



GTAM – GENESIS® Topology for ANSYS® Mechanical

安世亚太科技股份有限公司

目录

- VR&D简介
- GENESIS 结构优化
- 概述：GENESIS® Topology Extension for ANSYS® Mechanical (GTAM)
- 结构优化概念
- GTAM 功能
- 应用GTAM建立拓扑优化
- 实例展示
- GENESIS 拓扑优化应用
- 总结

VR&D简介

■ 公司背景

- Vanderplaats Research and Development, Inc. (VR&D) 由Gary Vanderplaats博士成立于1984
- 在上世纪70年代和80年代早期, Vanderplaats 博士开发的优化求解器在当时非常著名, 应众多用户的强烈需求, 成立了公司, 支持并改进优化器。
- 公司自当时开始便致力于开发最好的优化软件和优化技术。

■ VR&D 产品

- GENESIS - 结构优化
- Design Studio - GENESIS的前后处理
- GTAM - GENESIS Topology for ANSYS Mechanical
- VisualDOC - 多学科优化
- DOT - 优化器
- BIGDOT - 大规模优化器
- SMS - 快速特征值求解器
- ESLDYNA - Extend GENESIS optimization to LS-DYNA

GENESIS 结构优化

■ 分析选项

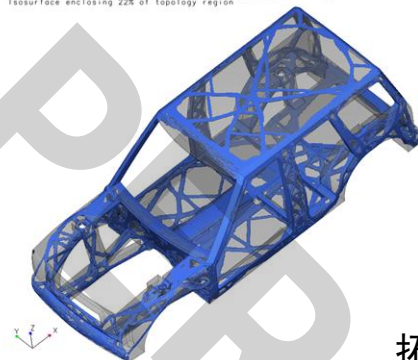
- 线性静态
- 惯性释放
- 正规模态
- 频率响应
- 热传递
- 屈曲分析
- 随机响应
- 非线性接触

■ 优化选项

- 拓扑
- 尺寸
- 自由尺寸
- 形状
- 形貌
- 自由形状

GENESIS 结构优化

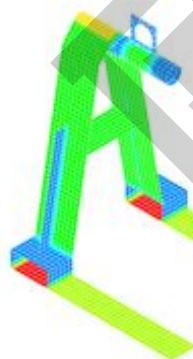
BODY IN WHITE SOLID ELEMENT TOPOLOGY BLANK MODEL
MP+D 10
TOPOLOGY DESIGN ELEMENT DENSITY: DESIGN CYCLE NUMBER = 28
Isosurface enclosing 22% of topology region



Cycle 28 Topology Result

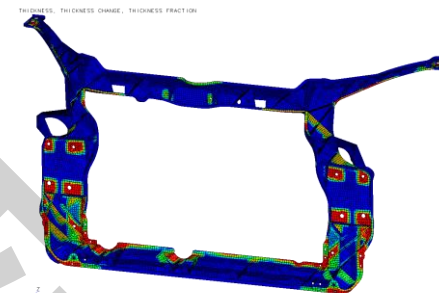
拓扑

PAUSET LIFTER FINITE ELEMENT MESH
THICKNESS, THICKNESS CHANGE, THICKNESS FRACTION



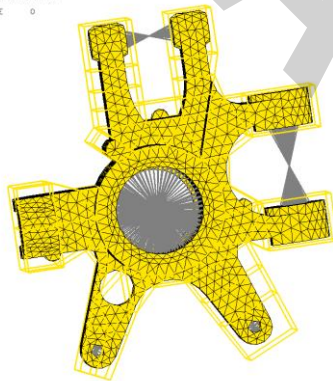
尺寸

THICKNESS, THICKNESS CHANGE, THICKNESS FRACTION



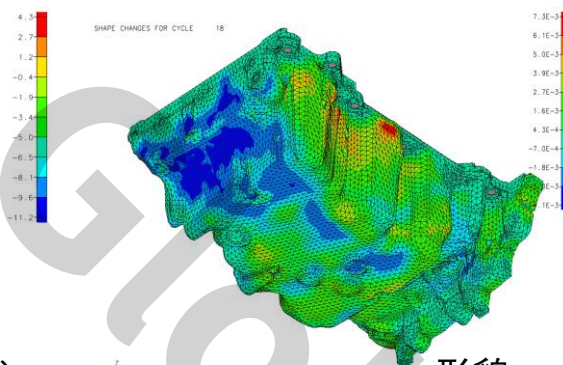
自由尺寸

GENESIS JOB, CREATED ON 25-AUG-00
SHAPE CHANGES FOR CYCLE 0



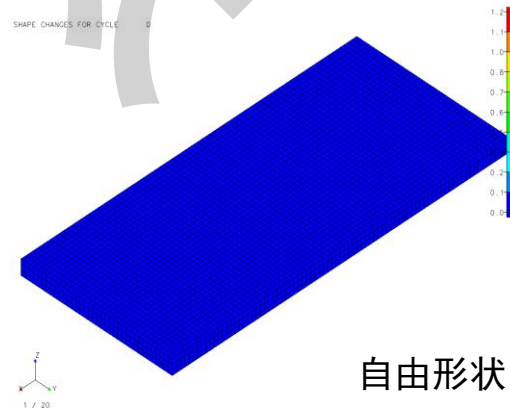
形状

SHAPE CHANGES FOR CYCLE 18



形貌

SHAPE CHANGES FOR CYCLE 0



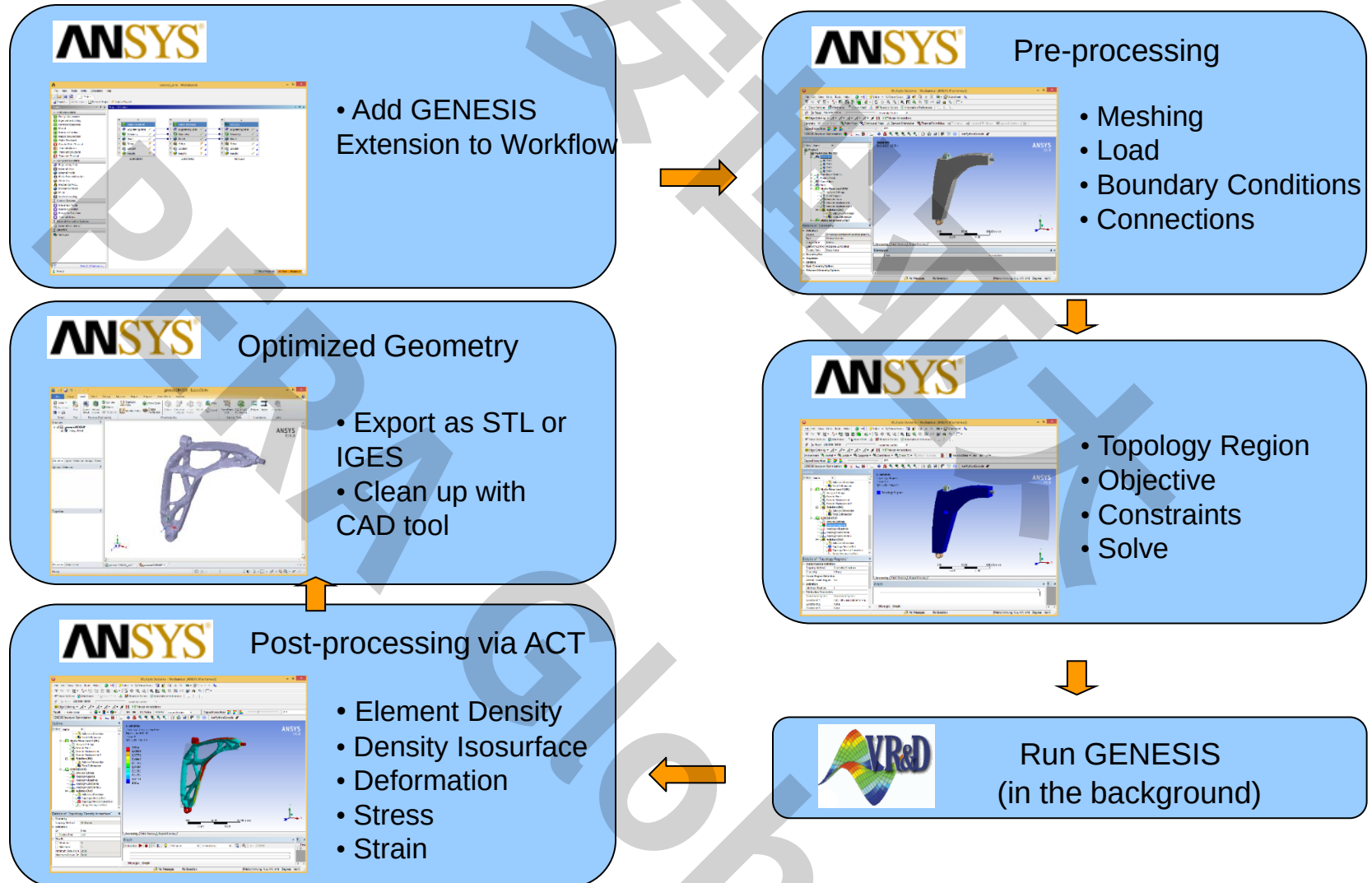
自由形状

概述：GENESIS Topology for ANSYS Mechanical (GTAM)

