用于复杂零件或者大型装配体



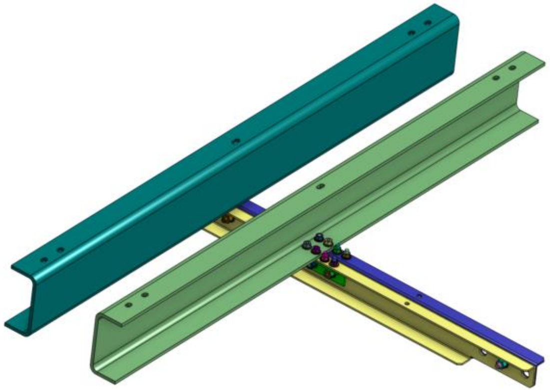
ISBN-13: 979-8467137773

Victor Apanovitch
Cofounder, President and CTO at SIMSOLID Corporation.
Formerly a professor at Belarus Polytechnic University.
The author of book “The Method of External Finite Element Approximations(1991初版,2021再版)“.

ALTAIR

SIMSOLID 求解分析步骤– 建模

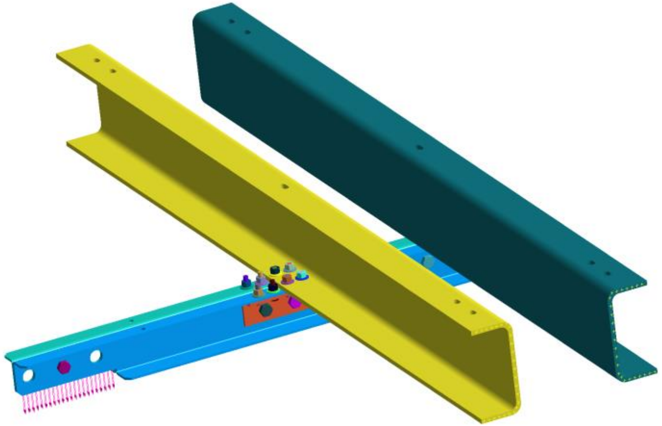
❶ 导入几何模型Process geometry
- 转换为面体组合faceted volumes



❷ 识别几何特征Classify geometry
- 螺栓，螺母，垫圈，弹簧 Bolts, nuts, washers, springs
- 薄壁件Thin sheets
- 通孔 Through holes

❸ 创建连接Create connections
- 自动搜索接触区域
- 自动定义接触类型 (bonding, sliding)
- 用户自定义接触类型 (separating)

❹ 求解设定Create analysis parameters
- 分析类型Analysis type
- 边界条件Boundary conditions
- 选定材料参数（也可以从CAD软件导入）
- 设定Solution adaption



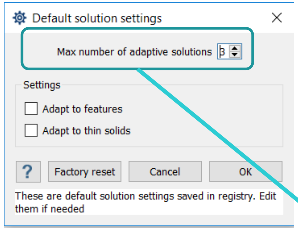
提交计算
Ready to analyze

ALTAIR

SIMSOLID求解分析步骤– 求解过程

❶ 几何评估
Geometry evaluation
- 特征识别与评估
  Feature recognition and evaluation
- 创建初始方程
  Create initial equations

结果方程 Result functions



根据最大的PASS次数自动循环计算
Loop and repeat based on specified # of max passes

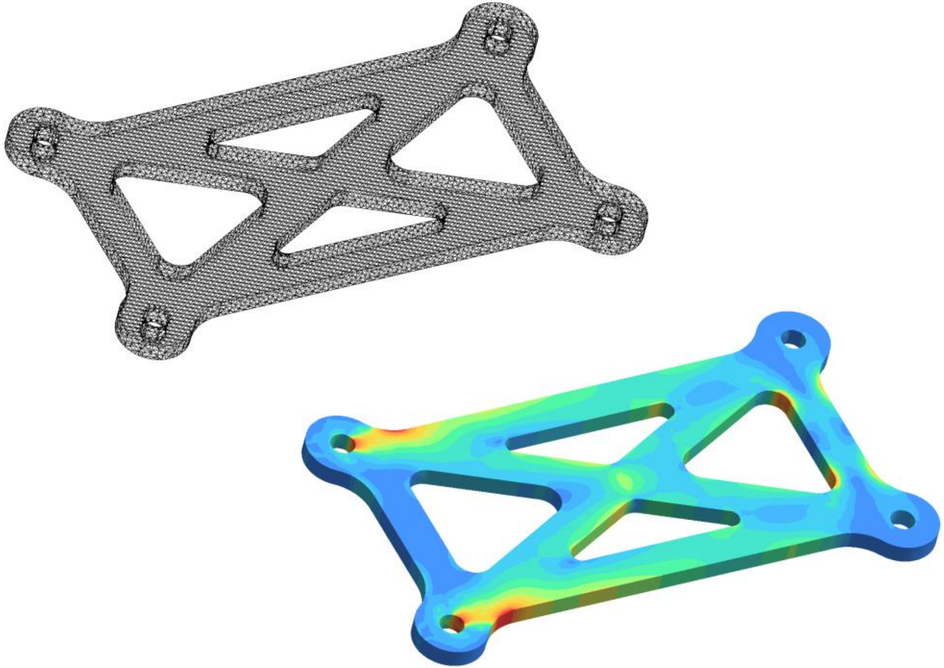
❷ 求解Solution pass
- 求解方程Solve equation set
- 获得应力应变Strain/stress recovery

❸ 误差分析Error analysis
- 约束位置的位移误差
  Displacement error analysis at constraints
- 在载荷位置与自由度的载荷误差
  Traction (force) error analysis at loaded or free surfaces of parts
- 位移与能量收敛性分析
  Displacement and energy convergence analysis

❹ Solution adaption
- 增加约束处的自由度
  Adding DOF locally at constraints
- 增加局部提结构的自由度
  Adding DOF locally in volumes
- 增加几何特征处的近似方程数
  Adding special approximation functions at features

ALTAIR

SIMSOLID 求解分析步骤 – 后处理



1 创建结果响应网格
- 用于把方程结果映射到几何上面
- 可以即时重新定义

2 可以读取任意点的结果数值
- 可以获得结果响应网格节点处的结果数值
- 提取速度快

3 输出反力
- 约束反力
- 接触反力
- 部件合力

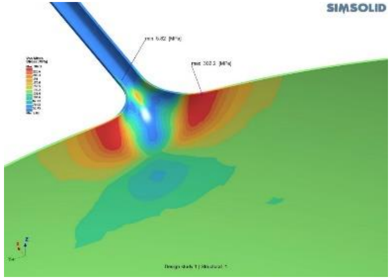
4 快速重分析Fast re-analysis
- SIMSOLID能利用之前设定响应网格与方程数量
- 重新计算快

5 高效耦合分析
- 某一个计算分析结果可以直接用到下一分析中
  Results of one analysis are directly used in analytical form in other analyses
- 如热固耦合分析，非线性分析，瞬态动力学分析
  Thermal-stress, nonlinear analysis, dynamics

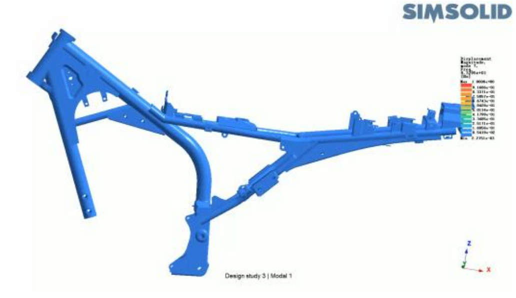
SIMSOLID 功能概述

SimSolid的分析功能

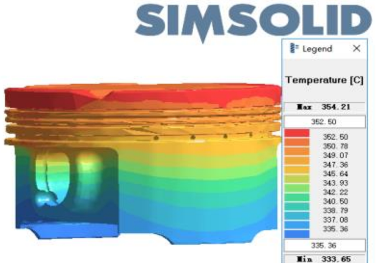
静强度分析



模态分析

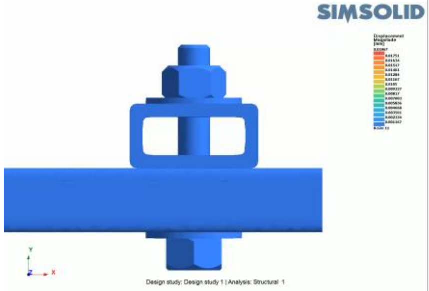


稳态/瞬态热分析

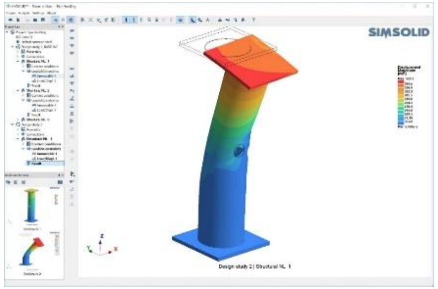


瞬态/频响/随机振动分析

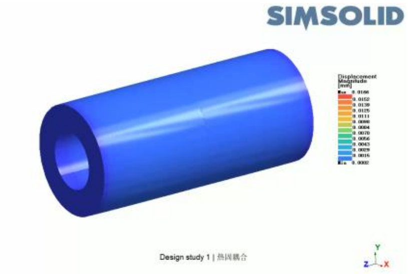
螺栓预紧序列工况



大变形/接触/弹塑性



热固耦合分析



疲劳分析

SimSolid2022 功能

分析类型

- 模态分析
- 线性静力学
- 非线性静力学
- 频率响应分析
- 线性瞬态分析
- 随机振动分析
- 热传导分析
- 热固耦合分析
- 惯性释放
- 螺栓预紧序列工况
- 线性叠加
- 部分动力学响应
- 疲劳分析
- 多工况分析

材料类型

- 各向同性
- 弹塑性
- 刚性体
- 流体物体
- 用户自定义
- 正交各向异性材料
- 疲劳材料曲线

连接方式

- 自动创建
- 绑定接触、滑移等
- 可分离接触
- 螺栓连接
- 点焊
- 铆接
- 焊缝（实体）
- 虚拟连接（弹簧、铰、销）
- 粘胶
- 接头（铰链等）
- 柔性远程质量

结果后处理

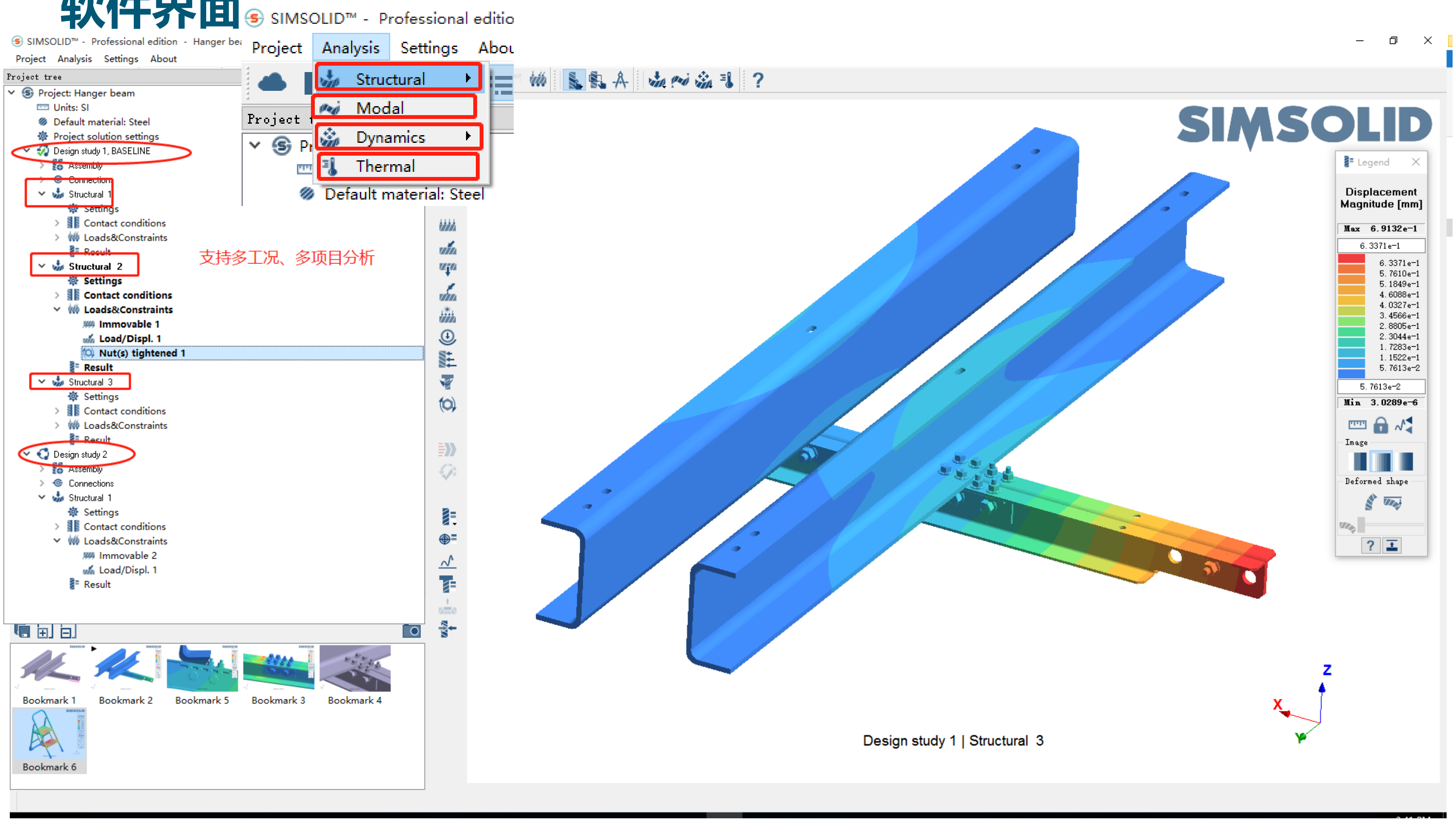
- 云图与动画
- 位移、应力、应变与应变能、安全因子
- 速度、加速度、平均辐射能
- 固有频率与振型
- XY曲线图
- 模态参与因子
- 接触力、约束反力
- 螺栓与焊接内力
- 显示最大最小值
- 显示面最大最小值
- 书签（PNG\AVI\GIF）
- 应力线性化

特色

- 快速对比分析
- 脚本建模与求解
- 中文界面与帮助文档

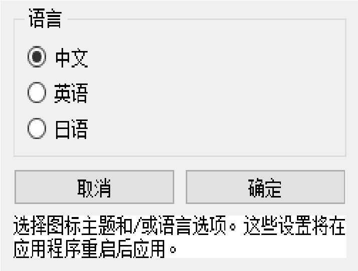
ALTAIR

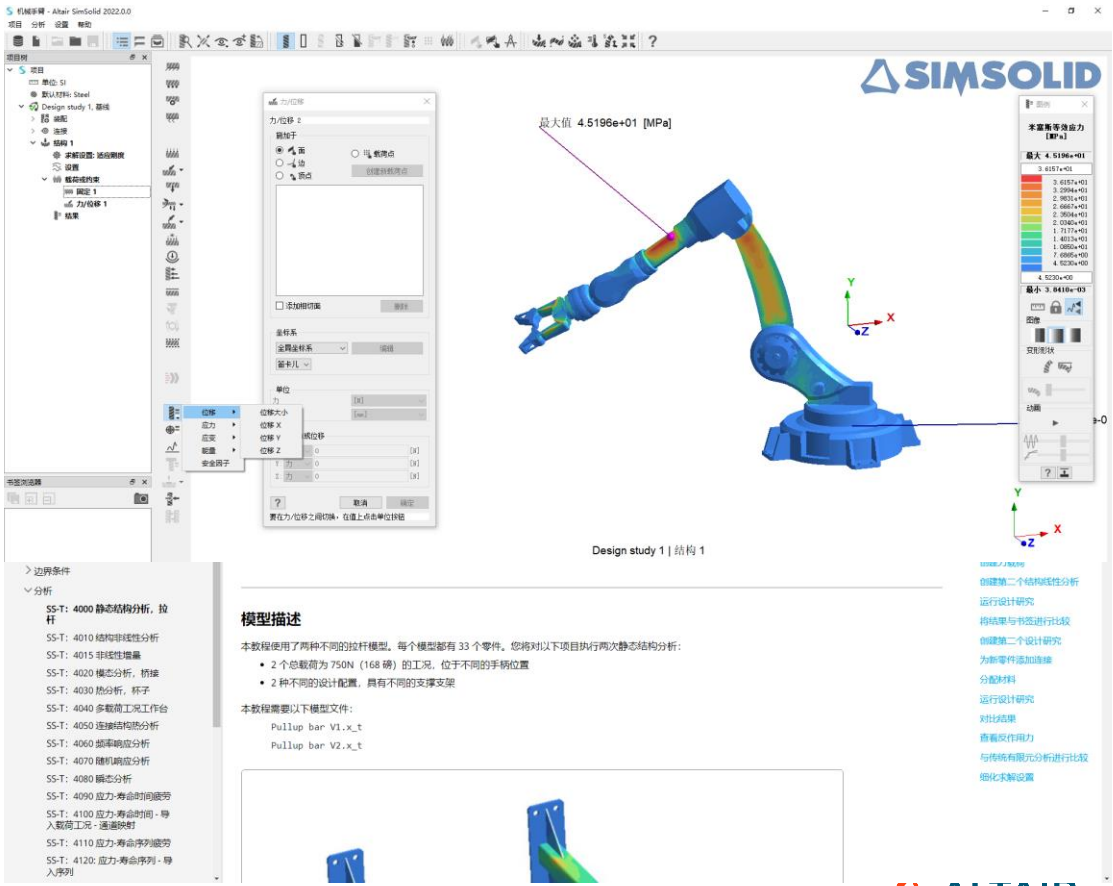
软件界面


SIMSOLID™ - Professional edition - Hanger beam
Project
Analysis
Settings
About
Structural
Modal
Dynamics
Thermal
Project tree
Project: Hanger beam
Units: SI
Default material: Steel
Project solution settings
Design study 1, BASELINE
Structural 1
Contact conditions
Loads&Constraints
支持多工况、多项目分析
Structural 2
Settings
Contact conditions
Loads&Constraints
Immovable 1
Load/Displ. 1
Nut(s) tightened 1
Result
Structural 3
Settings
Contact conditions
Loads&Constraints
Design study 2
Connections
Structural 1
Settings
Contact conditions
Loads&Constraints
Immovable 2
Load/Displ. 1
Result
Bookmark 1
Bookmark 2
Bookmark 5
Bookmark 3
Bookmark 4
Bookmark 6
SIMSOLID
Legend
Displacement Magnitude [mm]
Max 6.9132e-1
6.3371e-1
5.7610e-1
5.1849e-1
4.6088e-1
4.0327e-1
3.4566e-1
2.8805e-1
2.3044e-1
1.7283e-1
1.1522e-1
5.7613e-2
Min 3.0289e-6
Image
Deformed shape
Design study 1 | Structural 3

