高级整车级
前处理建模/高性能计算解决方案

IDAJ-China 上海技术部

目录

- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

CAE前处理及高性能并行计算的重要性

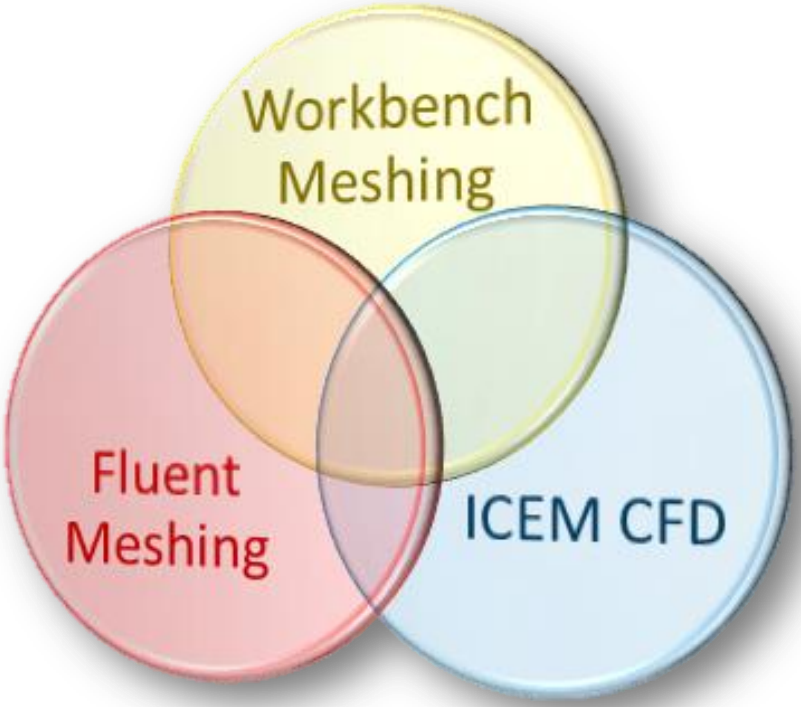
- 前处理的重要性
  - 网格是计算的基础
  - 高质量网格可以保证求解精度及稳健性
  - 采用不同前处理工具可以提高工作效率
- 高性能并行计算的重要性
  - 高质量网格可以使计算快速收敛
  - 提升计算效率，降低计算时间

目录

- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

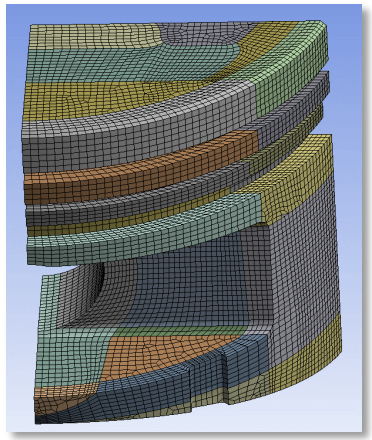
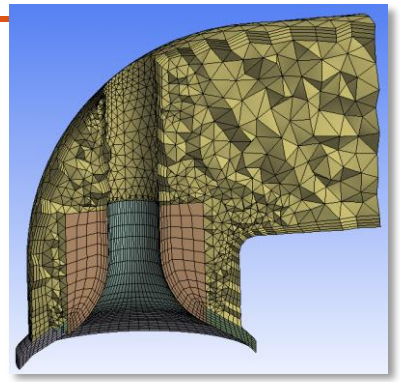
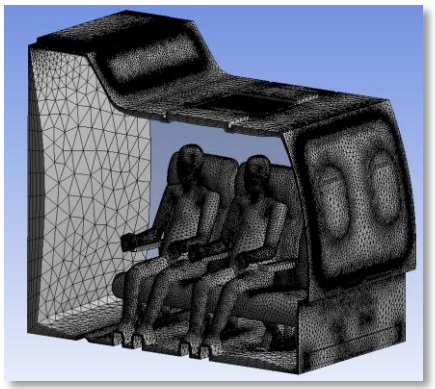
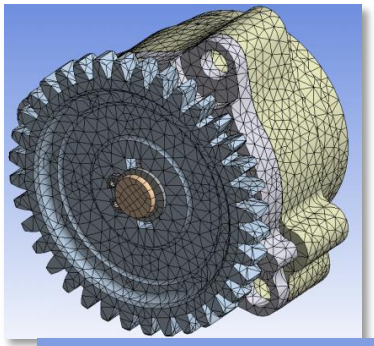
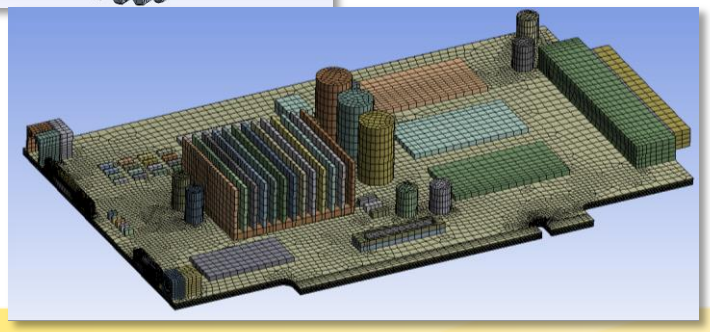
ANSYS前处理解决方案

- ANSYS Meshing
  - 最为一般的网格生成工具，应用范围广泛
  - 自动化网格生成技术
  - 可以参数化设计开发
  - 集成在workbench下
- ICEM CFD
  - 对于CFX用户是最好的网格工具
  - 可以进行几何诊断及修复
  - 高质量网格
  - 六面体网格
  - 内存共享并行网格
- Fluent Meshing
  - 对于Fluent用户是最好的网格工具
  - 可以进行几何诊断及修复
  - 高质量的包面功能
  - 分布并行网格
  - 支持日志和脚本文件



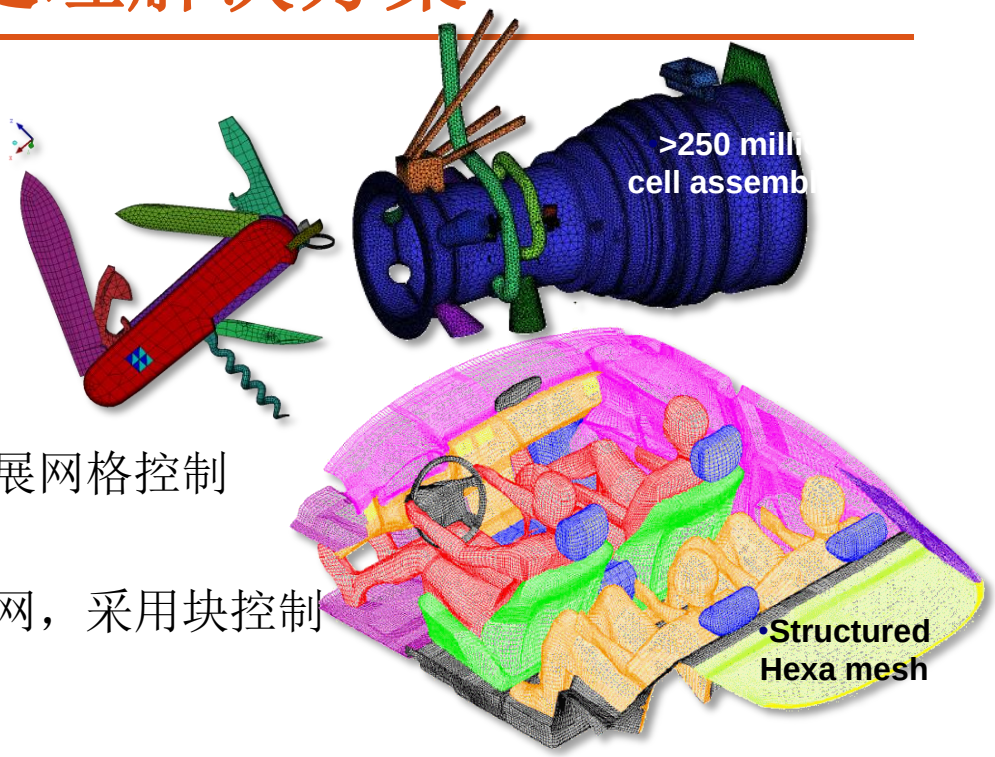
ANSYS前处理解决方案

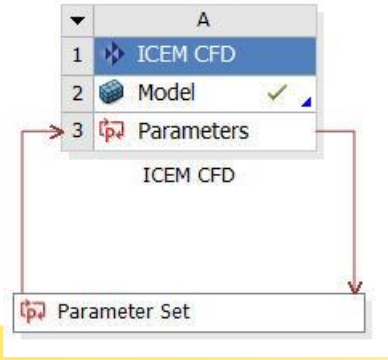
- ANSYS Meshing
  - 一般性的网格工具
  - 可以实现仿真驱动设计
    - ✓ 参数化，高度自动化，灵活性
  - 基于CAD生成网格对于中等复杂模型
  - 很强的自动化能力对于 tet/prism, hex, or hybrid tet/prism/hex
  - 可以向FBU、MBU及EBU提供网格

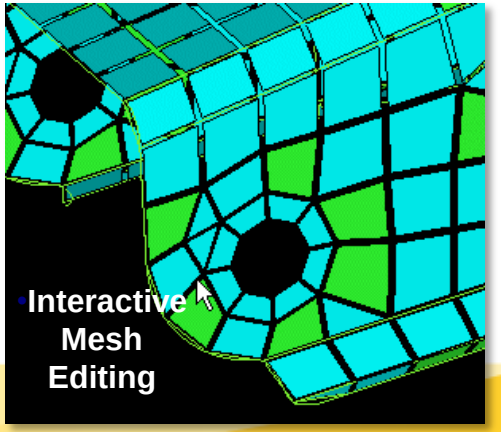






ANSYS前处理解决方案

- ICEM CFD
  - 扩展网格
  - 补丁独立技术
    - ✓ 脏CAD，第三方几何
    - ✓ 碎面数据，STL
    - ✓ CAD，碎面，网格的组合
  - 对于大且复杂模型可以采用扩展网格控制
  - 完整的几何网格功能
  - 自动划分结构或非结构六面体网，采用块控制
  - 灵活的网格诊断技术
  - 输出多种求解格式文件
    - ✓ CFD，FEA等
  - 支持workbench

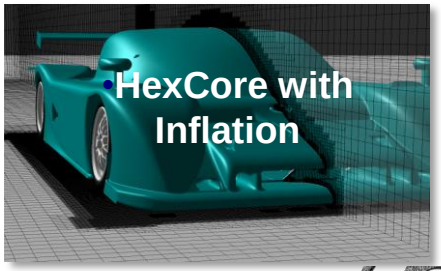


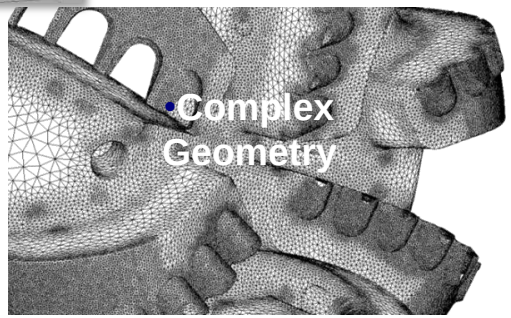


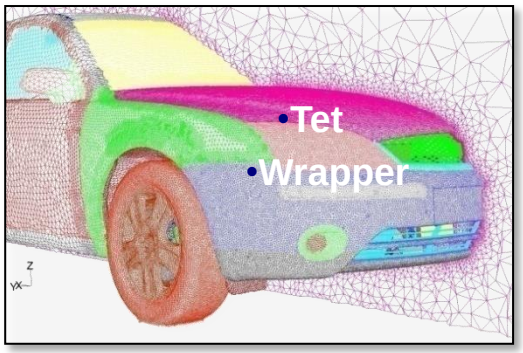


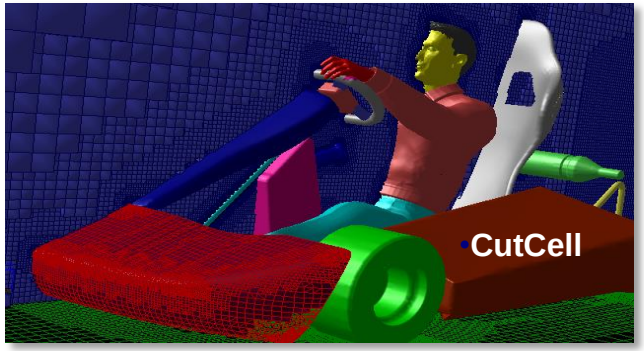
ANSYS前处理解决方案

- Fluent Meshing
  - 先进的CFD网格
  - 统一的网格生成器/求解器
  - 支持CAD，网格，STL的输入
  - 大尺寸，复杂网格
    - ✓ 大规模几何简化的包面技术以及面网格重构技术
    - ✓ 对于大规模网格支持并行边界层/四面体网格生成
    - ✓ 支持尺寸函数
    - ✓ 支持网格诊断，网格编辑，并且有洞/缝隙封闭工具
    - ✓ 支持脚本









目录

- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

ANSYS高级整车级前处理建模工作流程

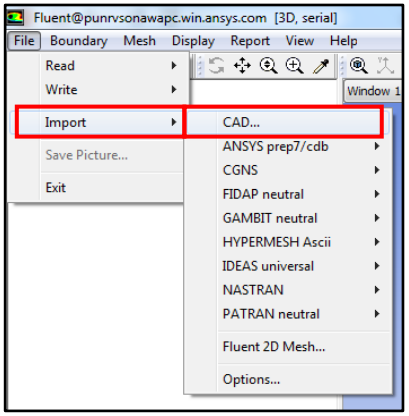
- 建模流程
  - 输入CAD
  - 采用诊断工具缝补孔洞
  - 定义材料点
  - 特征线抽取
  - 几何恢复
  - 定义尺寸函数
  - 修改脚本文件并运行
  - 添加换热器网格

ANSYS高级整车级前处理建模工作流程

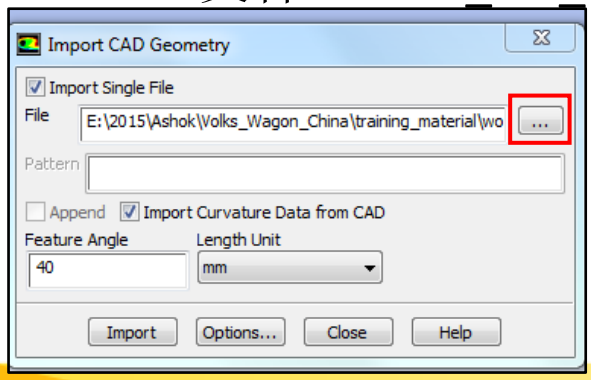
输入CAD

- 将SpaceClaim文件输入到Fluent Mesh中可按照如下步骤：

1. File→Import→CAD



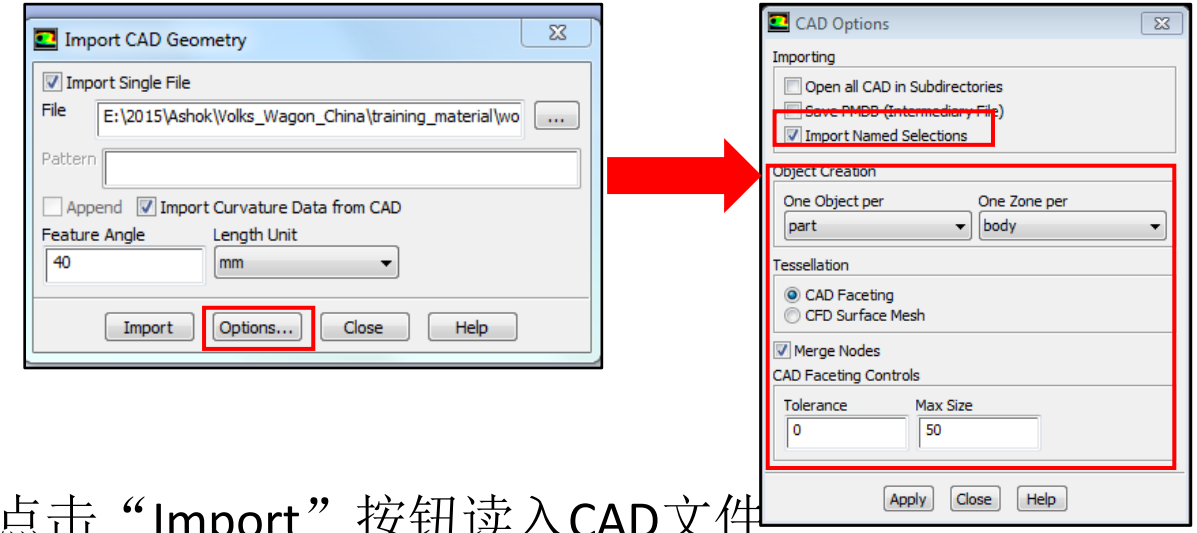
2. 选择SpaceClaim文件 “world_car_assly_final.scdoc”



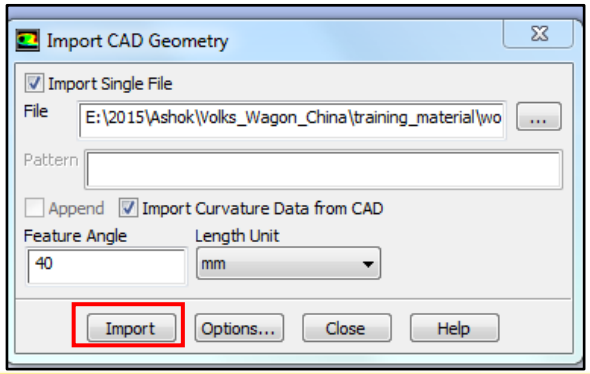
ANSYS高级整车级前处理建模工作流程

输入CAD

3. 选择 “Options…” 在 “Import CAD Geometry”