■ 集成GENESIS拓扑优化功能集成到ANSYS环境下

- 基于ACT开发 (ANSYS Customization Toolkit)
- 方便的将拓扑优化加入到既有的ANSYS Workbench工作流程中
- 方便快速的创建拓扑优化数据
- 方便快速的对拓扑优化结果进行后处理
- 导出优化后的几何为STL或IGES格式

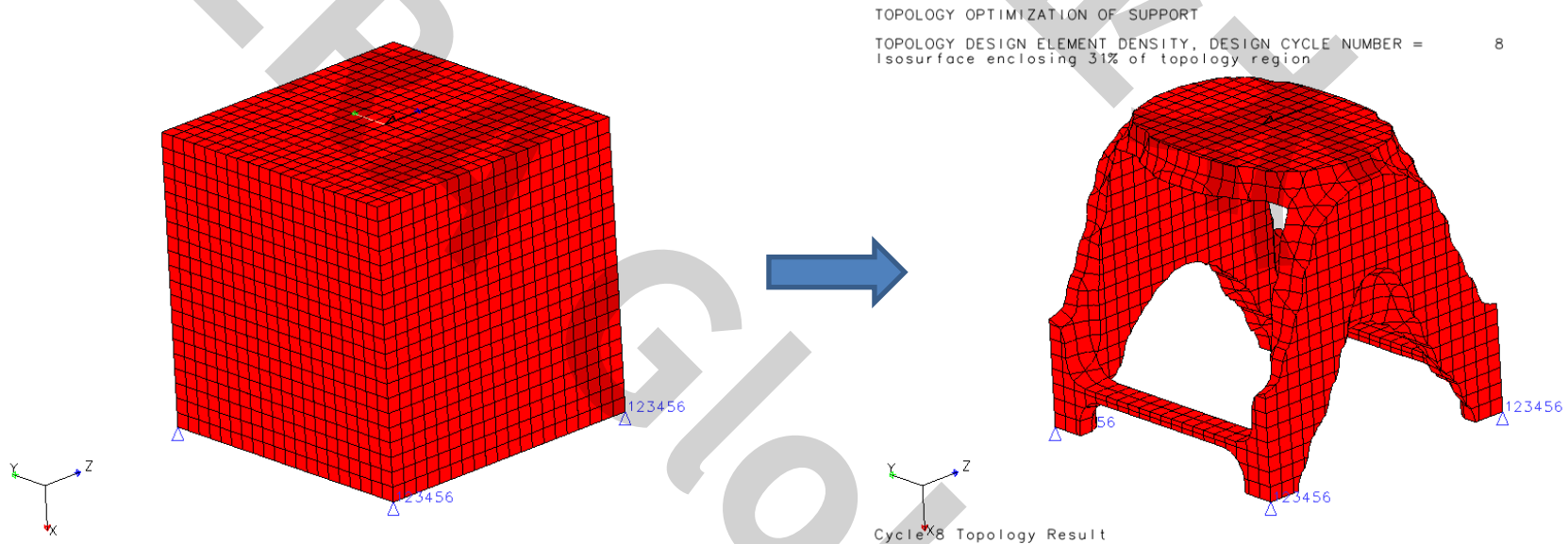
概述: GENESIS Topology for ANSYS Mechanical (GTAM)



拓扑优化概念

■ 什么是拓扑优化？

- 拓扑优化是这样一种优化，它允许从一块材料中切割出一个结构来。



拓扑优化概念

- 在GENESIS中，拓扑优化通过创建设计变量，使其与在设计空间内的所有单元的杨氏模量和密度相关联来实现。
- 设计变量从0.0 到1.0变化
 - 0.0 表示单元没有刚度和密度
 - 1.0 表示单元有正常的刚度和密度

GTAM 功能

■ 支持的分析类型

- 静态Static
- 模态Modal
- 线性屈曲Linear Buckling
- 谐响应Harmonic
- 随机响应Random

GTAM 功能

■ 支持的响应

- 应变能Strain Energy
- 位移Displacement
- 相对位移Relative Displacement
- 频率Frequency
- 屈曲载荷因子Buckling load factors
- 模态/直接/随机位移，速度和加速度Modal/direct/random displacement, velocities and accelerations
- 应力Stresses
- 惯性矩Moment of Inertia
- 质量分数Mass Fraction
- 接触压力Contact Pressure
- 接触间隙Contact Clearance

GTAM 功能

■ 支持的材料类型

- 各向同性Isotropic
- 正交各向异性Orthotropic
- 各向异性Anisotropic

GTAM 功能

■ 制造约束

• 铸造



FGZ



FBZ



FTZ

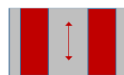


FSZ



F0Z

• 挤压和均匀分布



EZ

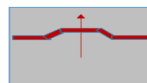


UZX

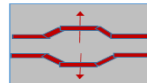


UXYZ

• 冲压

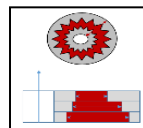


S1Z

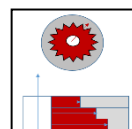


S2Z

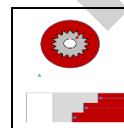
• 径向填充



RGZ

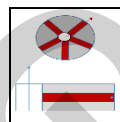


RBZ



RTZ

• 径向轮辐



KZ

• 平面对称和循环对称

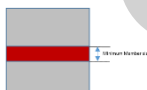


MZX



CX

• 最大&最小成员尺寸



应用GTAM建立拓扑优化设计

■ 定义拓扑数据

- 拓扑设计区域
- 拓扑目标函数
- 拓扑约束函数

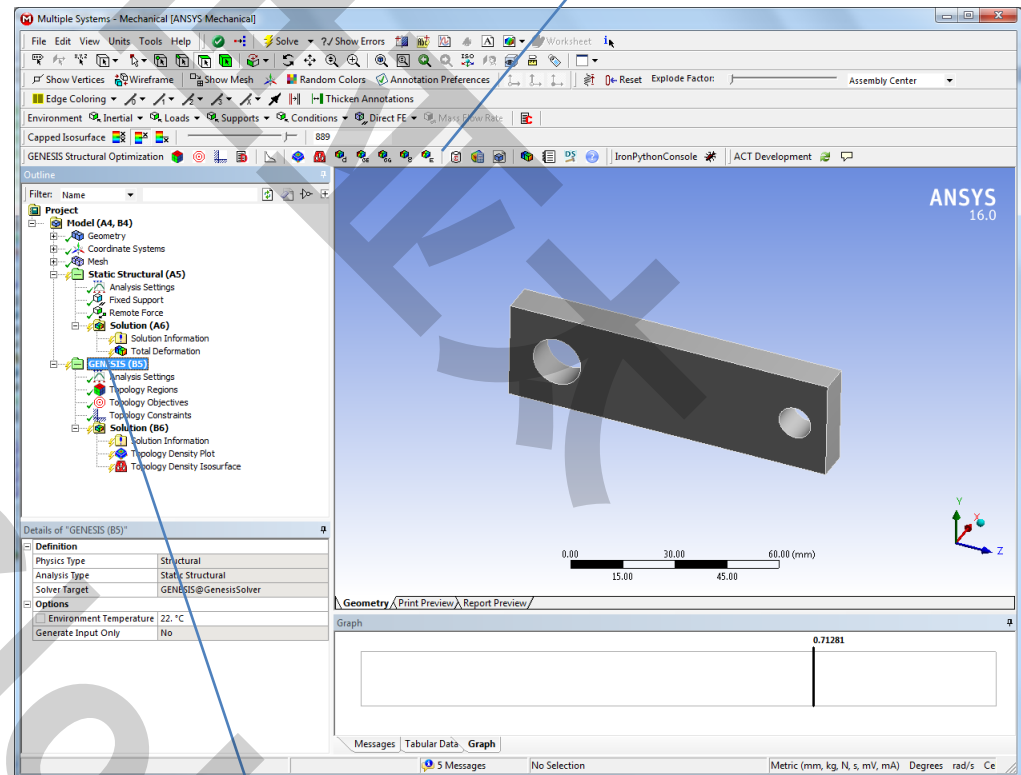
■ 检查求解过程

- 解的输出
- 实时设计历程输出

■ 后处理

■ 导出优化结果几何

GTAM 工具栏



GENESIS 系统

拓扑设计区域

GENESIS Structural Optimization

Details of "Topology Regions"