支持中文

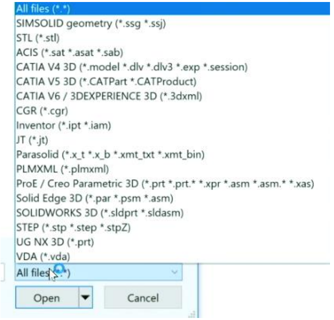
- 中文界面
- 中文帮助文档
- 文件中文命名



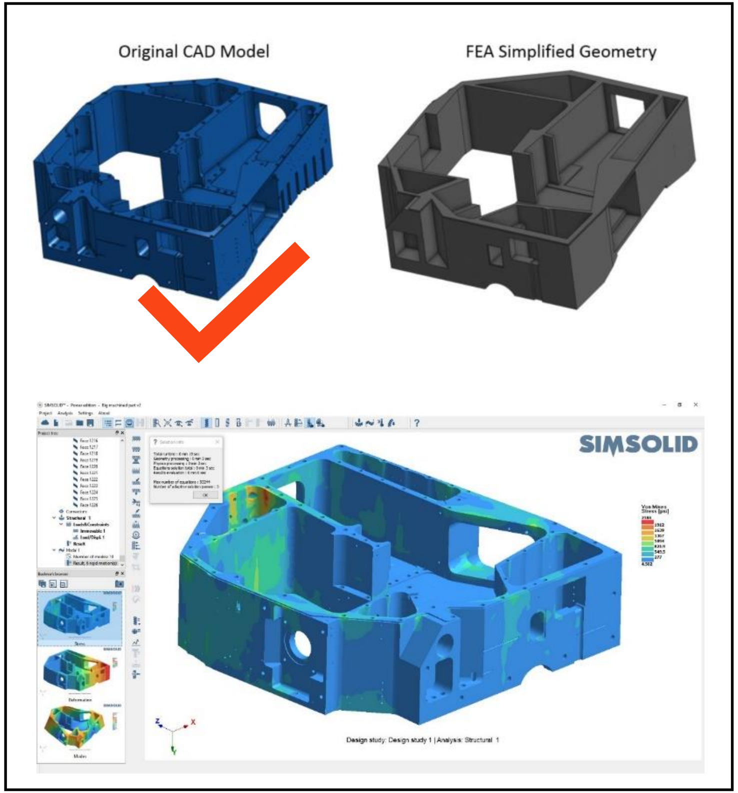


ALTAIR

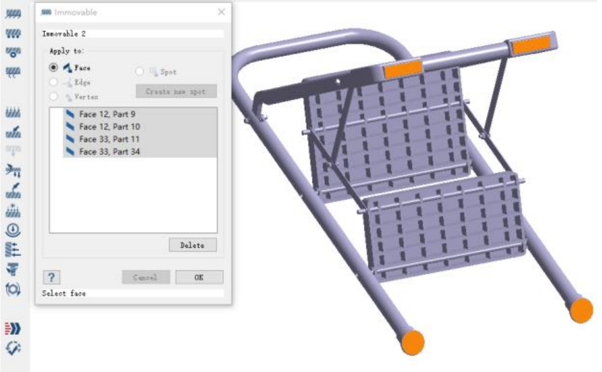
软件特点



支持的CAD格式

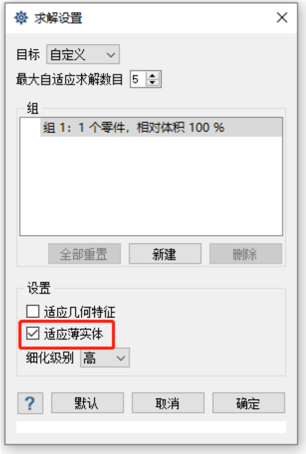


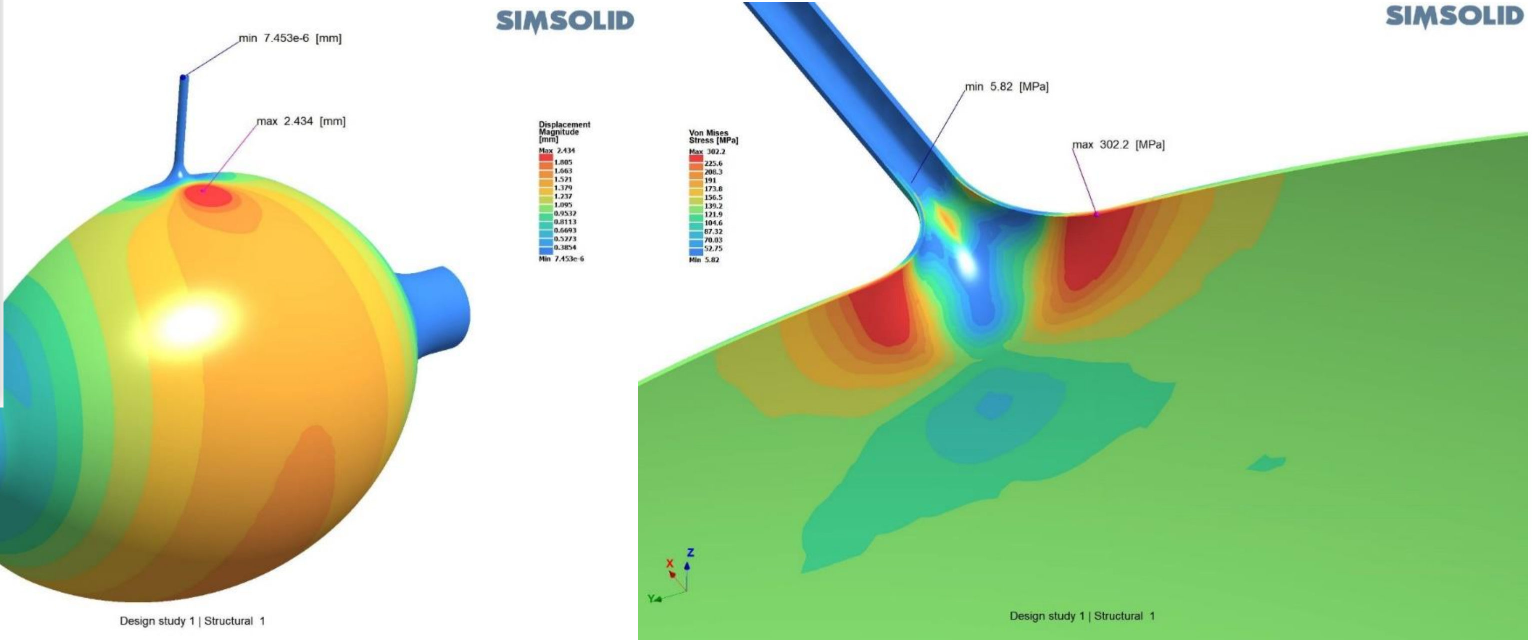
高保真几何
（不简化不几何合并）



载荷与约束施加在
几何上

多通道自适应求解——薄壁结构高精度求解



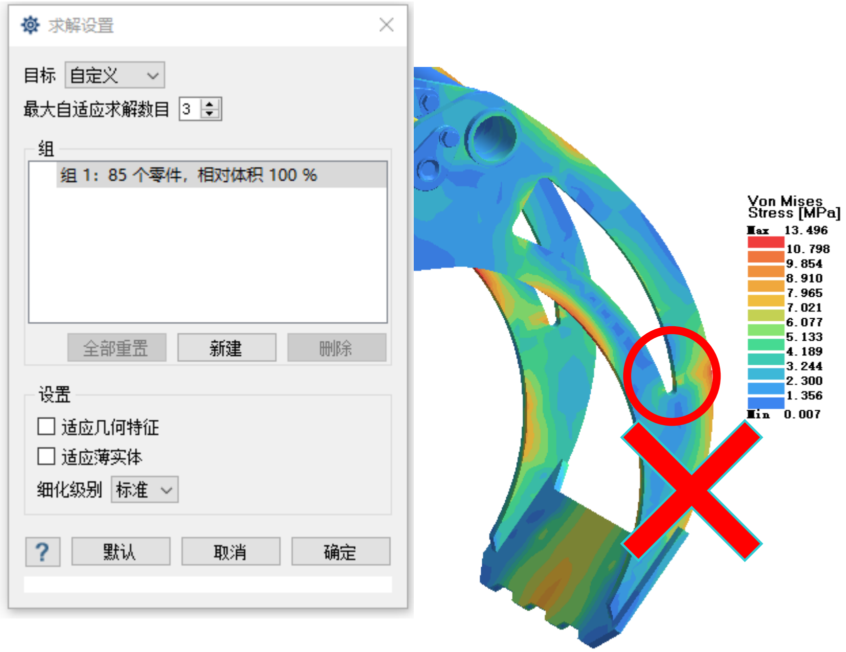


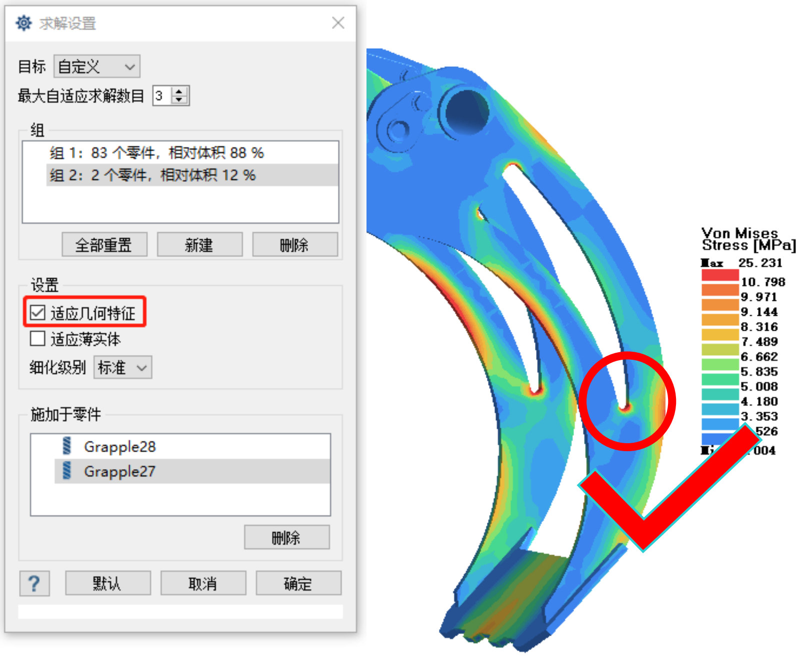
位移

应力

1 MPa 内压, 约束端部 (3 处), 其他边施加对称边界条件

多通道自适应求解——几何特征高精度应力捕捉



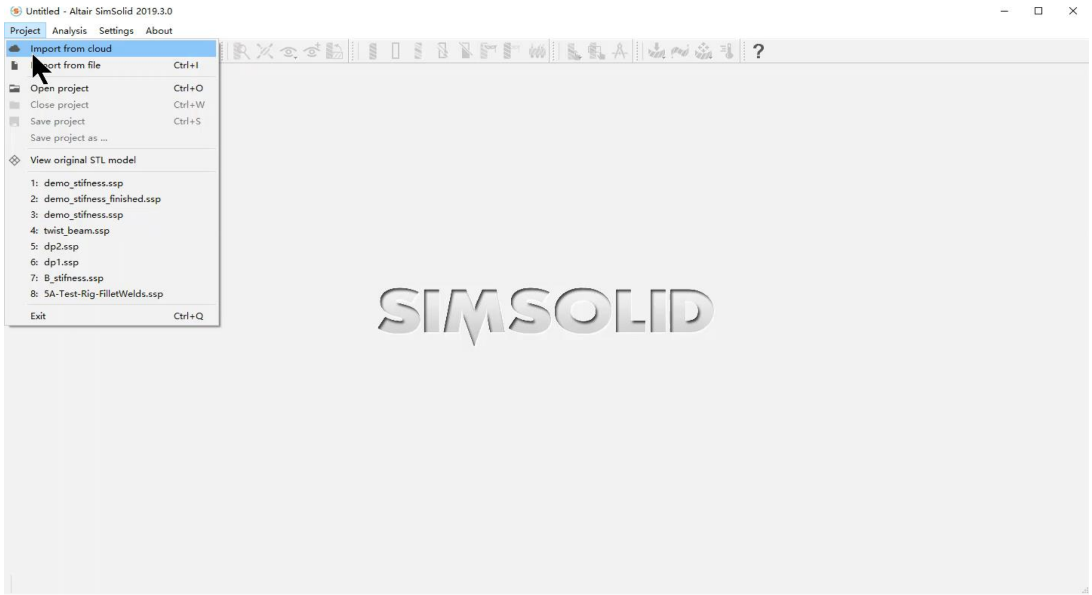


局部应用多通道自适应求解，获取高精度局部应力结果

SimSolid特色功能介绍

快速对比分析

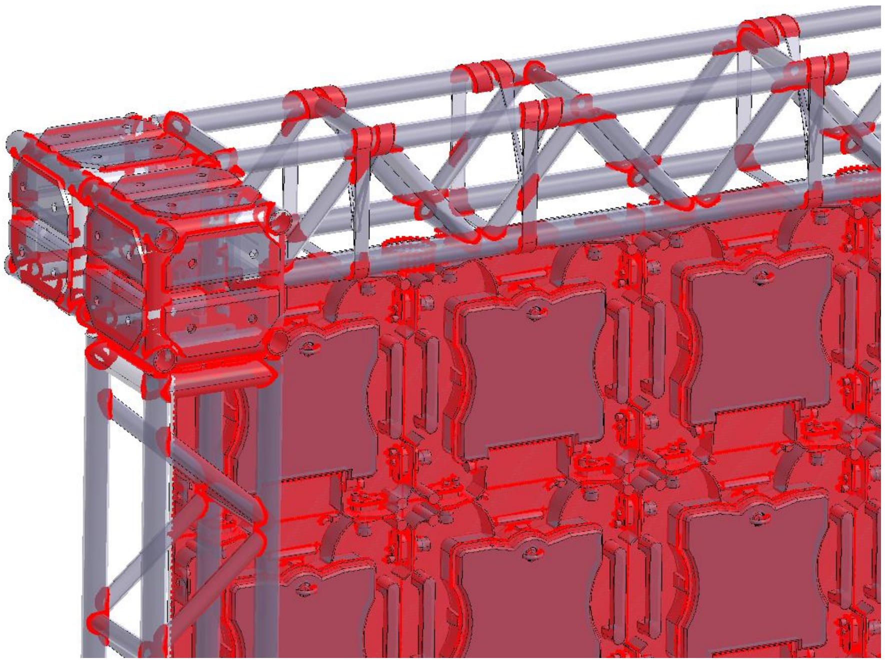
- 完成方案1分析模型；
- 导入方案2几何模型；
- 自动继承方案1的分析设置；
- 求解方案2分析模型；
- 同一页面中快速对比结果。

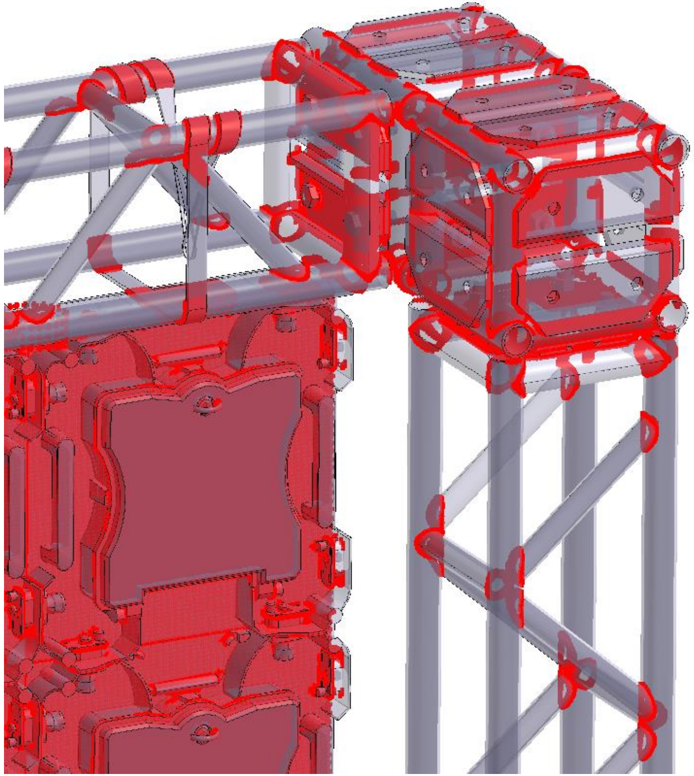


多方案快速对比分析——演示视频

SimSolid特色功能介绍

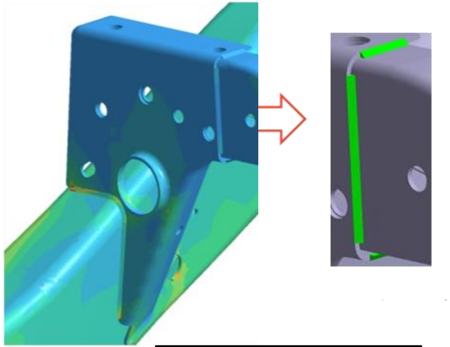
连接自动创建



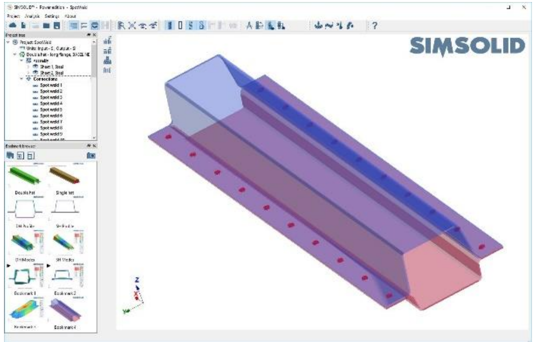


SimSolid特色功能介绍

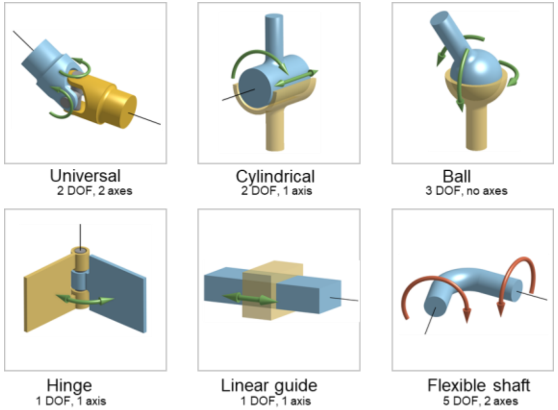
丰富的连接创建



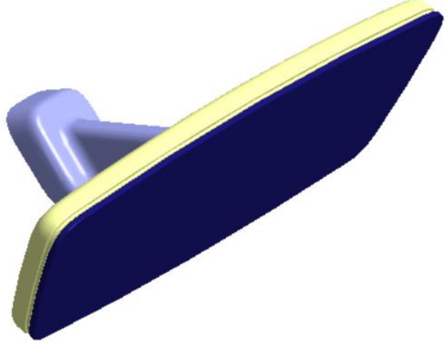
实体焊缝的创建



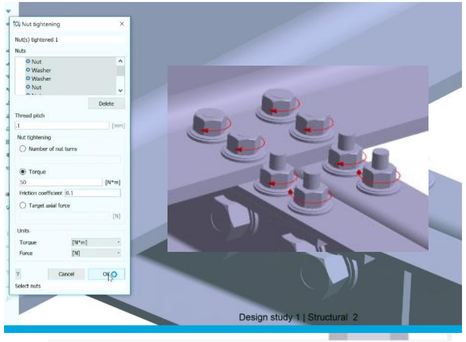
焊点批量导入



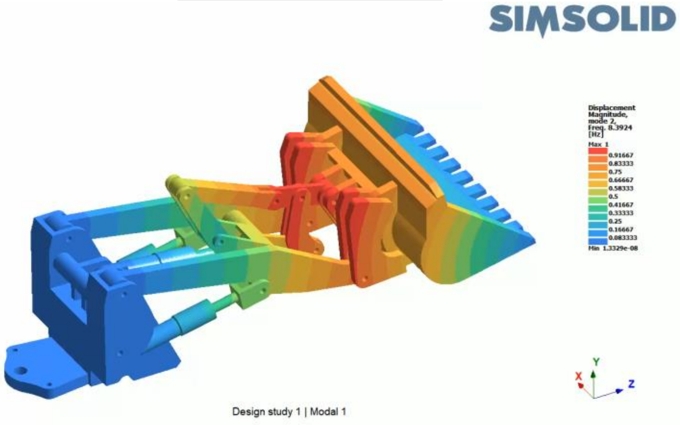
虚拟接头的创建



在镜子/框架间创建粘接接触

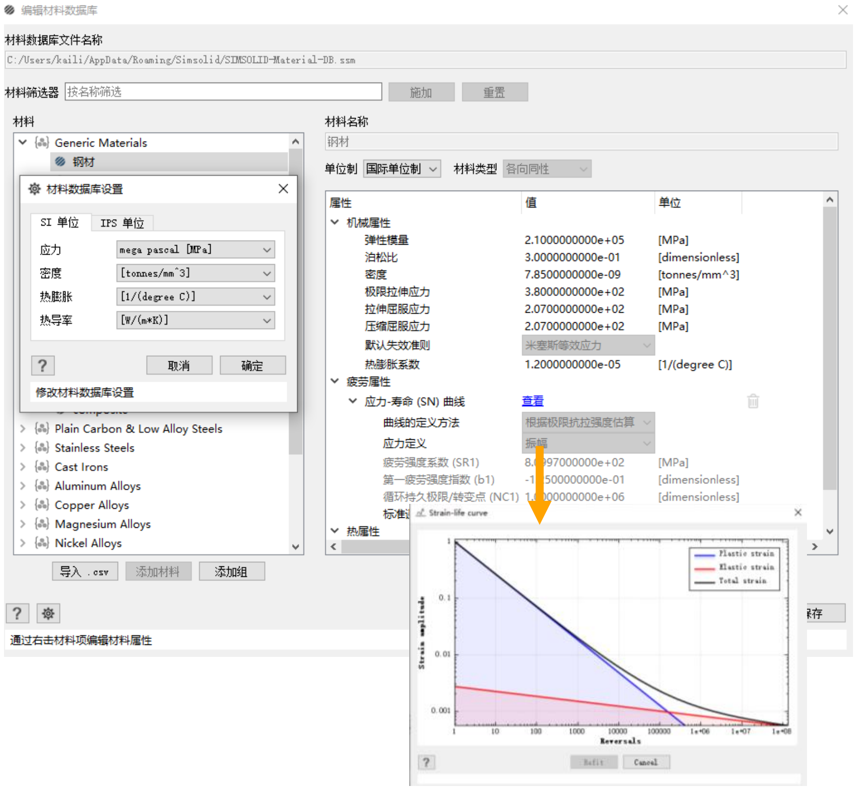


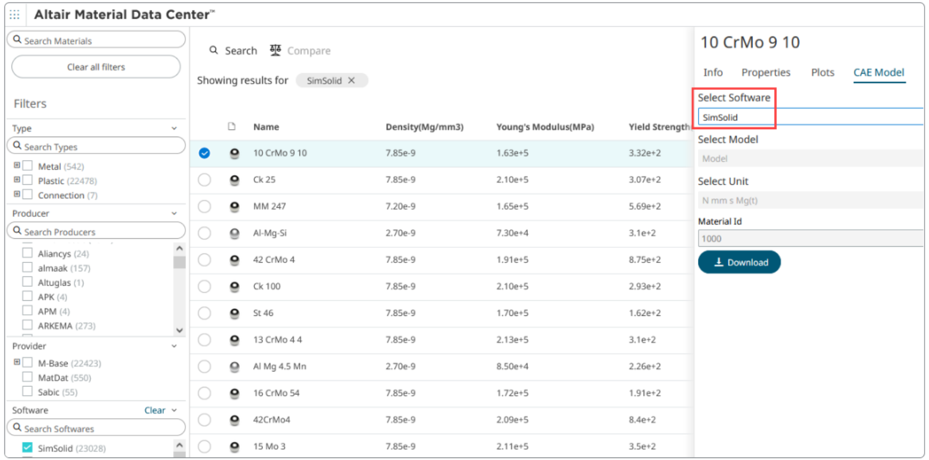
螺栓连接的创建



SimSolid特色功能介绍

可拓展的材料数据库





SIMSOLID 精度比对

Altair官方验证手册

NAFEMS官方验证精度

SimSolid in the News


BENCHMARK

January 2020 issue . . .

- Protection Against Lightning Impacts in Aircraft: The Role of Electromagnetic Simulation
- Let's do Better Than EDA
- Validation of Electromagnetics Simulations for Vehicle-to-Everything Applications using Measured Results
- Stochastic Challenge Problems - A Summary
- Designer Oriented Software - Is it Accurate?
- How Analysis Engineers Spend their Time
- Design of 5G mm-Wave Compatible Covers for High- End Mobile Phones
- Computational Electromagnetics for Nuclear Fusion Engineering and Design
- Insight from a NAFEMS Technical Fellow

THE INTERNATIONAL MAGAZINE FOR ENGINEERING DESIGNERS & ANALYSTS FROM **NAFEMS**

The Electromagnetics Issue

https://www.nafems.org/publications/resource_center/bm_jan_20_1/

Benchmark	Description	Quantity	Target Solution	SimSolid Results	Discrepancy
1	Pressure component	Von Mises stress	534MPa	532MPa	<1%
2	Coil spring	Spring rate	20.8N/mm	20.76N/mm	<1%
3	Skew plate	Maximum principal stress	0.82MPa	0.82MPa	<1%
4	Plate with hole	Maximum principal stress	314MPa	325.7MPa	3.7%
		Minimum principal stress	-114MPa	-117.9MPa	4.2%
5	U-shaped notch	Maximum principal stress	48.2MPa	47.6MPa	1.2%
6	Cantilevered plate	Mode 1	0.42Hz	0.42Hz	<1%
		Mode 2	1.02Hz	1.02Hz	<1%
		Mode 3	2.58Hz	2.56Hz	<1%
		Mode 4	3.29Hz	3.27Hz	<1%
		Mode 5	3.75Hz	3.72Hz	<1%
7	Cantilever under pure bending	Sxx	221MPa	221.7MPa	<1%
		Uz	0.0247m	0.0247m	<1%
8	Cantilever realistic support	S_{VM}	356.5MPa	366.5MPa	2.8%

A summary of results for all benchmarks(NAFEMS)

Check for Other Media Testimonials:

DE247 Digital Engineering

https://www.digitalengineering247.com/article/altair-simsolid-walkthrough/simulate

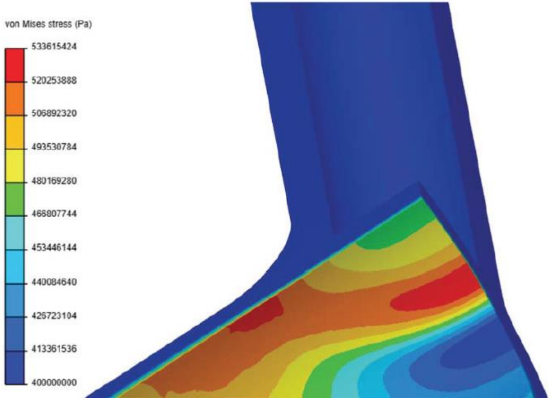
DEVELOP3D

https://www.develop3d.com/reviews/review-altair-simsolid-simulation-CAD-design-engineering

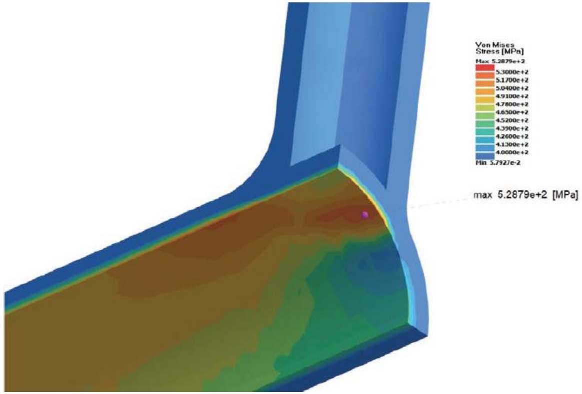
ALTAIR

NAFEMS官方验证精度——受压交叉圆管

受压交叉圆管



1/8模型 传统有限元 max 534

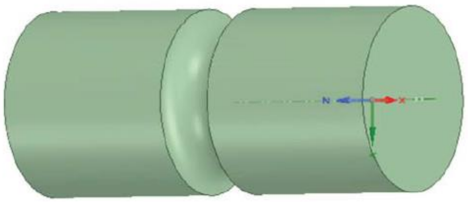


1/8模型 Altair SimSolid max 532
高应力区域位置一致

Benchmark	Description	Quantity	Target Solution	SimSolid Results	SimSolid Discrepancy
1	Pressure component	Von Mises stress	534MPa	532MPa	<1%

ALTAIR

NAFEMS官方验证精度——受拉U形缺口



受拉U形缺口

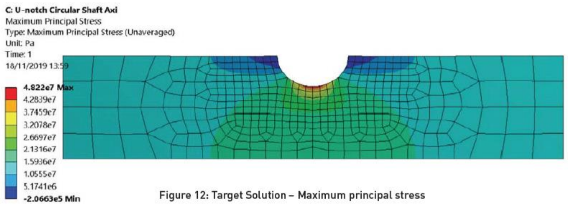
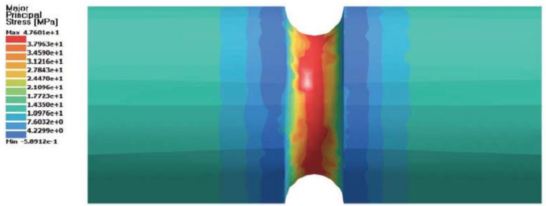


Figure 12: Target Solution – Maximum principal stress

传统有限元 max 48.2



SimSolid max（default solution setting） 42.1
SimSolid max (solution setting pass 8) 48.2

Benchmark	Description	Quantity	Target Solution	SimSolid	
				Results	Discrepancy
5	U-shaped notch	Maximum principal stress	48.2MPa	47.6MPa	1.2%

ALTAIR

SIMSOLID 用户与应用案例

SimSolid全球典型客户

Mercedes-Benz
TOYOTA
RENAULT
CHERY
JAGUAR
LAND ROVER
NISSAN MOTOR CORPORATION
NISSAN
DATSUN
VE COMMERCIAL VEHICLES
A VOLVO GROUP AND EICHER MOTORS JOINT VENTURE
TAFE
Hero
MARK EXHAUST SYSTEMS LIMITED
CEC CHUBU ENGINEERING CO., LTD.
JTEKT
JTEKT CORPORATION
Modern Coach Factory Raebareli

JOHN DEERE
CNH INDUSTRIAL
Andron HANDLING
SERAPID RIGID CHAIN TECHNOLOGY
TATA
TATA INTERNATIONAL DLT
L&T
MHPS
Lamprell
OMRON
LIUFENG
L&L Products
SOTECH design.build
HUAT
HUBEI UNIVERSITY OF AUTOMOTIVE TECHNOLOGY

SHARP
Panasonic
MITSUBISHI ELECTRIC Changes for the Better
DAIKIN
HAVELLS
KYOCERA
Amphenol
杭州安费诺飞凤通信部品有限公司
VIERHOUT Engineering
Driven by Engineering!
sabic
Sports Equipment Senoh
Teikyo University
WANHUA万华

ALTAIR

用户评价

用户评价

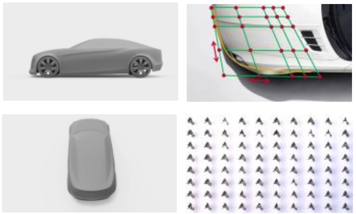
4 天 到 20 分钟	25x 到100x 速度提升	2 天 到 30 分钟
CNH INDUSTRIAL	RENAULT	DON-BUR

所有的用户评价:

https://web.altair.com/simsolid-media-testimonials

研发流程集成思路

仿真加速设计



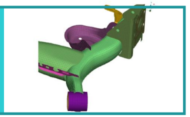
人工设计
参数建模
创成式设计

产品熟悉、经验丰富的设计人员
根据经验进行几何更新

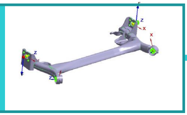
多个CAD设计模型

CAE分析筛选

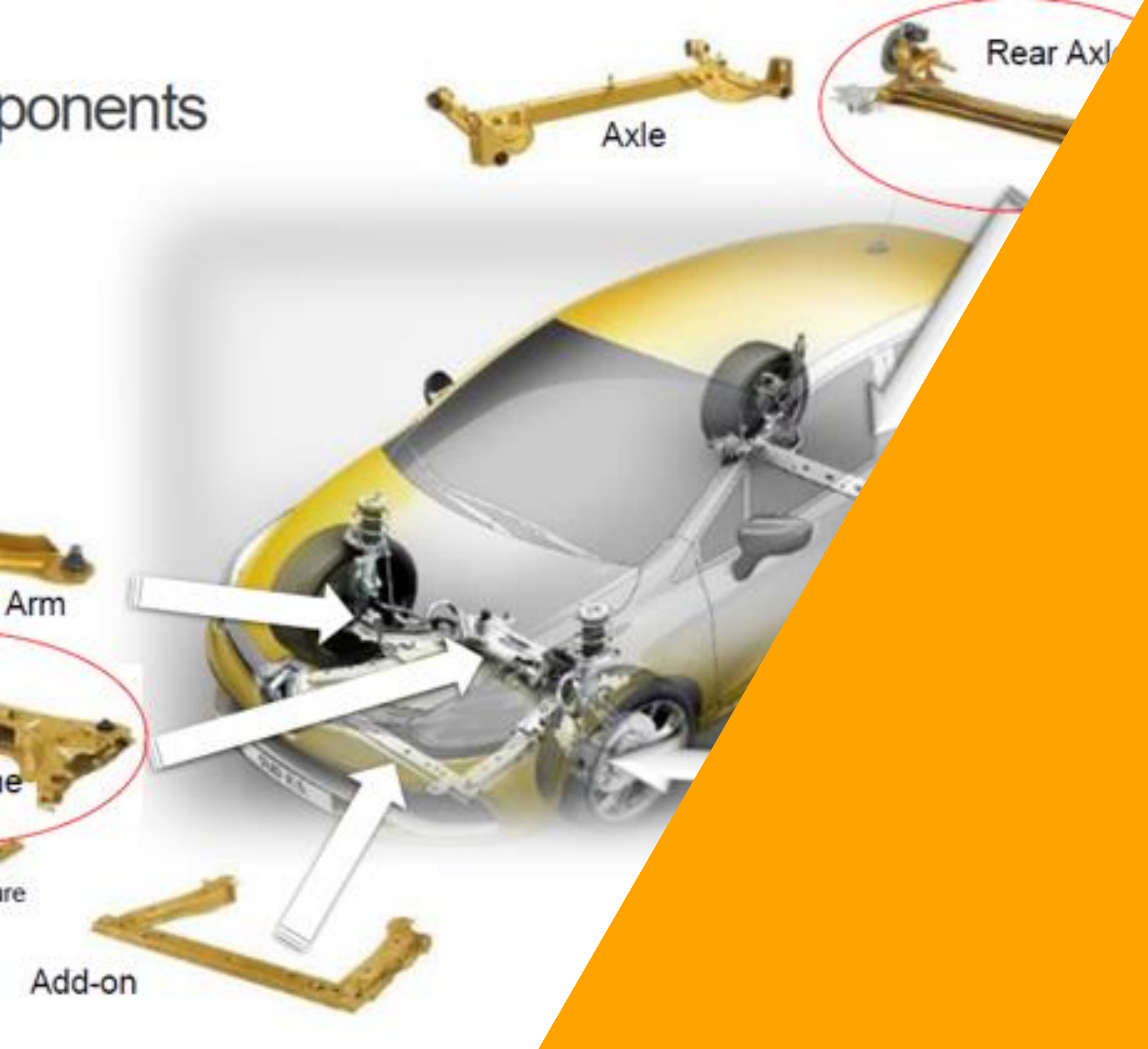
传统CAE工具
耗时耗力



无网格CAE工具
省时省力



可用传统CAE验证



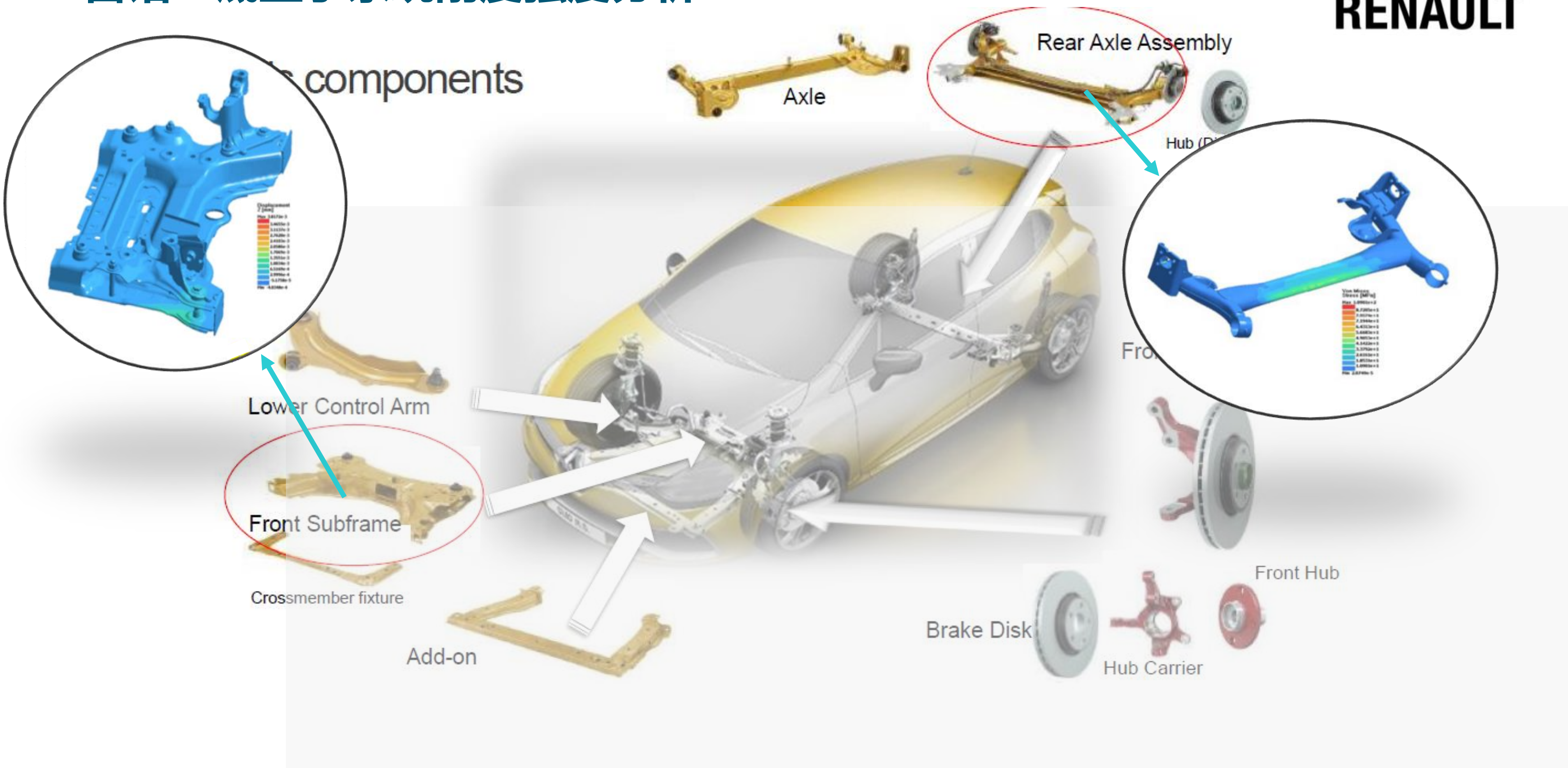
SIMSOLID车辆行业的应用

ALTAIR

SIMSOLID在底盘系统上的应用

ALTAIR

雷诺：底盘子系统刚度强度分析



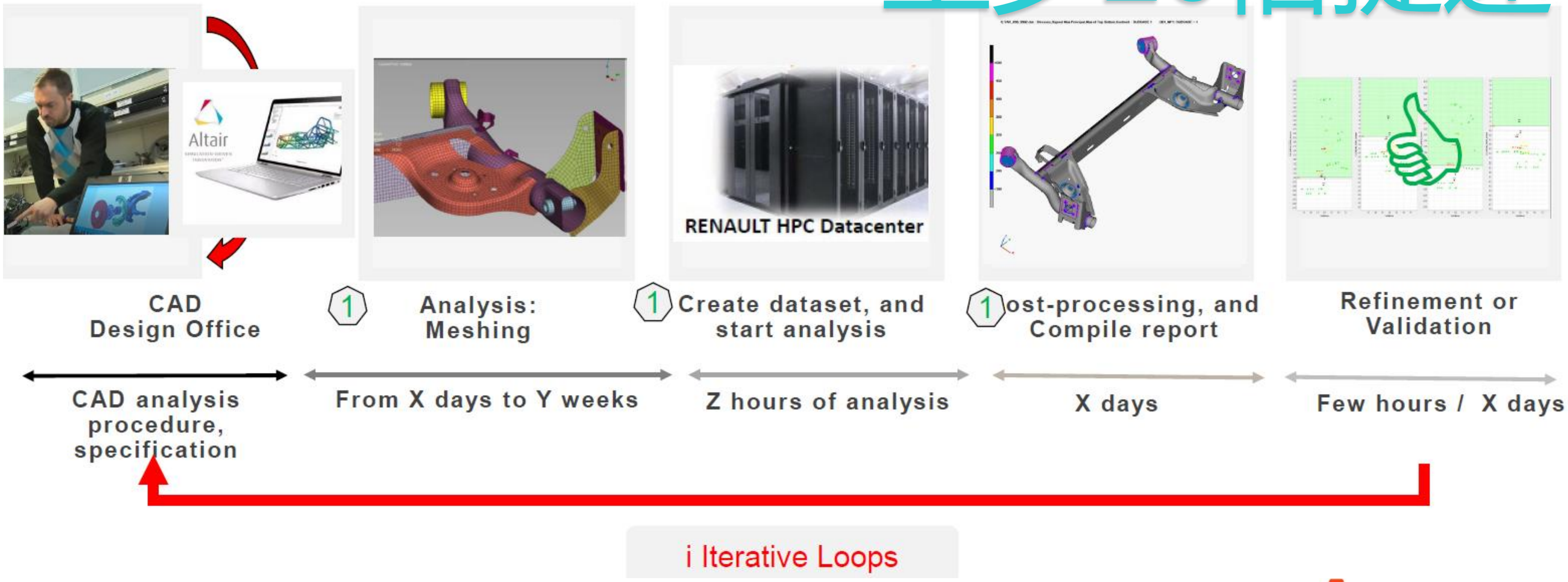
SimSolid应用效果

雷诺应用案例

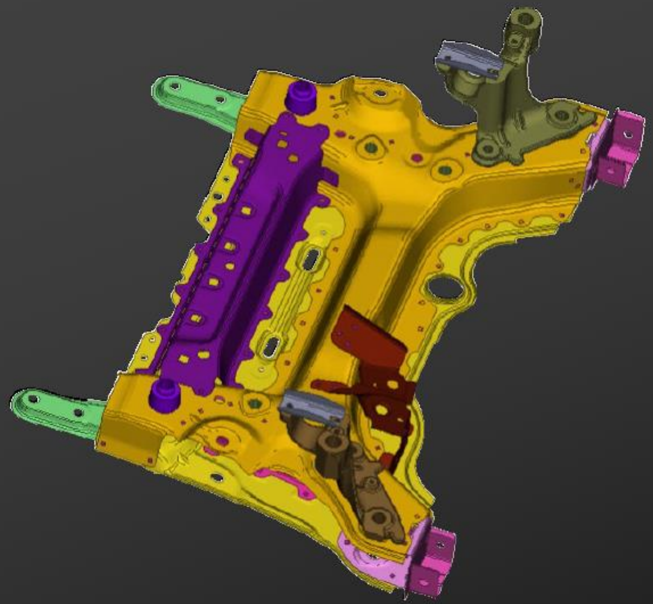
刚度误差5~12%
精度可以满足前期快速设计的要求

至少20倍提速

LEAD-TIME REDUCTION

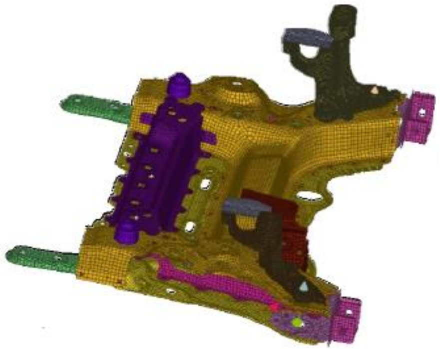


ALTAIR

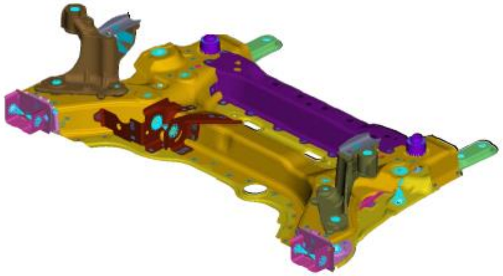


STANDARD METHOD

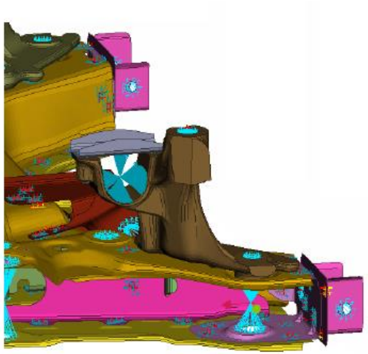
Finite Element



网格划分

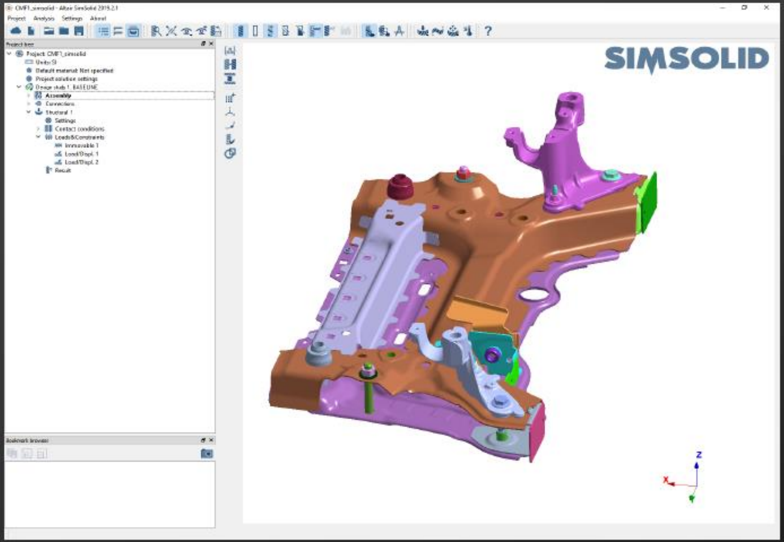


焊点连接与装配



加载和边界条件
线性静力学计算

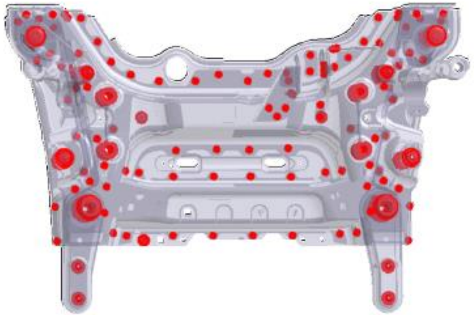
报告汇总



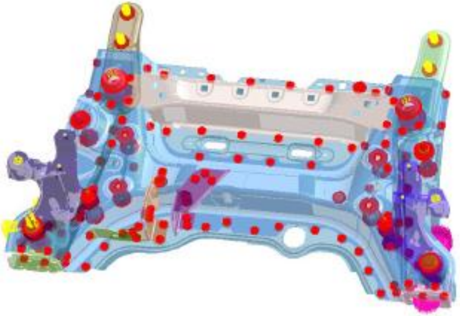
SIMSOLID™
simulation reinvented

METHOD

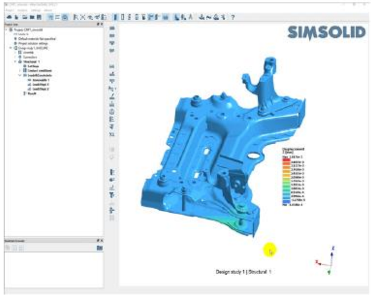
SimSolid求解器基于几何
无需网格划分



焊点连接与装配



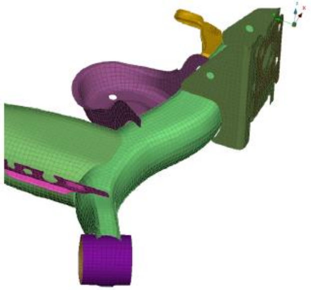
加载和边界条件



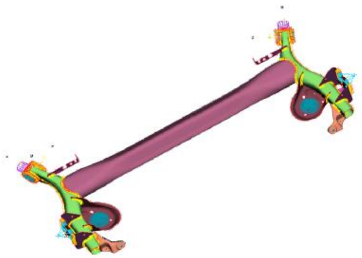
报告汇总

STANDARD METHOD

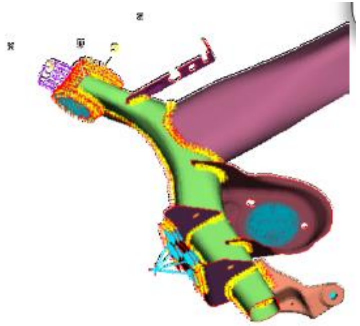
Finite Element



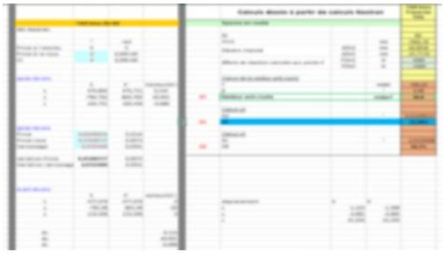
网格划分



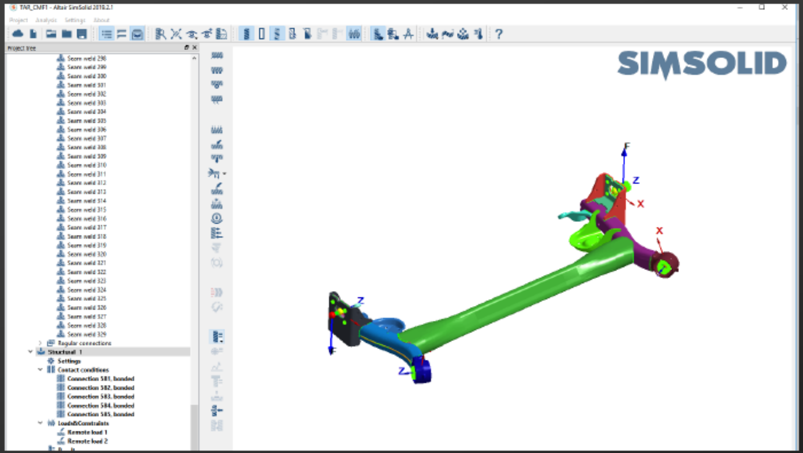
焊点连接与装配



加载和边界条件
线性静力学计算



报告汇总

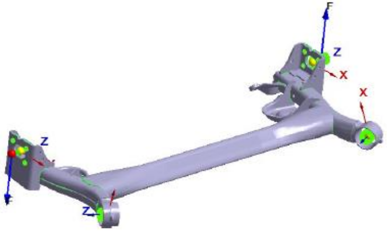


SIMSOLID™
simulation reinvented

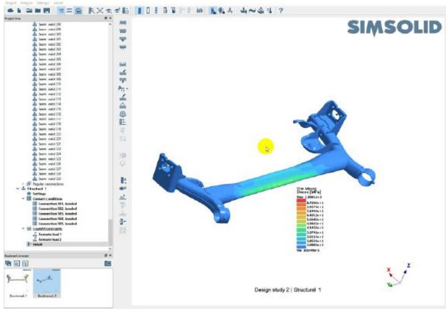
METHOD

SimSolid求解器基于几何
无需网格划分

焊点连接与装配

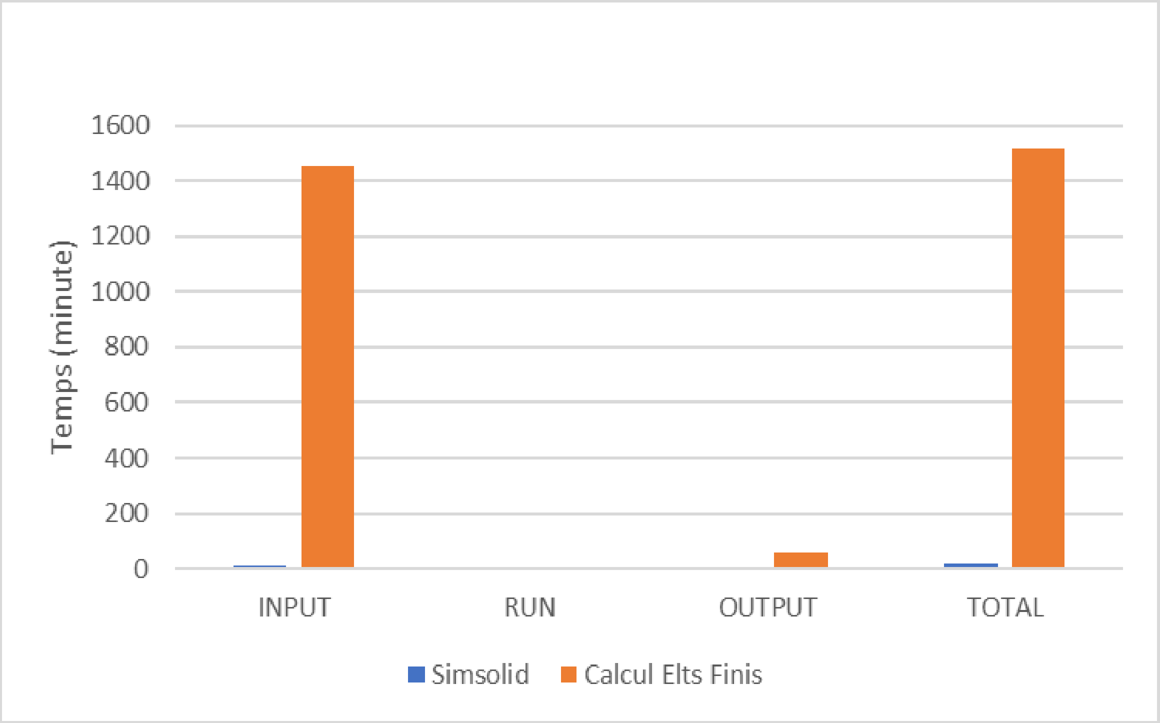


加载和边界条件



报告汇总

雷诺：传统方法和SimSolid方法的比较（开发周期和精度）



单次分析传统方法3天左右，SimSolid不到半小时

每个项目

3轮的设计迭代

传统有限元：1周或以上

SimSolid：2小时

90倍提速

刚度误差<5%

精度可以满足前期快速设计的要求

ALTAIR

雷诺总结的亮点功能

"To us, Altair SimSolid means **efficiency**. While the software quickly provides **accurate simulation** and optimization in one step it does **not require any expert knowledge**. No expertise in analysis is necessary and especially **no meshing** is required. Also, SimSolid helps our designer to shape chassis components with confidence based on the SimSolid simulation."

Anthony Reullier

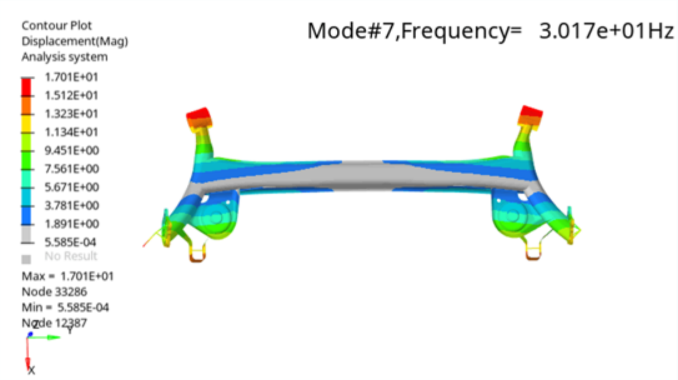
Digital simulation specialist and CAE leader at Renault

上海汇众汽车：底盘子系统刚度强度分析

第一阶模态结果对比

经典方法

第一阶模态 **30.17Hz**



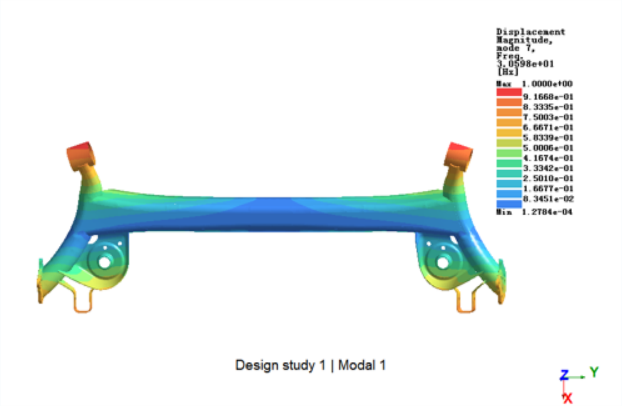
网格处理：6小时，HyperMesh中前处理

网格信息：网格大小4mm，网格数量108792

计算时长：20分钟

SimSolid无网格方法

第一阶模态 **30.59Hz**

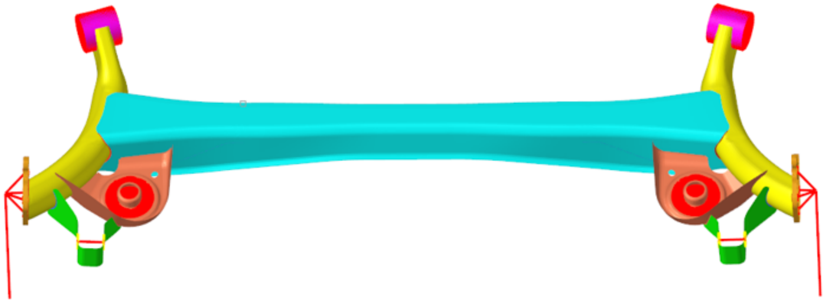


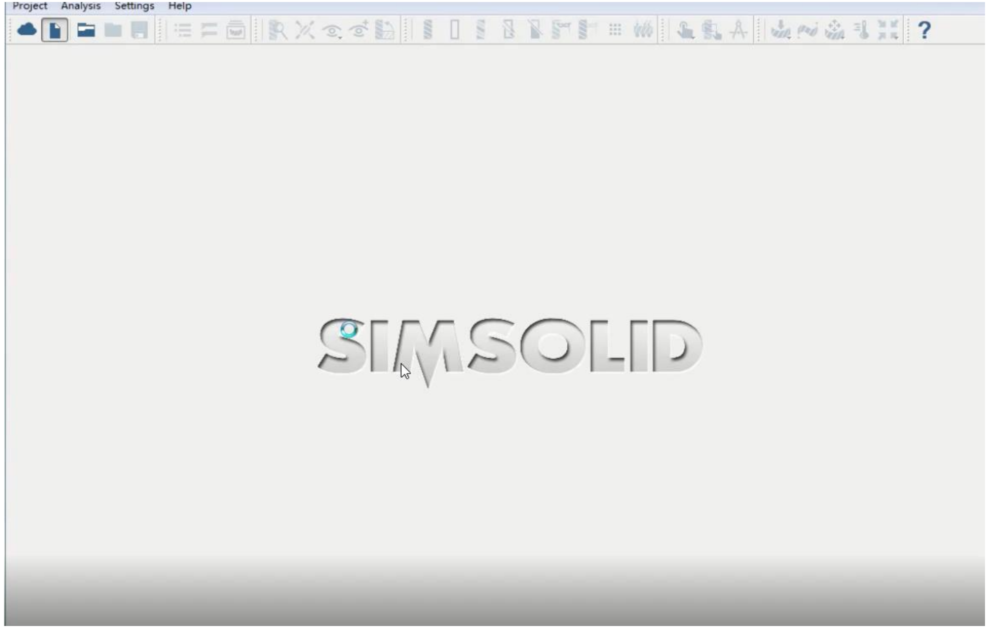
网格信息：无网格

设置时长：5分钟

计算时长：3分钟

上海汇众汽车：底盘子系统刚度强度分析

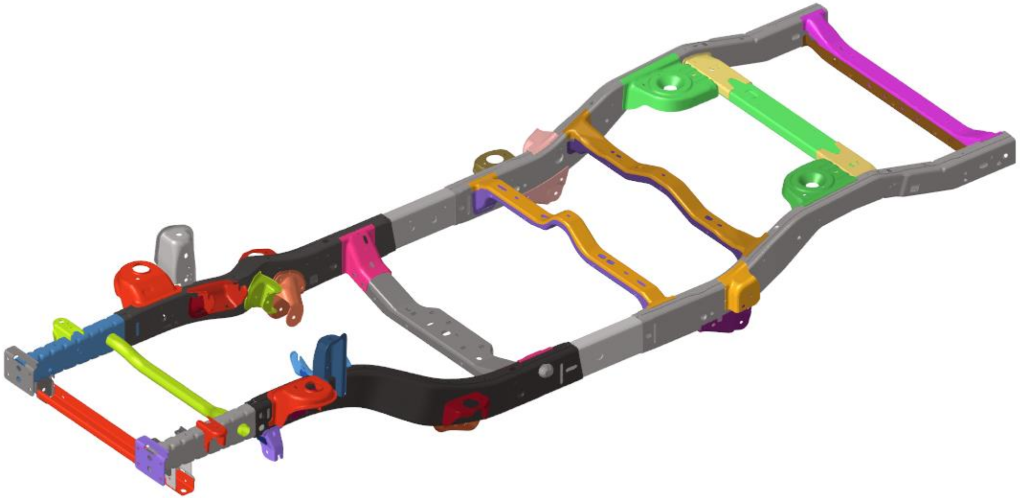




Altair车架的刚度和模态对比分析

车架的刚度和模态分析

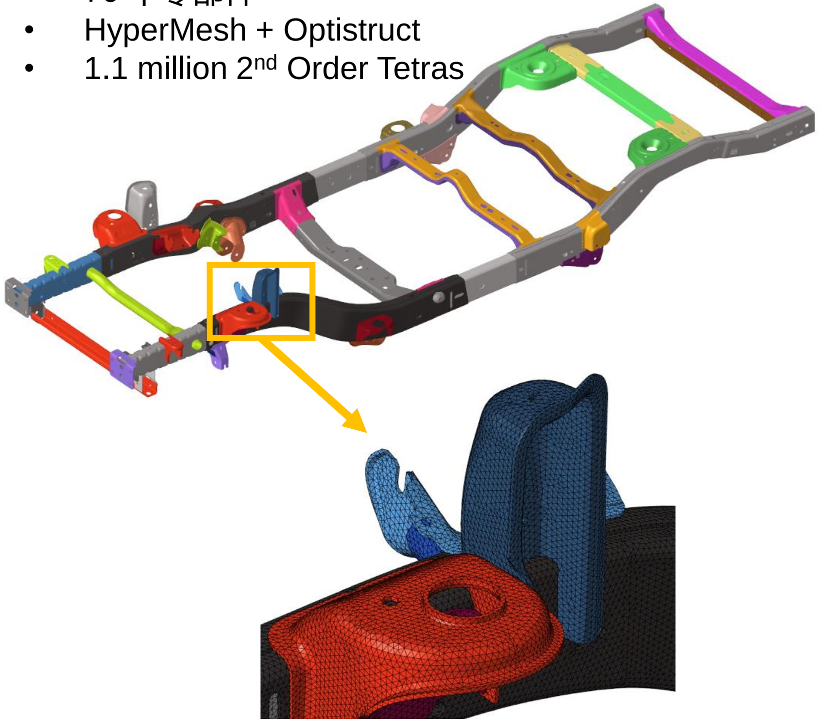
- 70 个左右的零部件
- HyperMesh + Optistruct
- 1.1 million 2nd Order Tetras