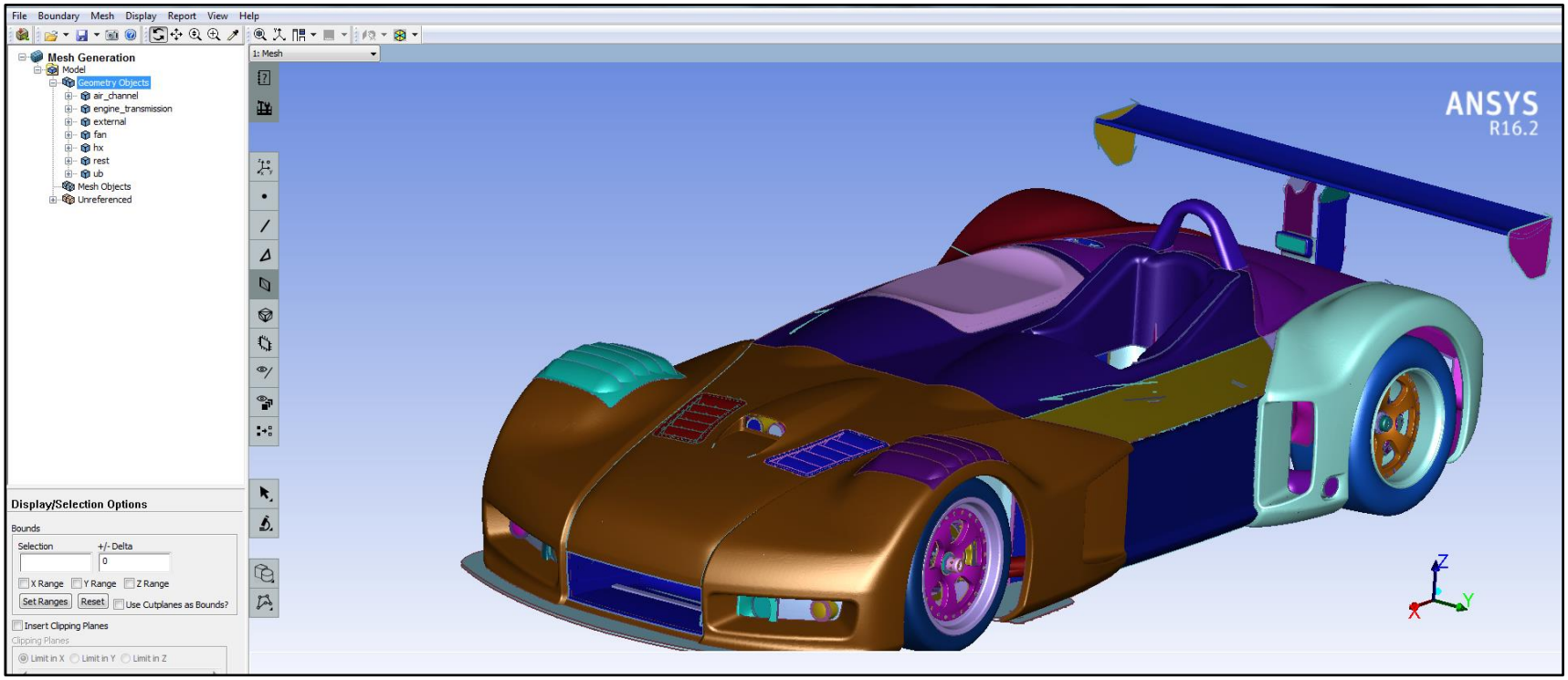
4. 点击 “Import” 按钮读入CAD文件



ANSYS高级整车级前处理建模工作流程

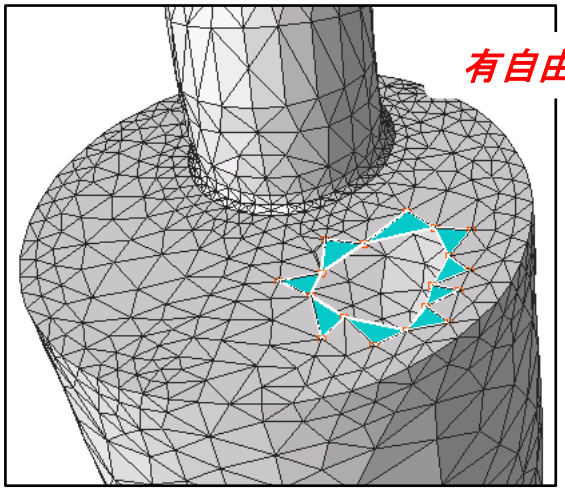
输入CAD

- 几何输入Fluent Meshing后需要进行漏洞修补



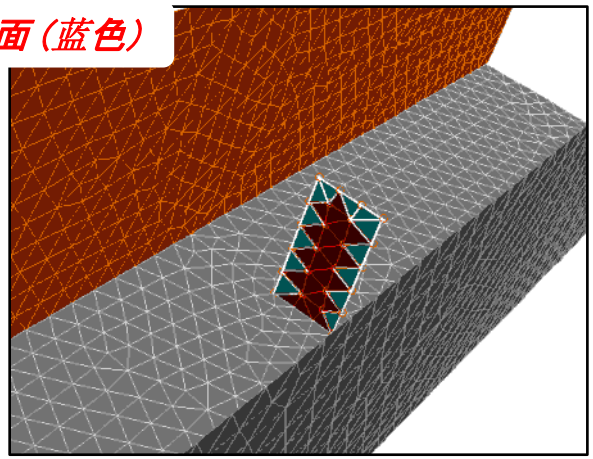
ANSYS高级整车级前处理建模工作流程

- 采用诊断工具缝补孔洞
  - 无泄漏边界
    - ✓ 边是由两个三角形或四边形连接
  - 自由边是网格面上的网格边
    - ✓ 网格上的孔洞
    - ✓ 零厚度壁面（baffle or fin）
    - ✓ 自由点是自由边上的点



由于网格中存在孔洞引起自由边
（需要修复）

有自由边的面（蓝色）

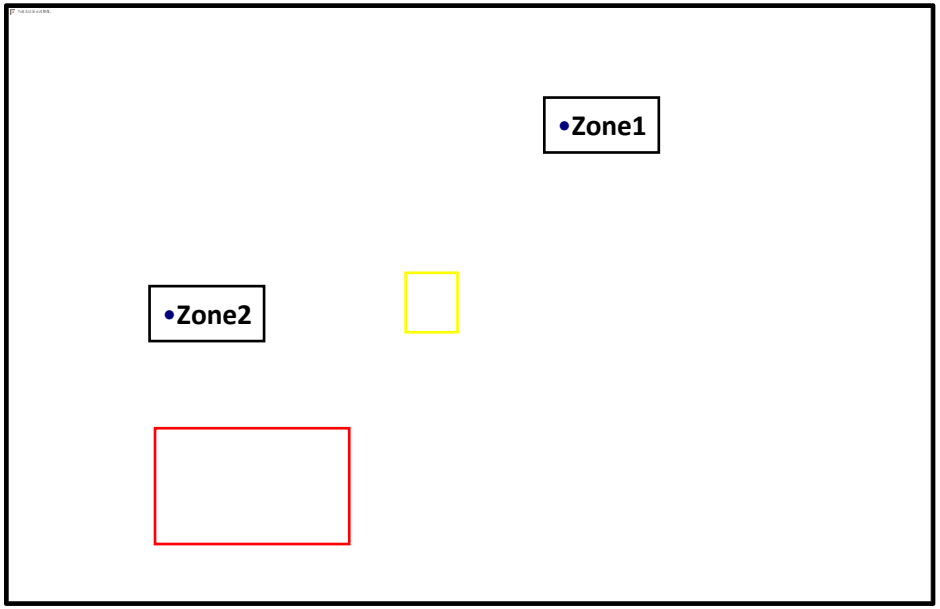


在管子翅片上存在自由边
（不需要修复）

ANSYS高级整车级前处理建模工作流程

定义材料点

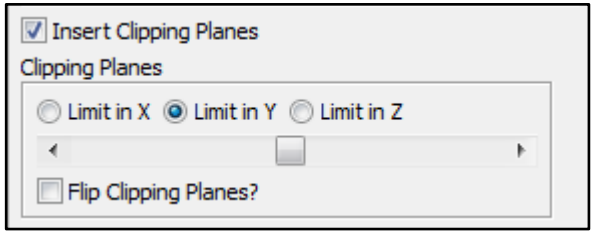
- 在封闭体内定义材料点
- 材料点可以通过指定点创建，或者通过亮点的中点创建



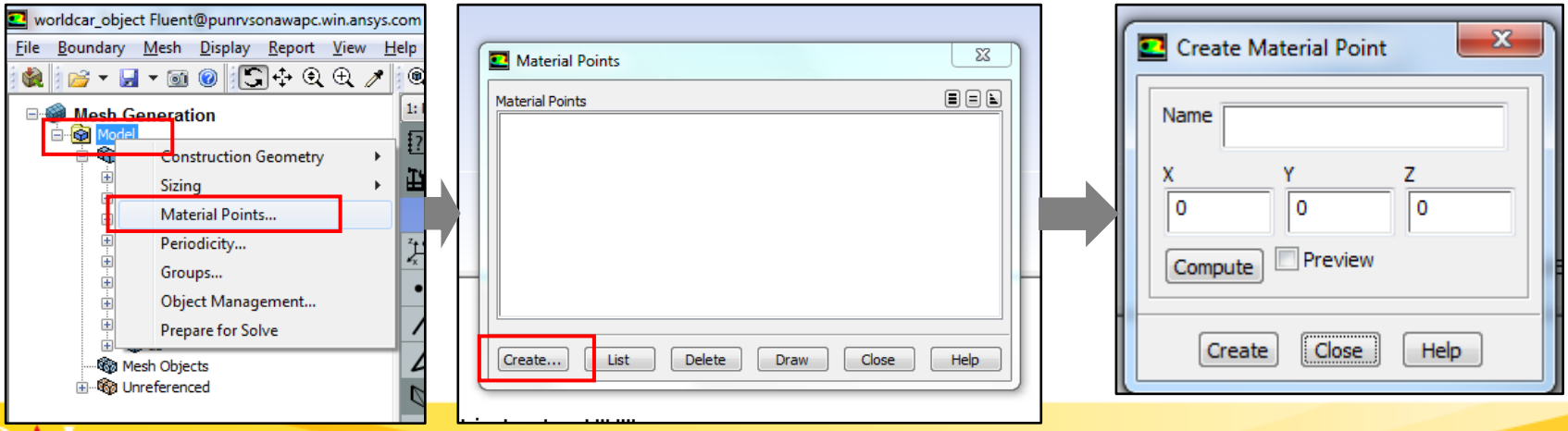
ANSYS高级整车级前处理建模工作流程

定义材料点

- 为外流场和MRF域定义材料点
- 定义“main_flow”材料点
  - ✓ 显示风洞
  - ✓ 打开“Insert Clipping Planes” 并选择“limit in Y”



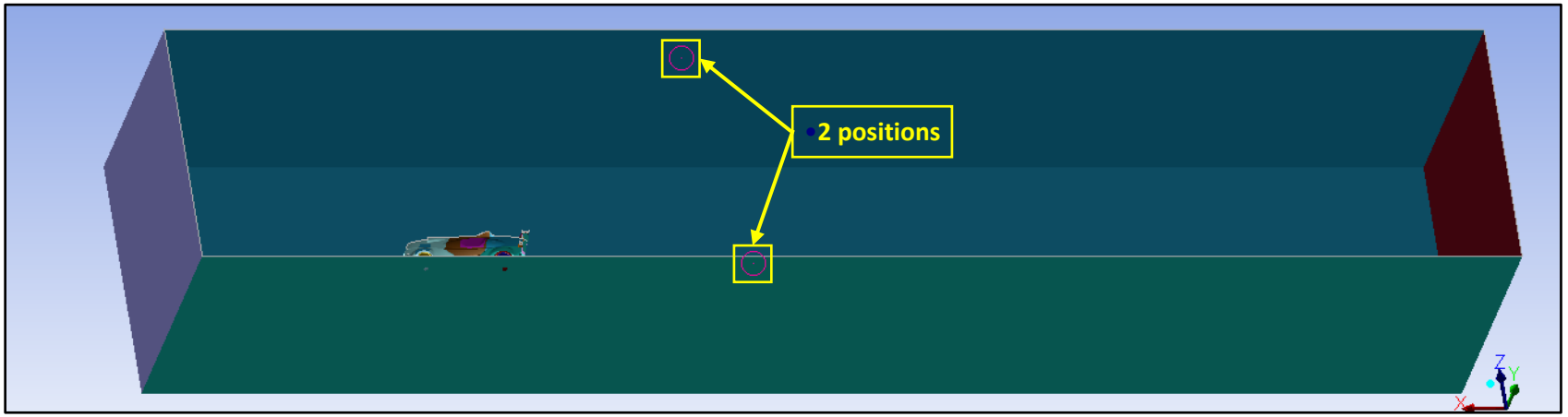
  - ✓ 右键点击Model进入材料点设置面板



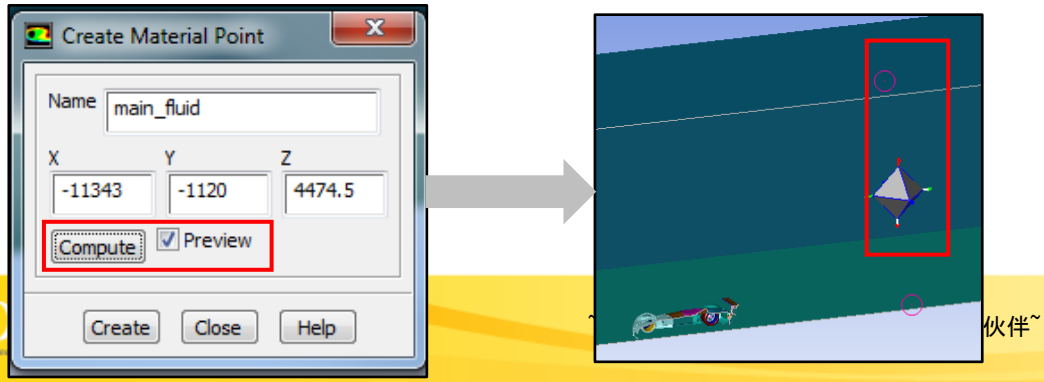
ANSYS高级整车级前处理建模工作流程

定义材料点

- ✓ 将材料点名字定义为”main_flow”
- ✓ 点击位置选择过滤器
- ✓ 选择两个点如下图



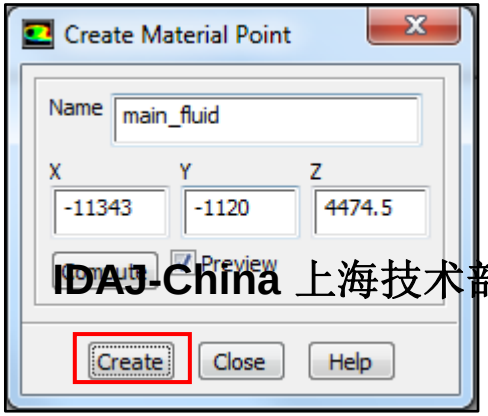
- ✓ 点击”Compute”在” Create material point “面板中，并可以预览这材料点

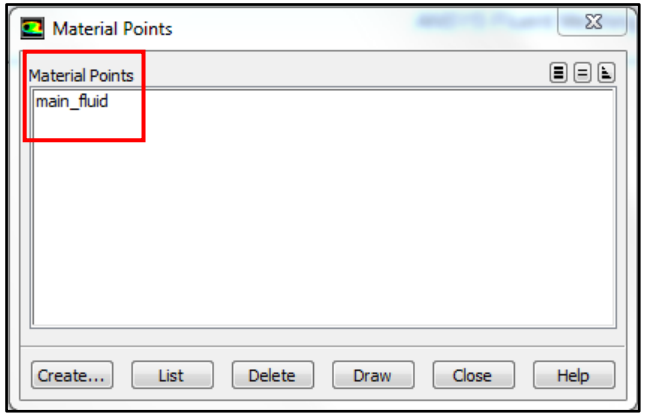


ANSYS高级整车级前处理建模工作流程

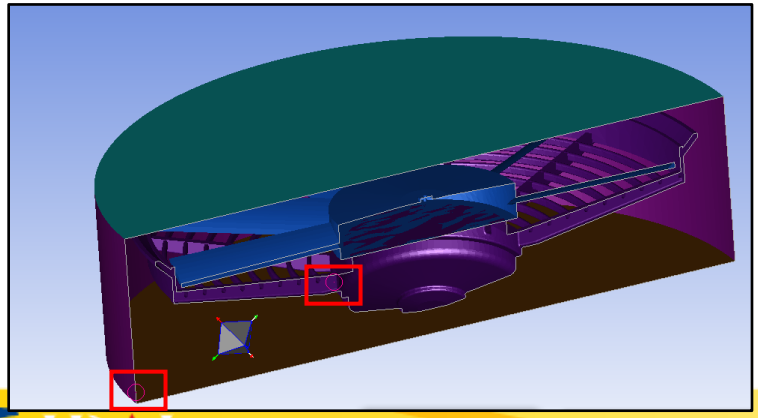
定义材料点

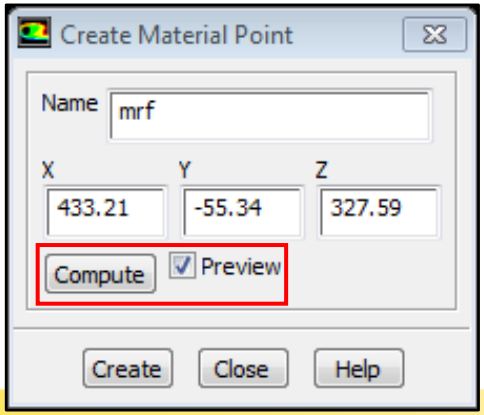
- ✓ 点击”creat”按钮在” Create Material Point”面板中，将创建”main_flow”材料点

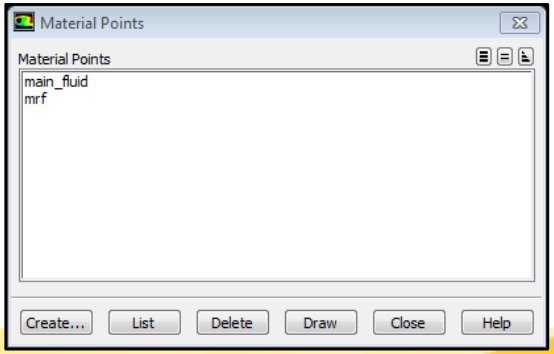




- ✓ 同理为MRF域创建”MRF”材料点

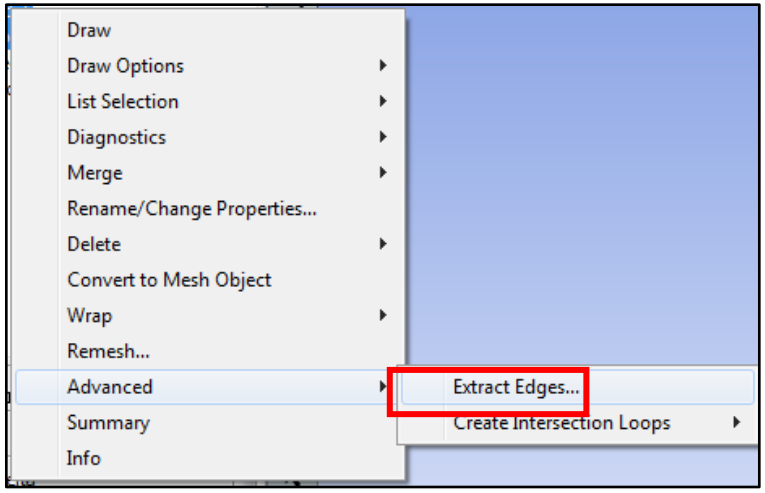






ANSYS高级整车级前处理建模工作流程

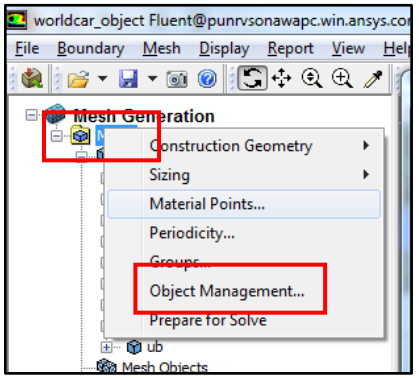
- 特征线抽取
  - 在包面过程中特征线可以达到印记目的
  - 按照角度抽取特征线
  - 对于抽取特征线有不同的方法获得