[-] Design Region Definition	
Scoping Method	Geometry Selection
Geometry	1 Body
[-] Frozen Region	
Define Frozen Region	Yes
[-] Frozen Region Definition	
Scoping Method	Geometry Selection
Geometry	2 Faces
[-] Definition	
Init Mass Fraction	0.3
[-] Fabrication Constraints	
Coordinate System	symmetry
Constraint 1	FOX : Fill X axis (plane to + and -)
Constraint 2	MZX : Mirror about XZ plane
Constraint 3	None
Minimum Size Control	Yes
Minimum Member Size	4 [mm]
Spread Fraction	0.5
Maximum Size Control	No

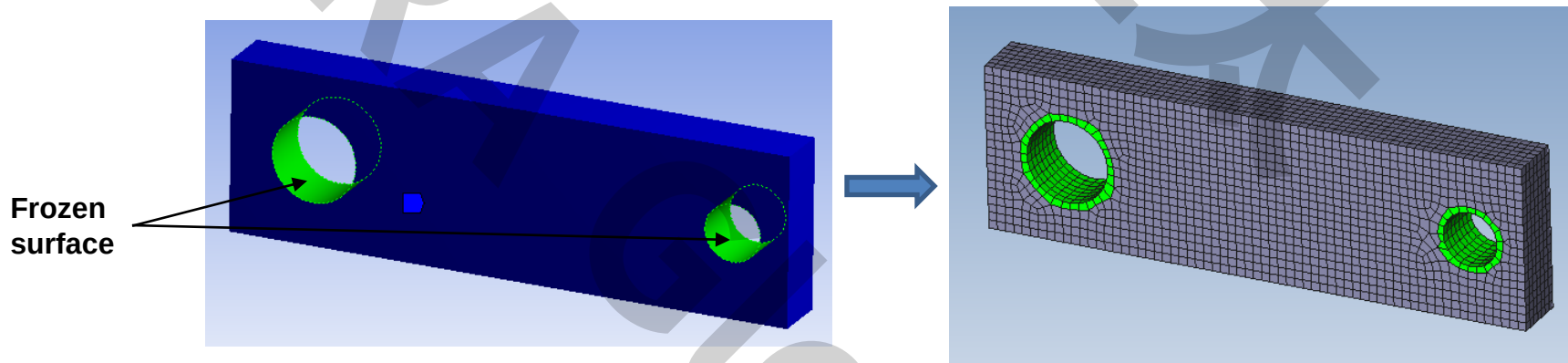
- 设计域
- 冻结域
- 制造约束
 - 对称Symmetry
 - 挤压Extrusion
 - 填充Filling
 - 对称填充Filling Symmetric
 - 冲压Stamping
 - 均匀分布Uniform
 - 径向填充和轮辐Radial Filling and Spokes
 - 在给定区域可以组合施加3种制造约束
- 最小成员尺寸
- 最大成员尺寸
- 可定义多个设计区域，每个设计域可施加不同的制造约束

15

拓扑设计区域

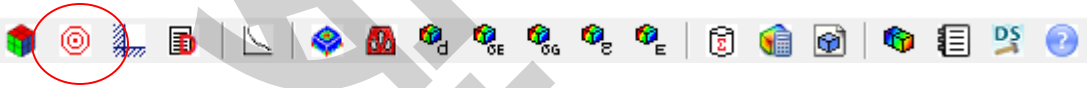
■ 冻结域

- 实体的表面或壳体的边可以被选作为冻结的
- 与冻结的实体表面或壳体的边相邻的一层单元会从拓扑设计域中去除，从而不参加拓扑优化

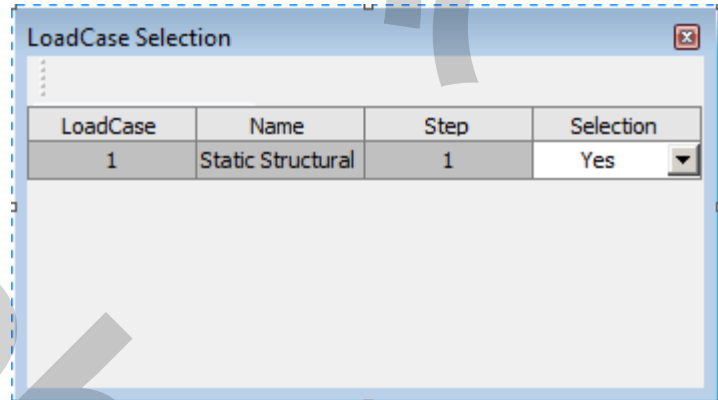
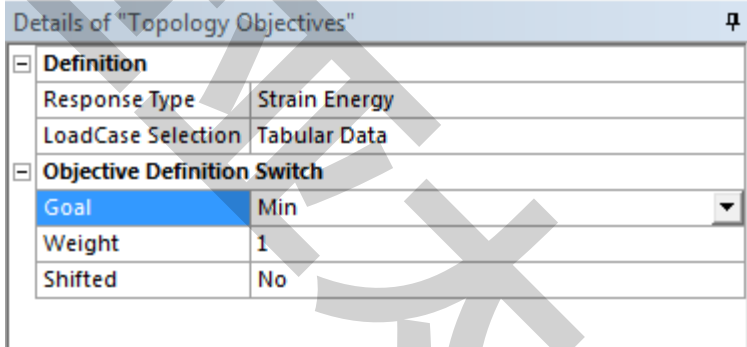


拓扑目标函数

GENESIS Structural Optimization



- 拓扑优化的目标函数是在拓扑优化过程中被选为目标函数的响应
 - 响应类型
 - Strain Energy应变能
 - Frequency Mode Number模态频率
 - Mass Fraction质量分数
 - Displacement位移
 - Relative Displacement相对位移
 - System Inertia系统惯量
 - Buckling Load Factor屈曲载荷因子
 - Dynamic Displacement动力位移
 - Dynamic Velocity动力速度
 - Dynamic Acceleration动力加速度
 - Contact Pressure接触压力
 - Contact Clearance接触间隙
 - 目标
 - Min最小化
 - Max最大化
 - Min-Max (Beta Method)最小-最大化
 - 支持多目标



拓扑约束函数

GENESIS Structural Optimization



• 拓扑约束是指在拓扑优化过程中被作为约束条件的响应

– 响应类型

- Strain Energy应变能
- Frequency Mode Number模态频率
- Mass Fraction质量分数
- Displacement位移
- Relative Displacement相对位移
- System Inertia系统惯量
- Buckling Load Factor屈曲载荷因子
- Dynamic Displacement动力位移
- Dynamic Velocity动力速度
- Dynamic Acceleration动力加速度
- Contact Pressure接触压力
- Contact Clearance接触间隙

– Constraint Bounds约束边界（上下限）

– Support Multiple Constraints支持多约束

– Bound Type边界类型

Details of "Topology Constraints"

Definition	
Response Type	Mass Fraction
Region	All Designed Groups
Constraint Bounds	
Lower Bound	None
Upper Bound	0.3
Bound Type	Actual

检查求解过程

■ 求解信息

- 时间步长，设计历程
- 错误信息

■ 实时设计历程输出

- 目标函数
- 最大约束偏离

求解信息

The screenshot displays the ANSYS Workbench interface during a topology optimization process. The 'Solve' button in the context menu is highlighted with a red circle. The 'Solution Information' table provides details about the solver output, and the 'ANSYS Workbench Solution Status' dialog box shows the progress of solving the mathematical model.