求解精度高–模态分析对比

求解概况

- 70 个零部件
- HyperMesh + Optistruct
- 1.1 million 2nd Order Tetras



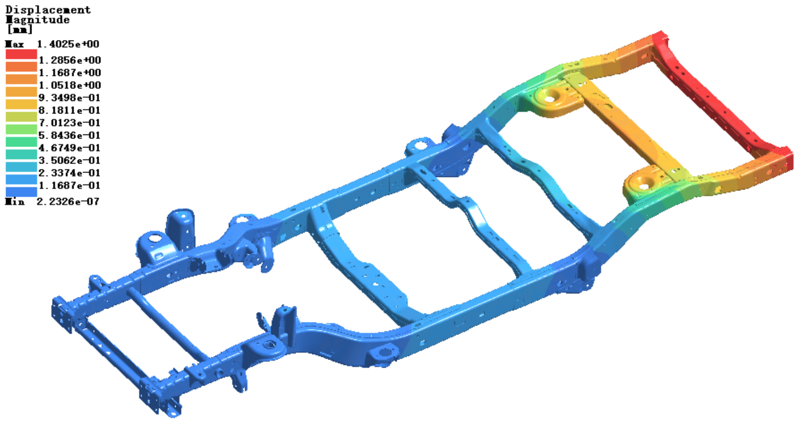
Task	SimSolid	Traditional FEA
Model preparation	5 minutes	5 hours
Model solve	14 minutes	30 minutes
TOTAL TIME	19 minutes	5.5 hours

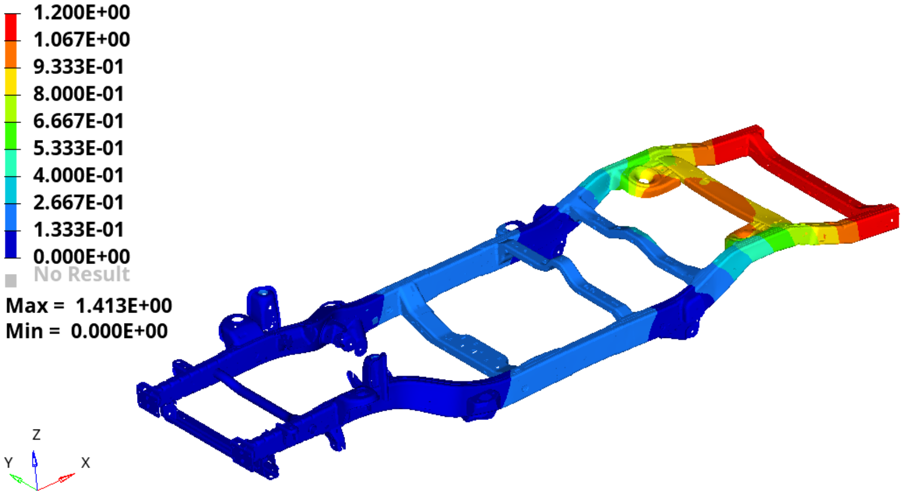
Mode	SimSolid	Optistruct	Error
1	34.031 Hz	33.749 Hz	0.84%
2	39.853 Hz	41.992 Hz	-5.09%
3	57.900 Hz	59.439 Hz	-2.59%
4	65.152 Hz	63.811 Hz	2.10%
5	80.082 Hz	80.317 Hz	-0.29%
6	81.066 Hz	81.729 Hz	-0.81%
7	95.578 Hz	95.598 Hz	-0.02%
8	102.22 Hz	103.20 Hz	-0.95%
9	110.02 Hz	117.88 Hz	-6.67%

求解精度高– 线性静力对比

SimSolid Max Disp ~ 1.40 mm

Optistruct Max Disp ~ 1.41 mm





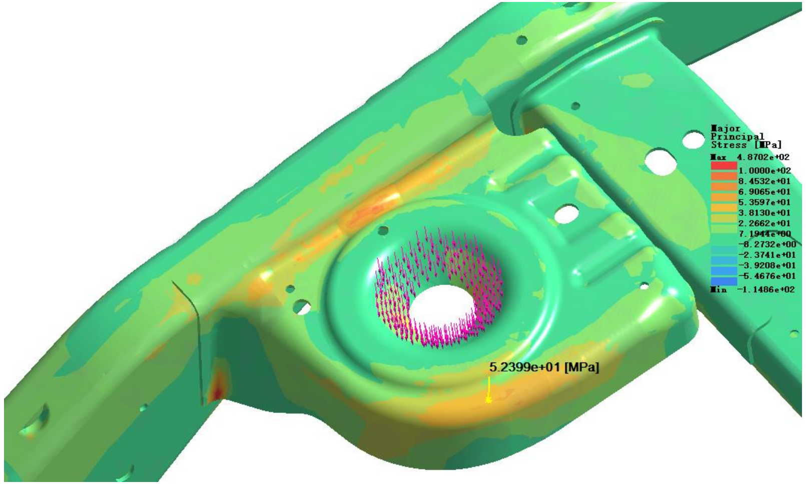
5 solution passes
Adapt to feature, Adapt to thin solids,

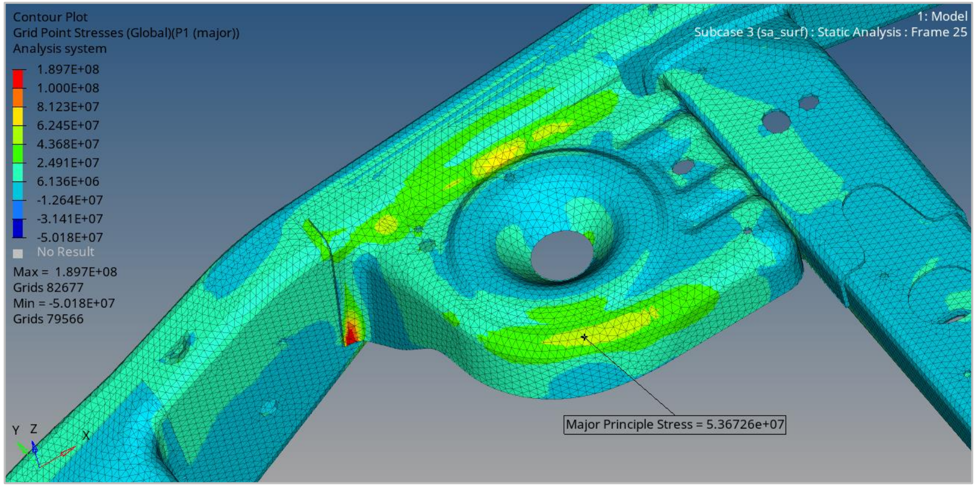
1.1 million 2nd Order Tetras

求解精度高– 线性静力对比

SimSolid Major Principal Stress

Optistruct Major Principal Stress





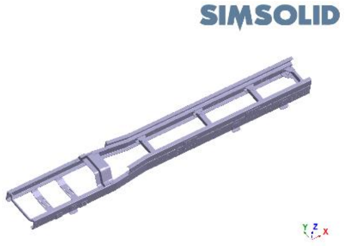
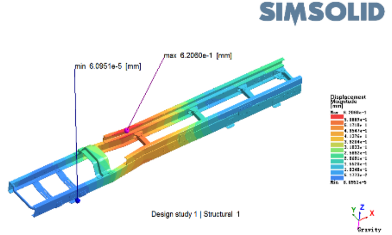
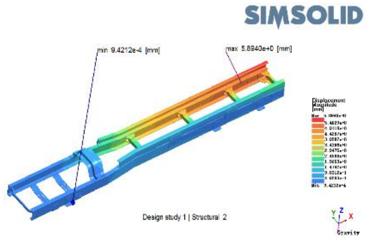
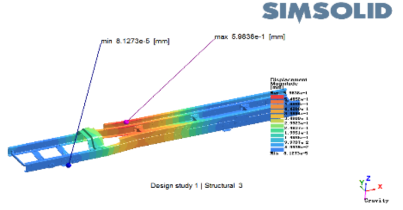
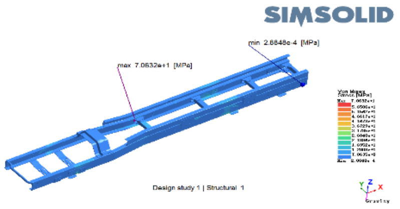
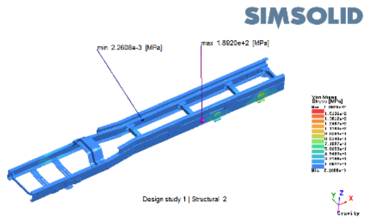
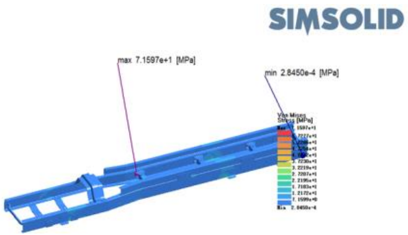
5 solution passes
Adapt to feature, Adapt to thin solids,

1.1 million 2nd Order Tetras

山东交通学院：载货汽车车架静载分析应用

20分钟内快速获得四种工况结果

校核多工况下设计方案性能

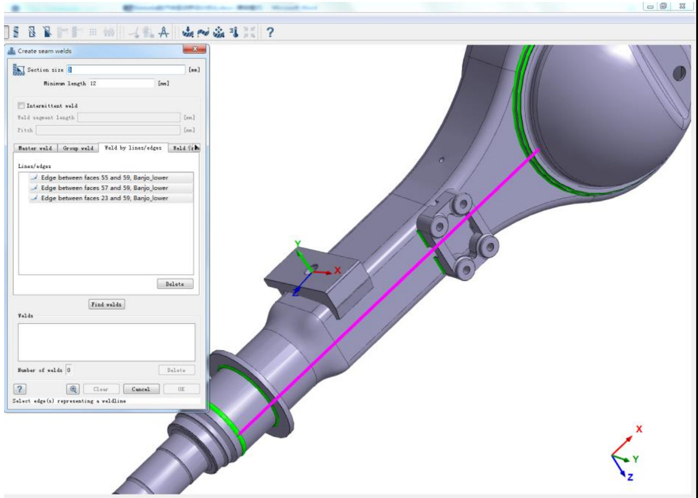
自由模态分析	货车满载弯曲工况	货车满载扭转工况	货车紧急制动工况
			
固有频率（Hz） 9.806 15.021 20.966 21.688 22.567 25.413			

合肥美桥：驱动桥桥壳开发设计

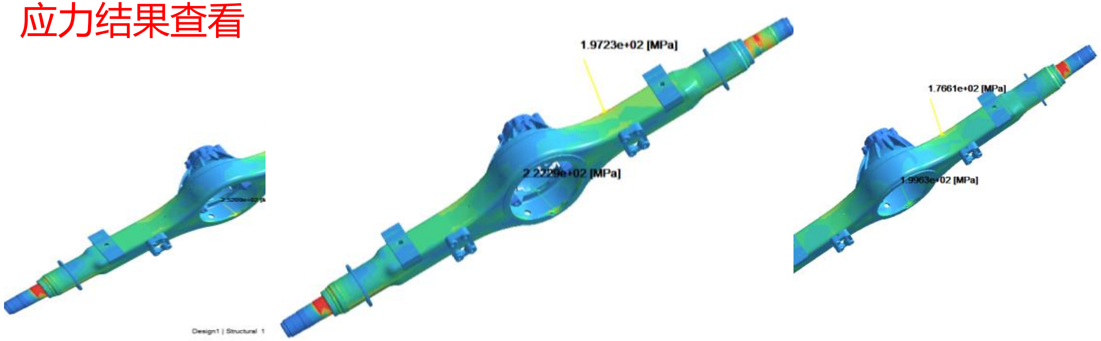
SimSolid辅助提升疲劳寿命

批量自动创建实体焊缝、与疲劳实验结果吻合

实体焊缝建模



应力结果查看

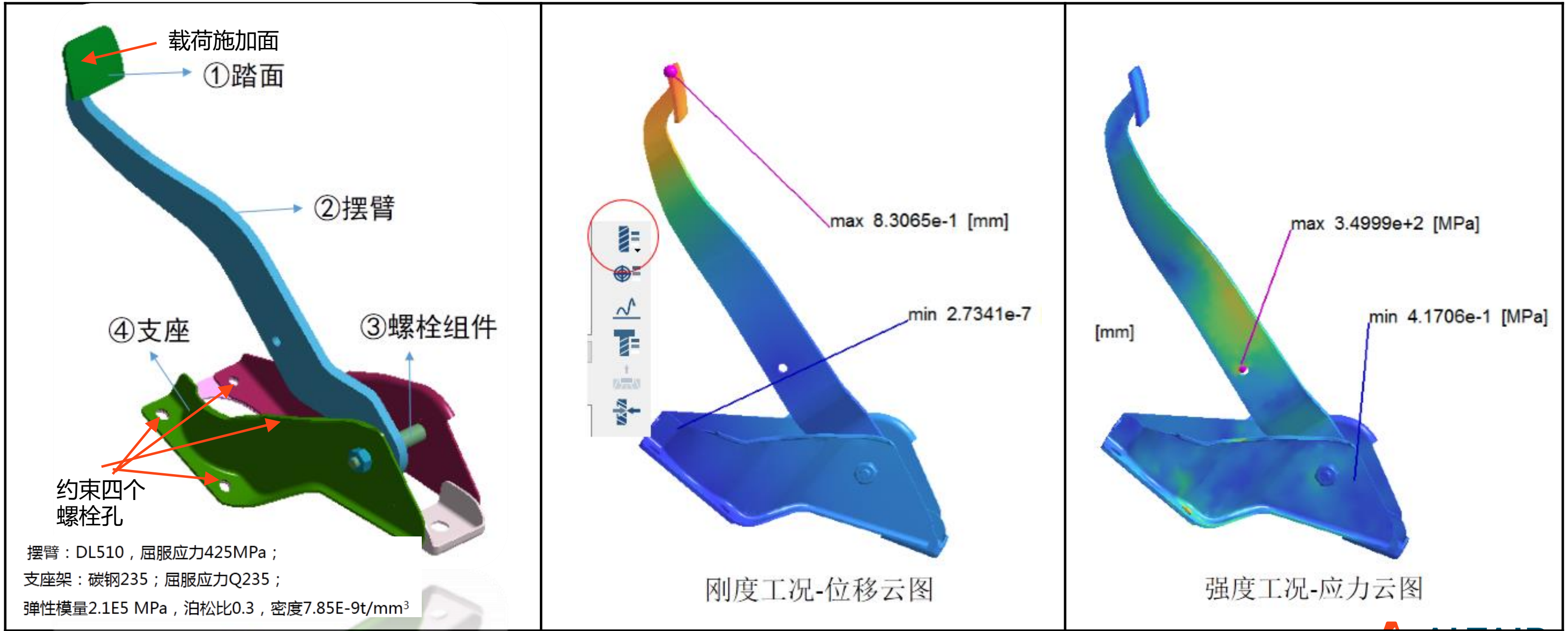


设计方案	整体变形 mm	应力 MPa		台架试验 Cycles	方案变更
		中心缺口	钢托侧		
Design1	3.284	252	204	52.1w	
Design2	3.224	222	197	65.2w	1 基础上增加一个 6mm 厚环
Design3	2.607	199	177	>80w	2 基础上在钢托下方增加内衬板

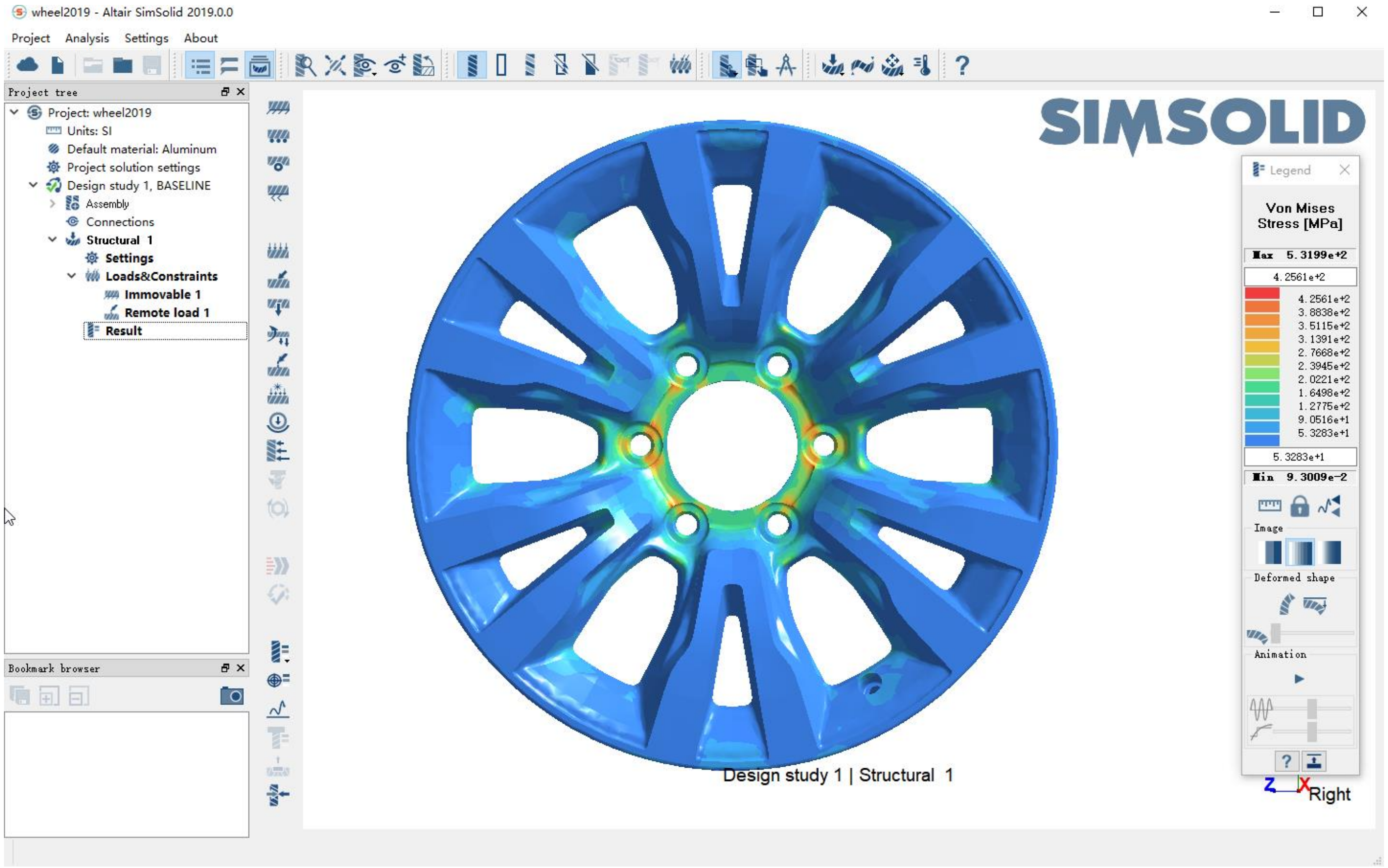
唐山工业职业技术学院：制动踏板的静强度刚度分析

SimSolid中10分钟完成制动踏板的静强度刚度分析

无需网格划分、自动创建接触



某知名轮毂企业：轮毂强度分析



考虑螺栓连接

接触非线性

滚动载荷

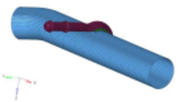
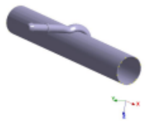
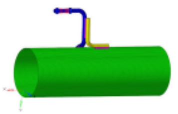
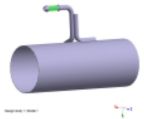
弯曲载荷

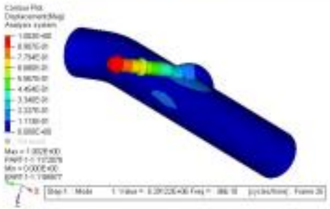
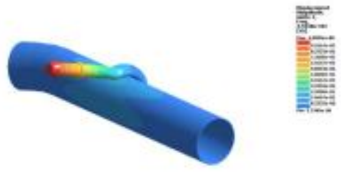
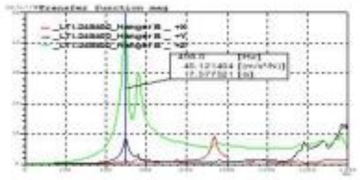
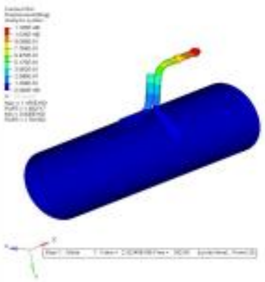
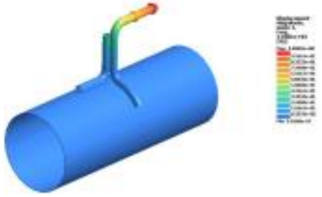
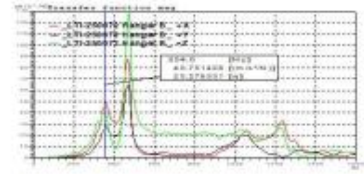
24个工况和FEA对标

应力和位移都在工程误差内

ALTAIR

佛吉亚：排气系统模态分析

	传统有限元求解器	SimSolid
模型 1： Pipe：直径 5，壁厚 1.0 Hanger：直径 12，实心 单位毫米	 pipe mesh：四面体片体网格（8105） Hanger mesh：六面体实心网格（1642） weld mesh：六面体实心网格（96） **预计工作时间：1000 秒**	 创建 connection:5 **预计工作时间：6 秒**
模型 2 Pipe：直径 60，壁厚 1.0 Hanger：直径 12，壁厚 1.5 Support：直径 12，壁厚 1.5 单位毫米	 Pipe mesh：四面体片体网格（15291）； Hanger mesh：四面体片体网格（3205） Weld mesh：六面体实心网格（144） Mass：配重点（1） **预计工作时间：1200 秒**	 创建 connection:5 Distributed mass **预计工作时间：20 秒**

传统有限元求解器	SimSolid	与实验结果做对比
 第一阶固有频率仿真结果： 366hz	 第一阶固有频率仿真结果： 456hz	 实验结果：498hz **SimSolid 仿真结果更贴近实验**
 第一阶固有频率仿真结果： 243hz	 第一阶固有频率仿真结果： 306hz	 实验结果：354hz **SimSolid 仿真结果更贴近实验**

SIMSOLID在车身系统上的应用

ALTAIR

菲亚特汽车公司：车辆耐久分析有限元方法与SimSolid无网格精度比对

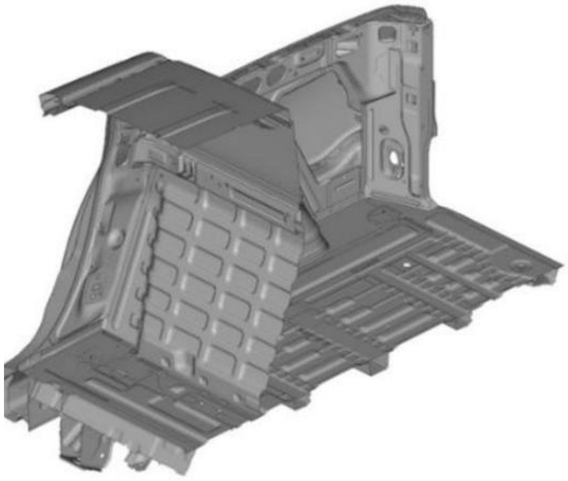


Figure 2. Car body of a pickup vehicle

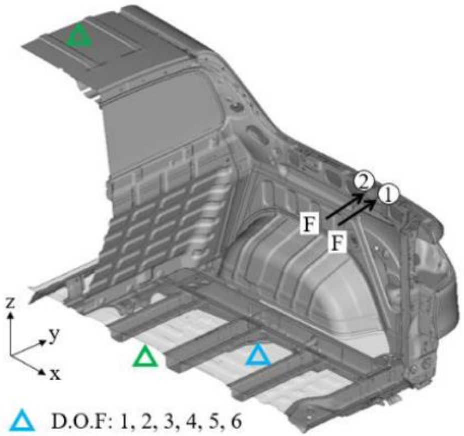


Figure 3. Car body boundary conditions

Table 1. Results of the stiffness analysis

Point	Experimental k [N/mm]	OptiStruct k [N/mm]	OptiStruct % error	SimSolid k [N/mm]	SimSolid % error
1	375.5	351.3	-6.4	393.7	+4.8
2	379.8	363.2	-4.4	387.1	+1.9

There are many methods which can be applied in the construction of shape functions. According to Liu [9], these methods fall into the following categories: integral representation methods (Smoothed particle hydrodynamics method, Reproducing kernel particle method, General reproducing method), series representation methods (Moving least square methods, Point interpolation methods, Partition of unit methods, Least squares methods, Finite element methods), differential representation methods (Finite difference method, Finite point method) and gradient smoothing methods.

It is normally used for the meshfree formulation the strong-form, weak-form or weakened-weak. Each one of them combined with the chosen method for shape function construction, gives to the meshfree method interesting and unique aspects.

The simulations of this paper applying the meshfree method were performed in the commercial software SimSolid [10], which type of method is not available for the users.

It is capable to run structural, dynamic and thermal analyses. To use meshfree method in this program, the model to be analyzed even in three-dimensions does not need to have simplifications, being thus closer to the real problem. In order to improve the accuracy, the shape function is defined in respect of the kind of problem besides the nodal distribution which is done using an adaptive algorithm that can enrich locally or globally the problem domain. This software can also automatically recognize the connections (e.g., welds, bolts) and the existing contacts on assemblies.

3 Simulations

In this section, three typical automotive durability analyses are presented comparing the results performed with the numerical methods related previously with the experimental data obtained from the bench tests. These simulations employ a problem of a linear and a nonlinear static nature and also a dynamic case applying a modal analysis.

These samples are all composed of steel materials with their sheet metals connected by spot welds, adhesives and bolts. The boundary conditions are shown with the fixed degrees of freedom (D.O.F), being 1, 2 and 3 the translations in the X, Y and Z directions, respectively, and 4, 5 and 6 the rotations in the same directions with all components of motion referred to the global axis of the figures. The applied load F is also shown in the static cases.

3.1 Linear static analysis

Figure 2 displays a car body rear compartment of a pickup vehicle, which has some holes to fix the cargo retaining hooks. A stiffness analysis was performed in two hook anchorage points, position 1 and 2, and using the boundary conditions as Fig. 3 illustrates. The sheet metal components were characterized by a Young´s modulus E = 205000 MPa and a Poisson´s ratio ν = 0.3.

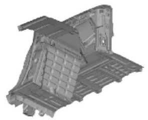


Figure 2. Car body of a pickup vehicle

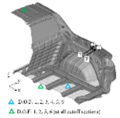


Figure 3. Car body boundary conditions

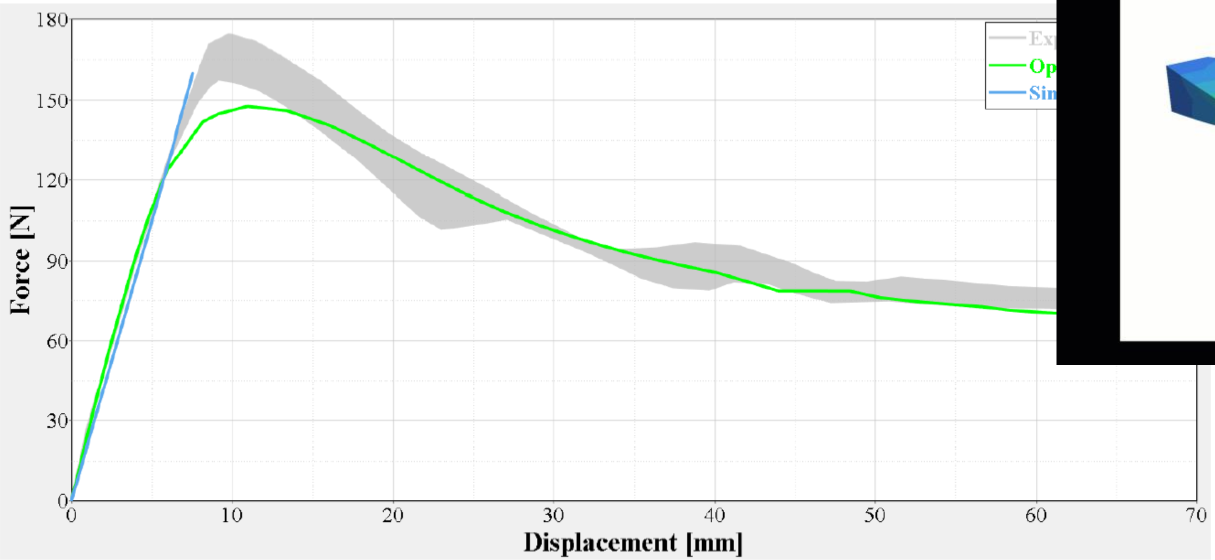
The results of this test are presented in Tab. 1 with the stiffness of each point denoted as the symbol *k*, and the percentage error from the numerical methods to the experimental test.