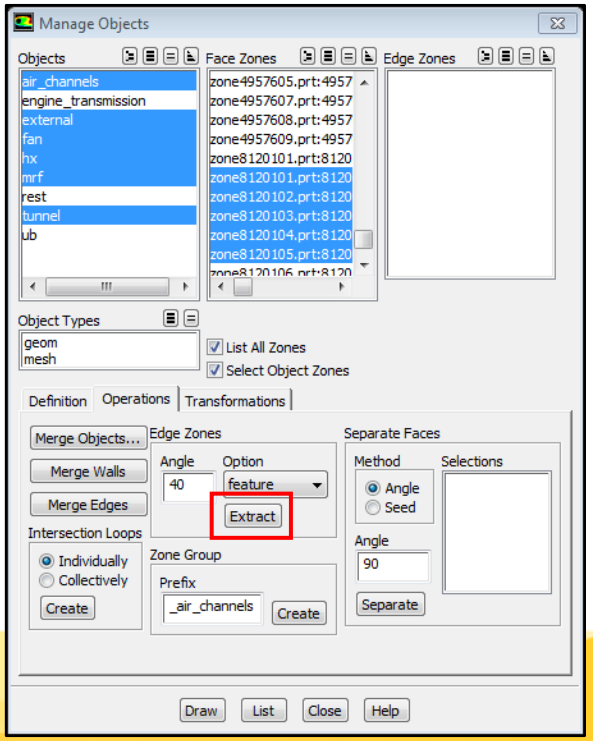
ANSYS高级整车级前处理建模工作流程

特征线创建

- 右键点击”Model”，进入”Object Management”面板

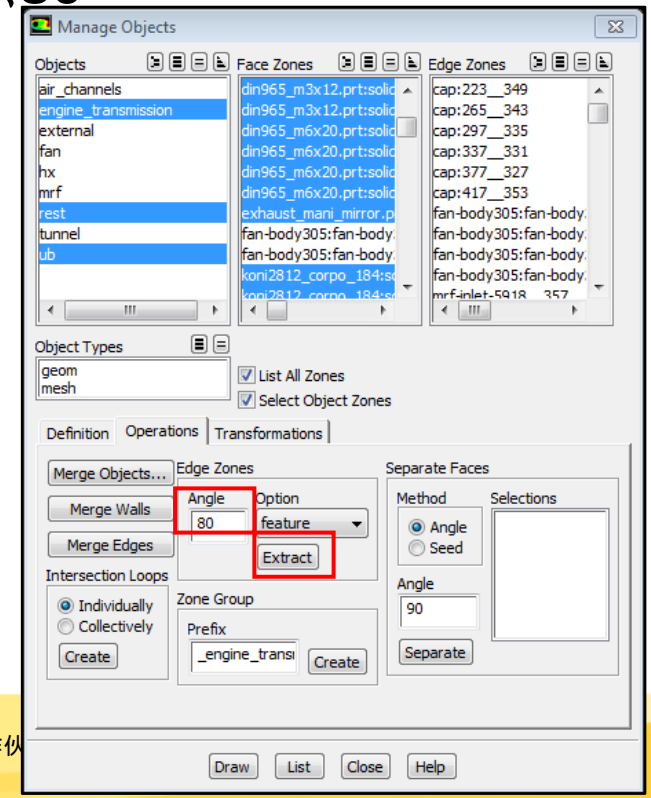


- 选择 air_channels, external, fan, hx, mrf, tunnel 对象并按照40° 角抽取特征线
- 注：上述对象中的全部边界在空气动力学中是非常重要的，这就是按照40° 角抽取特征线的原因



ANSYS高级整车级前处理建模工作流程

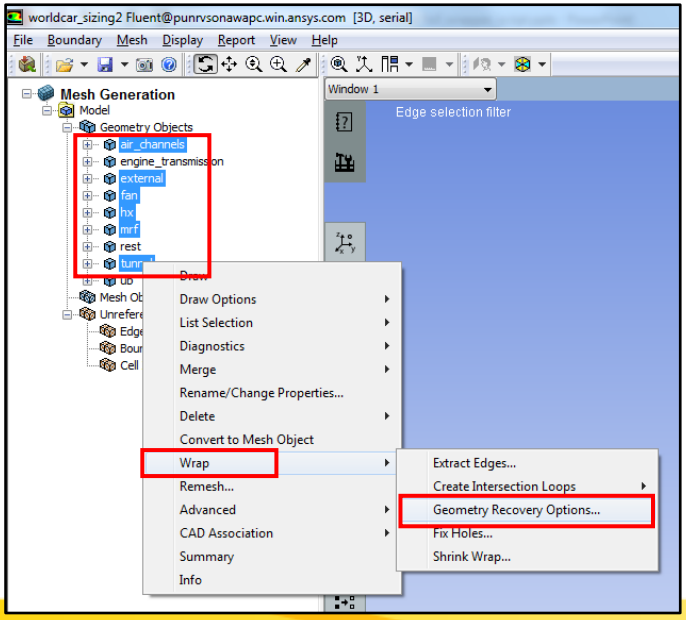
- 特征线创建
  - 其他对象按照80°角抽取特征线
  - 注：上述对象中的全部边界在空气动力学中不是很重要，这就是按照80°角抽取特征线的原因

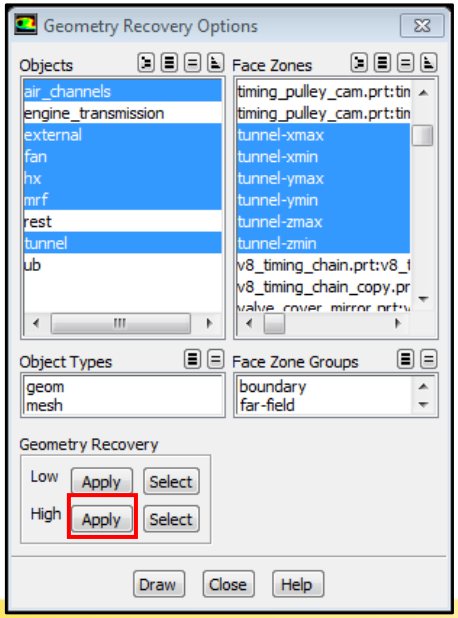


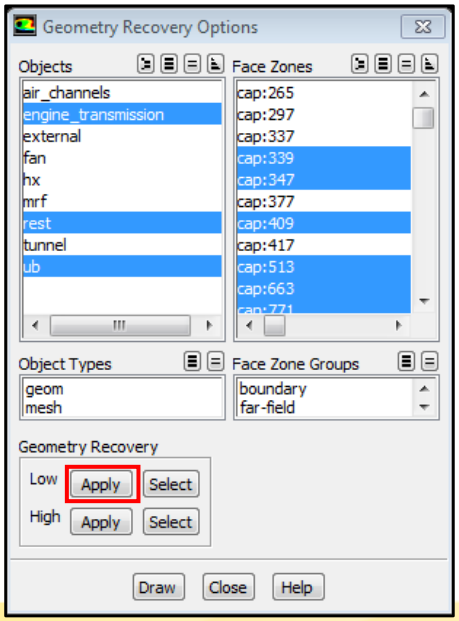
ANSYS高级整车级前处理建模工作流程

几何恢复

- 选择 air_channel, external, fan, hx, mrf and tunnel对象进入 Wrap→Geometry Recovery Options… 面板
- 点击“Apply” 在 “High”前面
- 选择engine_transmission, rest, ub从Geometry Recovery Panel面板
- 点击Apply按钮在Low前面

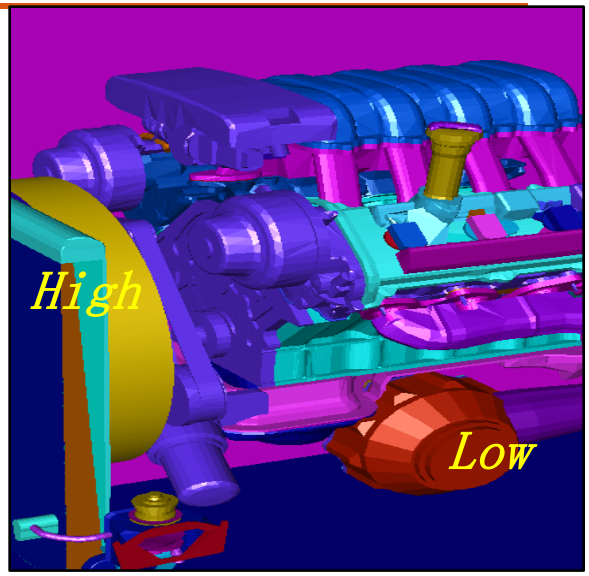


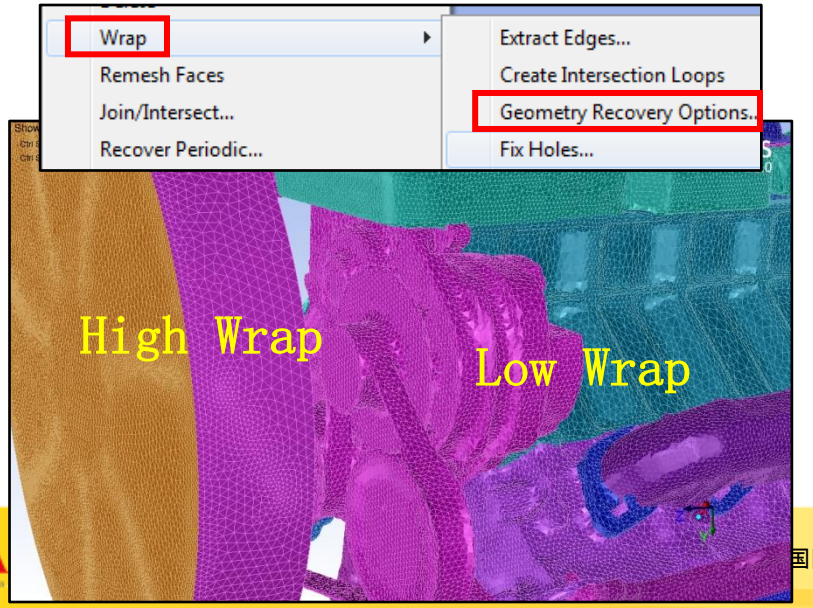


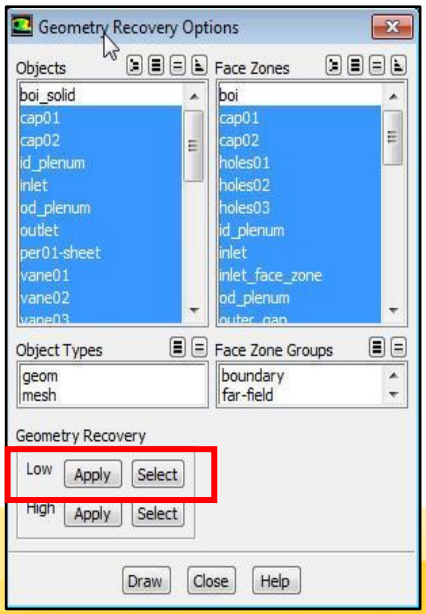


ANSYS高级整车级前处理建模工作流程

- 几何恢复
  - 混合包面
    - ✓带高和低分配的包面
      - 高：包含印记，精准的捕捉特征
      - 低：排除印记，加速周转时间和更好的初始网格质量
    - ✓默认所有对象采用高包面
      - 在树型图中使用几何恢复选项进入低包面设置





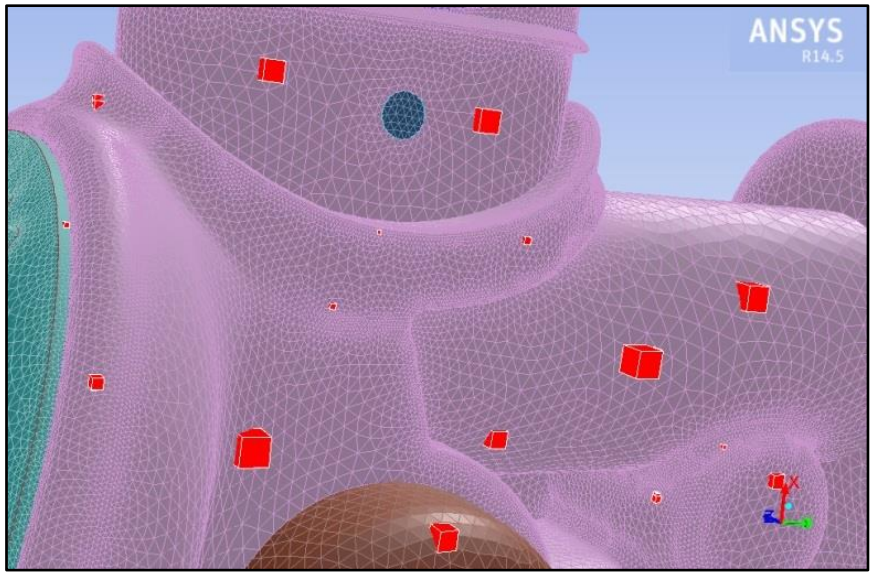


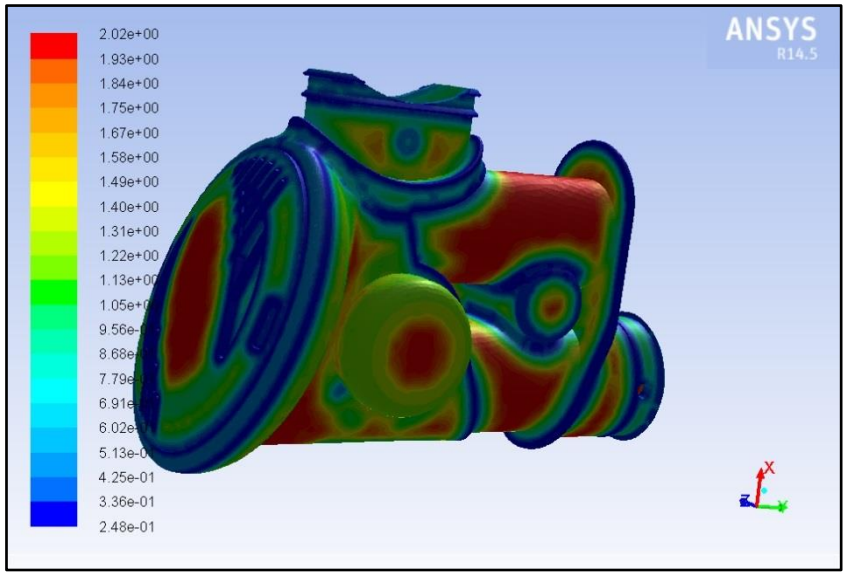
ANSYS高级整车级前处理建模工作流程

尺寸函数

尺寸函数驱动网格重构

- ✓在默认情况下，通过被计算的尺寸函数驱动一致性网格重构
- ✓指定尺寸函数控制和计算尺寸函数
- ✓通过探针或标量图查看尺寸分布



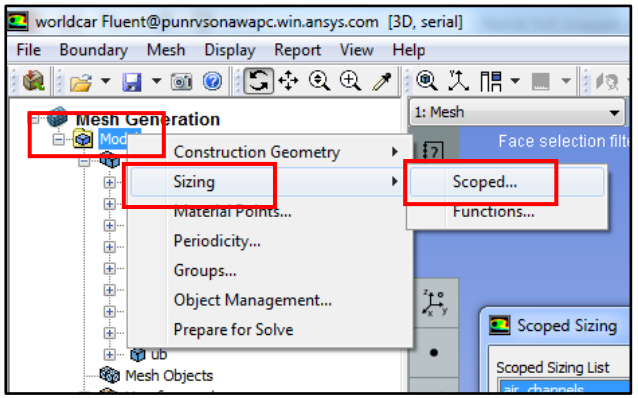


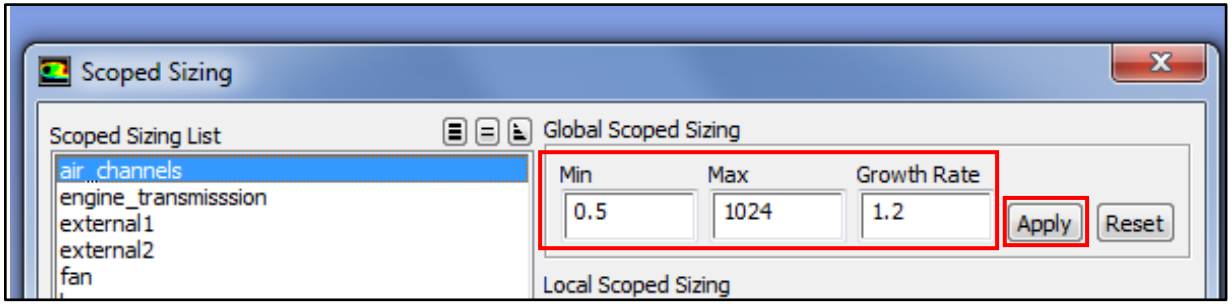
ANSYS高级整车级前处理建模工作流程

尺寸函数

尺寸函数的设置

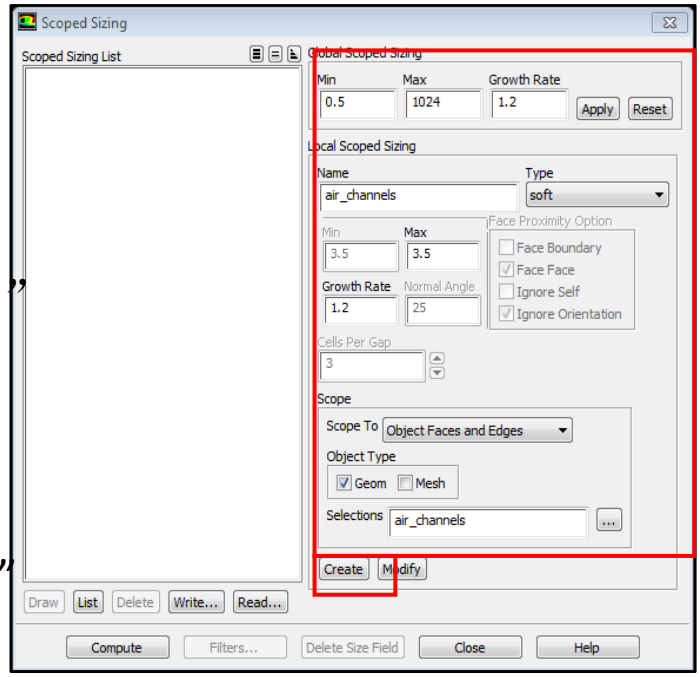
- ✓右键点击“Model”进入Sizing→Scoped
- ✓给定全局设置
  - 全局最小：0.5
  - 全局最大：1024
  - 增长率：1.2 点击Apply