Context Menu Options:

- Insert
- Solve
- Clear Generated Data
- Rename (F2)
- Open Solver Files Directory

Solution Information Table:

Details of "Solution Information"	
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All

ANSYS Workbench Solution Status Dialog:

Overall Progress...

Solving the mathematical model...

Stop Solution

Worksheet View Data:

```

NUMBER OF CHEXA (9-21 NODE) ELEMENTS: 3176
TOTAL NUMBER OF NON RIGID ELEMENTS: 11886
NUMBER OF RBE3 ELEMENTS: 4
NUMBER OF ELEMENT PROPERTIES: 2896
NUMBER OF MATERIALS: 2893
NUMBER OF DEGREES OF FREEDOM: 97228

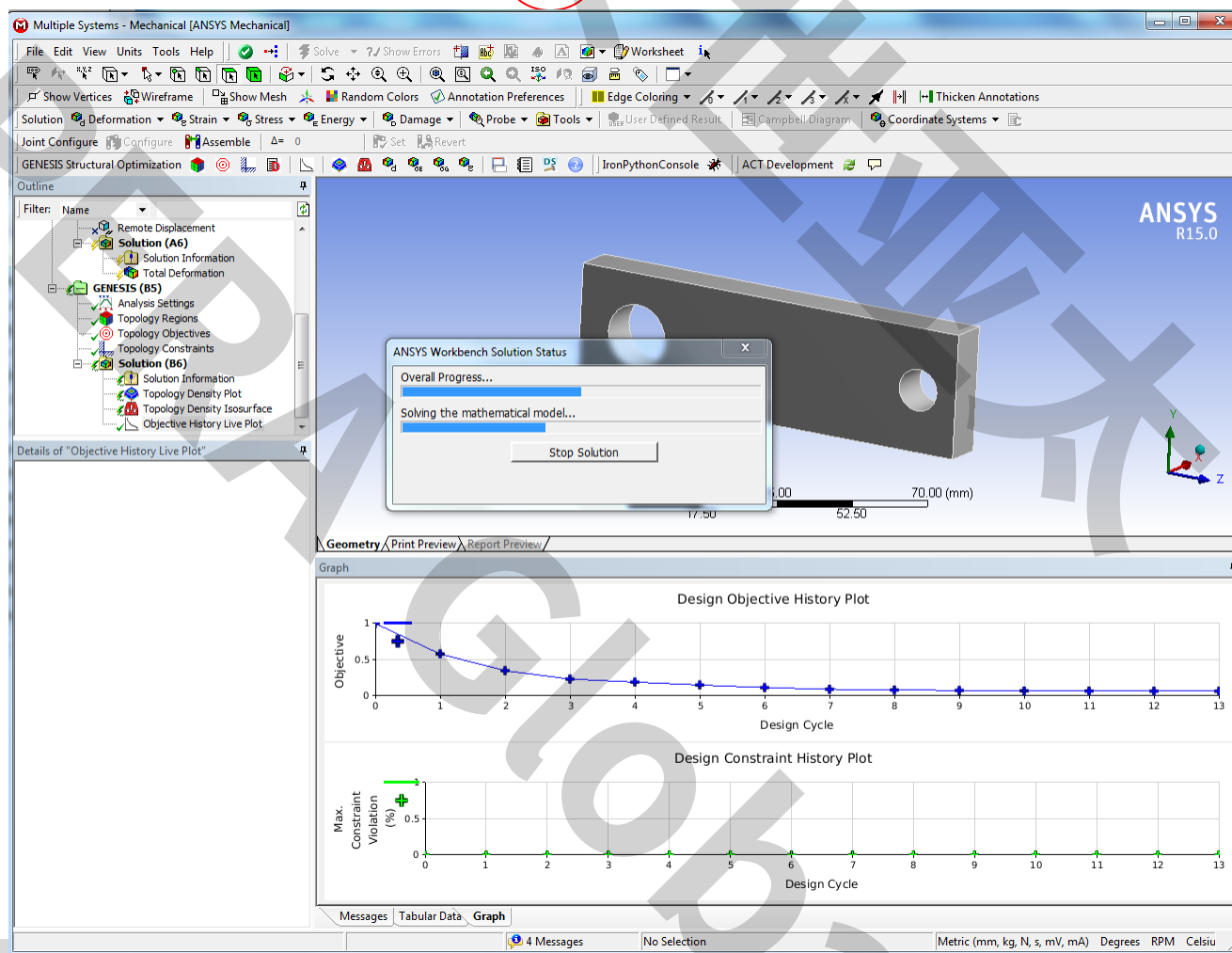
TOPOLOGY OPTIMIZATION PROBLEM SUMMARY
OBJECTIVE FUNCTION: MULTIPLE RESPONSES
TOTAL NUMBER OF TOPOLOGY INDEP. VARIABLES: 2080
NUMBER OF TOPOLOGY DEPEND. DESIGN VARIABLES: 2892
TOTAL NUMBER OF TOPOLOGY DESIGN VARIABLES: 4972
NUMBER OF TOPOLOGY DESIGNABLE ELEMENTS: 2892
NUMBER OF TOPOLOGY DIRECT RESPONSES: 2
NUMBER OF POTENTIAL TOPOLOGY CONSTRAINTS: 2

LOAD CASES SUMMARY
NUMBER OF STATIC LOAD CASES: 1
TOTAL NUMBER OF LOAD CASES: 1

CPU TIME SPENT IN MODULE PRINTI = 0.00 SEC, IN TOTAL = 17. SEC
ELAPSED TIME IN MODULE PRINTI = 0.00 SEC, IN TOTAL = 18. SEC
CPU TIME SPENT IN MODULE PRINTU = 0.00 SEC, IN TOTAL = 17. SEC
ELAPSED TIME IN MODULE PRINTU = 0.00 SEC, IN TOTAL = 18. SEC
CPU TIME SPENT IN MODULE ESL = 0.00 SEC, IN TOTAL = 17. SEC
ELAPSED TIME IN MODULE ESL = 0.00 SEC, IN TOTAL = 18. SEC
START DESIGN CYCLE 0
CPU TIME SPENT IN MODULE FEH = 0.00 SEC, IN TOTAL = 17. SEC
ELAPSED TIME IN MODULE FEH = 0.00 SEC, IN TOTAL = 18. SEC
CPU TIME SPENT IN MODULE ... = 0.00 SEC, IN TOTAL = 17. SEC
ELAPSED TIME IN MODULE ... = 0.00 SEC, IN TOTAL = 18. SEC
  