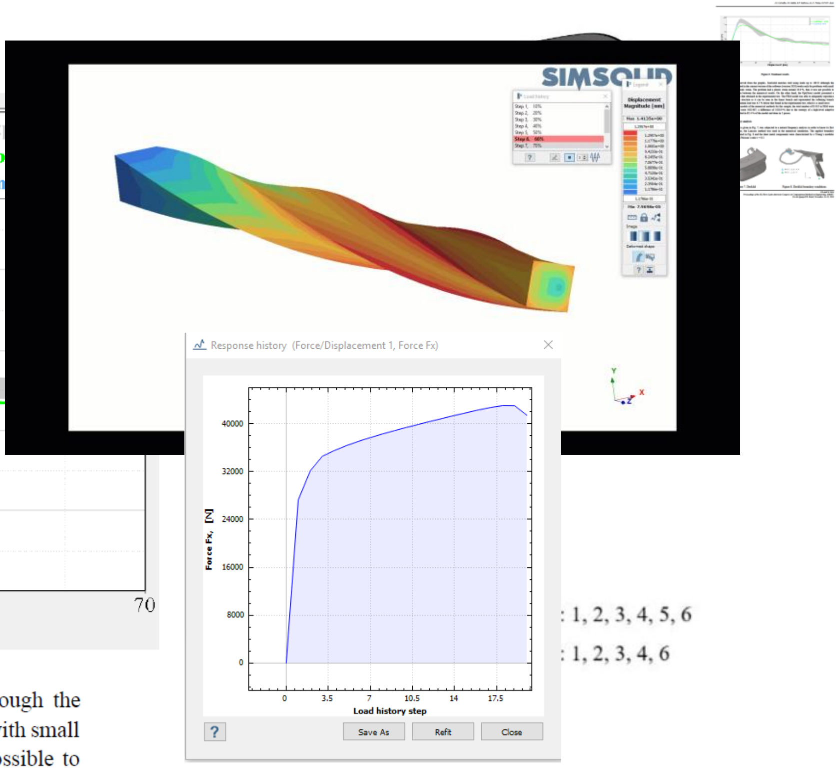
CILAMCE 2020
Proceedings of the XLI Ibero-Latin-American Congress on Computational Methods in Engineering, ABMEC
Foz do Iguaçu/PR, Brazil, November 16-19, 2020

ALTAIR

菲亚特汽车公司：车辆耐久分析有限元方法与SimSolid无网格精度比对

精度对比曲线





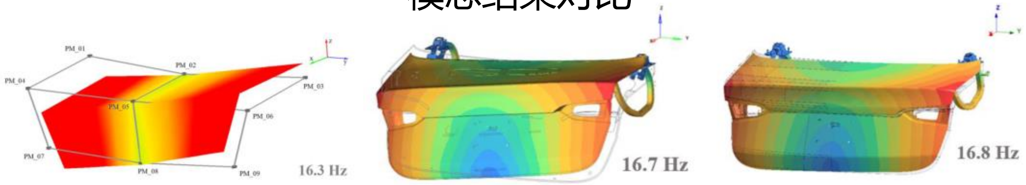
As it can be observed from the graphic, SimSolid matches well using loads up to 160 N although the implementation presented in the current version of the software (version 2020) works only for problems with small strain, i.e. 4.0 % of plastic strain. The problem had a plastic strain around 10.0 %, thus it was not possible to analyze the comparison between the numerical results. On the other hand, the OptiStruct model presented a behaviour very close to that obtained in the experimental test. The FEM model was able to adequately reproduce the linear state of the structure as it can be seen in the linear branch and represented the softening branch satisfactorily. The maximum load was 6.5 % below that found in the experimental test, which is a small error.

非线性比案例模型

ALTAIR

菲亚特汽车公司：车辆耐久分析有限元方法与SimSolid无网格精度比对

模态结果对比



Mode	Experimental	OptiStruct		SimSolid	
	[Hz]	[Hz]	% error	[Hz]	% error
First	16.3	16.7	+2.5	16.8	+3.1

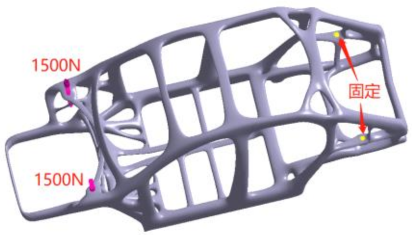
In linear static analysis, according to Tab. 1, the results of both softwares/methods are considered satisfactory, since the difference in comparison to the experimental test was within the acceptable range. In the nonlinear analysis, the FEM model reproduced the force-displacement curve of the experimental test almost completely (Fig. 6), only the maximum load was 6.5 % lower. Finally, in the natural frequency analysis, the numerical models were able to reproduce the first mode of the experimental test and also very accurate frequency values were found, as it can be seen in Tab. 3.

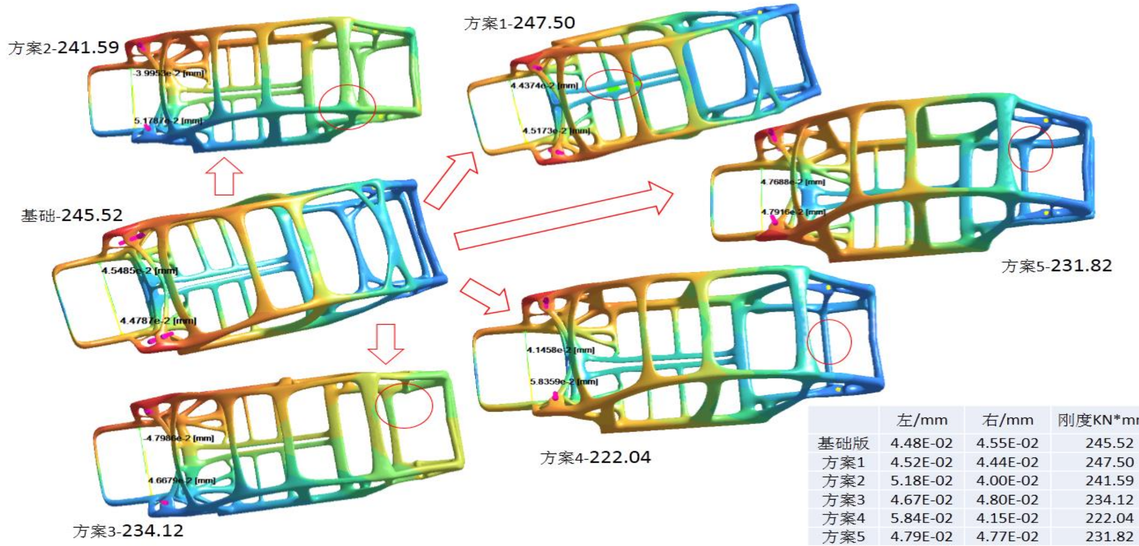
In general, the models analyzed with both softwares reproduced the tests well, presenting results similar to those obtained experimentally.

长城汽车股份有限公司：白车身拓扑方案快速验证分析

SimSolid15min内验证6个拓扑优化方案

加速拓扑方案的转化

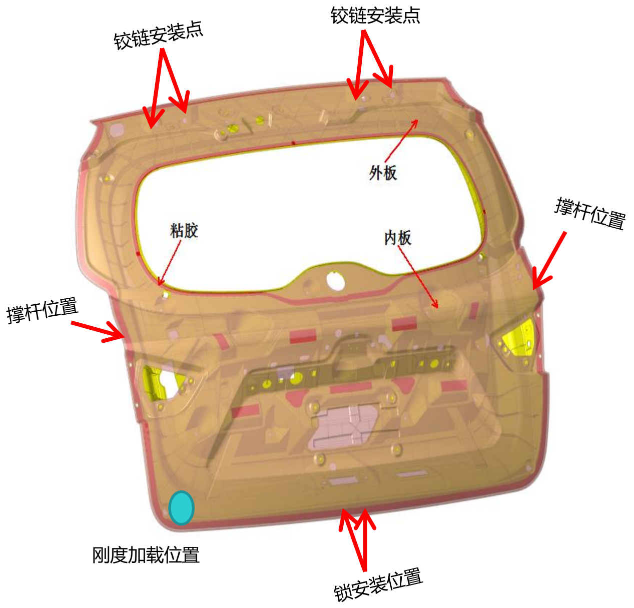




	左/mm	右/mm	刚度KN*mm/°
基础版	4.48E-02	4.55E-02	245.52
方案1	4.52E-02	4.44E-02	247.50
方案2	5.18E-02	4.00E-02	241.59
方案3	4.67E-02	4.80E-02	234.12
方案4	5.84E-02	4.15E-02	222.04
方案5	4.79E-02	4.77E-02	231.82

成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析

模型与工况



全塑尾门，以塑代钢。内板材料为PP+LGF40，外板材料为PP+EPDM-TD20，分析如下四种工况：

1、强度

铰链安装点、锁安装位置约束dof1~6，撑杆位置施加300N载荷，同时施加4.5G（向上）重力场载荷。

2、刚度

铰链安装点约束dof1~6，锁安装位置约束dof1~3；在左侧缓冲垫区域施加+X方向50N载荷。

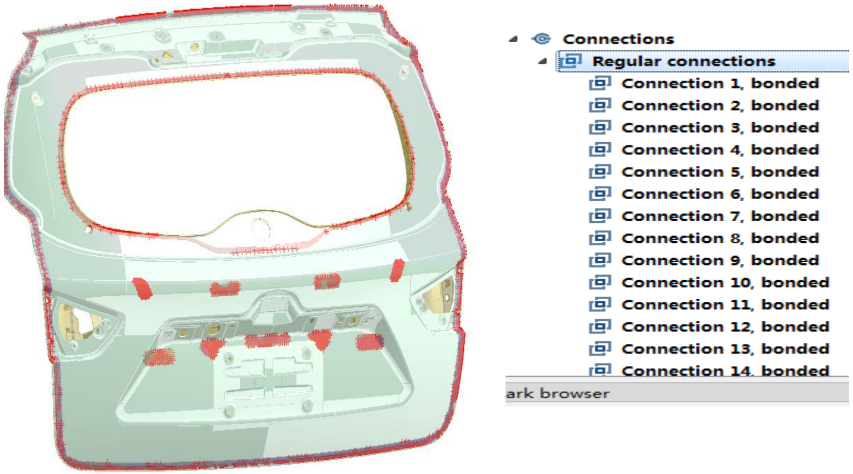
3、模态

分析前8阶自由模态。

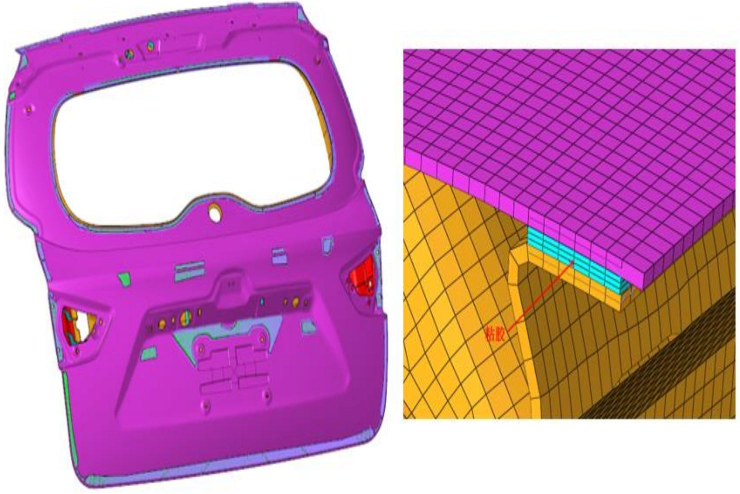
4、热变形

铰链安装点、锁安装位置约束dof1~6，施加温度载荷54℃。

成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析
仿真建模



SimSolid模型



OptiStruct模型

成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析
强度分析结果

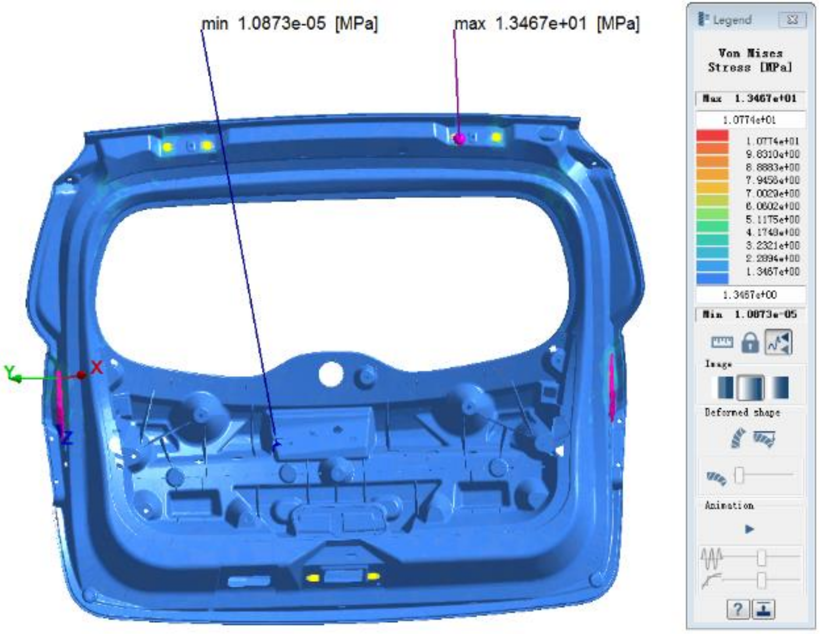
SimSolid

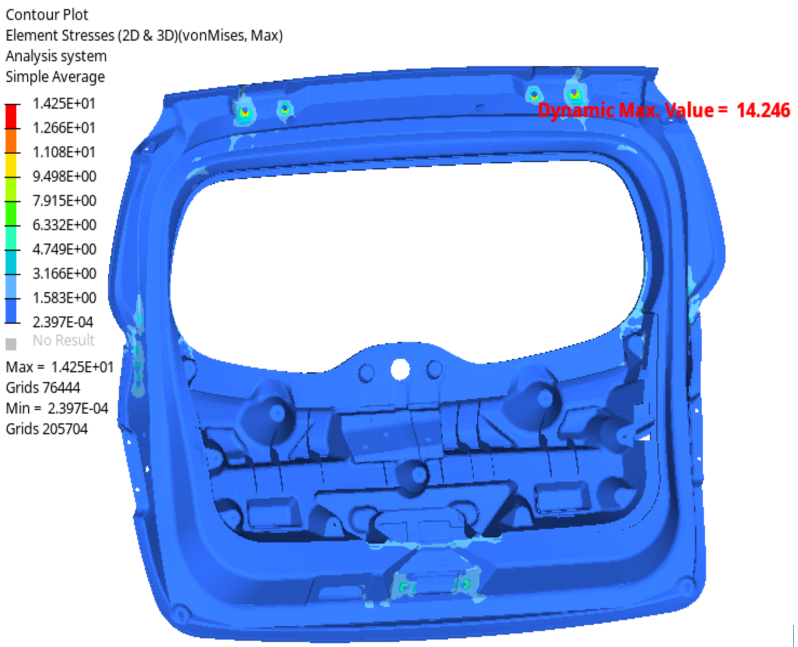
Max Von Mises 13.47 MPa

Error 5.8%

OptiStruct

Max Von Mises 13.47 MPa





成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析
刚度分析结果

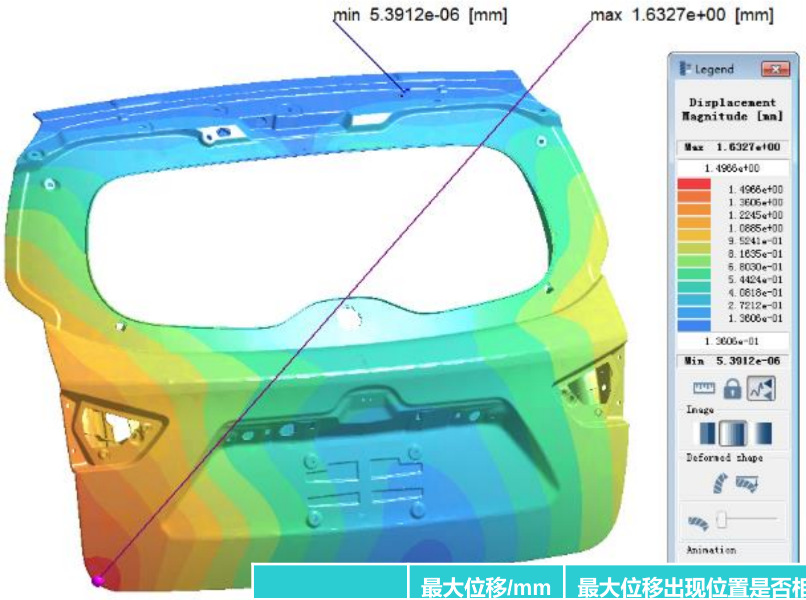
SimSolid

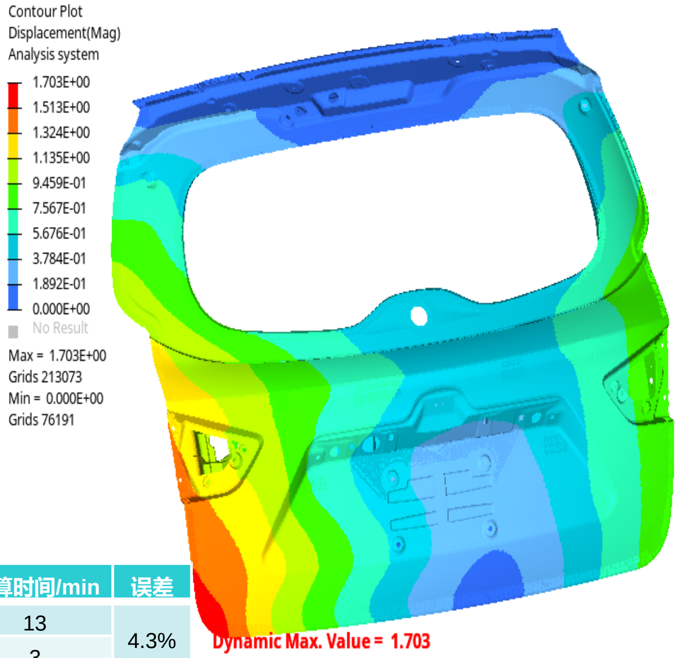
Max Displacement 1.633 mm

Error 4.3%

OptiStruct

Max Displacement 1.703 mm





	最大位移/mm	最大位移出现位置是否相同	计算时间/min	误差
SimSolid	1.633	是	13	4.3%
OptiStruct	1.703	是	3	
实验	1.65	是	/	

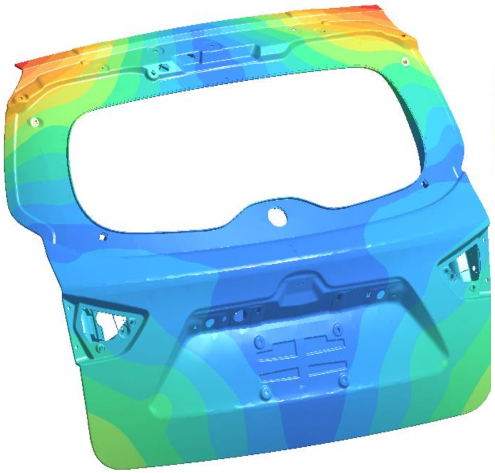
ALTAIR

成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析
模态分析结果

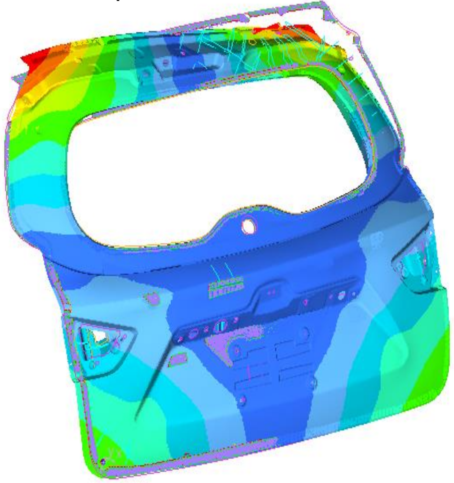
Error <4.5%

一阶扭转

SimSolid 27.4 Hz

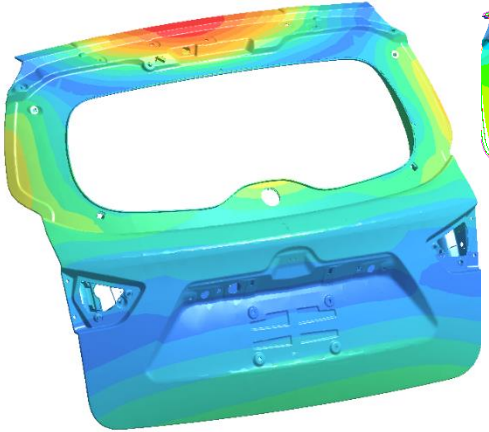


OptiStruct 29.0 Hz

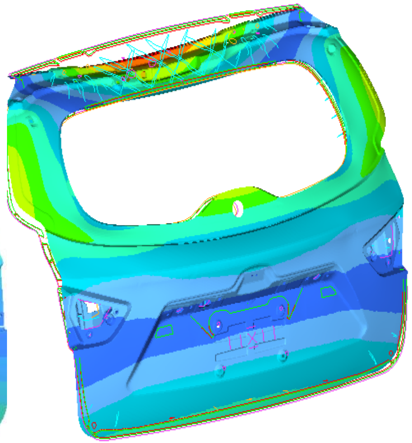


二阶弯曲

SimSolid 34.2 Hz



OptiStruct 35.3 Hz



成都航天模塑股份有限公司：基于SimSolid的全塑尾门力学分析
热分析结果

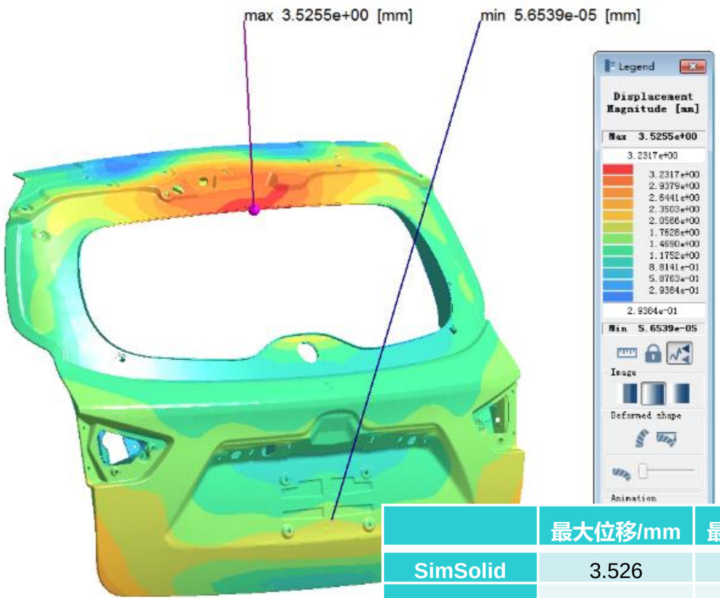
SimSolid

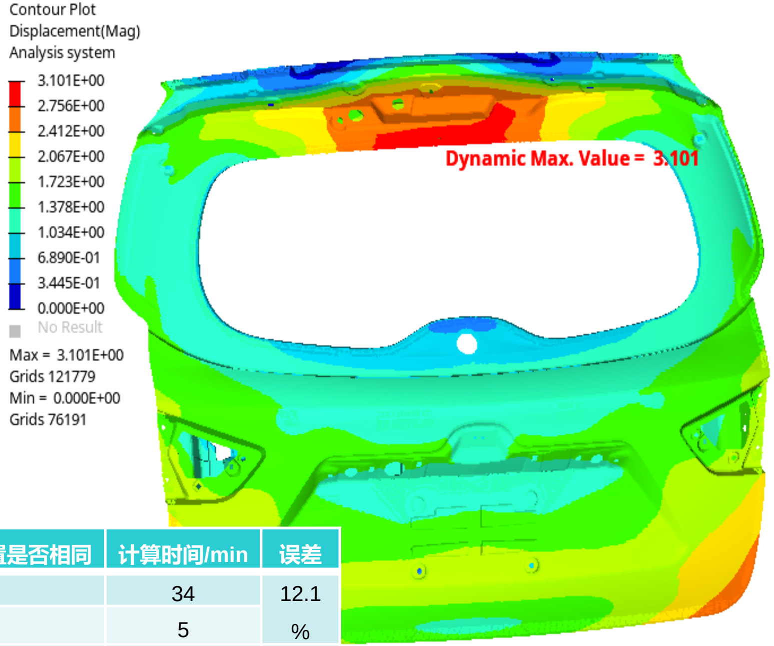
Max Displacement 3.526 mm

Error 12.1%

OptiStruct

Max Displacement 3.101 mm





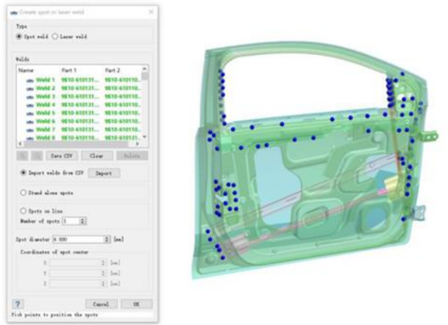
	最大位移/mm	最大位移出现位置是否相同	计算时间/min	误差
SimSolid	3.526	是	34	12.1 %
OptiStruct	3.101	是	5	
实验	/	/	/	/