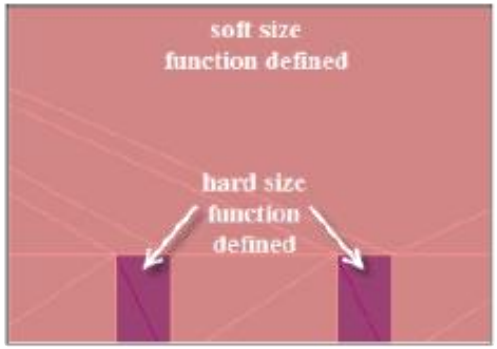
ANSYS高级整车级前处理建模工作流程

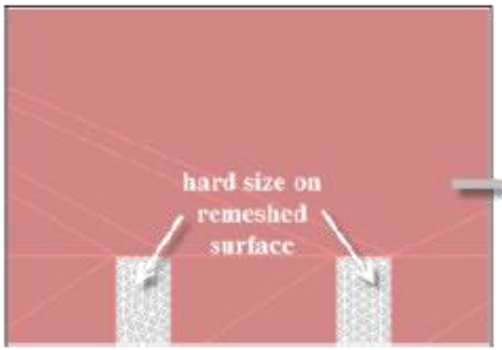
- 尺寸函数
  - 尺寸函数的设置
    - ✓风洞
      - 在局部尺寸中定义命名为“Air_channels”
      - 设置类型为“soft”
      - 设置最大尺寸为3.5
      - 范围
        - 设置范围为“Object Faces and Edges”
        - 选择对象类型为“Geom”
        - 在选择项中选择“Air_channels”对象
        - 点击“Create”

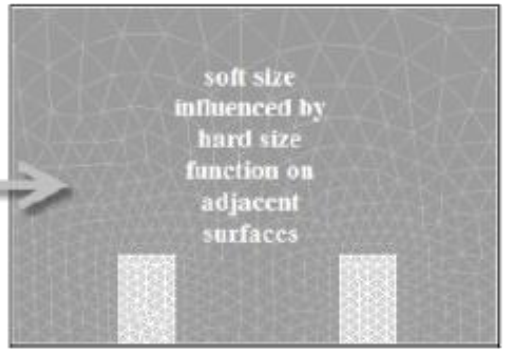


ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - Soft和Hard尺寸函数
    - ✓Hard尺寸函数设置固定尺寸
    - ✓Soft尺寸函数设置最大尺寸
    - ✓两者都可以运用在边和面上

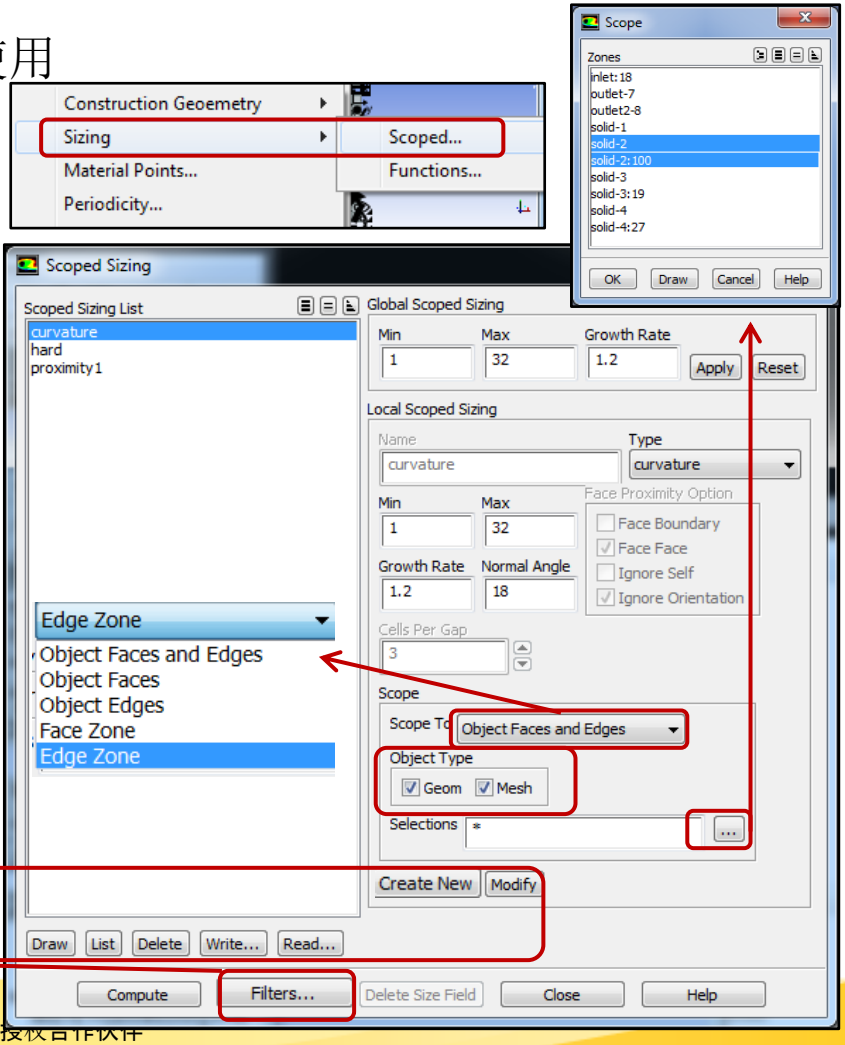






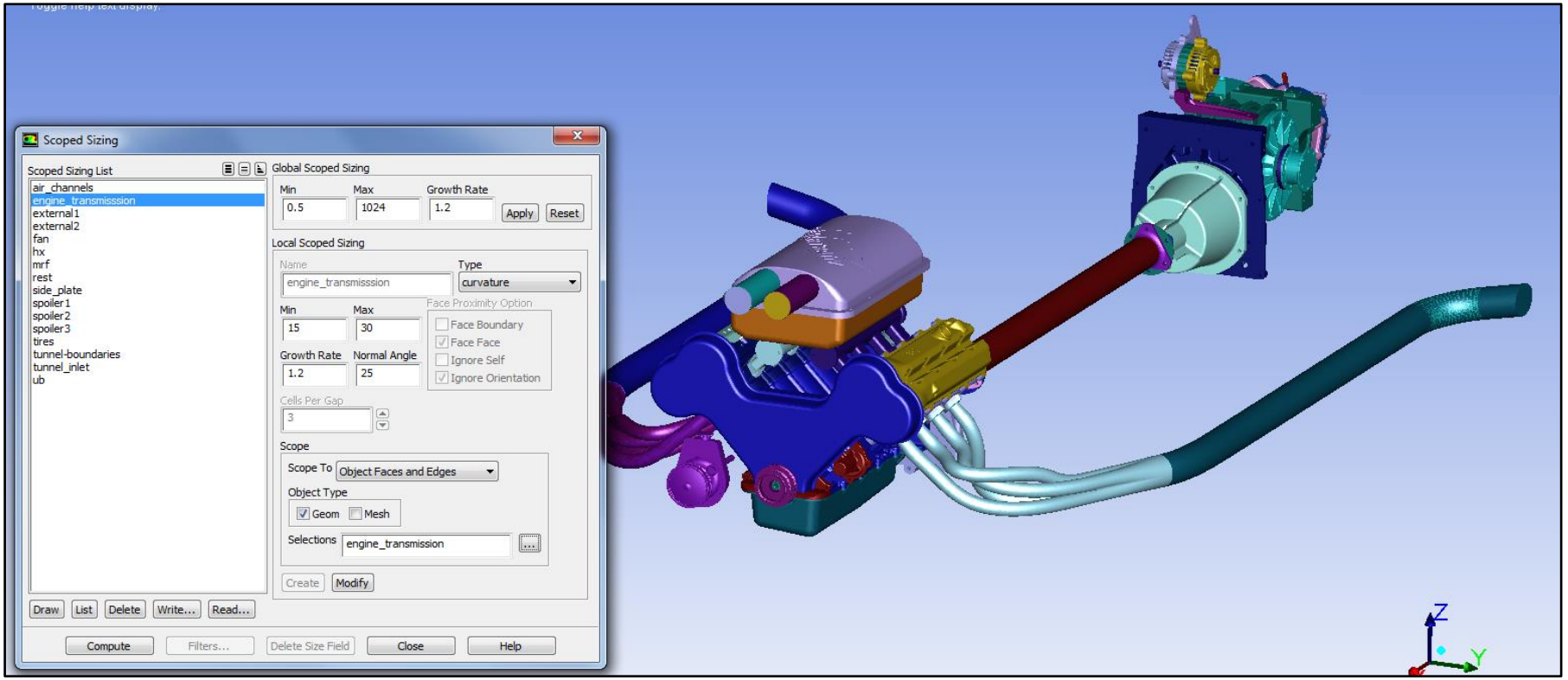
ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 采用scope sizing设置尺寸函数
    - ✓ 尺寸函数只能在对象定义的案例中使用
    - ✓ 全局和局部尺寸函数定义
    - ✓ 范围
      1. 对象或域
      2. 几何，网格及无引用类型
      3. 通配符字符串或选择
    - ✓ 可以进行创建，修改，删除
      显示，列表，写入或读入scope sizing
    - ✓ 可以计算或删除尺寸函数
    - ✓ 调用尺寸函数过滤器



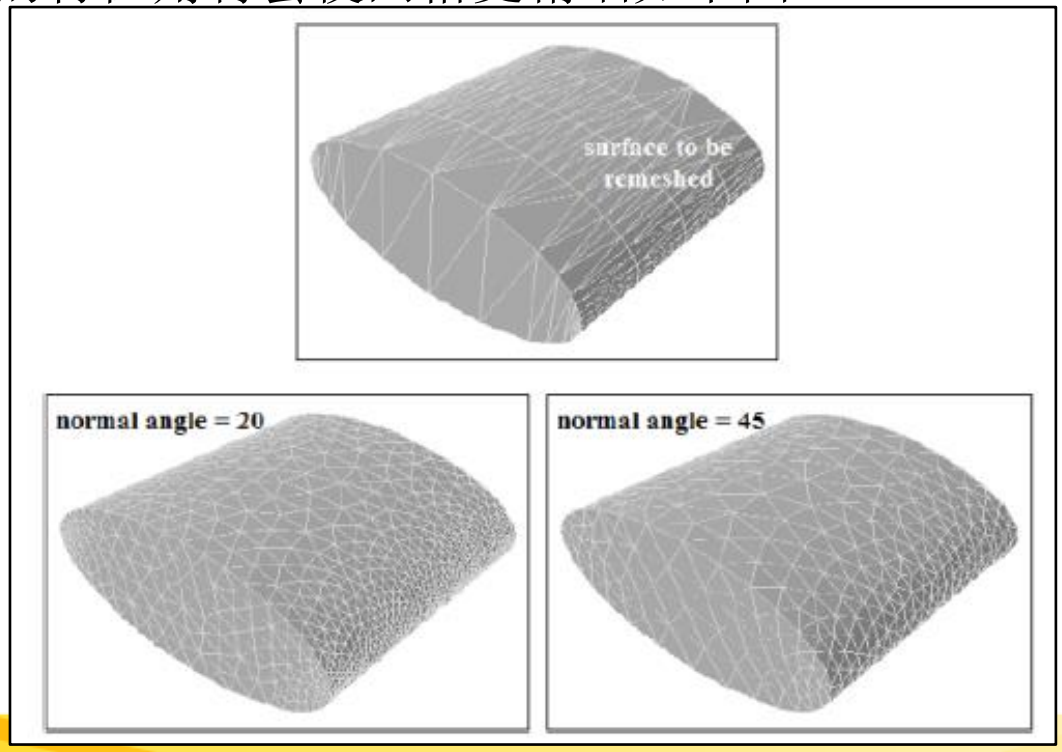
ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 发动机和变速箱



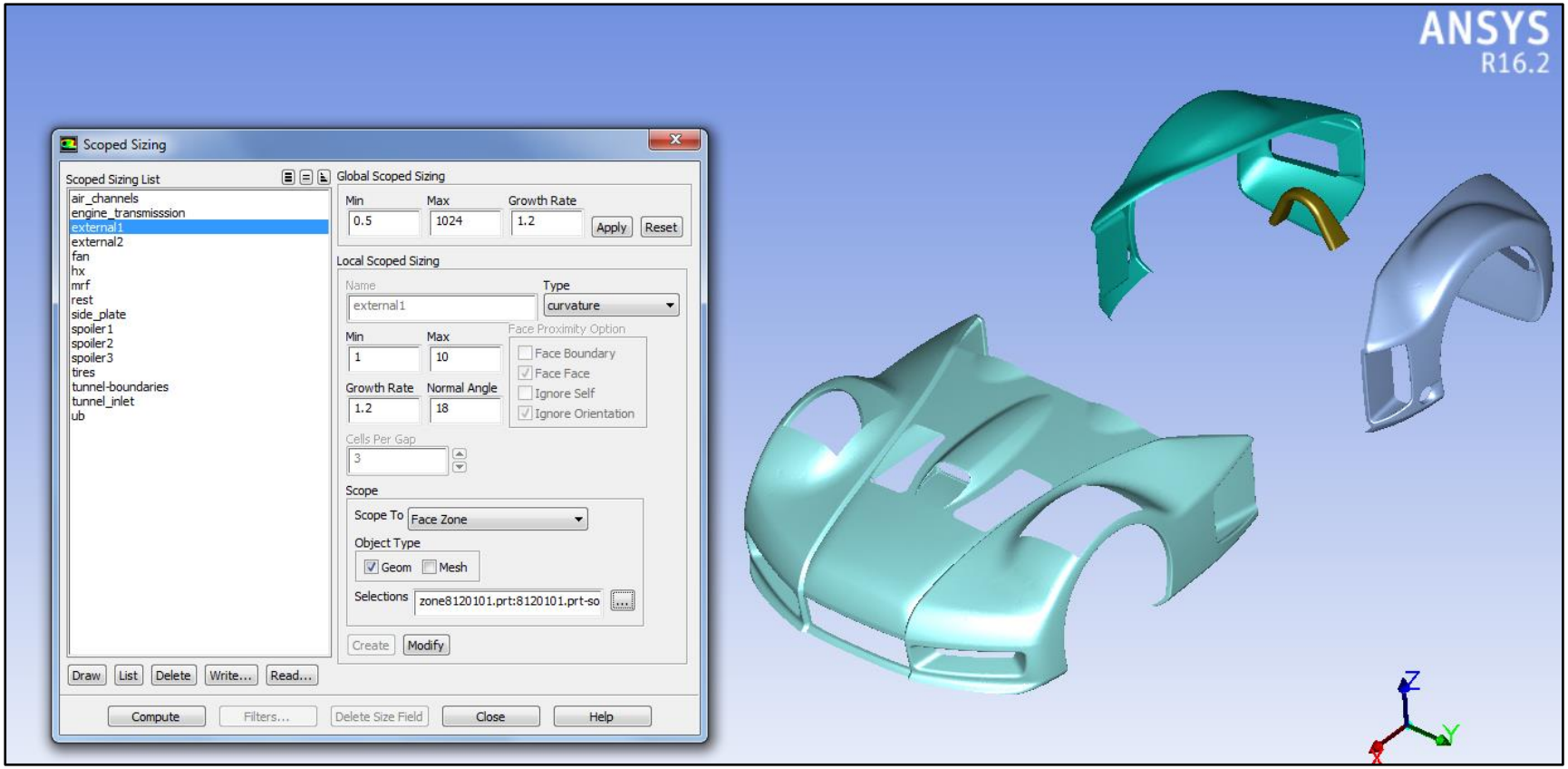
ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 曲率尺寸函数
    - ✓ 曲率尺寸函数是检查边或面上的曲率，
      它细化尺寸下限到用户定义最新尺寸来捕捉几何特征。
      较小的特征角将会使网格更精细如下图



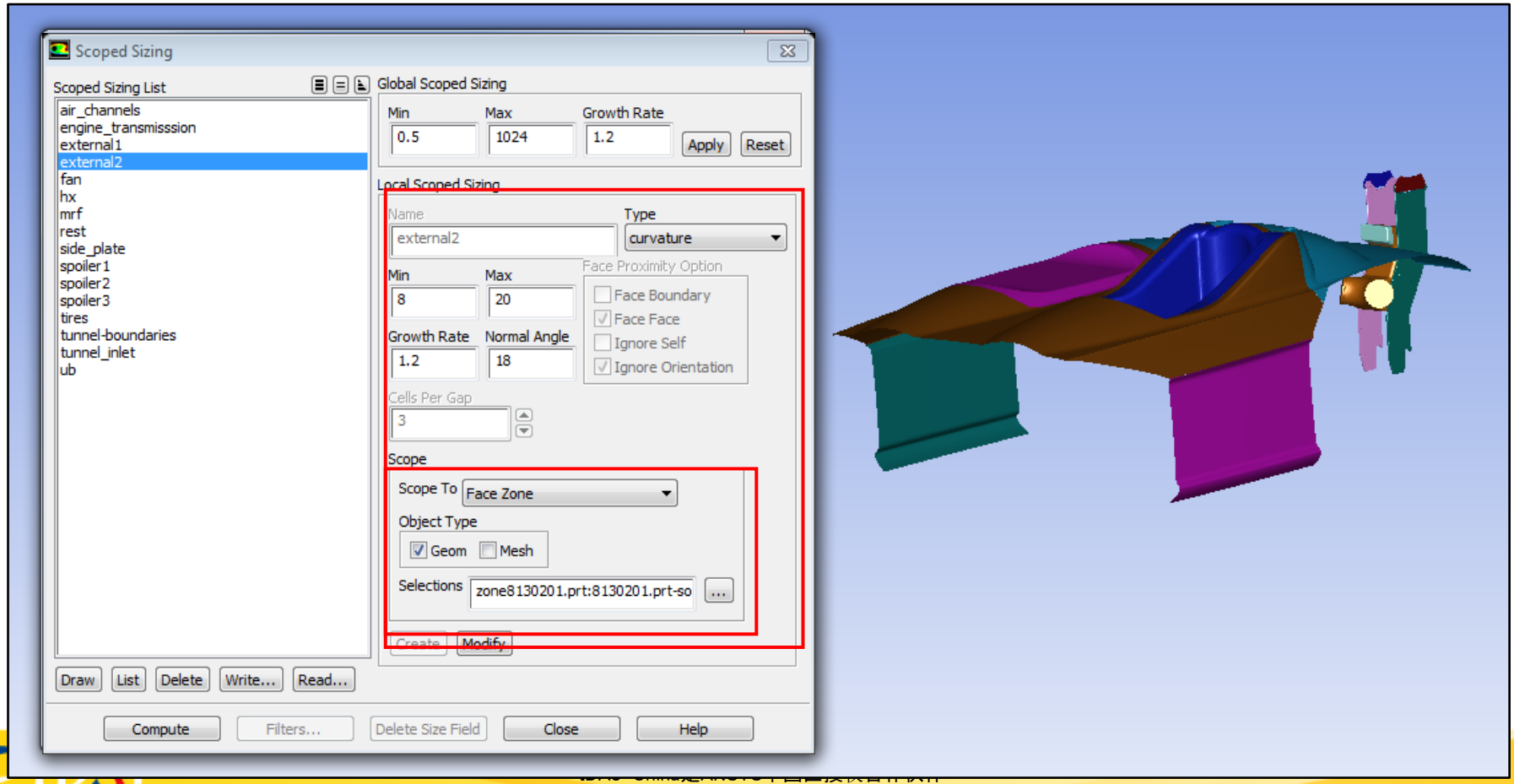
ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 外饰1