```

实时设计历程输出

GENESIS Structural Optimization

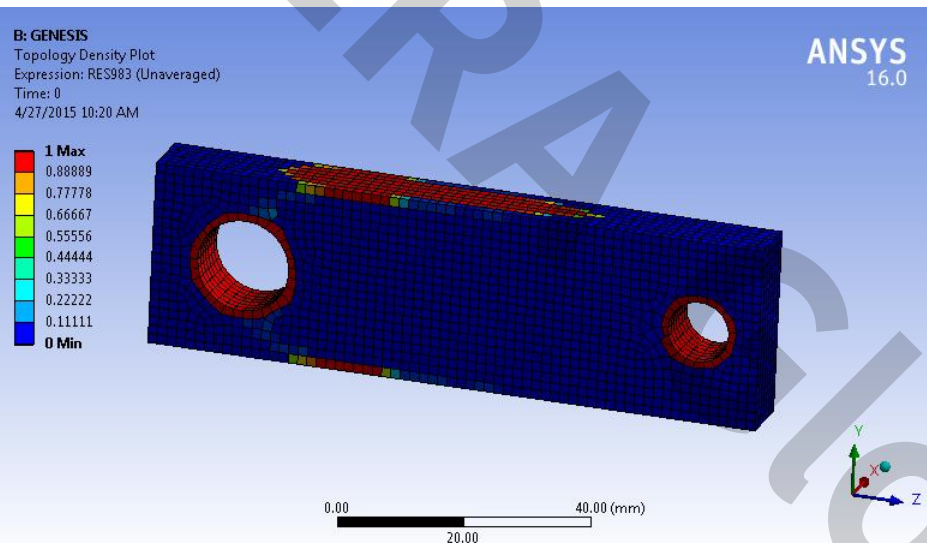


后处理

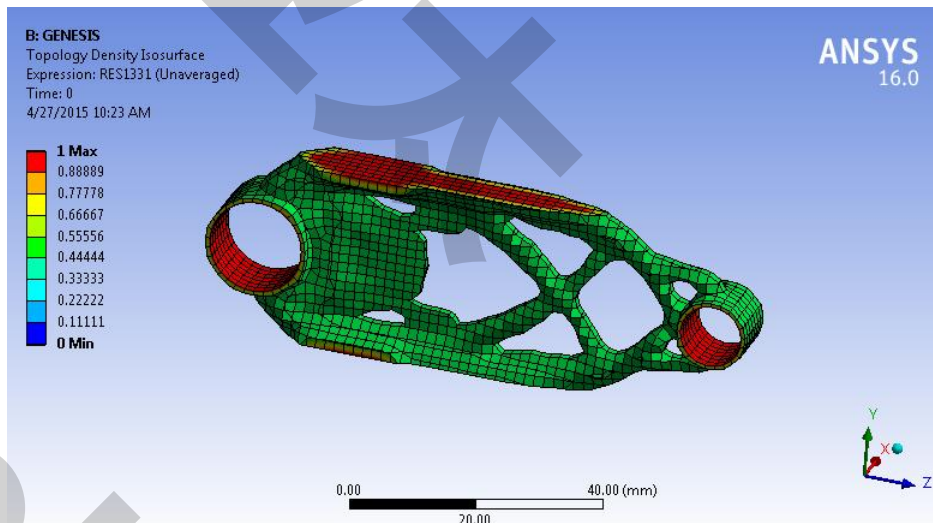
GENESIS Structural Optimization

■ 拓扑结果

- 单元密度
- 密度等值面



单元密度



密度等值面

后处理

GENESIS Structural Optimization



■ 分析结果

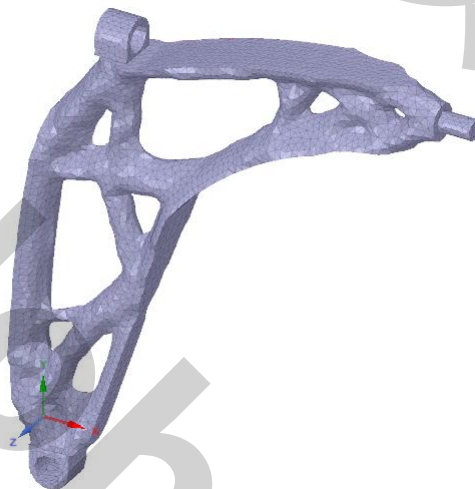
- 位移Displacement
- 单元应力Element Stress
- 节点应力Grid Stress
- 应变Strain
- 应变能Strain Energy

输出粗化表面

GENESIS Structural Optimization



- 以STL 或IGES 格式输出粗化表面
- Cutoff value截断值：： 指定一个密度值来创建拓扑密度等值面。
 - 如果一个单元的密度小于这个值，它将被从集合中删除。
- Level of details:控制输出的等值面的光滑程度
 - 0 代表最粗糙； 20 代表最光滑
- 只导出可见的组件



其他: 写出GENESIS输入文件, 启动Design Studio



■ 写出 Genes i s输入文件:

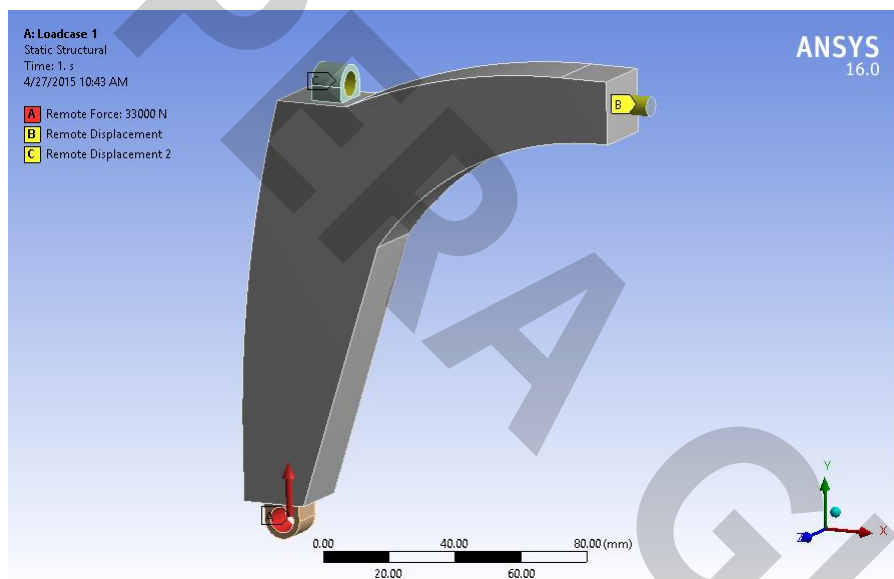
- 网格
- 材料属性
- 载荷及边界条件
- 连接
- 设计数据

■ 启动Design Studio For Genesis:

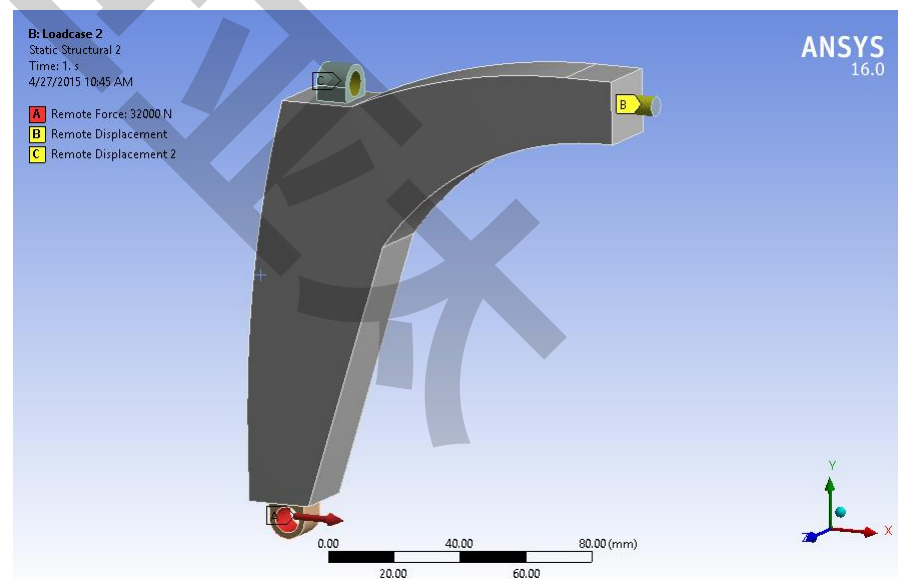
- 模型自动导入Design 中
- 进行其他功能如sizing, shape等