ALTAIR

凯毅德汽车系统有限公司：左前门总成静力及模态分析

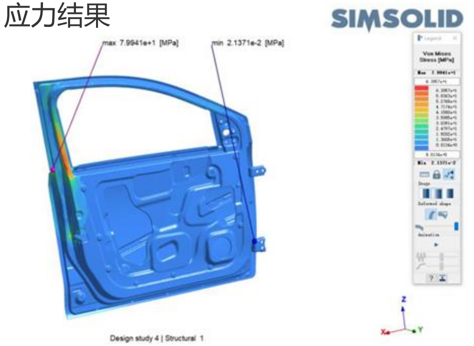
SimSolid中6分钟完成车门总成静力及模态分析

批量导入焊点、自动创建接触

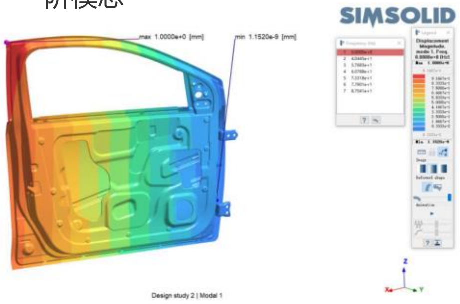


位移结果

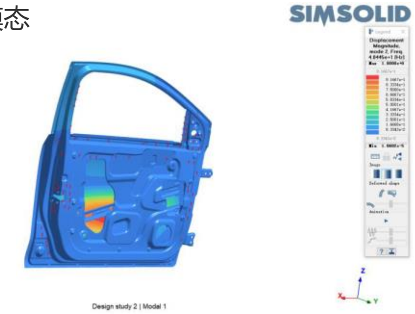
应力结果



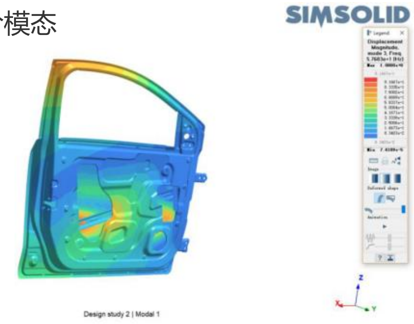
一阶模态



二阶模态



三阶模态



ALTAIR

项目背景

原始方案　　　　新方案

- 项目背景

车身B柱，材料为PP-lxs。该零件有一定刚度要求，若原始方案满足刚度要求，则去除部分加强筋变更为新方案，再次验证新方案刚度

零件	材料	密度 kg/mm³	弹性模量 MPa	泊松比
B上	PP-lxs	0.97	2050	0.36

- 分析步骤

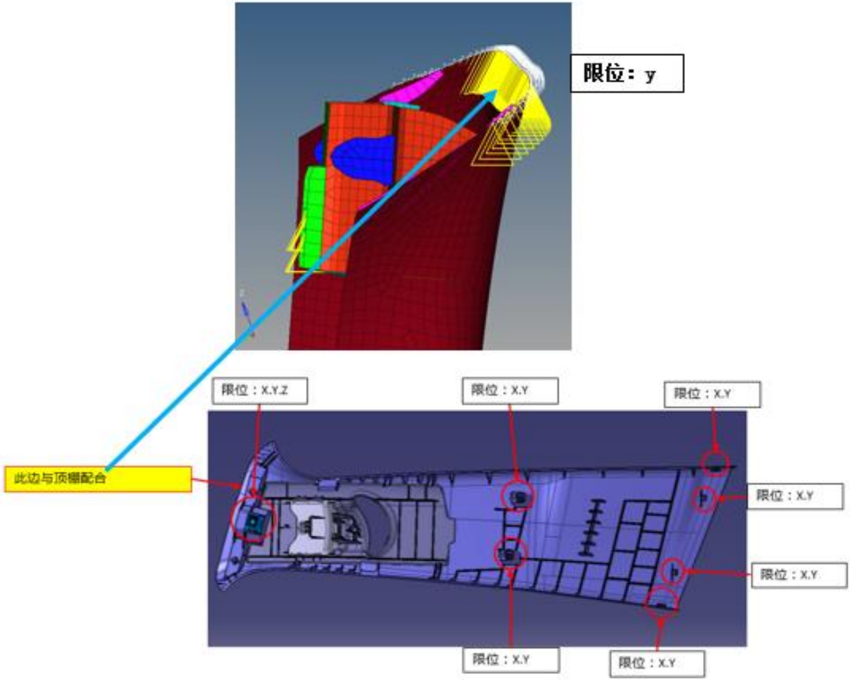
1、原始方案刚度分析，原始方案SimSolid与OptiStruct精度对比
2、新方案刚度分析，新方案 SimSolid与OptiStruct精度对比

- 约束与载荷

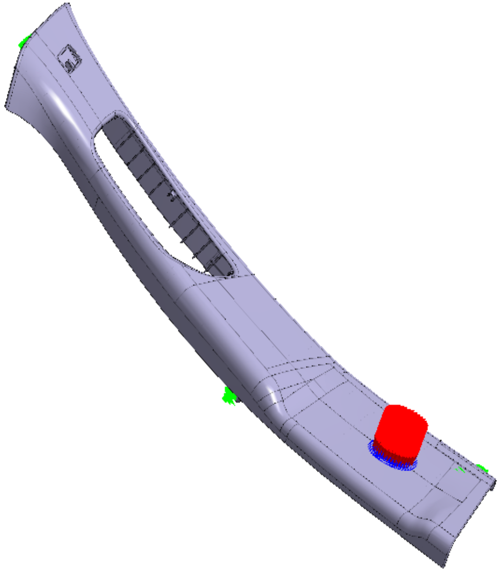
模型约束为卡扣和与顶棚的配合
实验施加工况为直径为50mm圆

ALTAIR

仿真建模



OptiStruct模型



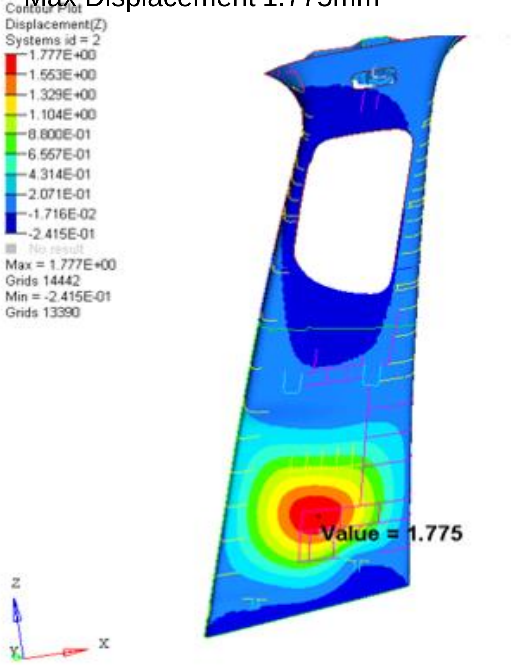
SimSolid模型

原始方案——刚度分析结果

Error 4.4%

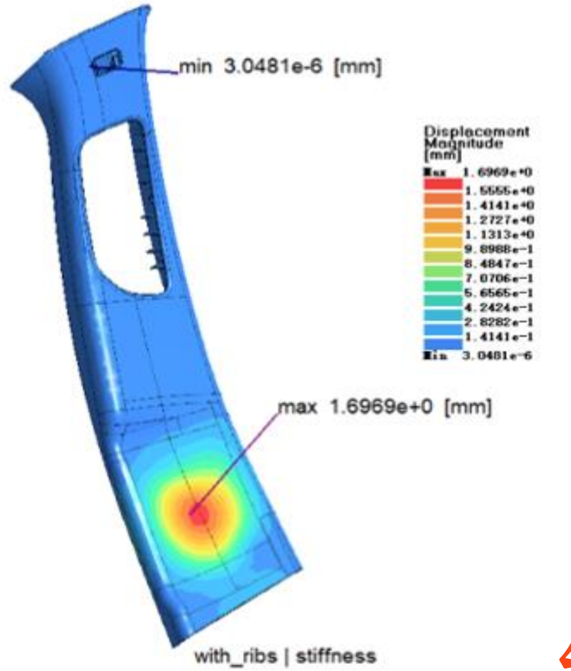
OptiStruct

Max Displacement 1.775mm



SimSolid

Max Displacement 1.696mm



新方案——刚度分析结果

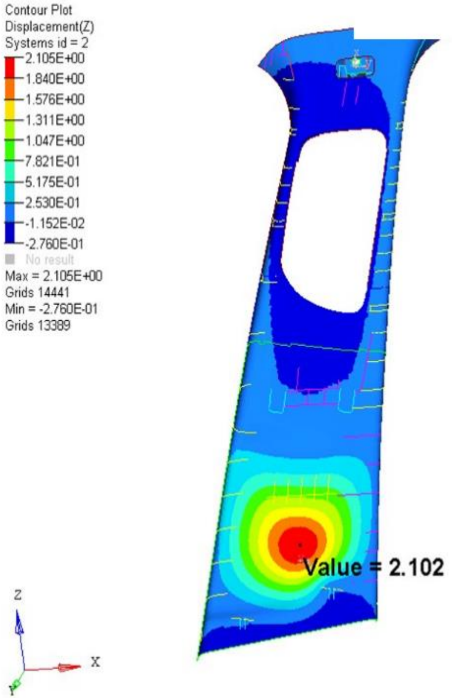
OptiStruct

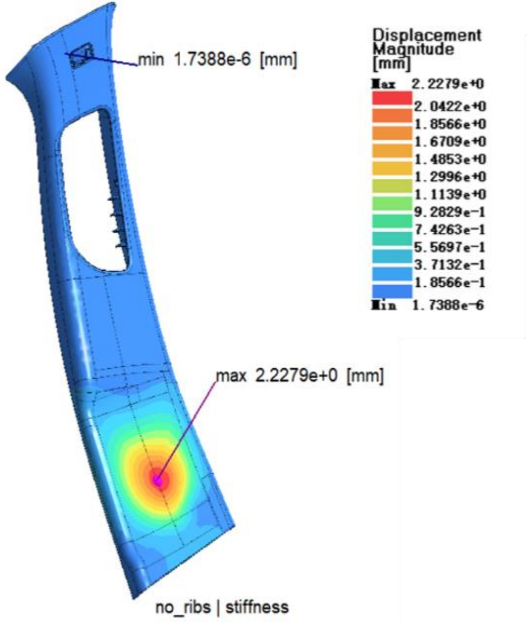
Error 5.9%

SimSolid

Max Displacement 2.102mm

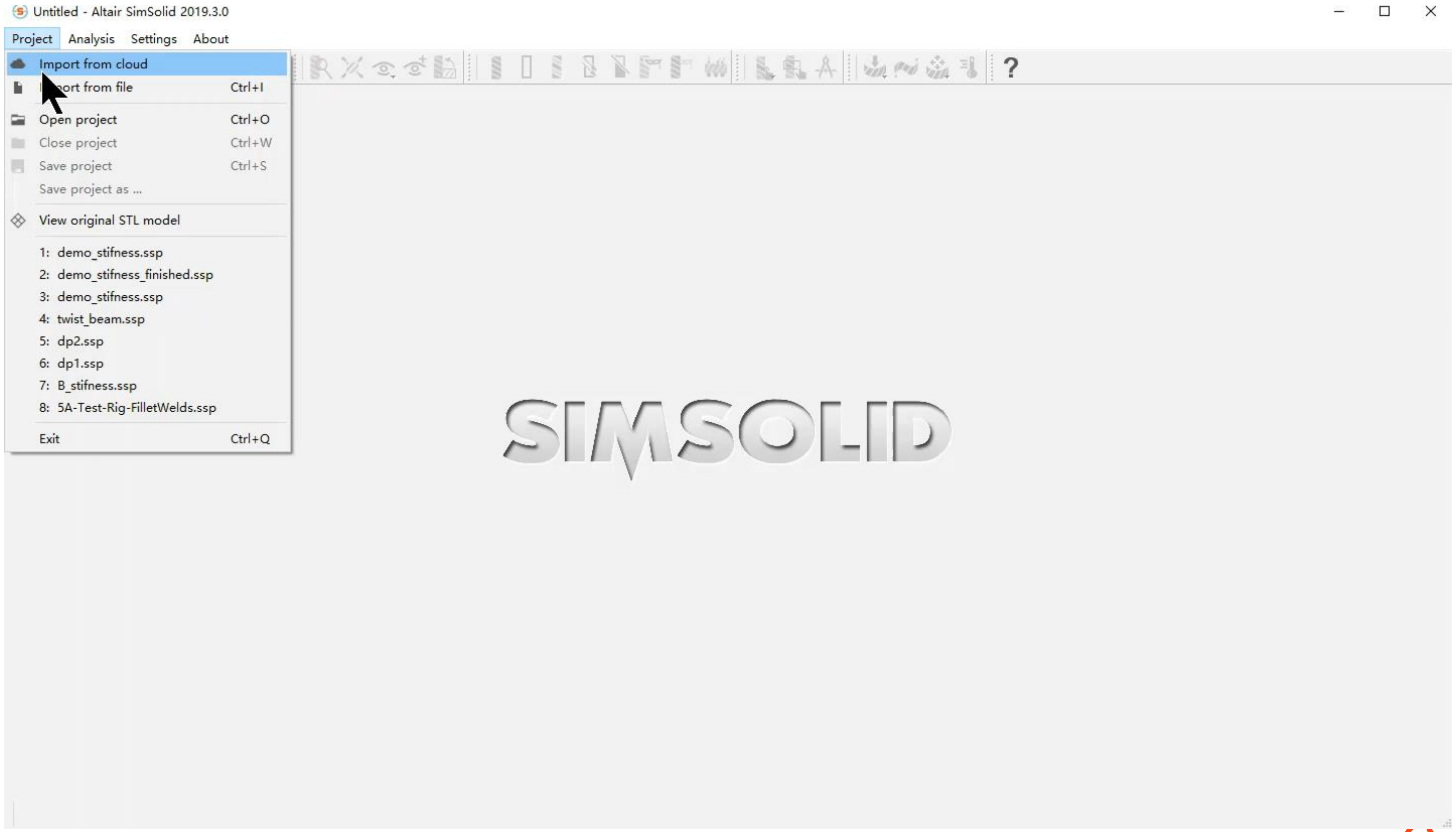
Max Displacement 2.227mm





ALTAIR

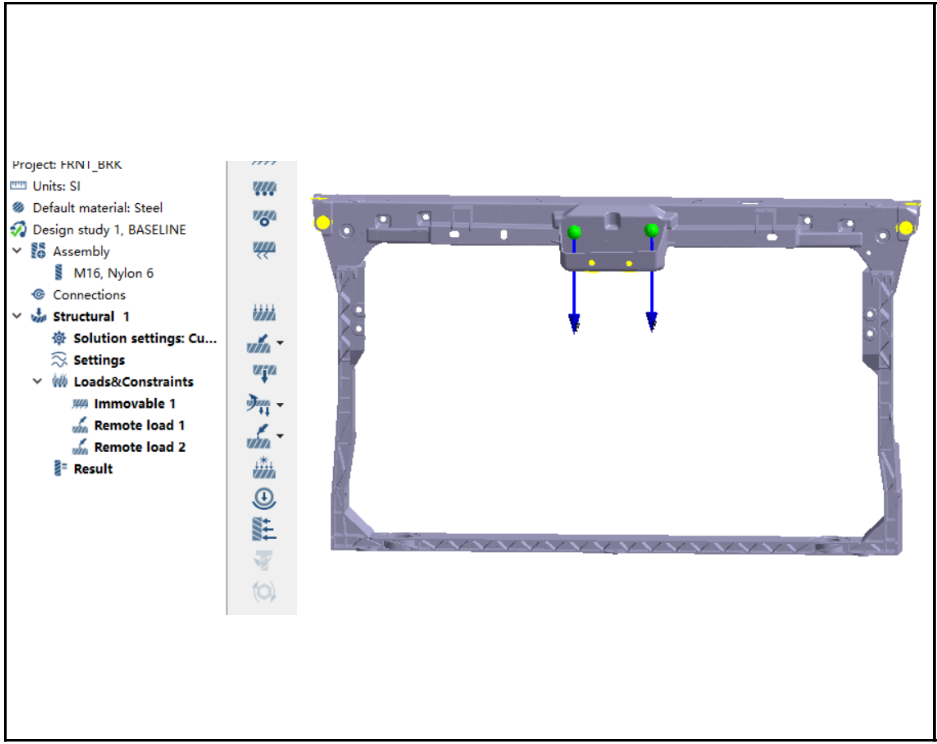
SimSolid特色快速对比分析功能

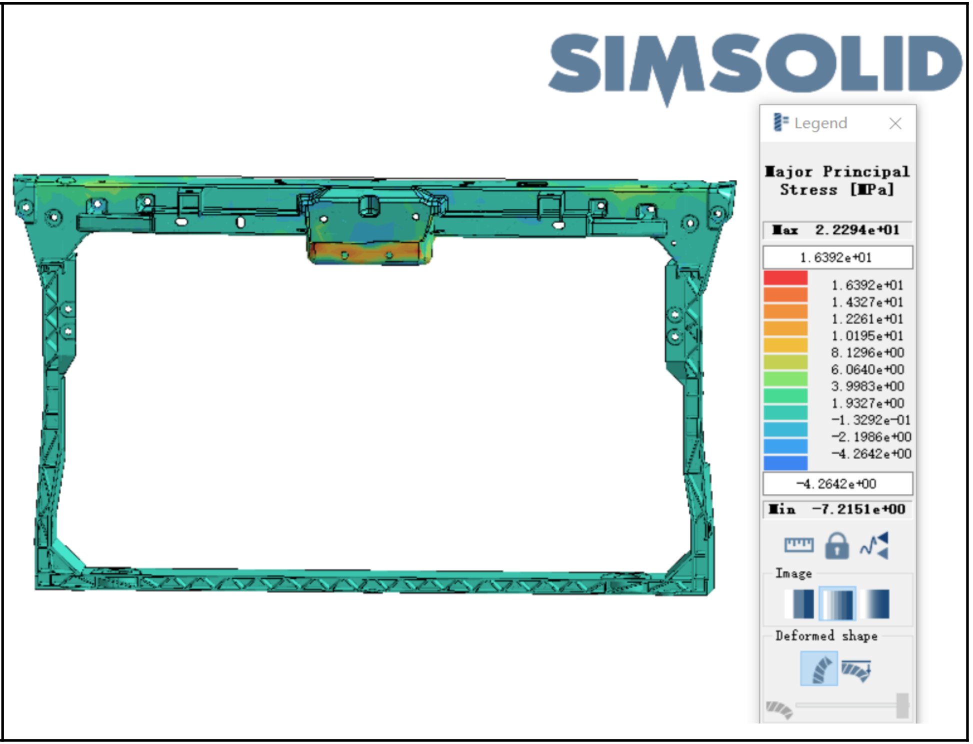


宁波华翔: 塑料前端框架的快速仿真

SimSolid快速仿真复杂塑料件

15分钟完成从导入几何到结果输出





Sabic 前端框架刚度与模态分析对比

Automotive Front-end Module – stiffness prediciton

Load case	OptiStruct - 2d modelling (normalized deformation)	SimSolid - adapt for stress (normalized deformation)	SimSolid – passes 6 (normalized deformation)	Deformation Shape
LC1: Lock force – 1 N (Max. deflection)	1	0.743	0.721	Bending of vertical members bottom area
LC1: Lock force – 1 N (lock point deflection)	1	0.832	0.831	Bending of vertical members bottom area
LC2: Radiator force – 1 N (Max. point deflection)	1	0.789		Bending of lower beam
LC2: Radiator force – 1 N (radiator center)	1	0.836		Bending of lower beam

SimSolid model shows higher stiff by ca. 20% than OptiStruct 2d mesh model.
Slightly predicted difference between advanced solution settings in SimSolid.

Sabic 前端框架刚度与模态分析对比

Automotive Front-end Module – frequency prediction

Mode	SimSolid (Hz)		OptiStruct (Hz)	Shape	Gap %	Shape	Remove mass difference Gap %
1	23.4	Bending of vertical members bottom area	23.07	Bending of vertical members bottom area	1.4%	Similar	-0.2%
2	36.4	Bending vertical member, lower beam moves in y direction	35.4	Bending vertical member, lower beam moves in y direction	2.8%	Similar	1.2%
3	43.1	Bending vertical member, upper beam moves/rotates along y direction	40.6	Bending vertical member, upper beam moves/rotates along y direction	6.2%	Similar	4.6%
4	45.9	Bending vertical in lower area, lower beam rotates along z.	45.1	Bending vertical in lower area, lower beam rotates along z.	1.8%	Similar	0.2%
5	49.1	Bottom beam bending	46.7	Bottom beam bending	5.1%	Similar	3.5%
6	60.5	Upper beam bending	57.2	Upper beam bending	5.8%	Similar	4.2%

Notes:

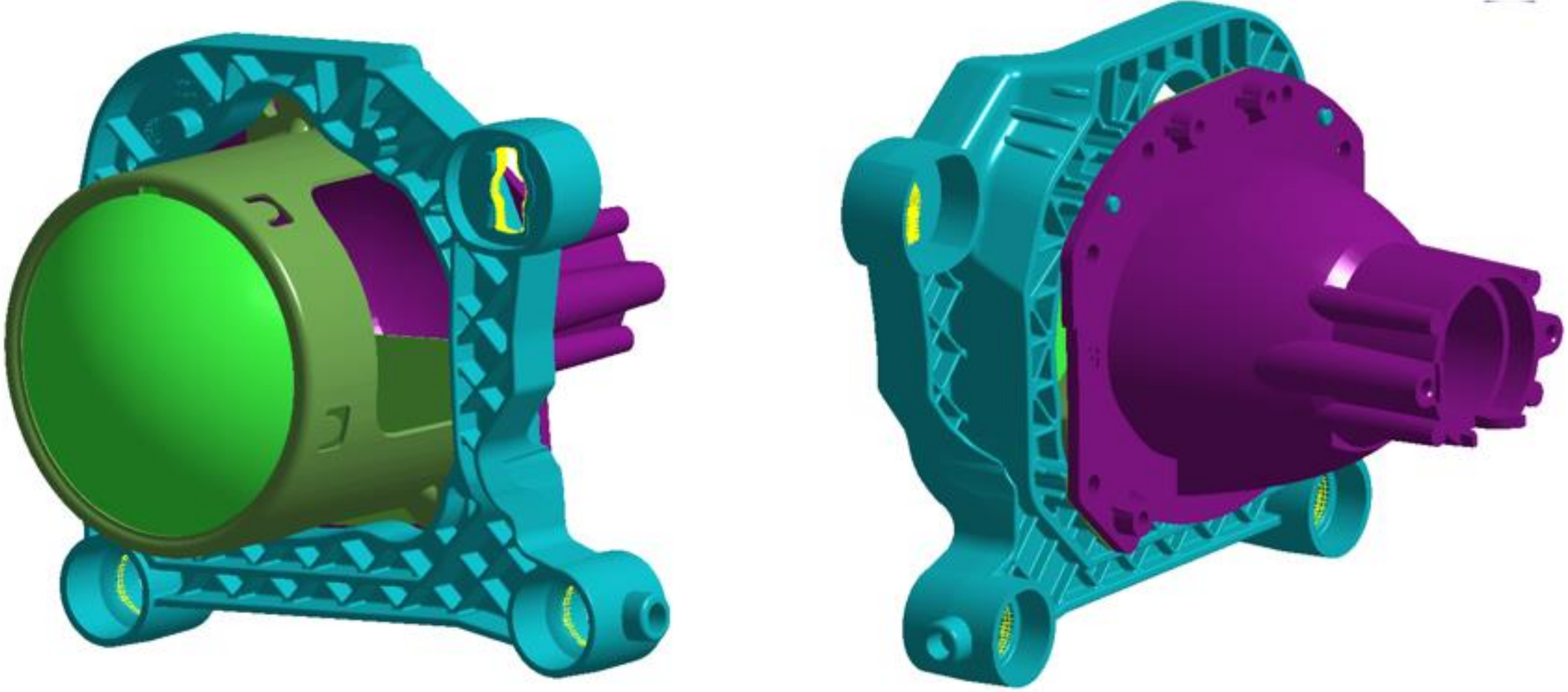
- OptiStruct model 3.2% more heavy. It is expected frequency in SimSolid is 1.6% higher than in OptiStruct. This is corrected for in the last column

Both frequencies and shapes are close each other.

ALTAIR

SIMSOLID在内外饰上的应用

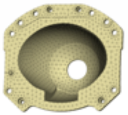
ALTAIR



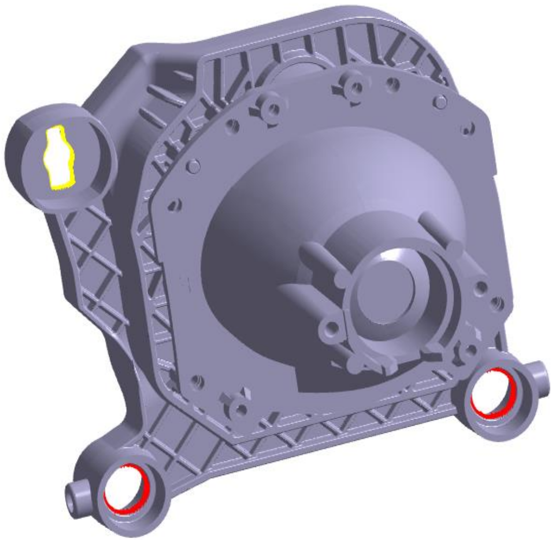
汽车车灯透镜组模态分析_SIMSOLID_VS_ABAQUS

ALTAIR

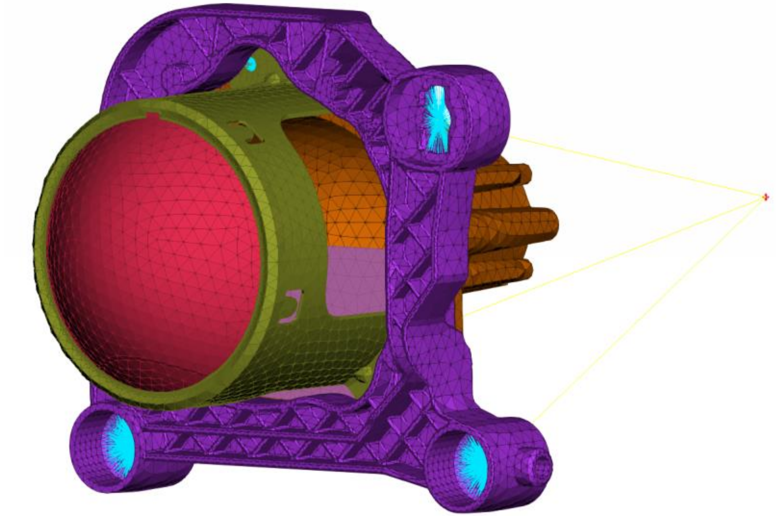
模型与材料

NO.	部件	部件名称	材料	密度(kg/m3)	弹性模量(MPa)	泊松比
1		透镜	GLASS	2520	72000	0.22
2		透镜支架	STEEL	7850	210000	0.3
3		反射镜	ALUM	2780	73000	0.33
4		透镜组支架	PBT-GF30	1310	9000	0.437
5		遮光板	STEEL	7850	210000	0.3