ANSYS高级整车级前处理建模工作流程

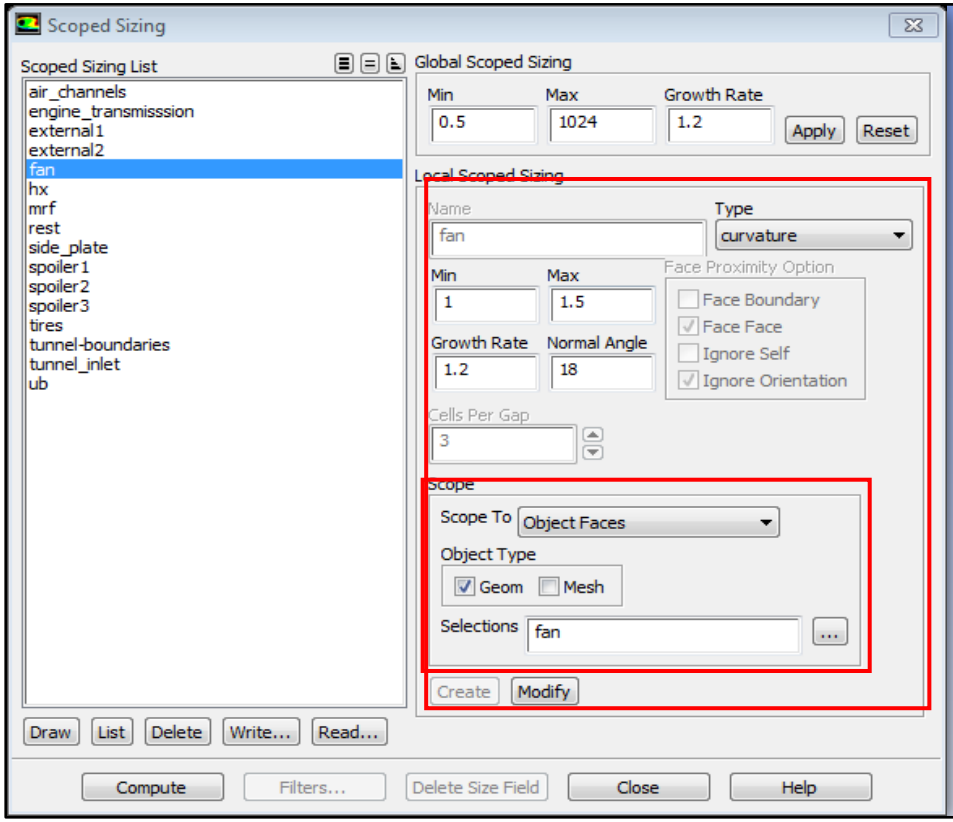
- 尺寸函数
  - 外饰2

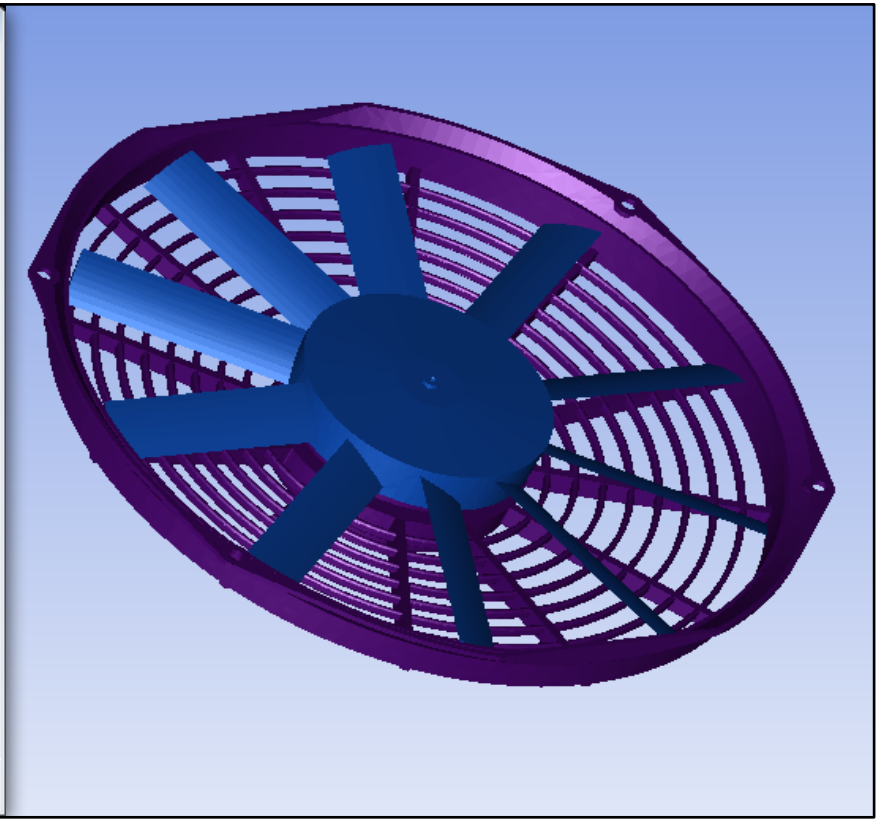


ANSYS高级整车级前处理建模工作流程

尺寸函数

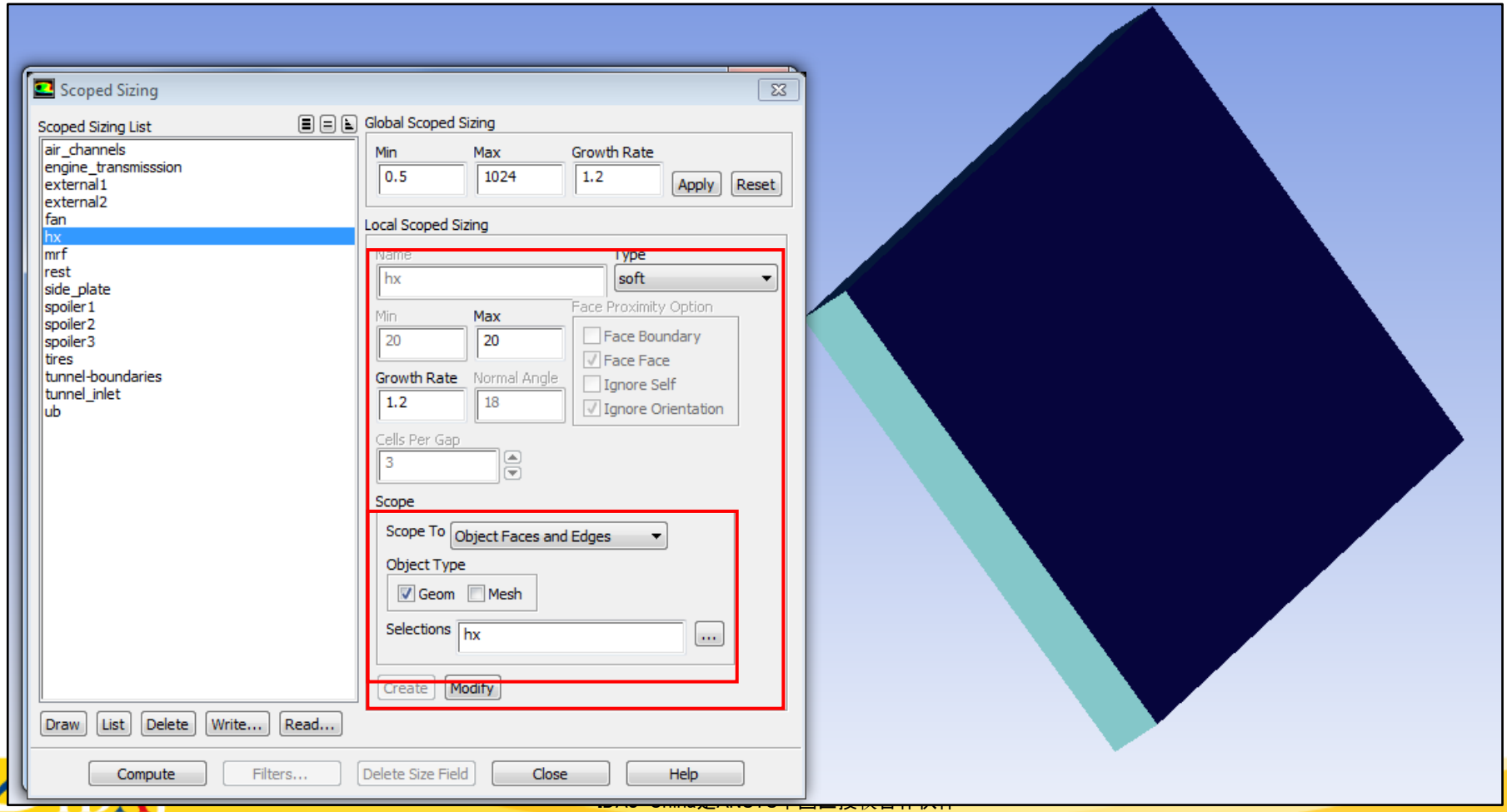
风扇





ANSYS高级整车级前处理建模工作流程

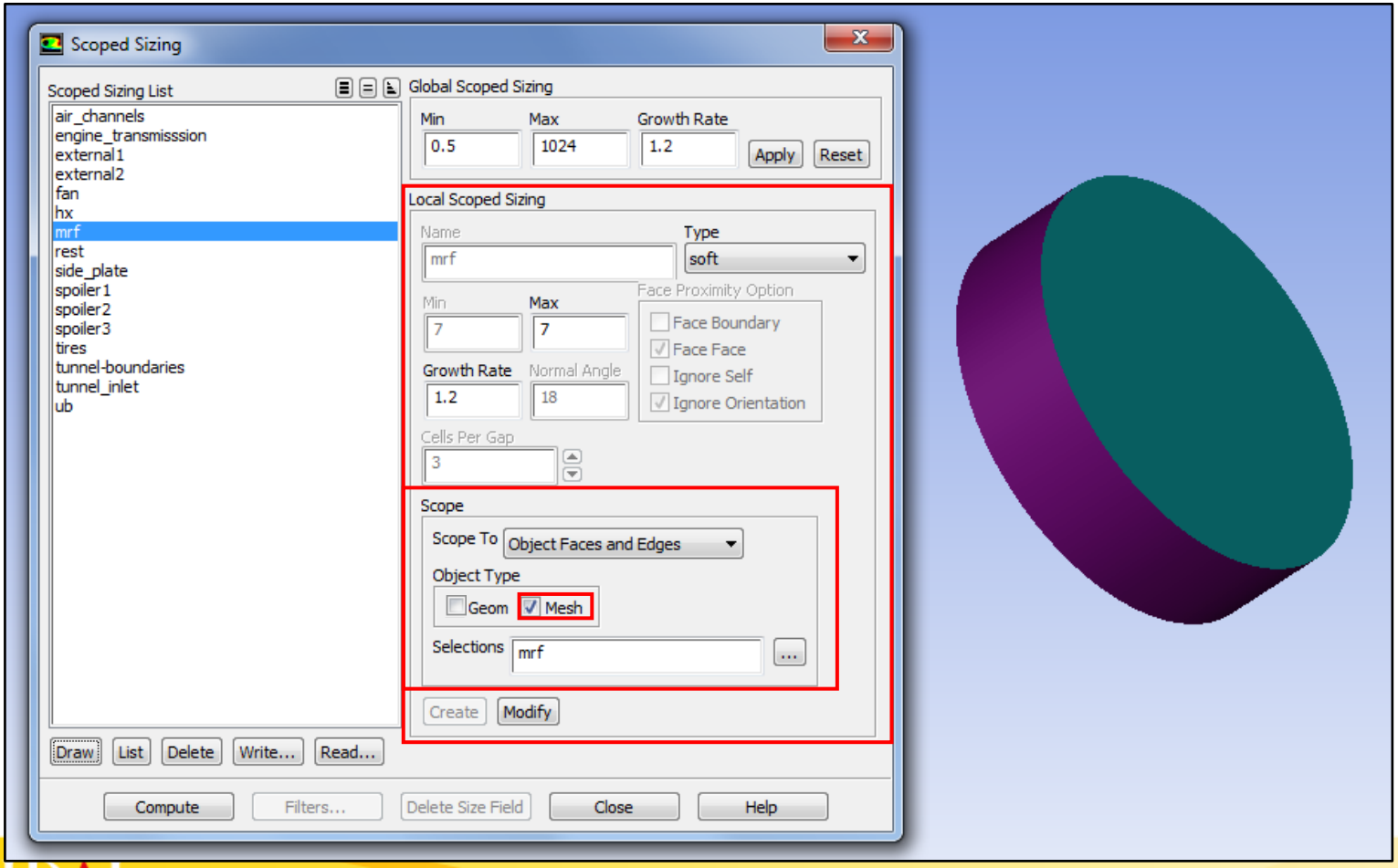
- 尺寸函数
  - 换热器



ANSYS高级整车级前处理建模工作流程

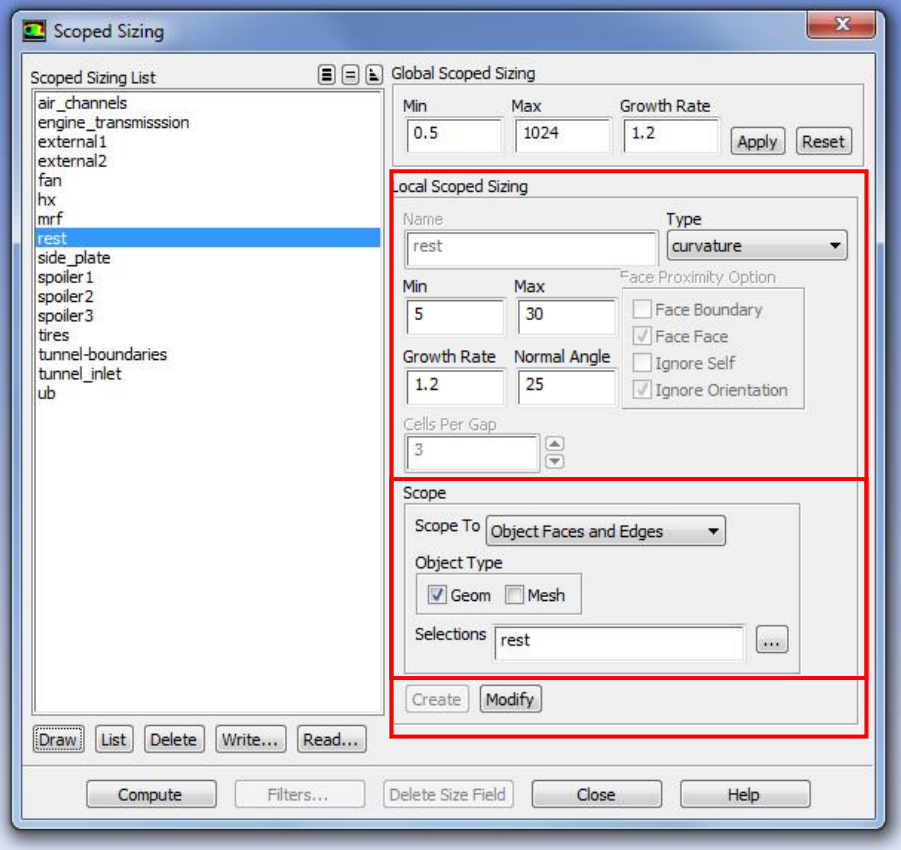
尺寸函数

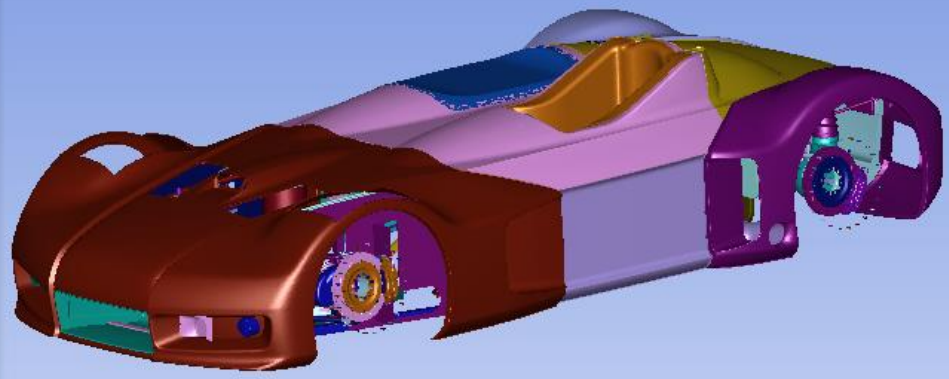
MRF



ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 其他

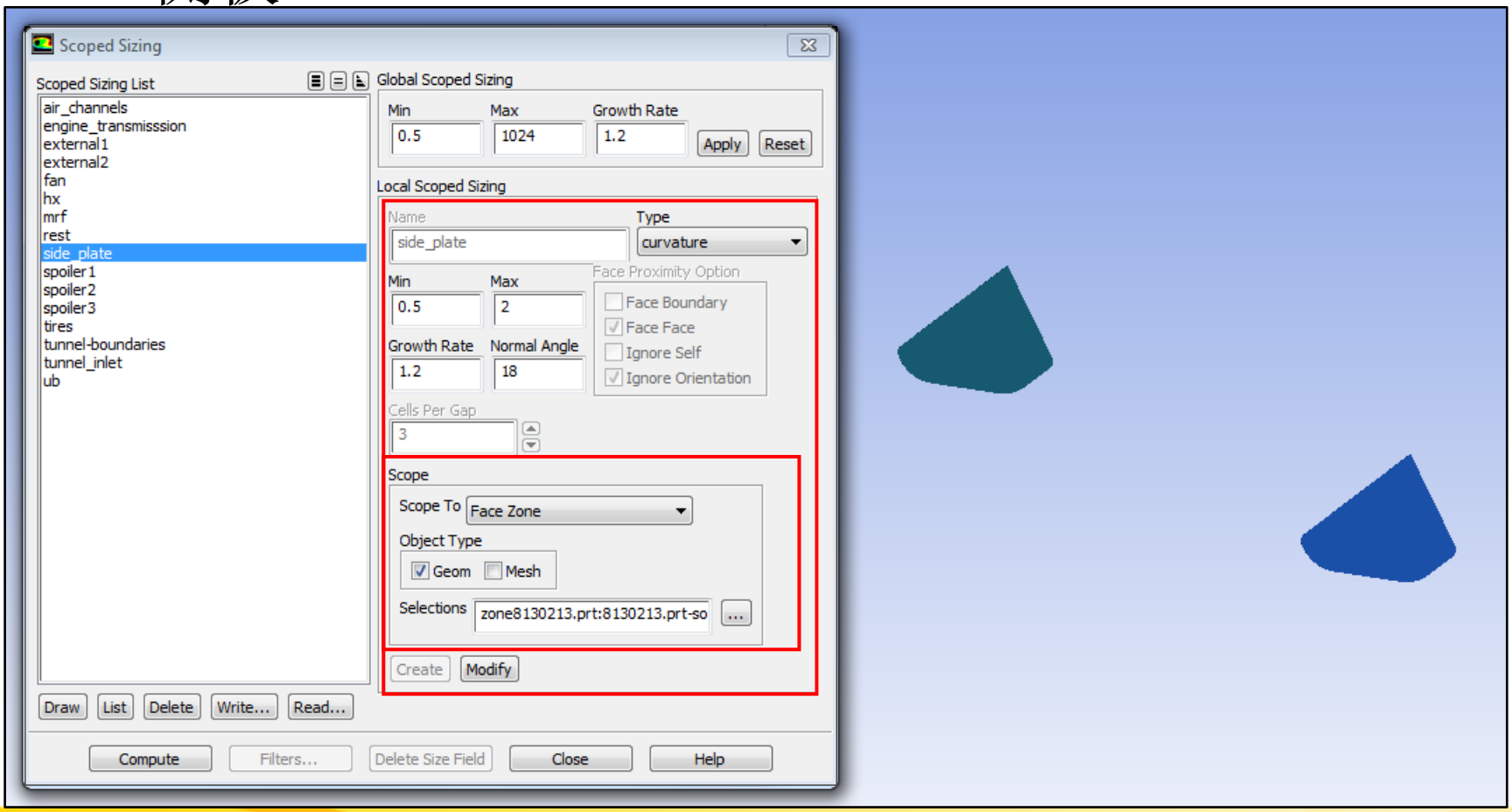




ANSYS高级整车级前处理建模工作流程

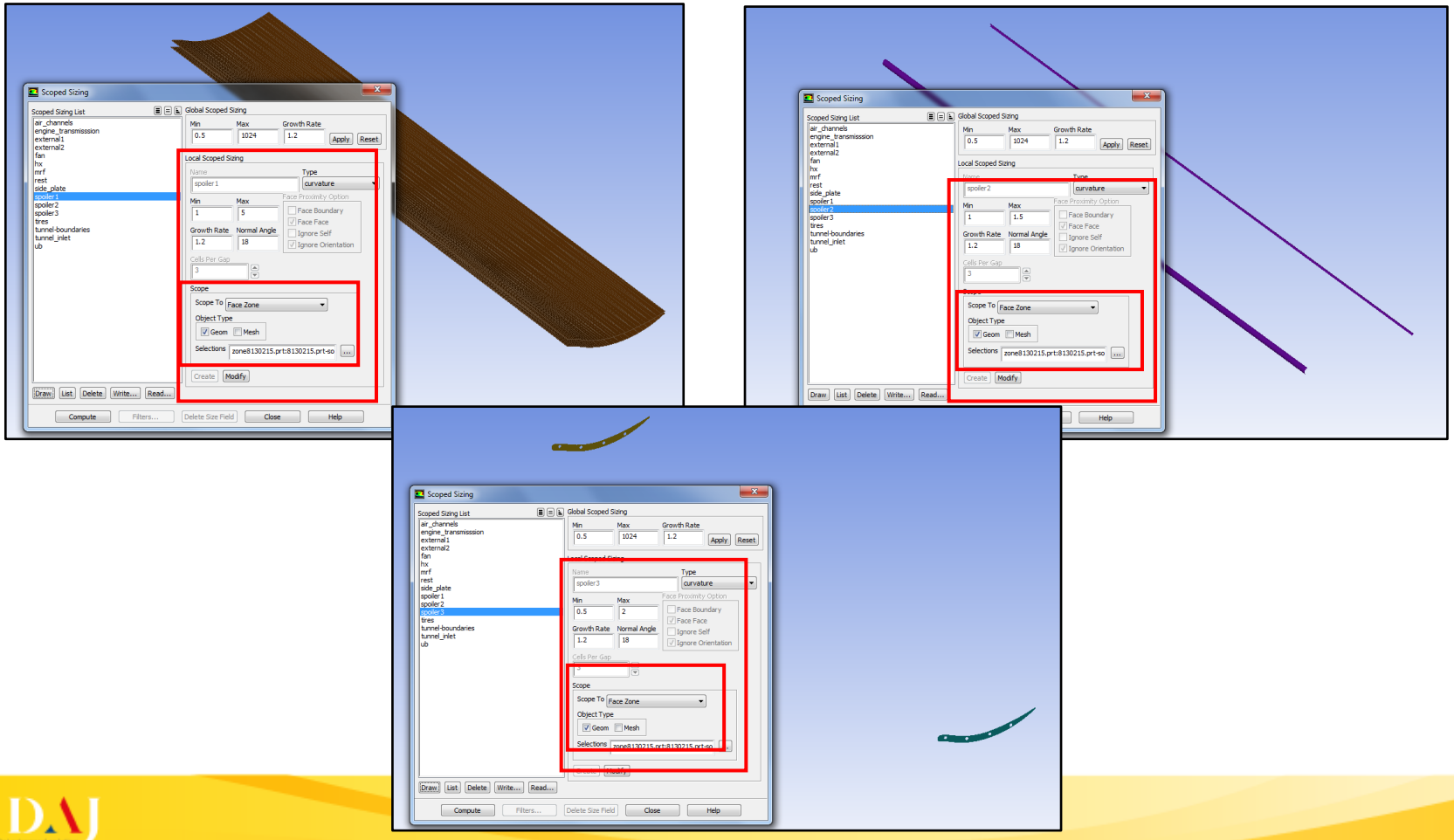
尺寸函数

侧板



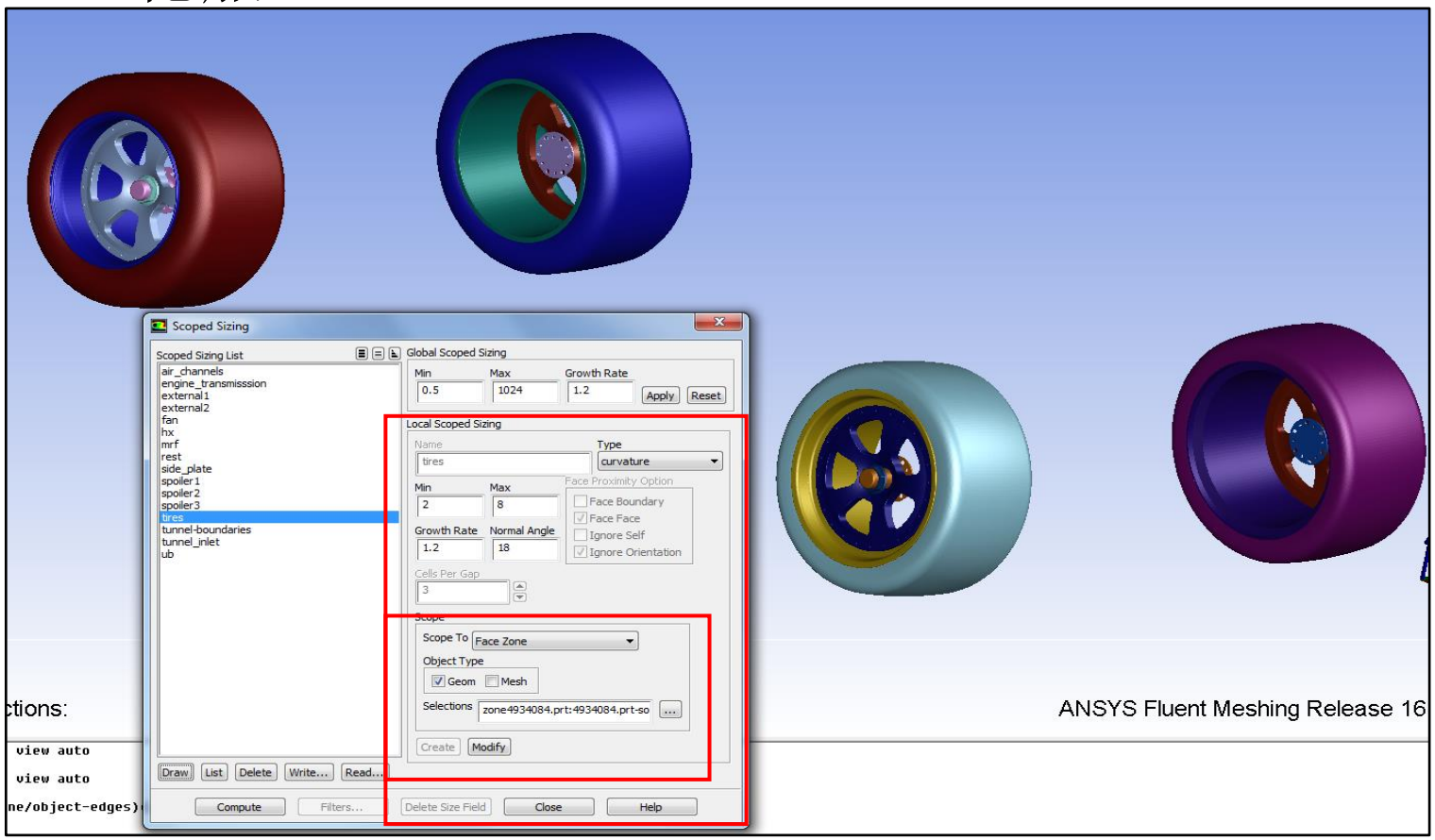
ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 扰流板



ANSYS高级整车级前处理建模工作流程

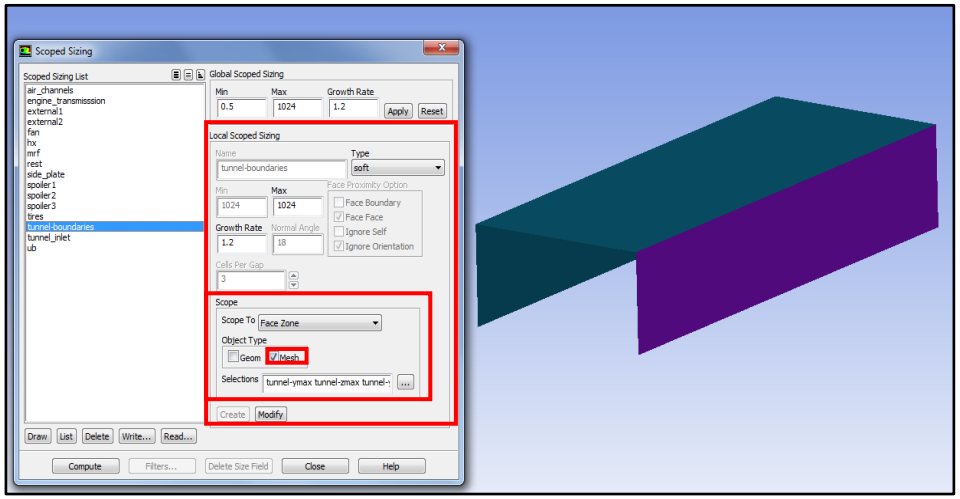
- 尺寸函数
  - 轮胎

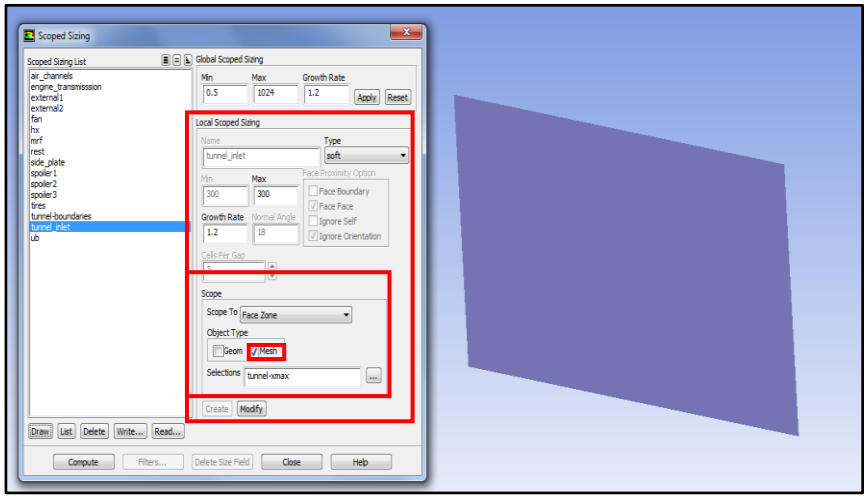


ANSYS高级整车级前处理建模工作流程

尺寸函数

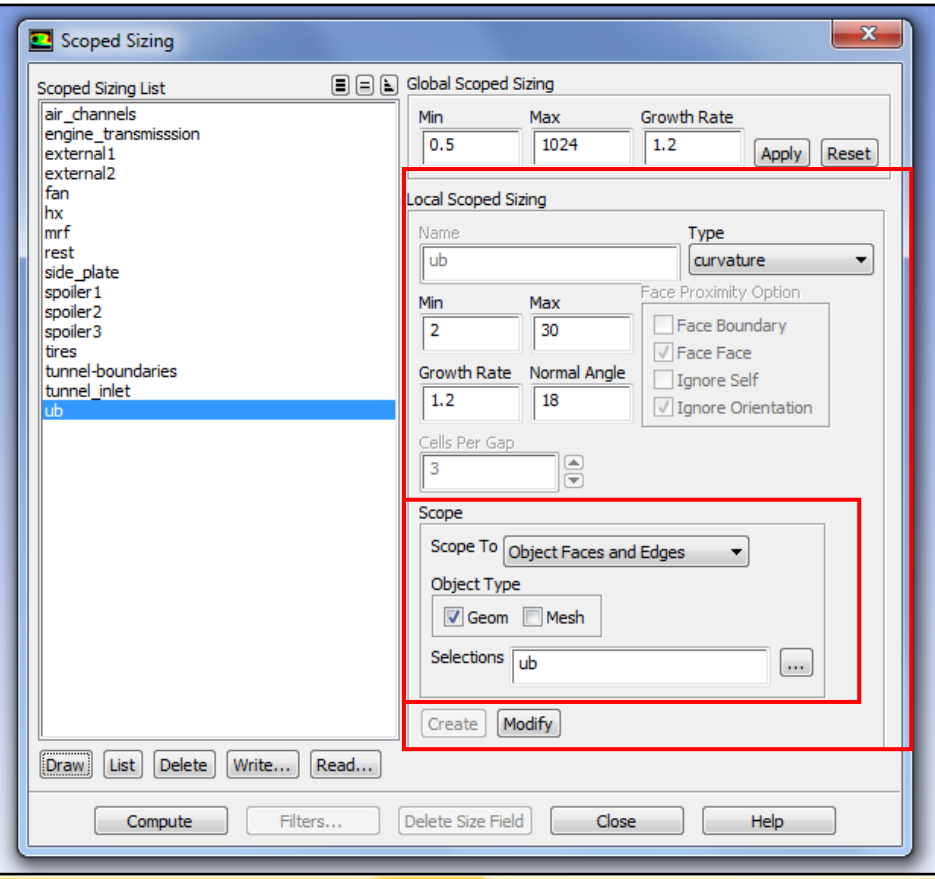
风洞

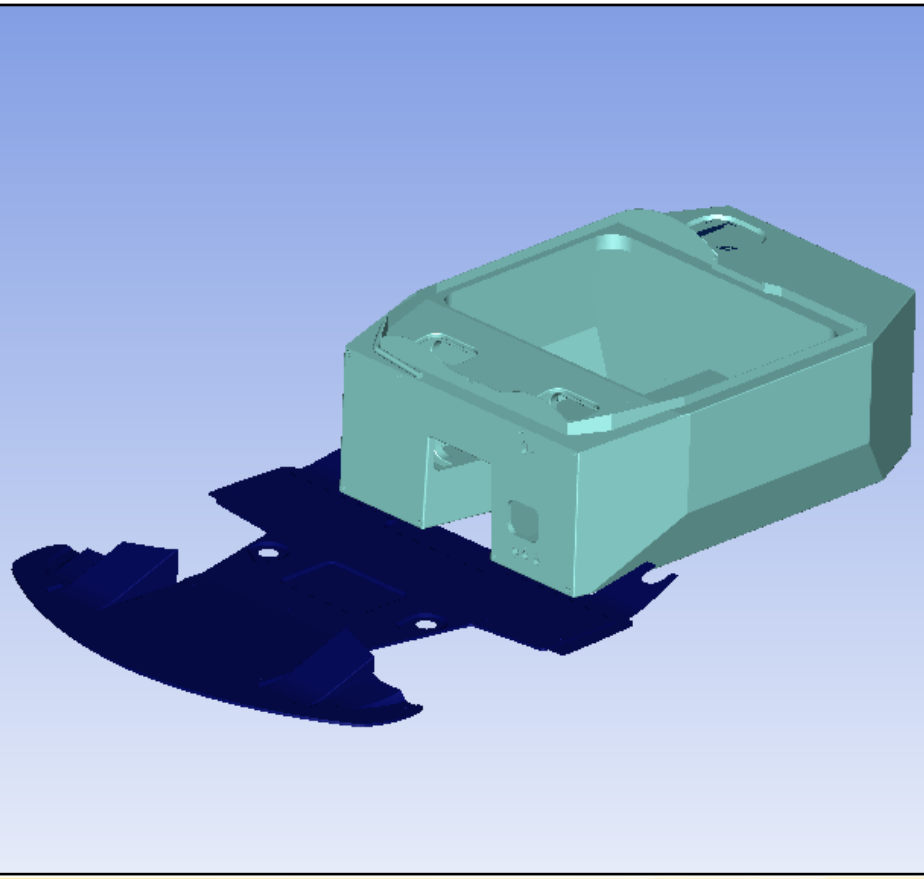




ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 下车体



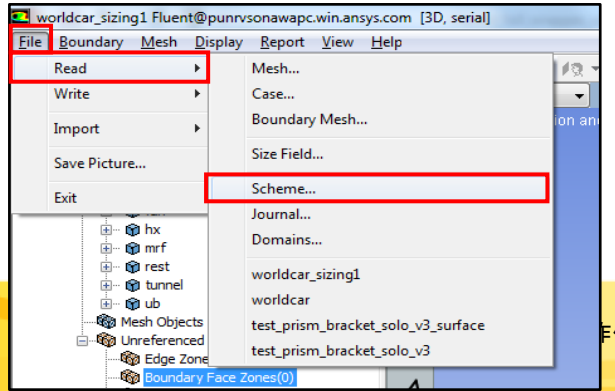


ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 将文件写为“worldcar_sizing1.msh.gz”
  - 用notepad++打开脚本“AdvWrapNPrisms_R16R17_v814_UserInputs.scm”
  - 将输入几何名字改为“worldcar_sizing1”
  - 保存scheme文件

```
;=============================================================================================================
; ================== USER INPUTS ==============================================================================
;=============================================================================================================
;
(define input_geometry "worldcar_sizing1")
;It will read <input_geometry>.msh.gz
;
(define compute_size_field "yes")
;If "no", it will read <input_geometry>-initial.sf for wrapping and <input_geometry>-target.sf for surface re
;If yes, it will compute size field using <input_geometry>-initial.szcontrol for wrapping and <input_geometry
;
(define initial_over_target_size_ratio 0)
;If initial_target_size_ratio is set as 0, it will use <input_geometry>-target.szcontrol for surface remesh.
;If initial_target_size_ratio is set as 1, it will use <input_geometry>-initial.szcontrol for surface remesh
;If initial_target_size_ratio is set greater than 1, it will use initial_target_size_ratio * <input_geometry
;
;IMPORTANT: For complex models, the initial size cannot be too different from target size -- surface remesh
;
```