实例：Topology Optimization of an Automobile Control Arm

■ 载荷和边界条件



工况 1



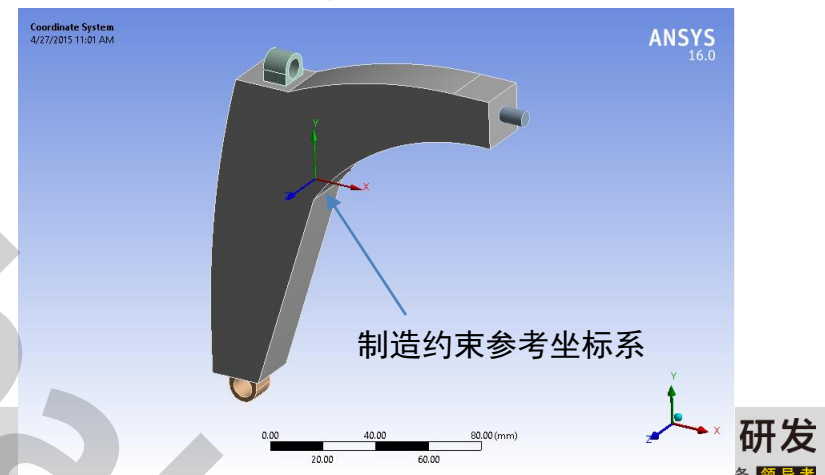
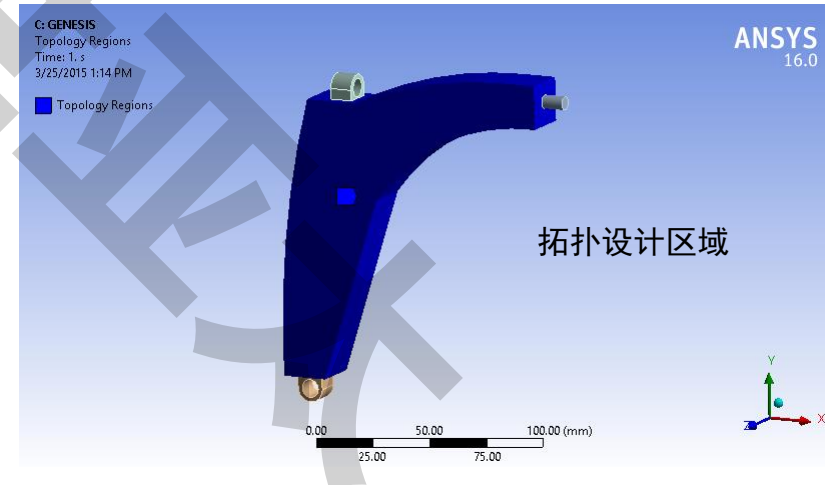
工况 2

实例：Topology Optimization of an Automobile Control Arm

■ 目标：得到轻质而又刚的结构

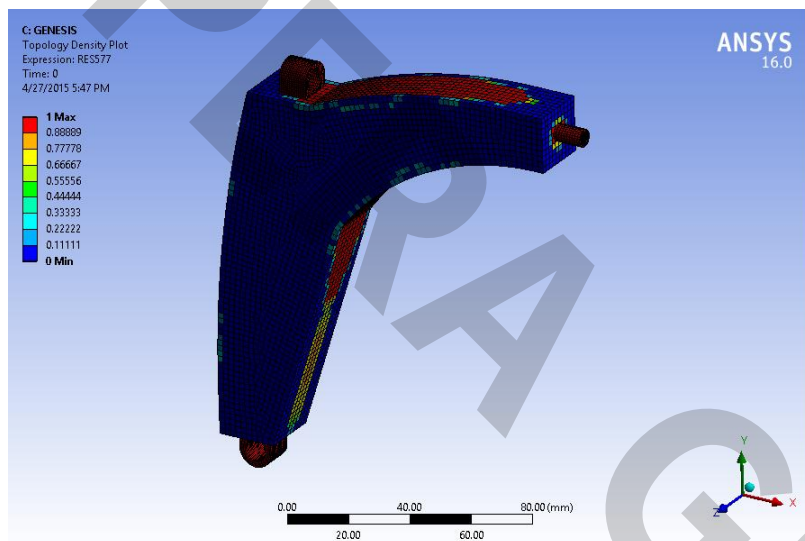
■ 问题建立

- 目标
 - 最小化质量
- 约束
 - Displacement for loadcase 1 $< 1.2 \text{ mm}$
 - Displacement for loadcase 2 $< 10 \text{ mm}$
- 设计域
 - 制造约束
 - F0Z (Casting symmetrically)