仿真建模



SimSolid模型



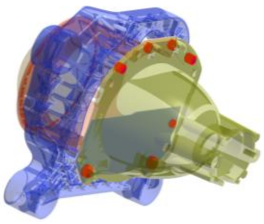
OptiStruct模型

ALTAIR

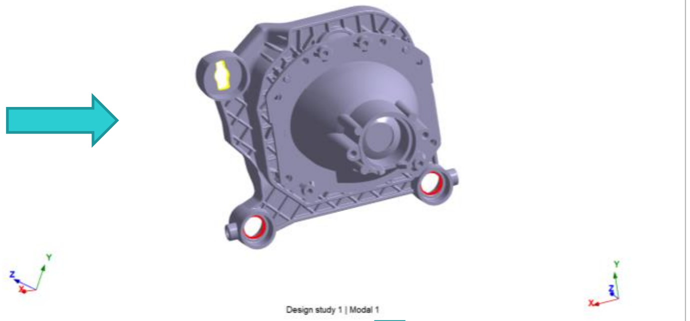
Altair 结构快速验证—— SimSolid建模分析步骤

1、导入几何

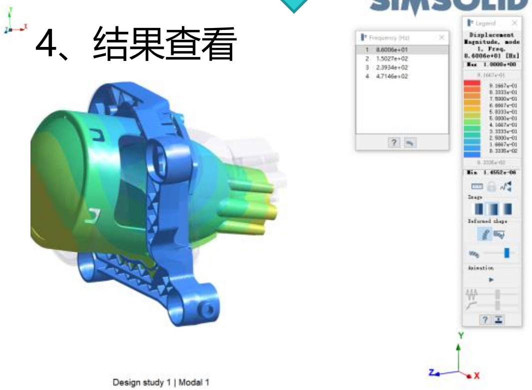
2、连接创建



3、约束施加

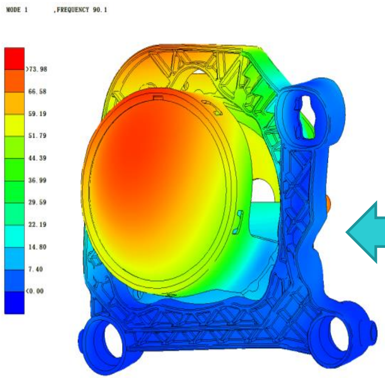


4、结果查看



第一阶用户对标

Error 4.4%



优势：同步设计的仿真

缩短设计周期

仿真流程更稳健

模态分析结果

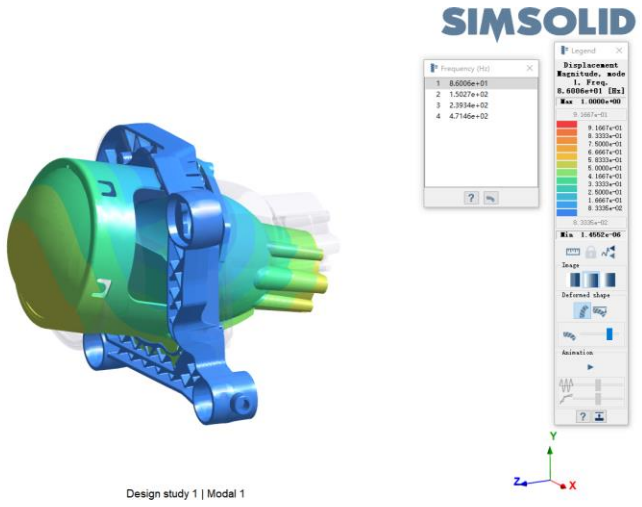
SimSolid

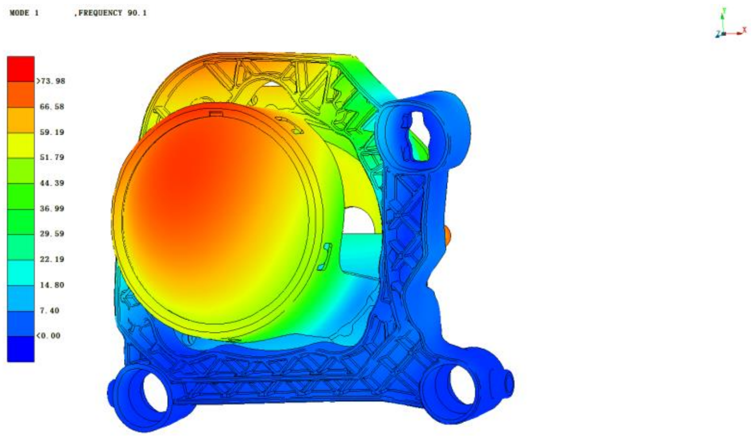
第二阶 86 Hz

Error 4.4%

Abaqus

第一阶 90 Hz





模态分析结果

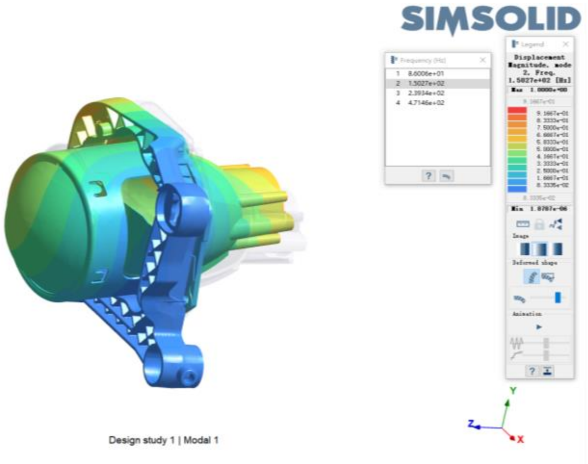
SimSolid

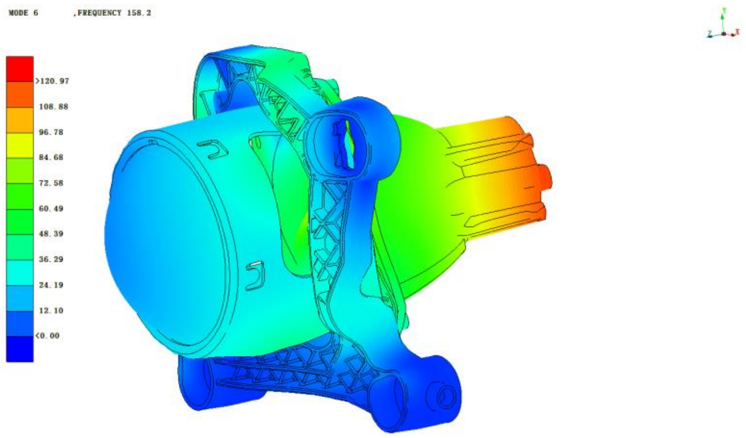
第三阶 150 Hz

Error 5.1%

OptiStruct

第一阶 158 Hz





模态分析结果

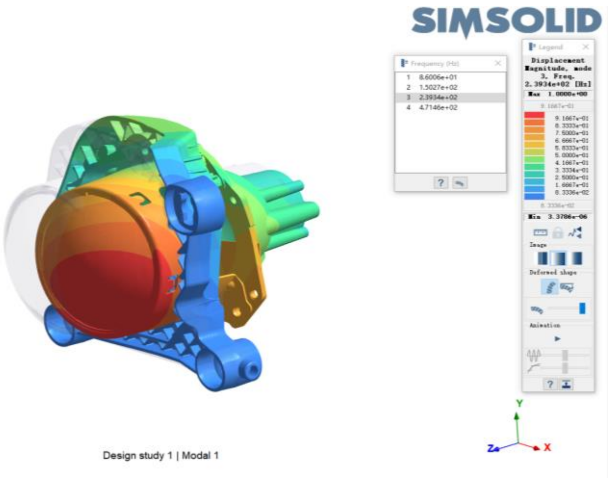
SimSolid

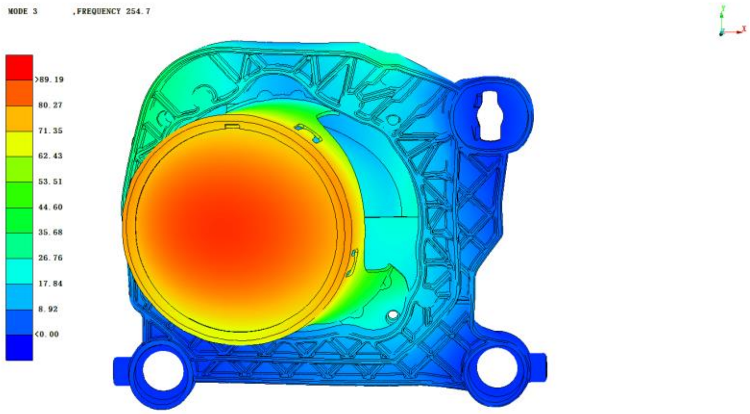
第四阶 239 Hz

Error 6.3%

OptiStruct

第一阶 255 Hz





ALTAIR

模态分析结果

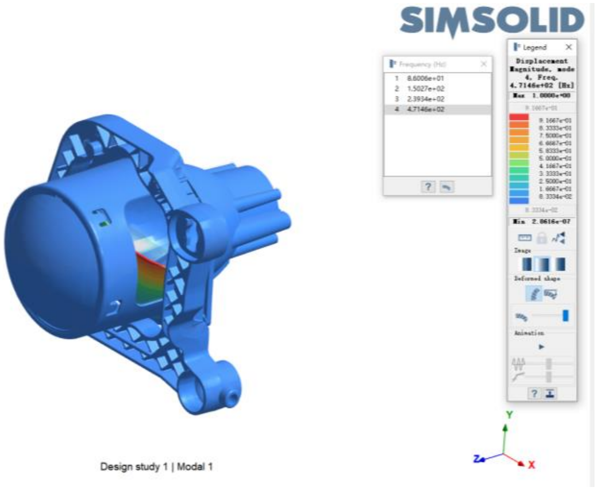
SimSolid

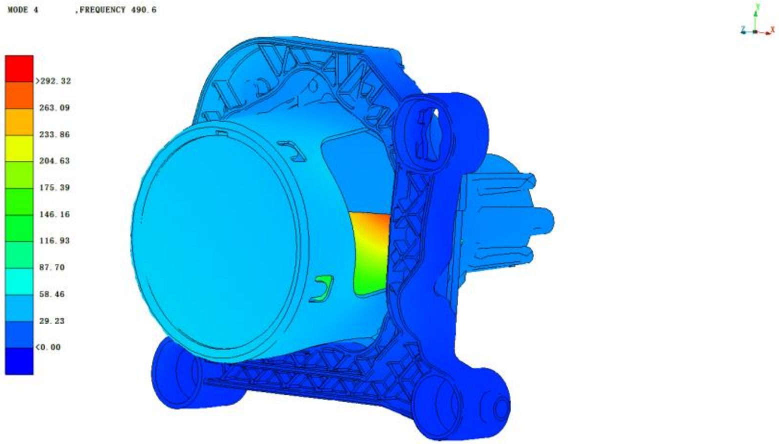
第五阶 471 Hz

Error 4.1%

OptiStruct

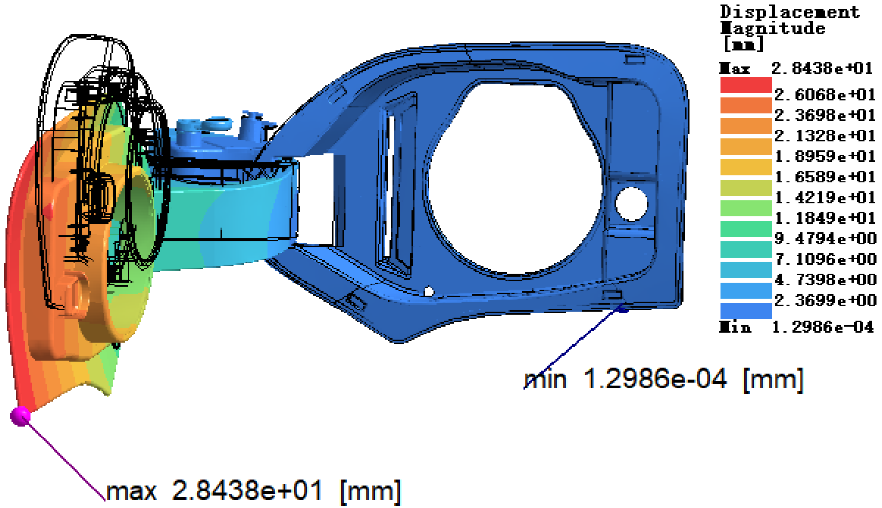
第一阶 491 Hz





宁波华翔: 充电口盖总成刚度分析

SimSolid快速仿真复杂塑料件
求解时间不到1分钟



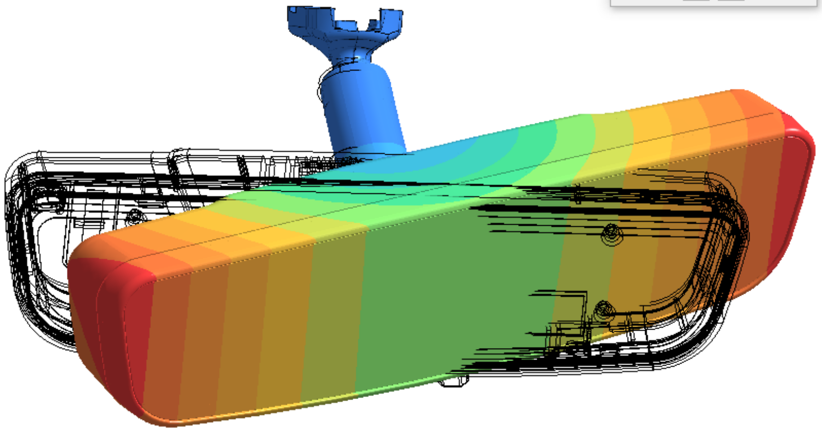
样品编号 Sample No.	测试结果 Results
S1912060101A02（01-02）	1、 F=50N;下沉量 29.60mm

最大位移	F=50N 下沉量28.4mm

宁波华翔: 后视镜模态分析

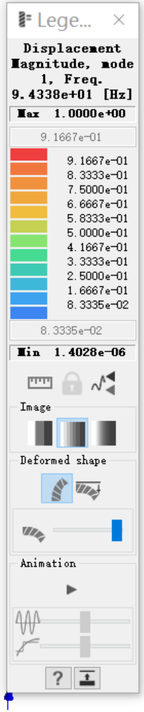
SIMSOLID

第一阶固有频率
94HZ



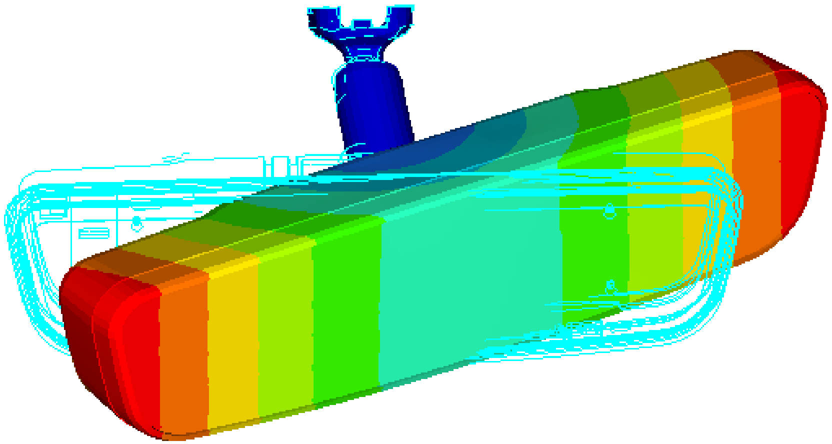
Frequency (Hz)

1	9.4338e+01
2	1.2016e+02
3	1.4363e+02
4	3.1114e+02
5	3.9531e+02
6	4.1877e+02



第一阶固有频率
90HZ

Subcase 1 (loadstep1) : Mode 1 - F = 9.052456E+01
Frame 25



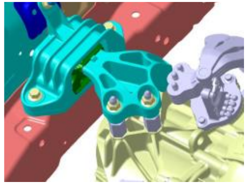
SIMSOLID动力系统上的应用

ALTAIR

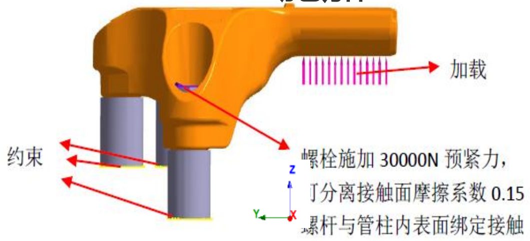
众泰汽车：汽车动力总成悬置支臂的静力与模态分析

SimSolid在3min内取得收敛结果

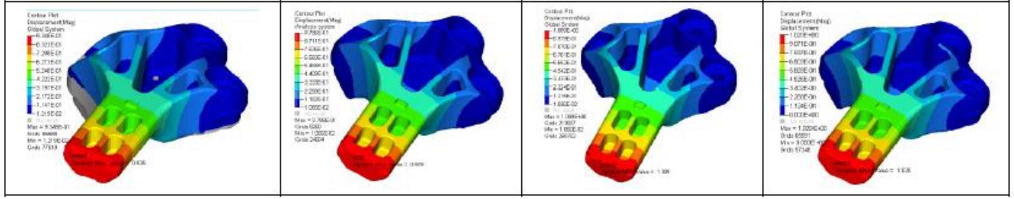
直观快速施加螺栓预紧力、一键设置精度提升



Z施加8000N



OptiStruct 计算结果-位移			
单元 3mm	单元 2mm	单元 1mm	3mm 二阶
0.935 mm	0.979 mm	1.009 mm	1.02 mm



SimSolid 计算结果-位移			
Max number of solutions 3	Max number of solutions 3 + Adapt to features	Max number of solutions 6	Max number of solutions 6 + Adapt to features
1.071mm	1.063 mm	1.067 MPa	1.055 mm

众泰汽车：汽车动力总成悬置支臂的静力与模态分析

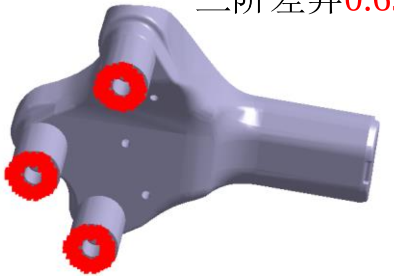
SimSolid在3min内取得准确模态分析结果

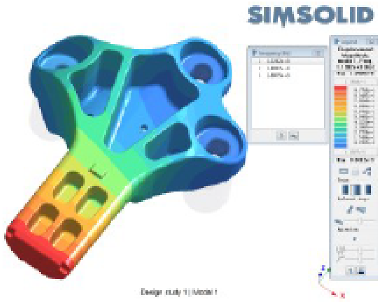
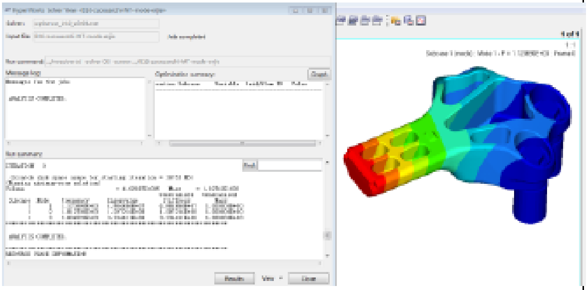
一键自动创建连接、免网格划分

一阶差异0.36%

二阶差异0.22%

三阶差异0.69%



SimSolid 模态	OptiStruct 模态
一阶：1128 Hz	一阶：1124 Hz
二阶：1809 Hz	二阶：1813 Hz
三阶：2886 Hz	三阶：2906 Hz
	

中国一汽无锡油泵油嘴研究所：涡轮增压器配机试验减振分析

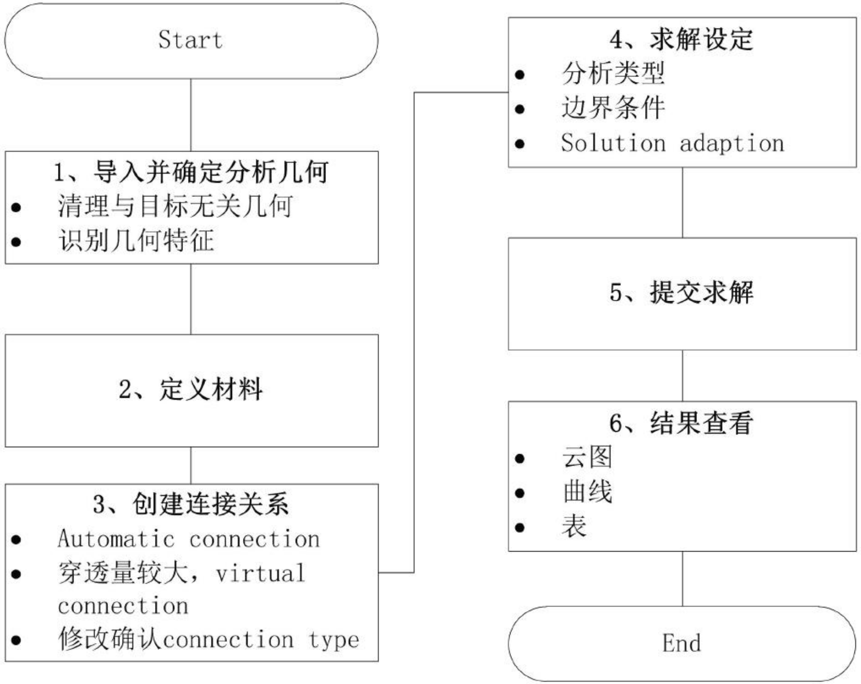


图 2 基于 SimSolid 分析流程

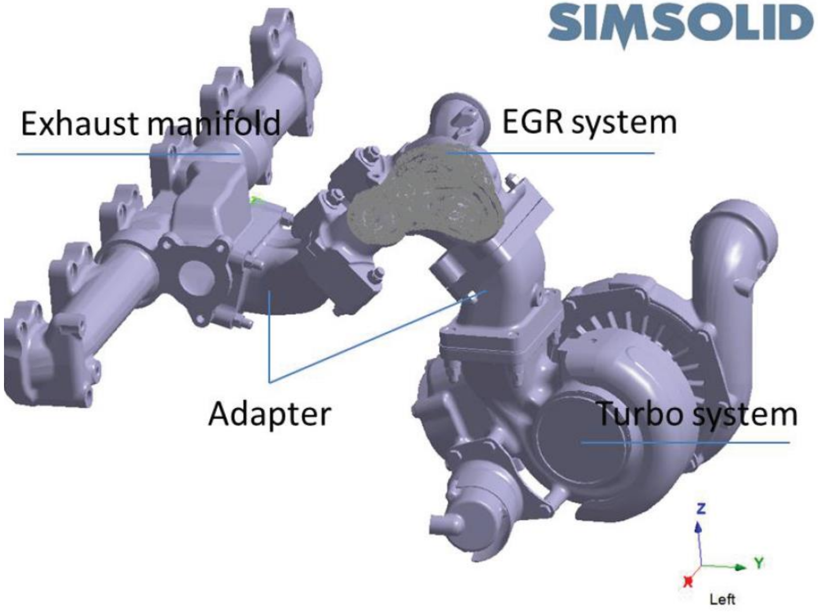
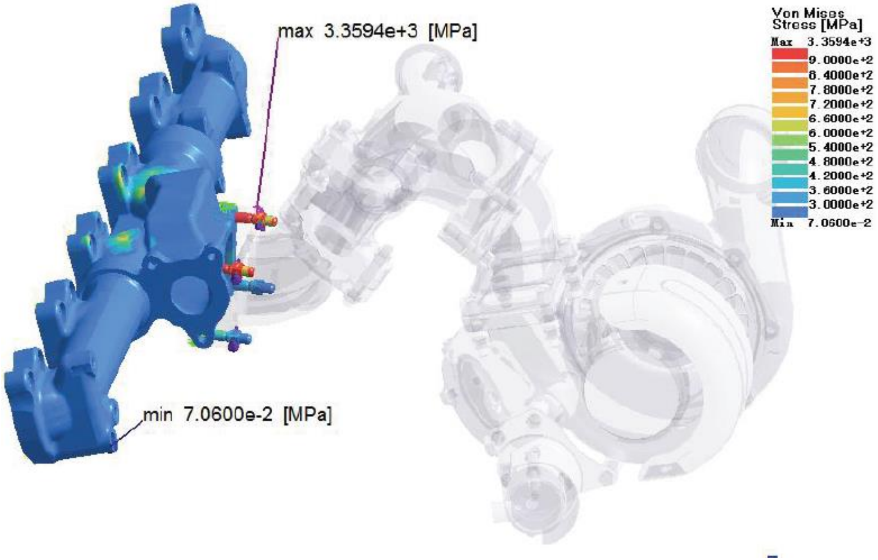


图 1 无网格分析模型

ALTAIR

中国一汽无锡油泵油嘴研究所：涡轮增压器配机试验减振分析



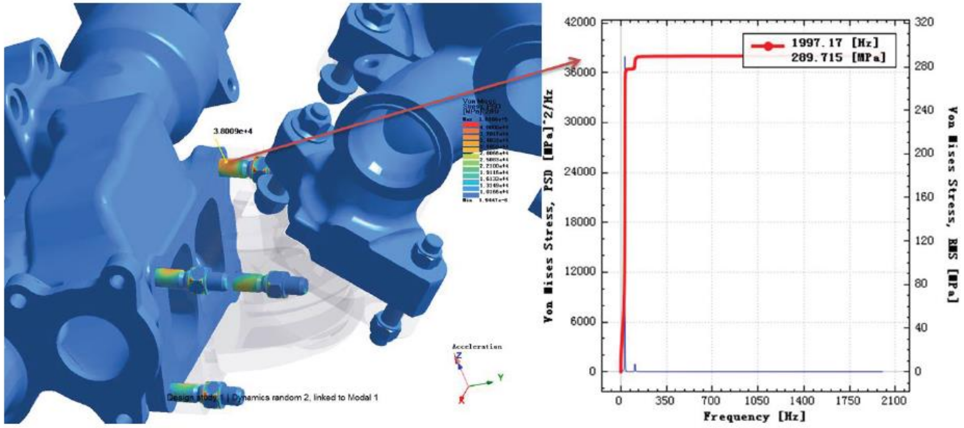


图 6 case 1：无支承方案螺栓应力有效值：290MPa

静力工况：对结构整体施加竖直向下（-Z 向）加速度10g
结果：最大应力发生在适配器与排气歧管结合处的螺栓上表面
风险：螺栓有剪切断裂失效的危险

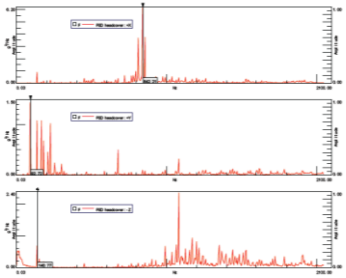


图 5 发动机额定工况激励信号 PSD 曲线

ALTAIR

中国一汽无锡油泵油嘴研究所：涡轮增压器配机试验减振分析

表 2 支承概念设计方案

编号	概念设计方案	约束	刚度设置
case 1	无支承	歧管约束	
case 2	单点 Z 向支承	歧管约束+支承位置 1	Kz=1.05e11 N/m Kx=Ky=0
case 3	单点三向支承 1	歧管约束+支承位置 1	Kz=1.05e11 N/m; Kx=Ky=Kz/10
case 4	单点三向支承 2	歧管约束+支承位置 1	Kz=1.05e11 N/m; Kx=Ky=Kz/2
case 5	两点支承	歧管约束+支承位置 1 +支承位置 2	支承点 1: Kz=1.05e11 N/m; Kx=Ky=Kz/10 支承点 2: Kx=Ky=Kz=1.05e10 N/m

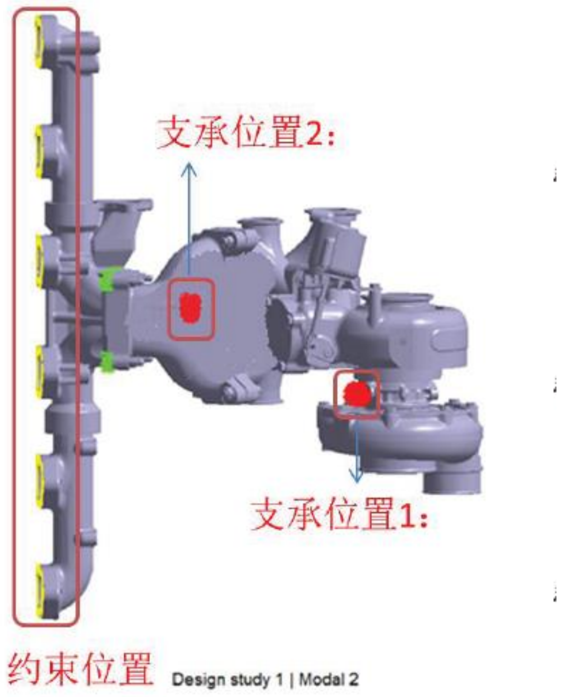


图 4 支承方案概念设计示意图

ALTAIR

中国一汽无锡油泵油嘴研究所：涡轮增压器配机试验减振分析

基于随机响应应力有效值利用三区间法进行寿命评估

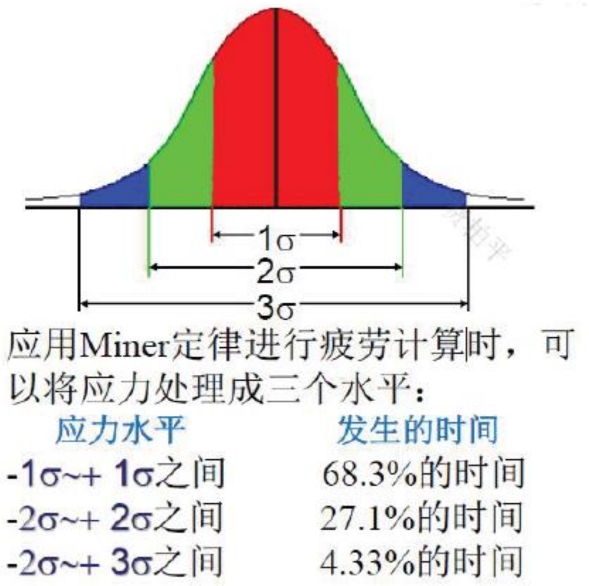


图 7 基于 PSD 随机响应疲劳寿命三区间估计法

图 8 10.8 级高强度螺栓 SN 曲线

ALTAIR

中国一汽无锡油泵油嘴研究所：涡轮增压器配机试验减振分析

表 3 各设计方案疲劳寿命对比

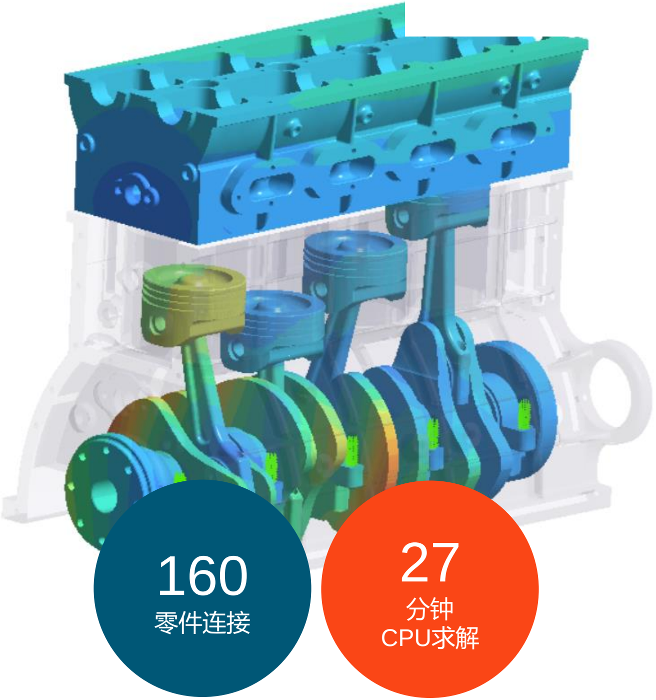
方案	螺栓位置应力	RMS Mises Stress			Life: repeats	成本	整体模态最低固有频率：Hz
		1σ	2σ	3σ			
	发生概率	68.30%	27.10%	4.33%			
case 1	无支承	290	580	870	75189	低	37.1
case 2	单点Z向支承	175	350	525	3926097		65.7
case 3	单点三向支承1	52	104	156	infinite		256.3
case 4	单点三向支承2	46	92	138	infinite		328.8
case 5	两点支承	14	28	42	infinite	高	275.1