  - 加载脚本文件“AdvWrapNPrisms_R16R17_v814_Main_Load.scm”到Fluent Meshing通过File→Read→Scheme



ANSYS高级整车级前处理建模工作流程

尺寸函数

用户输入scheme文件

- ✓Scheme文件是用来定义局部设置或BOI设置

```
;==========user input for refinement regions==============================
(define exclude_zones "tunnel*")
(define zones_to_offset '(
;Refinement     Zone                  BL          BL              BL            Wake Total      Wake            Cross-Wake
;Region Name    List                  Height      Divisions       Min SF        Expansion       Divisions       Total Expansion
;                                                                               Factor                          Factor
("_boi_full"    "*"                   192             1               48            2               0               1.5)

))
(define upstream_boi_min_size 24)
(define upstream_boi_levels 2)
(define upstream_boi_height_2_expansion_ratio 1)
;
(define RefinementRegionGrowthRate 1.5)
;Usage: During size controls definition, type: generate_bois
```

ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 用户输入Scheme文件修改
    - ✓ 打开AdvWrapNPrisms_Rxx_v814_AdvSettings.scm
    - ✓ 修改BOI设置

```
;==========user input for refinement regions==============================
(define exclude_zones "tunnel*")
(define zones_to_offset '(
;Refinement     Zone                BL          BL              BL              Wake Total      Wake            Cross-Wake
;Region Name    List                Height      Divisions       Min SF          Expansion       Divisions       Total Expansion
;                                                                               Factor                          Factor
("_boi_full"    "*"                 256             3               64              4               6               2.2)

))
(define upstream_boi_min_size 24)
(define upstream_boi_levels 6)
(define upstream_boi_height_2_expansion_ratio 1)
;
(define RefinementRegionGrowthRate 1.5)
;Usage: During size controls definition, type: generate_bois
```

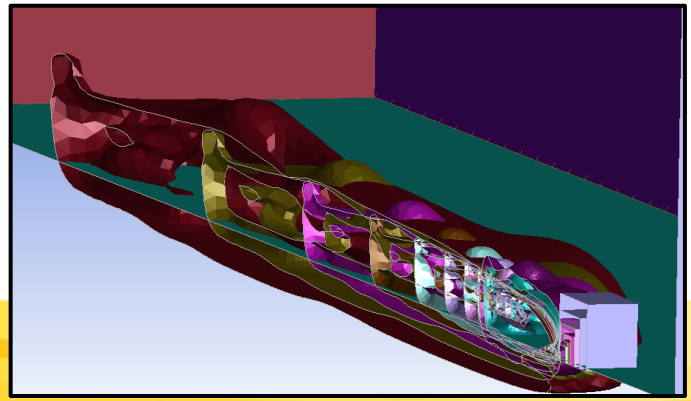
    - ✓ 保存Scheme文件

ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 尺寸范围设置
    - ✓使用TUI输入“generate_bois”，在车周围创建BOIs

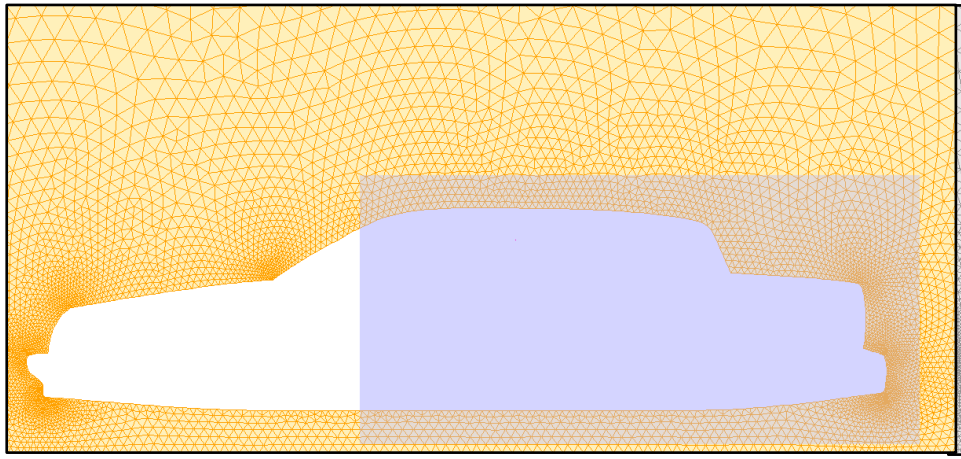
```
Loading "E:\2016\Mohib\SVW_worldcar_adv_wrapper_workshop\AdvWrapNPrisms_R16R17_v814_Main_Load.scm"
Fast-loading ".\AdvWrapNPrisms_R16R17_v814_Main.bin"
Loading ".\.\AdvWrapNPrisms_R16R17_v814_UserInputs.scm"
Done.
Loading ".\.\AdvWrapNPrisms_R16R17_v814_AdvSettings.scm"
Done.
Done.
Done.
()

Meshing> generate_bois
```

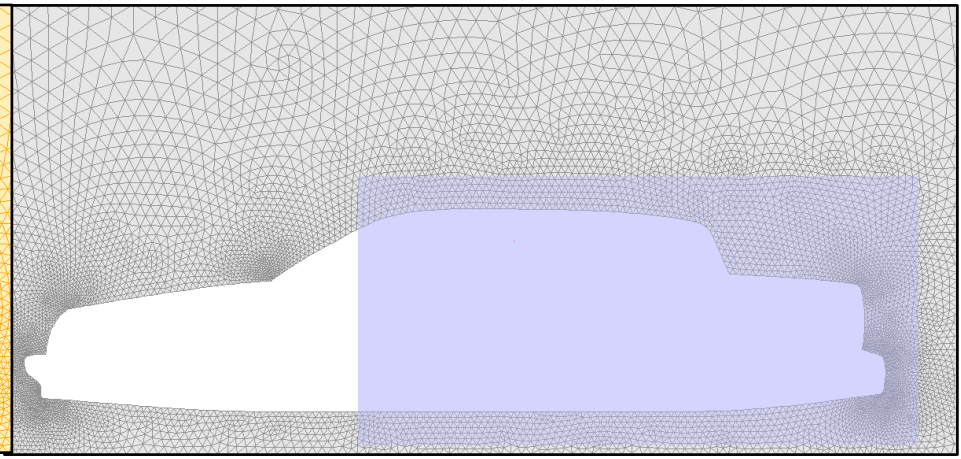
    - ✓可以看到这个域创建在“Unreferenced”
    - ✓右键点击Model进入Sizing→Scoped，BOI尺寸自动定义在sizing scope面板



ANSYS高级整车级前处理建模工作流程

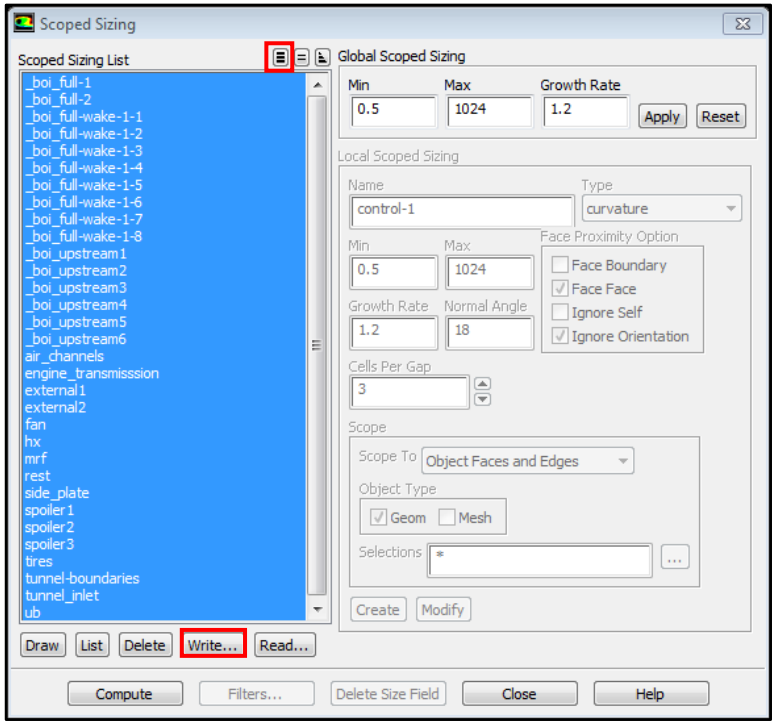
- 尺寸函数
  - BOI尺寸函数
    - ✓BOI是用来限制体域内边界网格尺寸的
    - ✓如下面例子，面网格采用BOI限制车尾部增长





ANSYS高级整车级前处理建模工作流程

- 尺寸函数
  - 尺寸范围设置
    - ✓ 我们用尺寸2控制1在包面或网格重构过程
    - ✓ 点击“Write”写入尺寸函数并命名为“worldcar_target.szcontrol”



ANSYS高级整车级前处理建模工作流程

- 脚本运行
  - 包面和边界层生成需要4个scheme文件和1个bin文件

1. AdvWrapNPrisms_Rxx_UserInputs.scm ⟷ 网格/包面设置是通过这个文件设定
2. AdvWrapNPrisms_Rxx_v814_AdvSettings.scm ⟷ 局部/先进设置是通过这个文件设定
3. AdvWrapNPrisms_Rxx_v814_Run.scm ⟷ 用来运行脚本文件
4. AdvWrapNPrisms_Rxx_v814_Main_Load.scm ⟷ Userinput和advsettings是通过这个文件加载

5. AdvWrapNPrisms_Rxx_v814_Main.bin ⟷ 包面和网格生成过程是通过这个文件来完成的

  - 上述5个文件中Userinput和Advsetting需要修改对于输入目的来说
  - 其余的3个文件不要改动

ANSYS高级整车级前处理建模工作流程

- 脚本运行
  - 从scheme文件中提供输入

```
;=============================================================================================
; ================= USER INPUTS ===============================================================
;=============================================================================================
(define input_geometry "worldcar")
;If entered <name>, or <name.msh.gz>, it will read <name.msh.gz> and zip (.msh.gz) all output mesh files.
;If entered <name.msh>, it will read .msh and write unziped (.msh) all output mesh files.
;If C: drive is full, set the system environment variable "TEMP" to point to a drive that has enough space.

;
(define compute_size_field "yes")
;If "no", it will read <input_geometry>-initial.sf for wrapping and <input_geometry>-target.sf for surface remesh
;If yes, it will compute size field using <input_geometry>-initial.szcontrol for wrapping and <input_geometry>-target.szcontrol for surface remesh.
;
(define initial_over_target_size_ratio 0)
;If initial_over_target_size_ratio is set as 0, it will compute <input_geometry>-initial.szcontrol for wrapping and <input_geometry>-target.szcontrol for surface remesh.
;If initial_over_target_size_ratio is set as 1, it will compute <input_geometry>-initial.szcontrol for wrapping and surface remesh.
;If initial_over_target_size_ratio is set greater than 1, it will compute <input_geometry>-initial.szcontrol for wrapping and use initial_over_target_size_ratio * <input_geometry>-initial.sf for surface remesh.
;
;IMPORTANT: For complex models, the initial size cannot be too different from target size -- surface remesh cannot properly "walk over" small details. Keep the factor below 2.0.
```

  - 在这里我们提供网格文件的名字，它是我们的起始文件对于运行脚本文件来说
  - 我们提供了尺寸函数输入，注意如果已经存在被计算的尺寸函数我们应该(define compute size_field “no”)，如果你想写出尺寸控制那么改变设置为“yes”
  - (define initial_over_target_size_ratio X)在这里可以定义缩放尺寸对于后处理包面和面网格重构操作。

ANSYS高级整车级前处理建模工作流程

- 脚本修改
  - 打开“AdvWrapNPrisms_R16R17_v814_UserInputs.scm”脚本

```
;=============================================================================================
; ================= USER INPUTS ===============================================================
;=============================================================================================
(define input_geometry "worldcar")
;If entered <name>, or <name.msh.gz>, it will read <name.msh.gz> and zip (.msh.gz) all output mesh files.
;If entered <name.msh>, it will read .msh and write unziped (.msh) all output mesh files.
;If C: drive is full, set the system environment variable "TEMP" to point to a drive that has enough space.

;
(define compute_size_field "yes")
;If "no", it will read <input_geometry>-initial.sf for wrapping and <input_geometry>-target.sf for surface remesh
;If yes, it will compute size field using <input_geometry>-initial.szcontrol for wrapping and <input_geometry>-target.szcontrol for surface remesh.
;
(define initial_over_target_size_ratio 0)
;If initial_over_target_size_ratio is set as 0, it will compute <input_geometry>-initial.szcontrol for wrapping and <input_geometry>-target.szcontrol for surface remesh.
;If initial_over_target_size_ratio is set as 1, it will compute <input_geometry>-initial.szcontrol for wrapping and surface remesh.
;If initial_over_target_size_ratio is set greater than 1, it will compute <input_geometry>-initial.szcontrol for wrapping and use initial_over_target_size_ratio * <input_geometry>-initial.sf for surface remesh.
;
;IMPORTANT: For complex models, the initial size cannot be too different from target size -- surface remesh cannot properly "walk over" small details. Keep the factor below 2.0.
```

  - ✓ 改变输入几何名字为“worldcar”
  - ✓ 定义计算尺寸为“yes”
  - ✓ Initial_over_target_size_ratio to “0”

ANSYS高级整车级前处理建模工作流程

脚本修改

- 在这里我们提供了对象名称，其要求在面之间有好的特征捕捉，例如external对象，fan对象，tunnel等

```
(define sharp_angle_object_list '(air_channels external fan))
;List objects that require better feature capturing between their face zones. Examples:
;1. (define sharp_angle_object_list '(aero))
;2. (define sharp_angle_object_list '(aero tunnel))
;3. (define sharp_angle_object_list '())
;4. (define sharp_angle_object_list '(*)) --> all objects
;This operation will force the sharing wrapper nodes onto the geometry.
;
```

- 在这里我们定义对象需要去特征并且也为这个对象定义尺寸

```
(define dirty_objects '())
;List dirty objects that need automatic leakage detection and closure.
;Format: (define dirty_objects '(("object" leakage_size_threshold))) --> it will close all leakage smaller than leakage_size_threshold
;Example: (define dirty_objects '(("engine" 16) ("trans" 32)))
;
;If there is not leakage, enter: (define dirty_objects '())
;
```

ANSYS高级整车级前处理建模工作流程

- 脚本修改
  - 输入需要更好捕捉特征的对象名，例如air_channel, external, fan

```
(define sharp_angle_object_list '(air_channels external fan))
;List objects that require better feature capturing between their face zones. Examples:
;1. (define sharp_angle_object_list '(aero))
;2. (define sharp_angle_object_list '(aero tunnel))
;3. (define sharp_angle_object_list '())
;4. (define sharp_angle_object_list '(*)) --> all objects
;This operation will force the sharing wrapper nodes onto the geometry.
;
```

ANSYS高级整车级前处理建模工作流程

- 用户输入脚本
  - 我们提供了在包面之前需要修补的材料点，包面是不能进入内部的

```
(define dead_regions '())
;Other way to close holes is to provide a material point inside. This is useful if there are not a single object defining that region.
;For example, there is no single region defining the cabin of a car. If you don't want any tet mesh inside the cabin, enter a material point inside.
;It will automatically detect and close holes from these material points to the main fluid region.
;Each material point must be entered in quotes like: (define dead_regions '("oil_pan" "cabin"))
;
;If there are no such dead zones and if you are not expecting any leakages, enter: (define dead_regions '())
;
```

  - 我们提供了想要包面的流体材料点

```
(define live_regions '("main_fluid" "mrf"))
;List all the material points for regions in which you want to put tet/tet+prism mesh (live)
;
```

  - 我们提供了材料点和体填充方法

```
(define volume_fill_regions '(("main_fluid" prism tet) ("mrf" no_prism tet)))
;These are the material points in which volume mesh to be generated. Examples:
;(define volume_fill_regions '(("fluid_main" prism hexcore) ("fluid_rims" no_prism tet))) ==>
;"fluid_main" will have prism layers and volume-filled with hexcore. "fluid_fan" will NOT have prism layers and will be volume-filled with tet.
;Note: avoid creating hexcore in thin regions.
```

  - 我们提供了全局边界层设置，这个适用于所有域

```
;============================global prism layer settings ============================
(define PrismLayers 2)
(define prism_layer_method "aspect_ratio")              ;enter "aspect_ratio" or "first_height_and_last_ratio" or "uniform"
(define last_ratio_percentage 40)                       ;used for "first_height_and_last_ratio"
(define prism_layer_first_height .0254)                 ;used for "first_height_and_last_ratio" and "uniform"
(define PrismAspectRatio 5)                             ;used for "aspect_ratio"
(define PrismLayerGrothRate 1.2)                            ;used for "aspect_ratio" and "uniform"
;
```

ANSYS高级整车级前处理建模工作流程

- 用户输入脚本
  - 输入我们想要包面的main_fluid流体材料点

```
(define wrap_regions '("main_fluid"))
;List all the material points for regions in which you want to put tet/tet+prism mesh (live)
;
```

  - 我们只在车表面生成边界层，在MRF里面只填充四面体单元

```
(define volume_fill_regions '(("main_fluid" prism tet) ("mrf" no_prism tet)))
;These are the material points in which volume mesh to be generated. Examples:
;(define volume_fill_regions '(("fluid_main" prism hexcore) ("fluid_rims" no_prism tet))) ==>
;"fluid_main" will have prism layers and volume-filled with hexcore. "fluid_fan" will NOT have prism layers and will be volume-filled with tet.
;Note: avoid creating hexcore in thin regions.
```

  - 我们提供的全局边界层设置，其适用于全部域

```
;============================global prism layer settings ============================
(define PrismLayers 2)
(define prism_layer_method "aspect_ratio")          ;enter "aspect_ratio" or "first_height_and_last_ratio" or "uniform"
(define last_ratio_percentage 40)                   ;used for "first_height_and_last_ratio"
(define prism_layer_first_height .0254)             ;used for "first_height_and_last_ratio" and "uniform"
(define PrismAspectRatio 5)                         ;used for "aspect_ratio"
(define PrismLayerGrothRate 1.2)                        ;used for "aspect_ratio" and "uniform"
;
```