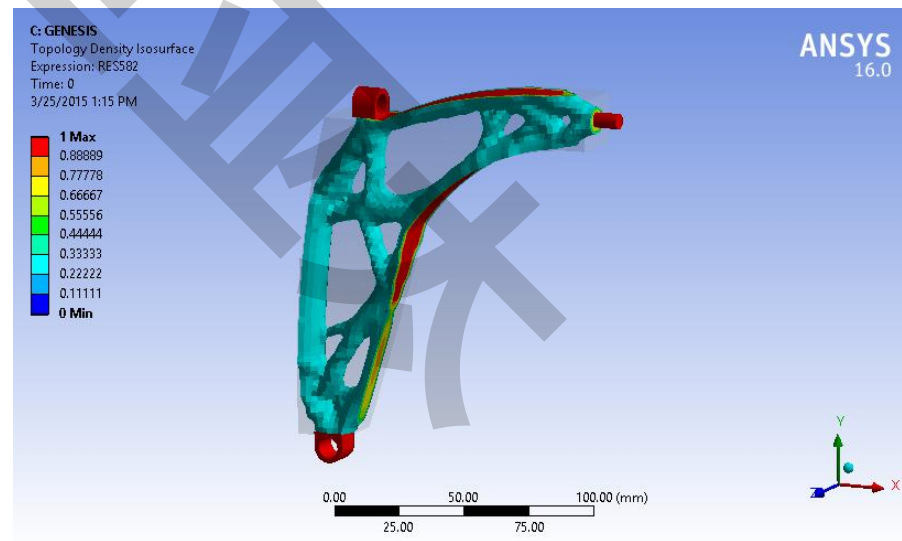


实例：Topology Optimization of an Automobile Control Arm

■ 拓扑结果



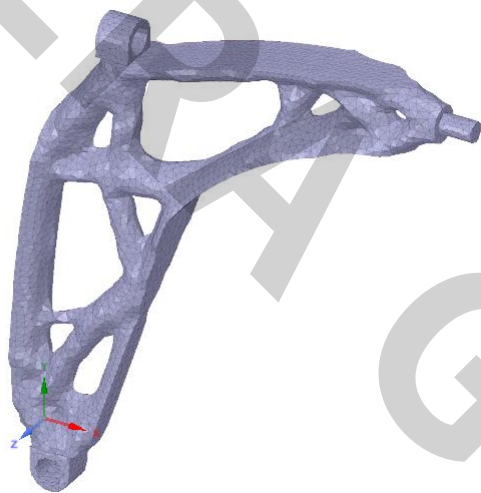
单元密度



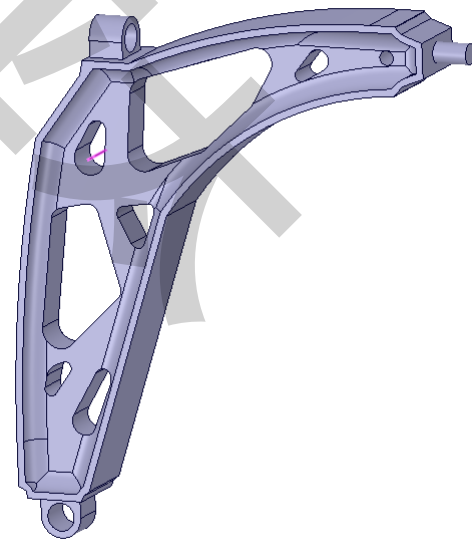
密度等值面

实例：Topology Optimization of an Automobile Control Arm

- 导出优化后的几何
- 应用SpaceClaim进行几何清理



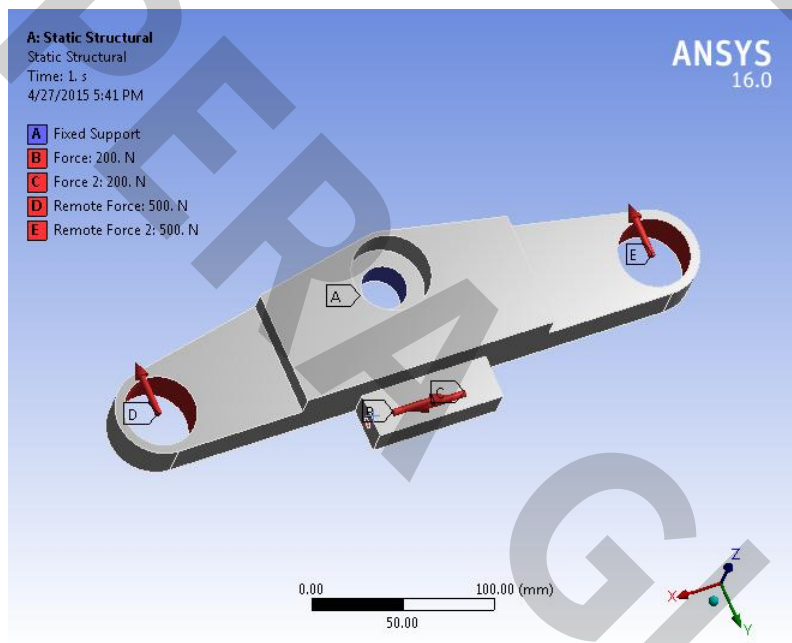
STL 模型



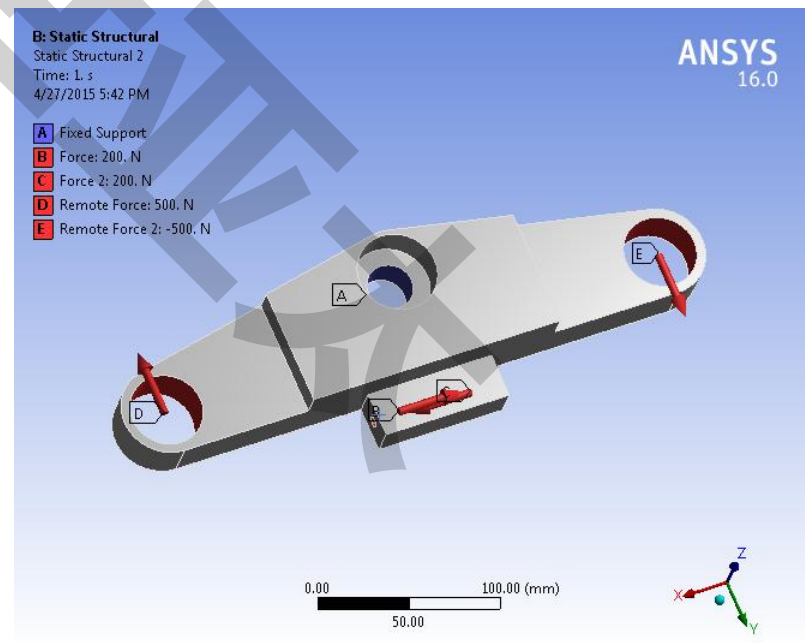
CAD 模型

实例：Topology Optimization of A Clamping Bracket

■ 载荷和边界条件



工况 1



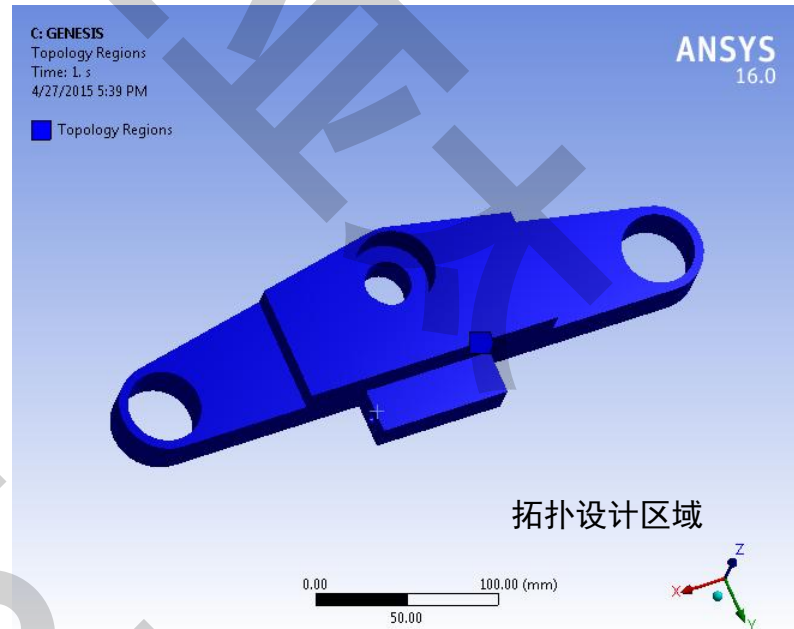
工况 2

实例：Topology Optimization of A Clamping Bracket

■ 目的：得到轻质而刚的结构

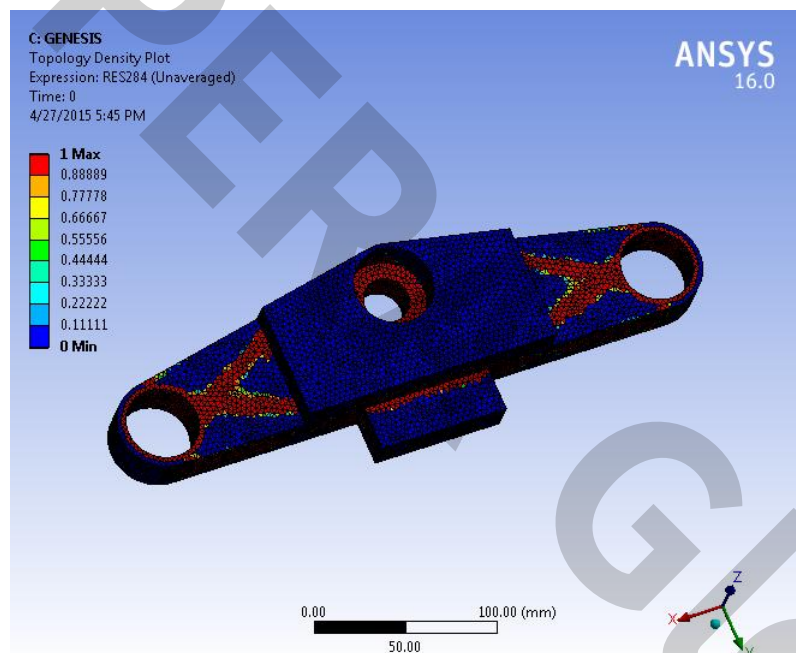
■ 问题建立

- 目标
 - 最小化应变能
- 约束
 - 质量分数 < 30%
- 设计域
 - 制造约束
 - FGZ (Casting in Z direction)
 - 关于YZ平面对称