1）原始结构即无支承方案危险位置螺栓应力值较大，在高转速工况运行下会短时间内发生螺栓断裂失效。
2）仅在增压器端作Z 向支承，仍不能实现无限寿命设计，有较大的失效风险。
3）单点三向支承及两点支承方案能大幅降低随机振动响应应力，在给定的激励曲线下能满足无限寿命设计。
4）两点支承方案较单点支承方案低阶固有频率提高幅值不显著，且布置较难成本较高，在都能满足无限寿命设计要求下，推荐采用方案4单点三向支承。

ALTAIR

发送机模态分析

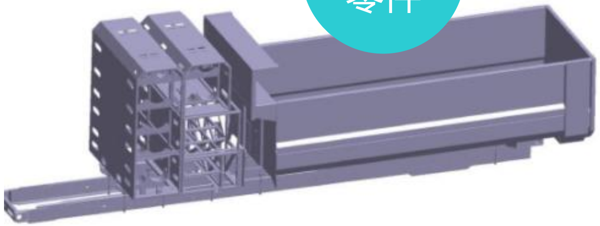
发送机—50阶模态



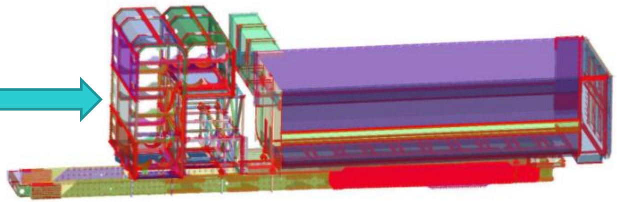
中山大学：燃料电池自卸车结构性能分析

Step1:几何导入

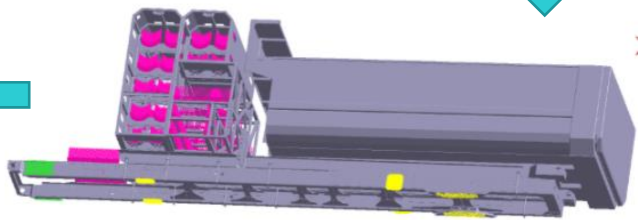
476
零件



Step2:连接批量创建

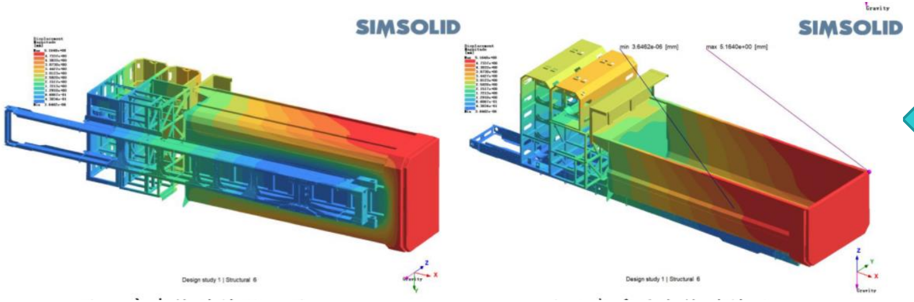


Step3:载荷施加



5分钟求解

Step4:结果查看



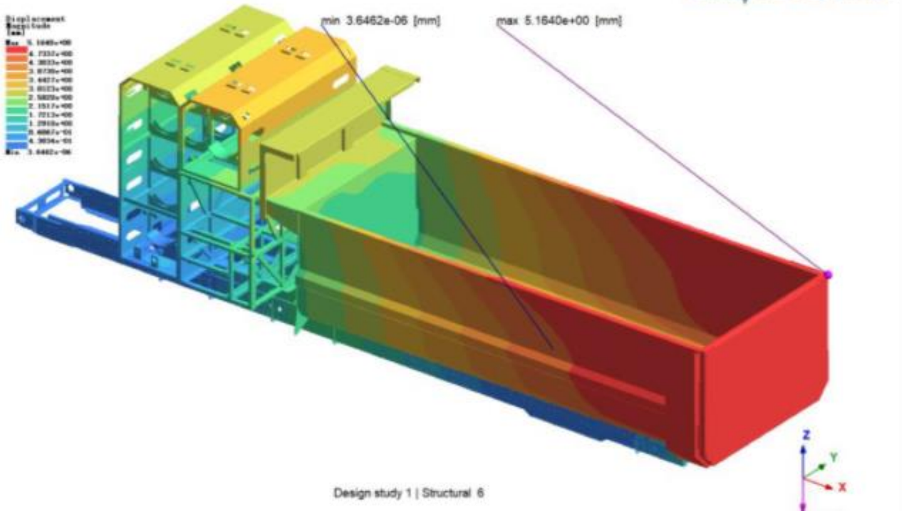
中山大学：燃料电池自卸车结构性能分析

SimSolid与OptiStruct位移比较

Error 2.9%

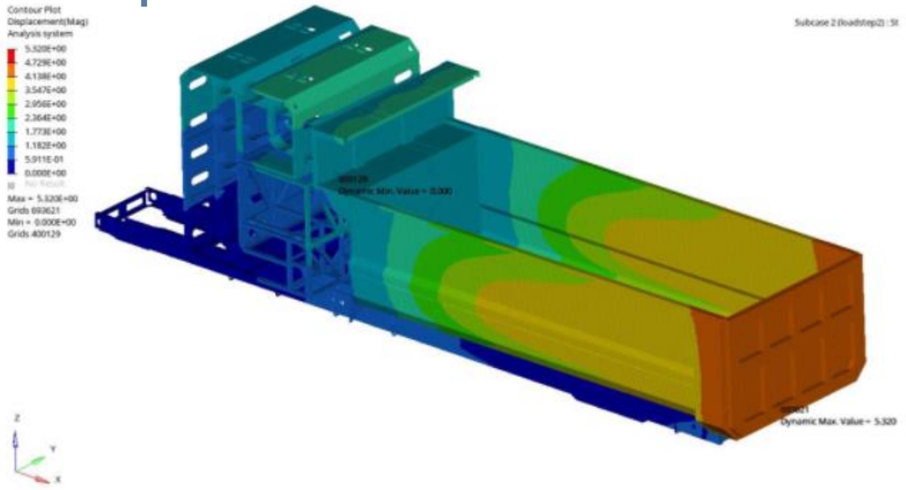
Max 5.164mm

SIMSOLID



Max 5.32mm

OptiStruct



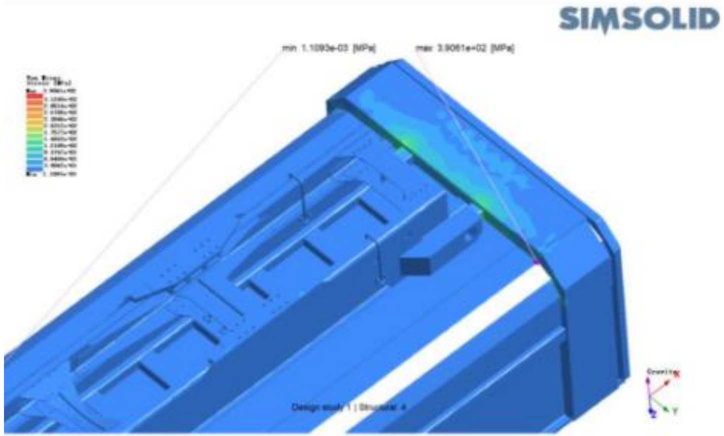
中山大学：燃料电池自卸车结构性能分析

SimSolid与OptiStruct位移比较

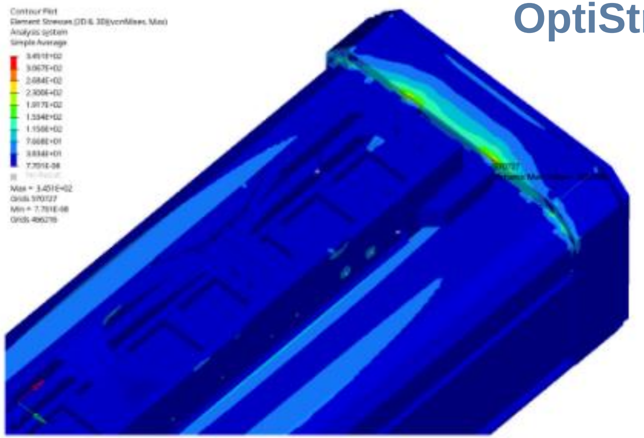
Error 13.19%

Max 390.61MPa

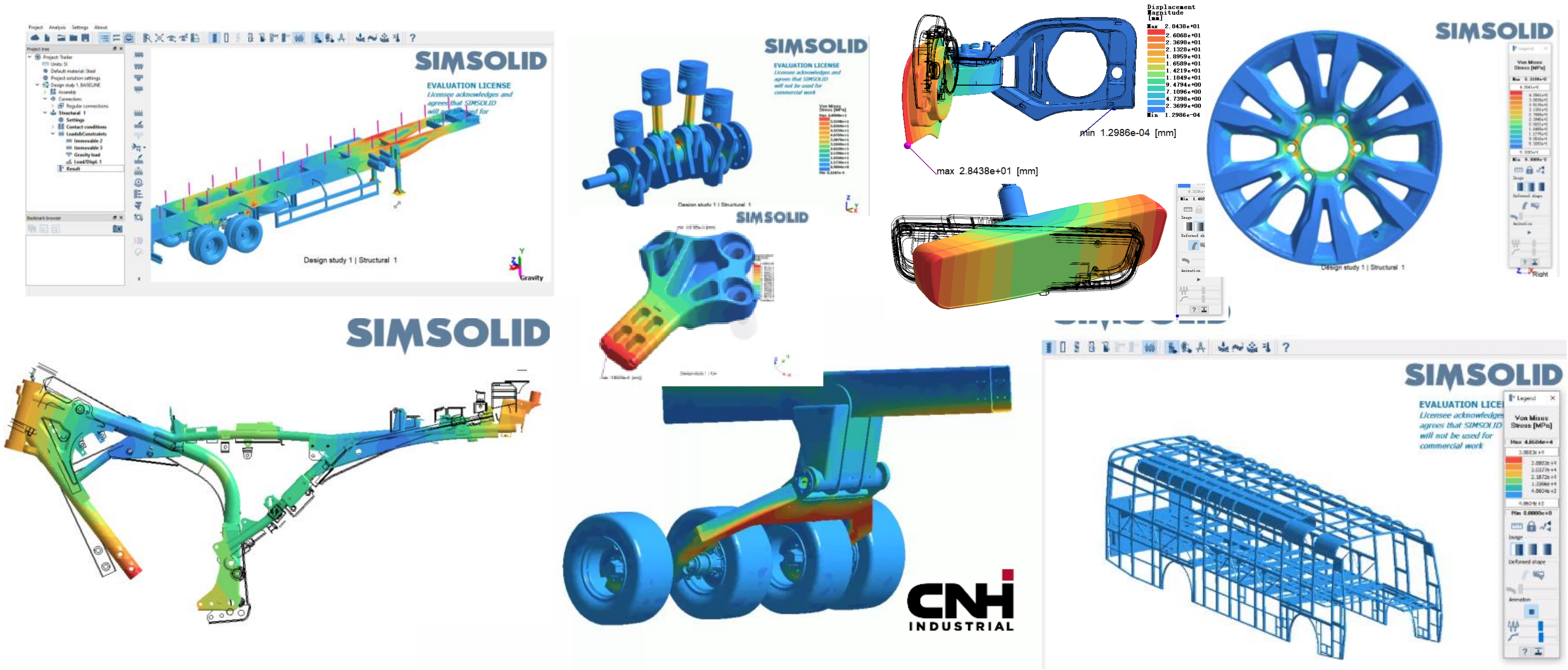
Max 345.1MPa



OptiStruct



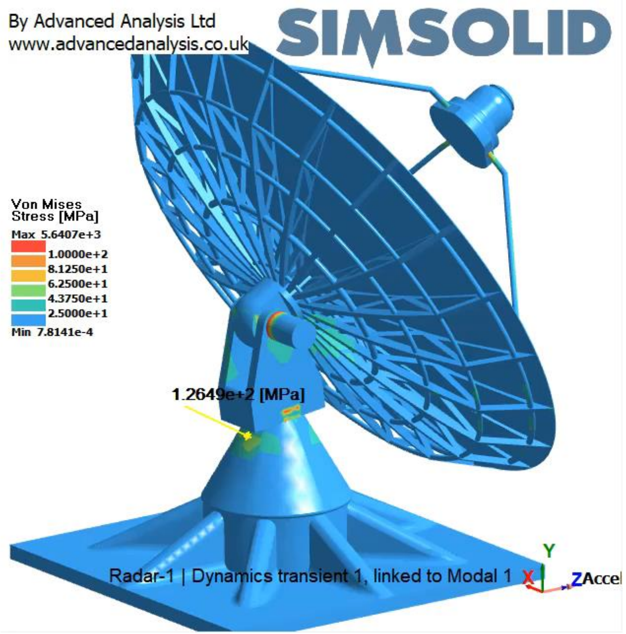
SimSolid车辆应用案例



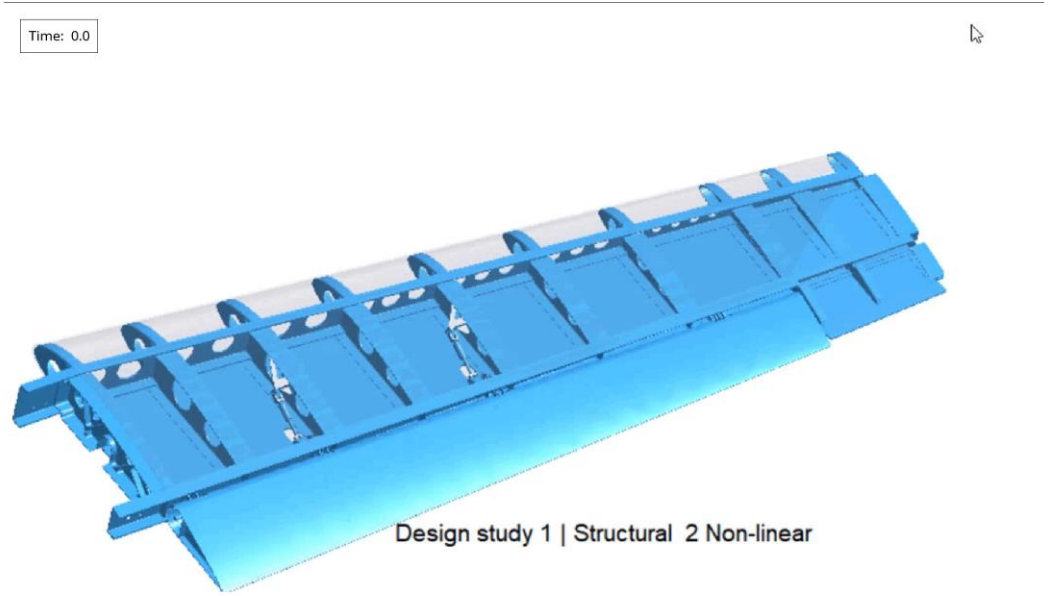
SIMSOLID在非汽车行业的应用

ALTAIR

SimSolid航空航天工程应用案例

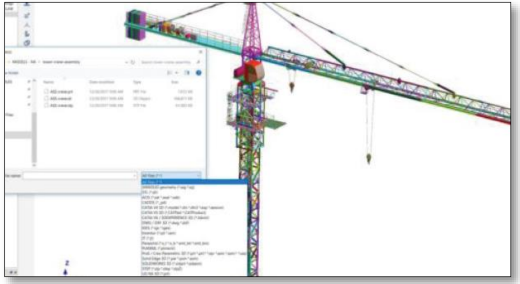


雷达瞬态分析

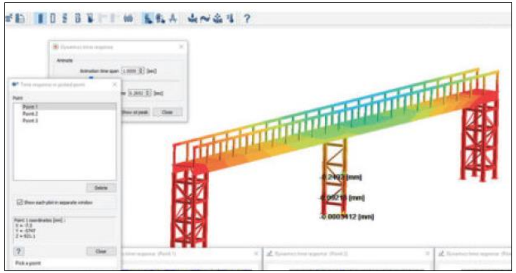


机翼非线性分析

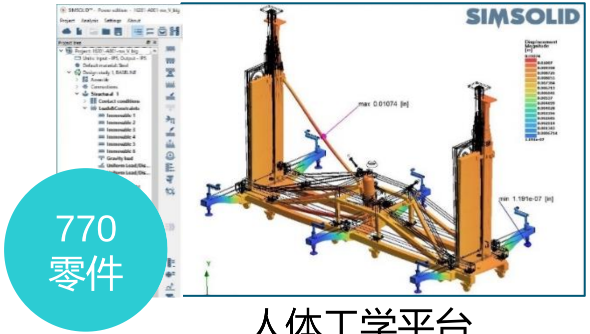
SimSolid工程器械应用案例



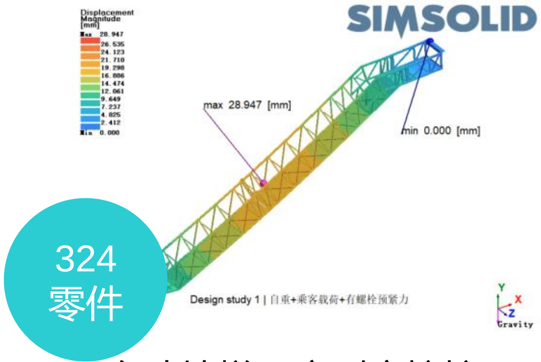
建筑用升降平台



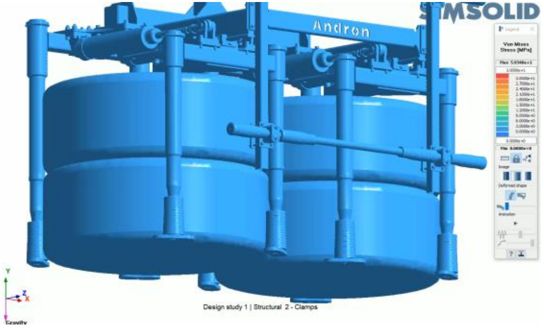
振动分析



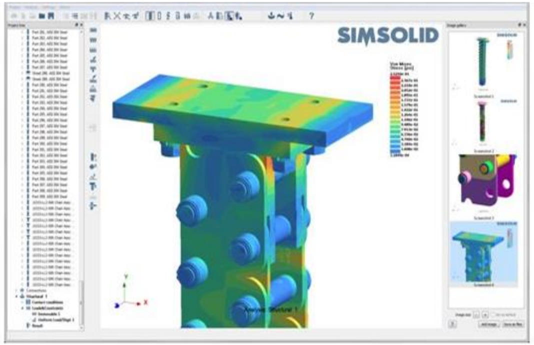
人体工学平台



自动扶梯强度刚度校核



测试装备

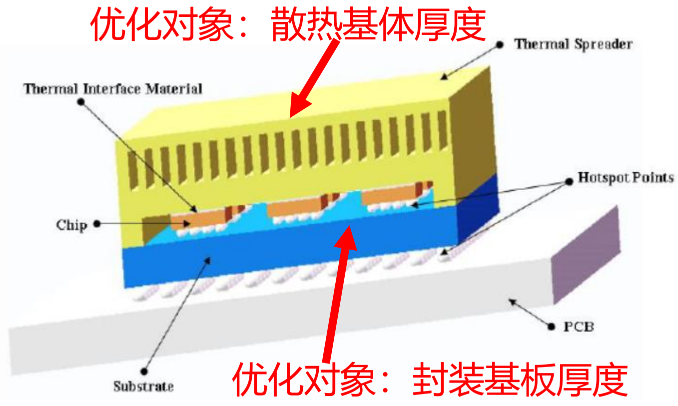


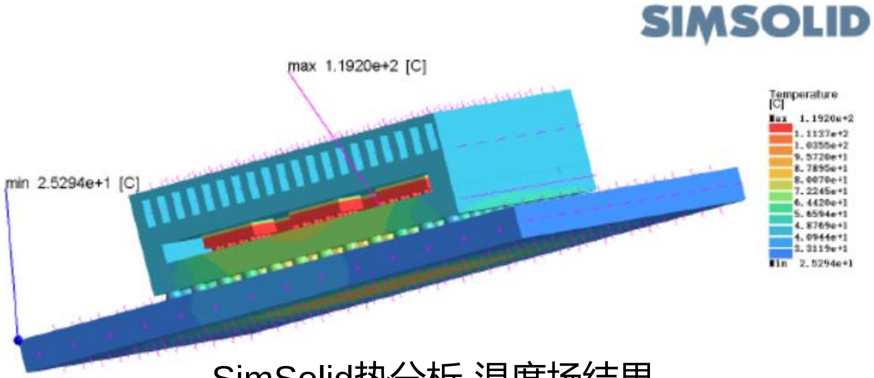
传动承载

ALTAIR

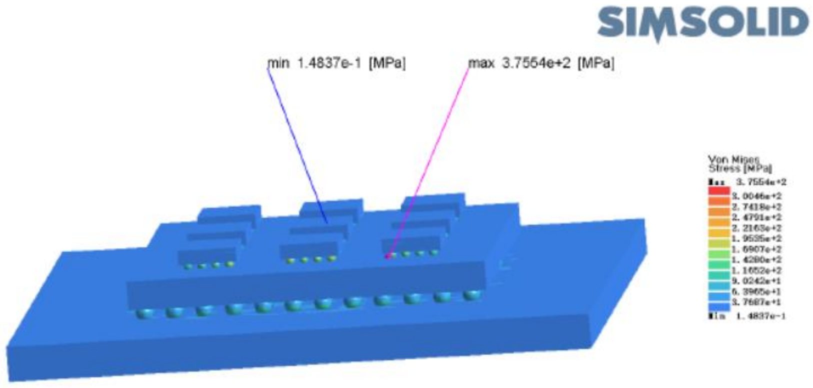
电子电路：MCM-BGA 封装体运行温度及热应力分析与优化设计

项目背景：MCM-BGA封装体共包含 9个芯片，每个芯片表面的热功率为 50W/cm^2，通过封装体表面和散热外壳冷却通道散热。优化封装基体厚度和散热基体厚度，确保芯片工作温度和避免热应力导致的开裂。





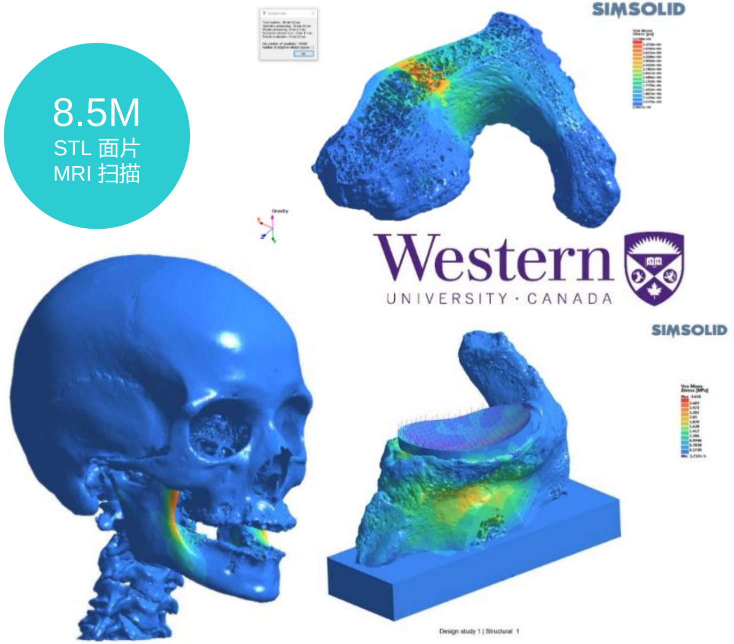
SimSolid热分析-温度场结果



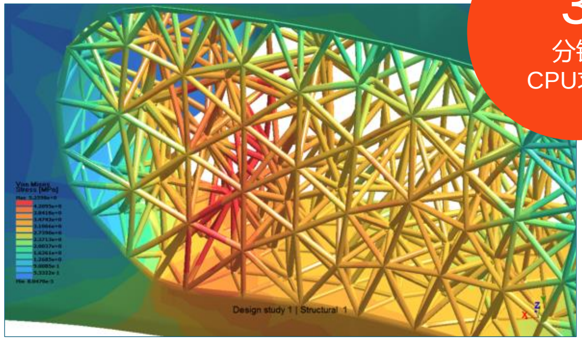
SimSolid热力耦合分析-应力场结果

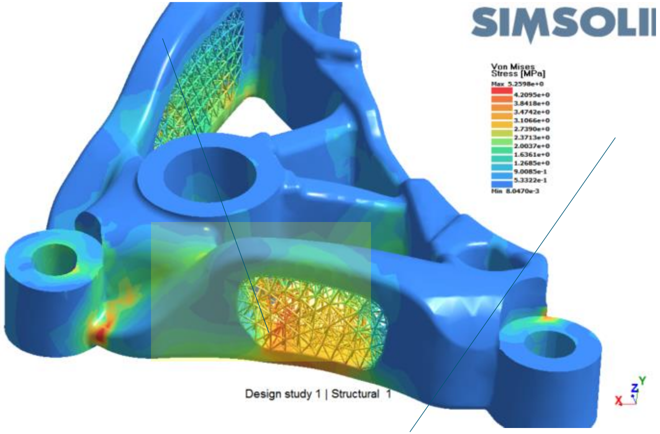
SimSolid增材制造与医疗器械应用案例

8.5M
STL 面片
MRI 扫描



3
分钟
CPU求解





ALTAIR

SimSolid的优势来自哪里?

大型装配体分析速度快

- 无需网格
- 自动识别装配

精度满足要求

- 线性静力学、模态等分析精度可以满足前期设计的要求

学习周期短

- 传统有限元几何简化、网格质量控制、建模简化等需要大量工程经验
- SimSolid纯基于完整几何模型建立高保真的仿真

硬件资源成本底

- SimSolid只需要传统CPU和桌面级的配置即可高效运算

SimSolid桌面级配置
可满足大型装配体的求解需求

Altair技术支持及资料

- 官方技术支持电话： 400-619-6186　　官方技术支持邮箱： support@altair.com.cn

官方微信：
AltairChina

官方博客：
blog.altair.com.cn

官方资料库：
https://nas.altair.com.cn:5001/fsdownload/DhVLPcodc/Altair-KnowledgeBase

B站：
https://space.bilibili.com/478537404

ALTAIR

THANK YOU

altair.com

#ONLYFORWARD

ALTAIR