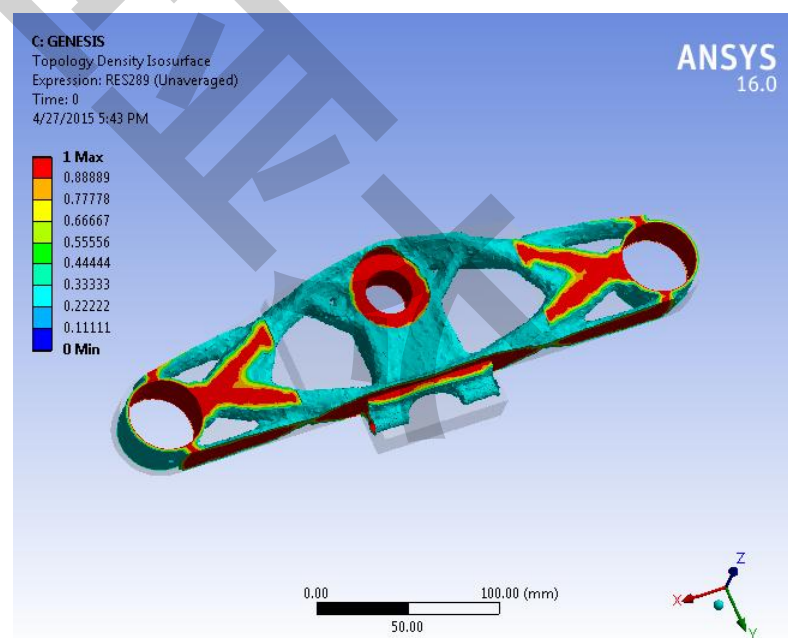


实例：Topology Optimization of A Clamping Bracket

■ 拓扑结果



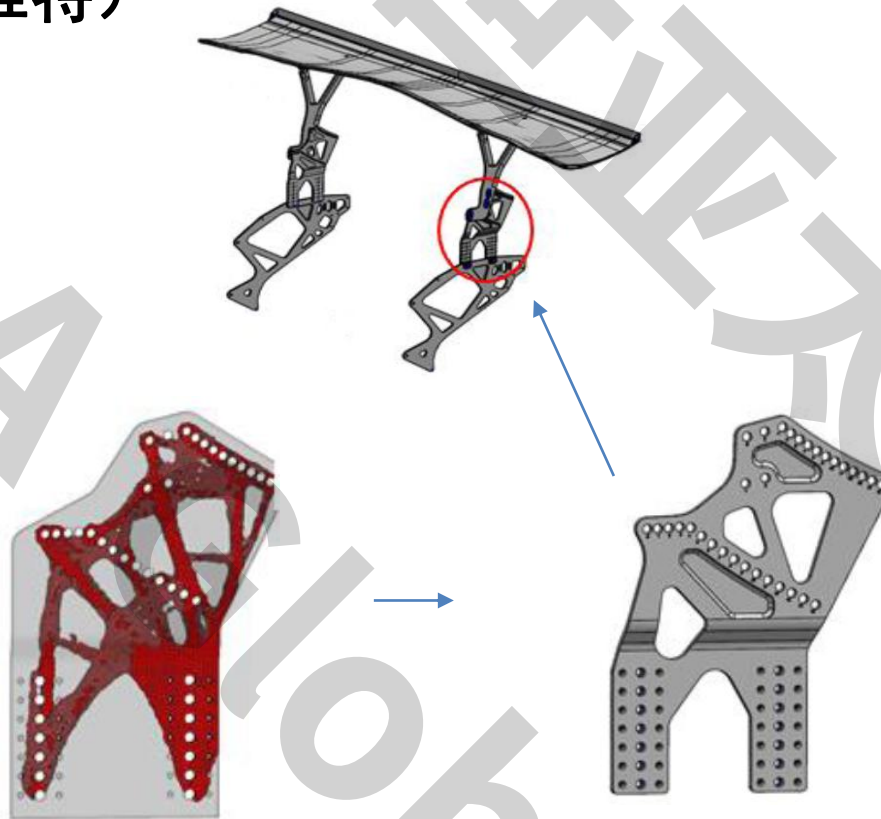
单元密度



密度等值面

GENESIS 拓扑优化应用

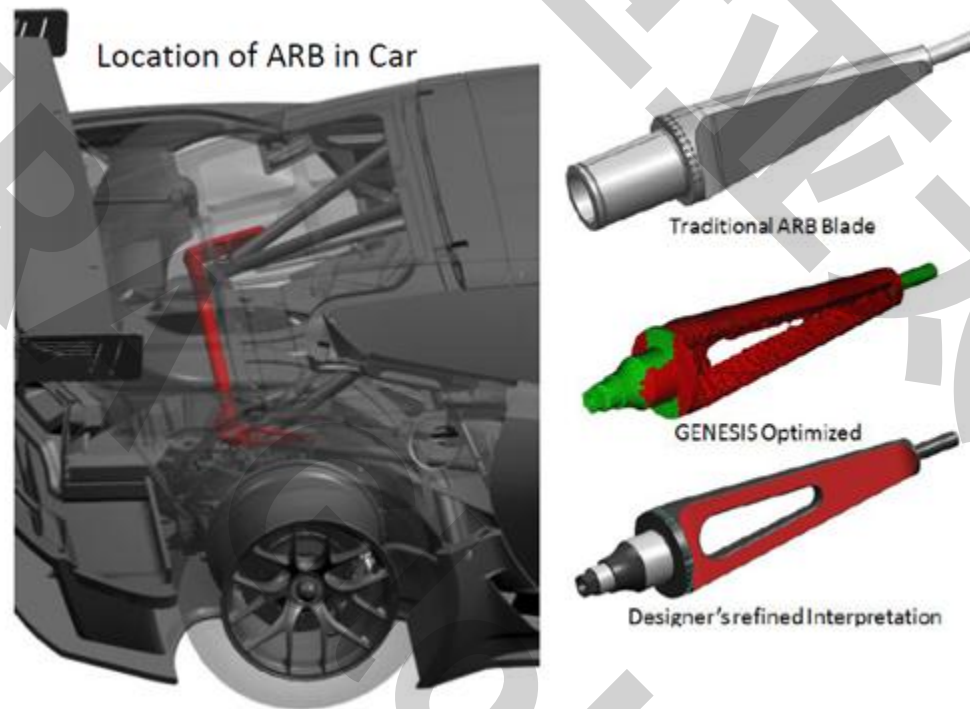
■ Topology Optimization of C6.R Tail Frame Assembly (雪佛兰克尔维特)



C6.R Tail Frame Assembly
(Courtesy of Pratt & Miller Engineering)

GENESIS 拓扑优化应用

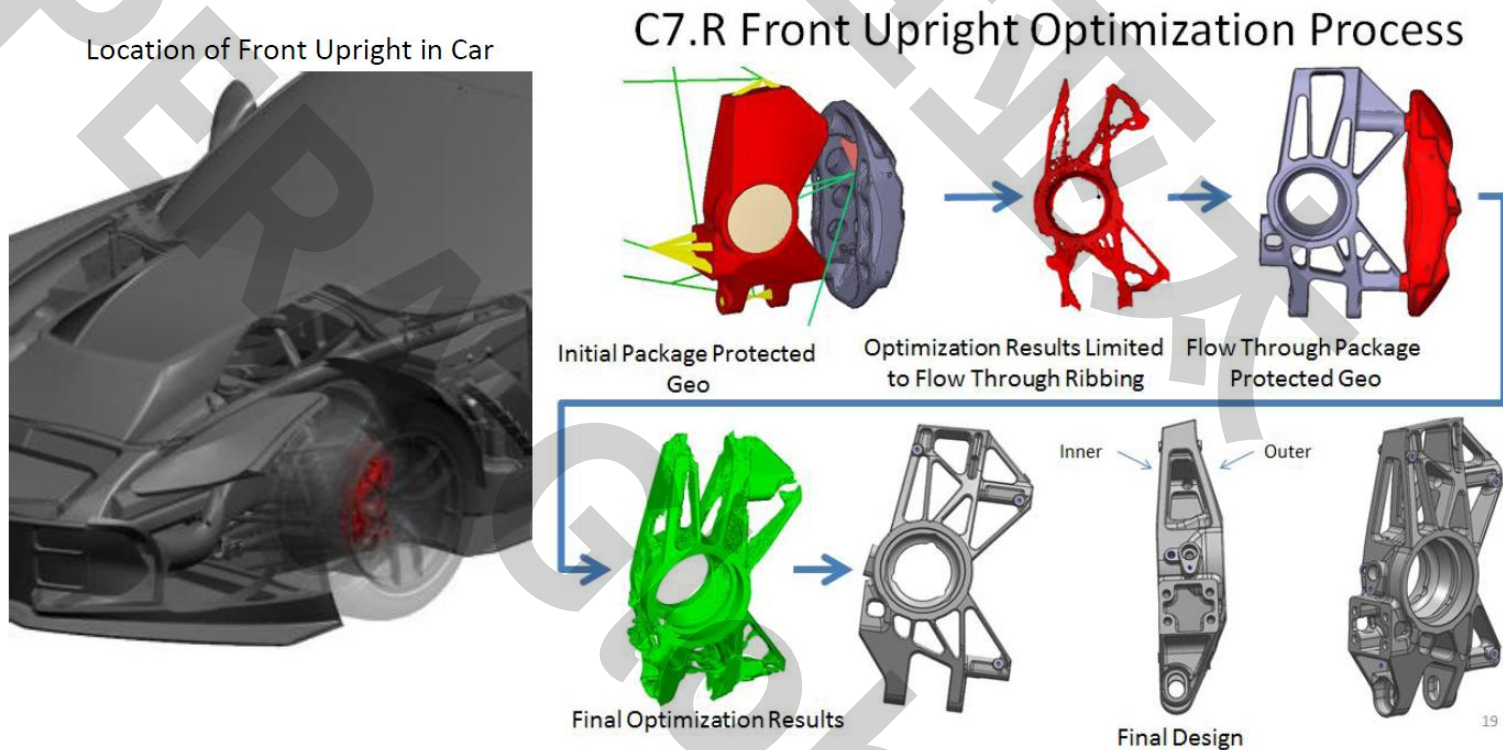
■ Topology Optimization of Anti-roll Bar (防滚架)



Anti-roll Bar
(Courtesy of Pratt & Miller Engineering)

GENESIS 拓扑优化应用

■ Topology Optimization of C7.R Front Upright (雪佛兰克尔维特赛车)



C7.R Front Upright
(Courtesy of Pratt & Miller Engineering)

总结

- GTAM 允许用户在ANSYS环境下容易的建立拓扑优化问题，并进行优化，后处理，导出优化后的几何。
- 设计者们将在一个可靠、稳健且易于应用的界面下很容易的获得自动生成的新的设计方案。