  - 保存scheme文件

ANSYS高级整车级前处理建模工作流程

- 用户输入脚本
  - 如果你打算为CHT案例做固体网格，那么你可以给出这样的设置

```
;========================== Parallel Solid Meshing Journals and Bat file. ============================
(define solid_meshing "no")                          ;enter "yes" if solids to be meshed. They will be wrapped from outside.
(define solid_meshing_number_of_cores 4)             ;number of cores to be used for solid meshing.
(define solid_resolution_factor 1.2)                 ;size field for surface mesh of solids = solid_resolution_factor * target.sf
(define solid_meshing_max_skew 0.985)                ;target skewness for solids
(define solid_meshing_volume "yes")                  ;enter "yes" if you want volume mesh, enter "no" if you want only surface mesh.
(define solid_meshing_R16_installation_path "C:\Program Files\ANSYS Inc\v170\fluent\ntbin\win64\fluent.exe")
                                                     ;installation path for the bat file.
;Usage: Right after size function is created, a .bat file is created. Double-click on .bat file to start meshing the solids.
;============================Separate contact pairs ============================
(define fluid_fluid_angle_tolerance 5)               ;separate contact faces between fluids if angle between them is less than this value (degrees)
(define fluid_fluid_distance_scaled_tolerance 0.125)     ;separate contact faces between fluids if scaled distance between them is less than this value
(define other_angle_tolerance 20)                    ;separate contact faces between fluids and solids and between solids and solids (for mapping) if angle between them is less than this value (degrees)
(define other_distance_scaled_tolerance 0.25)        ;separate contact faces between fluids and solids and between solids and solids if scaled distance between them is less than this value
(define separate_ring_around_ff_pairs "no")          ;separating a ring of faces outside the fluid2fluid mesh interfaces and they (_ff_pairs_ring-*) will be excluded from mesh adaptation.
(define seprate_neighbors_around_ff_pairs "no")      ;renaming adjacent zones for fluid2fluid mesh interfaces and they (_ff_pairs_neighbors-*) will be excluded from mesh adaptation.
;
;Usage: (1) change zone type to non-walls (e.g. symmetry) if you don't want them paired (example: heat exchanger sides),
;(2) type: separate_contact_pairs, (3) to setup mesh interfaces in the solver load out_solver_f2f_set_nci.jou.
;Mapping (fluid:solid, and solid:solid journals and.csv files will also be created.
;
```

ANSYS高级整车级前处理建模工作流程

- 用户输入脚本
  - 这个脚本用来局部设置或BOI设置

```
;=============================================================================================================
;============================ADVANCE SETTINGS=================================================
;=============================================================================================================
;==========user input for refinement regions============================
(define exclude_zones "tunnel*")
(define zones_to_offset '(
;Refinement     Zone                    BL          BL              BL              Wake Total      Wake            Cross-Wake
;Region Name    List                    Height      Divisions       Min SF          Expansion       Divisions       Total Expansion
;                                                                                   Factor                          Factor
("_boi_full"    "*"                     256             3               64              6               8               1.5)
))
(define upstream_boi_min_size 32)
(define upstream_boi_levels 6)
(define upstream_boi_height_2_expansion_ratio 1.2)
;
(define RefinementRegionGrowthRate 1.5)
;Usage: During size controls definition, type: generate_bois
;=============end user input for refinement regions ============================
```

ANSYS高级整车级前处理建模工作流程

用户脚本设置

- 这部分脚本是用来附加包面设置

```
;============================Wrapping Settings ==========================================================================
(define exact_node_projection "no")
;only in r16
;
(define size_adaptation_factor 1)
;Initial wrapper will be coarsened by size_adaptation_factor before imprint. This will lead to faster imprint and final remesh.
;The larger this factor the more loss of features during imprint.
;
(define size_adaptation "yes")
(define skinning_type "skin")
;"shadow" or "skin"
(define size_adaptation_only_on_geom "no")
(define objects_with_gaps '())
;Enter object name, max size.
;
;Format: (define objects_with_gaps '(("object_name1" max_gap_size_to_cover1) ("object_name2" max_gap_size_to_cover2) ... ("object_namen" max_gap_size_to_covern)))
;max_gap_size_to_cover: max gap size.  This is the size at which wrapper will cover all the important gaps.
;Example: (define objects_with_gaps '(("ext" 8)))
;If plan is not to cover those gaps, enter: (define objects_with_gaps '())
(define separate_sharp_angle "no")
;set to "no" for complex geometry
(define sharp_angle_separation_angle 40)
;feature cleanup
(define size_adaptation_using_target_size "no") ; If yes, it will lead to loss of features, but is faster (~20%)
(define merge_overlapping_edges "yes")
(define feature_merge_factor 0.125)
;any features that are in proximity to each other within feature_merge_factor * local_size_field, it will be merged. After merge it leads to faster imprint and remesh.
(define coarsen_by_remesh_features_after_delete_overlapping_edges "no")
(define edge_curvature_growth_rate 2)
(define edge_curvature_rate 10)
(define merge_lows "yes")
(define high_wrap_live_regions "no")
;only used for version 15.
;
(define standard_low_wrap "no")
(define standard_inner_wrap "yes")
(define coarsening_curvature_angle 35)
;used for coarsening and remeshing of surfaces that don't have any feature edges defined.
;The smaller this number the more curvature will be captured, but with reduced quality.
;============================End Wrapping Settings ======================================================================
```

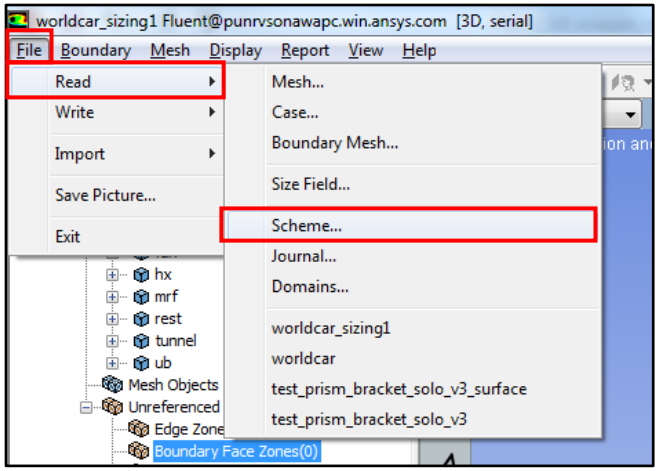
ANSYS高级整车级前处理建模工作流程

- 先进设置脚本
  - 这部分脚本是用来捕捉特征的，网格重构和优化设置

```
;============================Feature Capturing ====================================
(define no_of_imprints_off_geom 1)
(define no_of_imprints_on_geom 2)
(define no_of_rezones_for_sharp_angle_object 1)
(define no_of_rezones_for_sharp_angle_object_skinning 1)
(define feature_threshold_tol_rest 2)
(define feature_cleanup_tol_off_geom 2)
(define feature_threshold_tol_off_geom 2)
(define feature_cleanup_tol_on_geom 0.5)
(define feature_threshold_tol_on_geom 0.5)
(define sharp_angle_capturing_for_skinning_of_dirty_objects "no")
;
(define off_geom_relative_dist_imprint 0.1)
(define off_geom_angle_imprint 90)
(define off_geom_ring_imprint 2)
(define on_geom_imprint_after_merge "no")
;============================End Feature Capturing ================================
;============================Surface Remesh ======================================
(define surf_remesh_method 0) ;enter 0, 1 or 2 (most robust and fast method is 1)
(define surface_remesh_method_1-off_geom_coarsening_factor 1.1)
(define on_geom_surface_remesh_angle 40)
(define on_geom_surface_remesh_corner_angle 60)
(define edge_cleanup_angle 40)
(define edge_cleanup_count 4)
(define off_geom_relative_dist 0.25)
(define off_geom_angle 90)
(define off_geom_ring 1)
(define hybrid_wrap "yes")
(define alternating_quad_split_for_wrapper "no")
(define standard_wrap_n_remesh "yes")
(define off_geom_aggressive_improve "yes")
(define coarsen_using_surface_remesh "yes")
(define surface_remesh_curvature_angle 40)
(define surface_remesh_corner_angle 60)
;;
;============================End Surface Remesh ===================================
;============================Surface Improve ======================================================================
(define contact_face_count 6)
;any contact face count less than 6 will be removed.
(define remove_sharp_angle_global "yes")
(define remove_sharp_angle_global_iterations 1)
(define remove_sharp_angle_ignored_base_aggressive_improve "no")
(define remove_sharp_angle_ignored_base_gentle_improve "no")
(define remove_sharp_angle_ignored_base_iterations 1)
(define remove_sharp_angle_ignored_base_smooth_only "no")
(define min_dihedral_angle 16.0)
(define aggressive_collapse_skew 0.9)
;compulsive collapse of faces having skew larger than this value after all automatic options are exhausted
;
(define MaxFaceSkew 0.4)
;Faces that cause bad prisms will be separated. It will use this max skew value to collapse them.
;
(define MaxFaceSizeChange 1.5)
;Faces that cause bad prisms will be separated. It will use this max size change to collapse them.
;
(define max_island_face_count 500)
;Island clusters are deleted if their face count is smaller than or equal to 200.
```

ANSYS高级整车级前处理建模工作流程

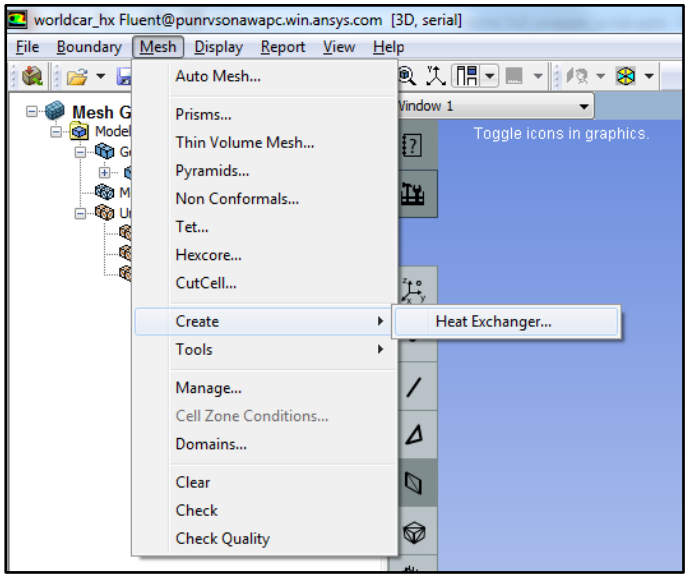
- 运行脚本
  - 打开Fluent Meshing，提供网格文件和脚本文件存放路径
  - 进入File→Read→ Scheme

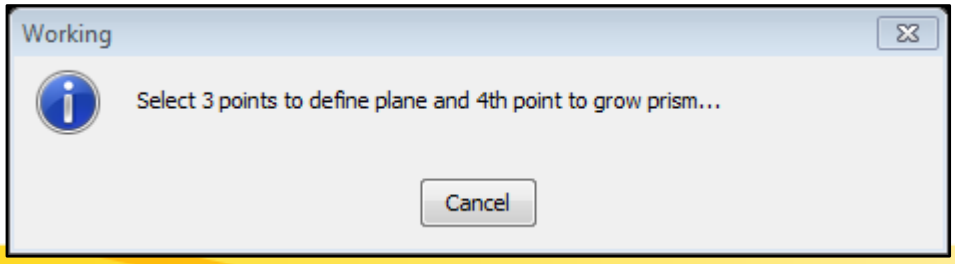


  - 选择“AdvWrapNPrisms_R16R17_v814_Run.scm” 脚本文件，它将开始运行

ANSYS高级整车级前处理建模工作流程

- 添加换热器网格
  - 打开Fluent Meshing
  - 进入File→Read→Mesh
  - 读入“worldcar_hx.msh.gz” 文件
  - 进入to Mesh→Create→Heat exchanger
  - 进入换热器对象，右键显示对象
  - 在“Heat Exchanger Mesh” 面板中，参数设置为局部点，并由尺寸创建
  - 点击“Select nodes..”按钮在“Heat Exchanger Mesh” 面板中
  - 它要求选择3个点并第4个点是增长方向

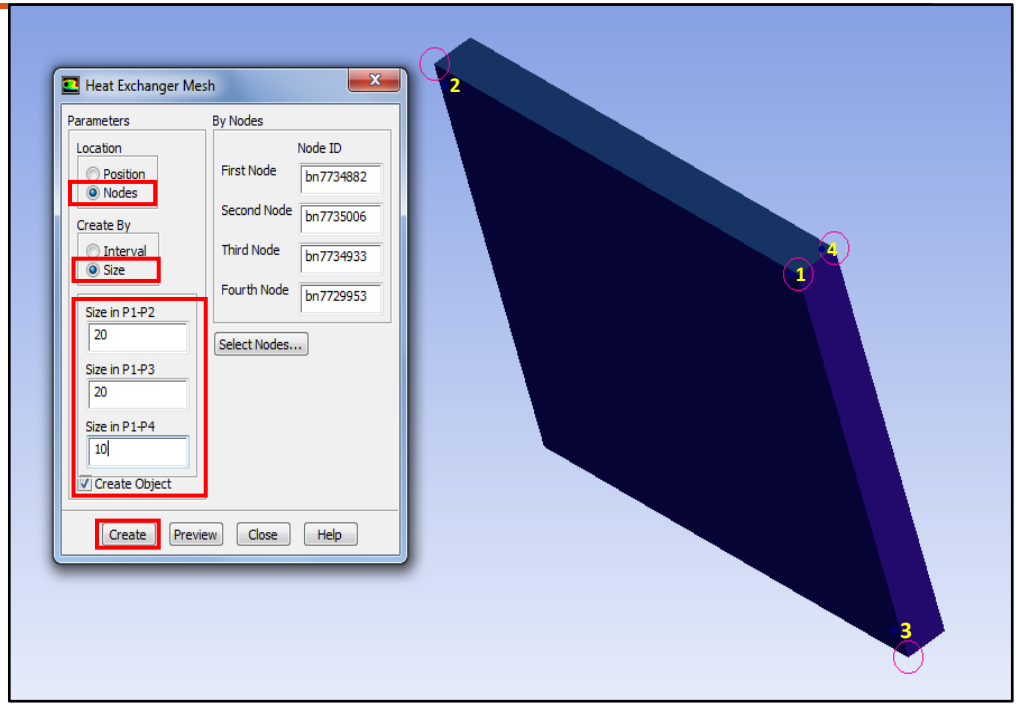


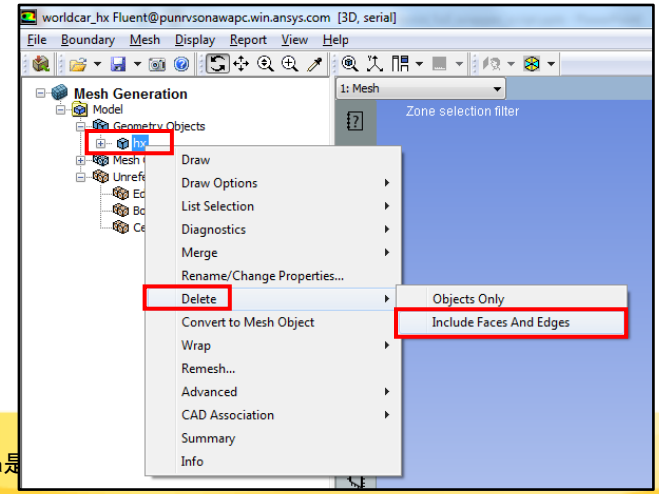


ANSYS高级整车级前处理建模工作流程

- 添加换热器网格
  - 选择点如图
  - 创建换热器网格
  - 写入网格文件为

“worldcar_hx_mesh.msh.gz”



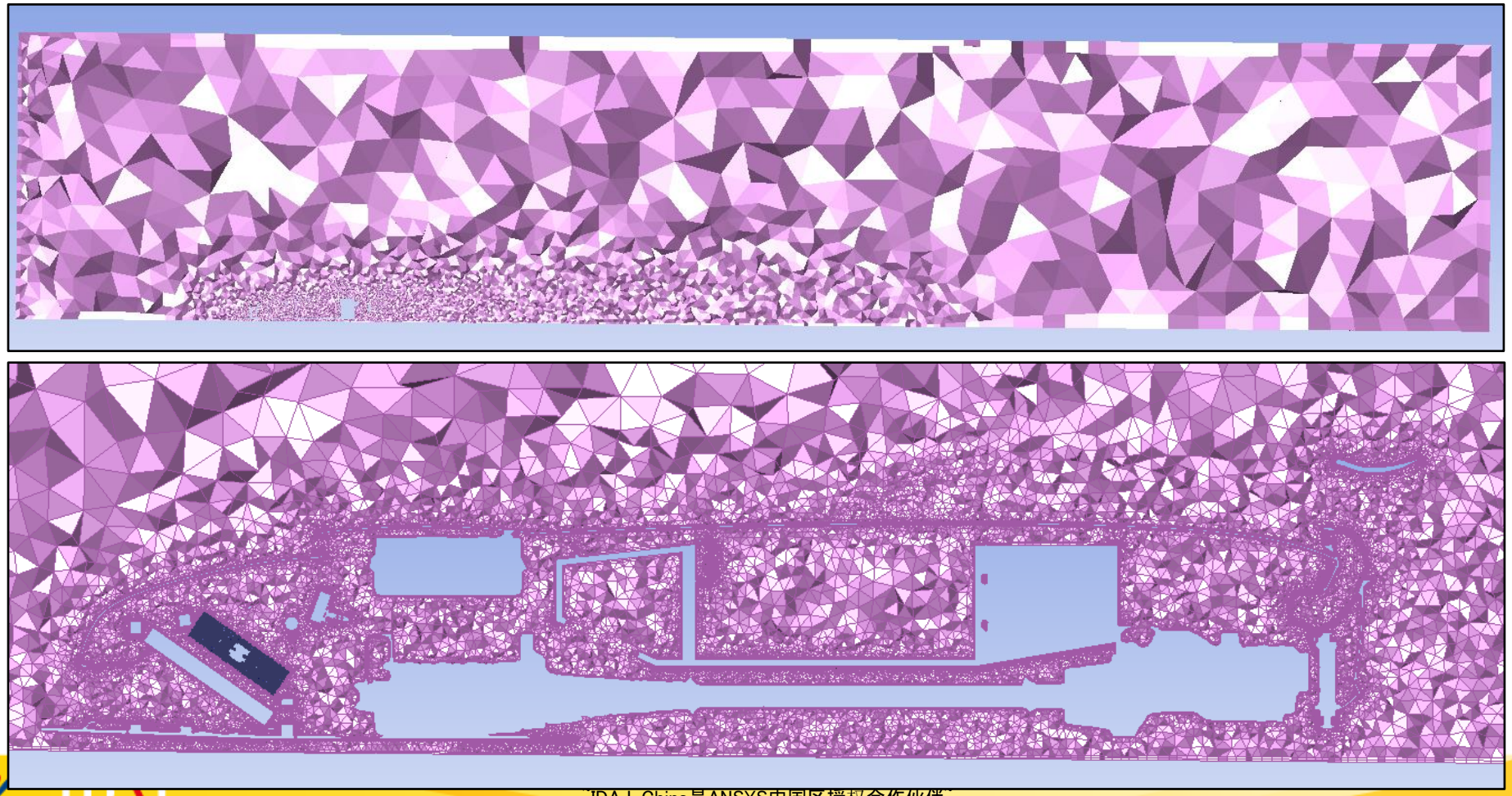


目录

- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

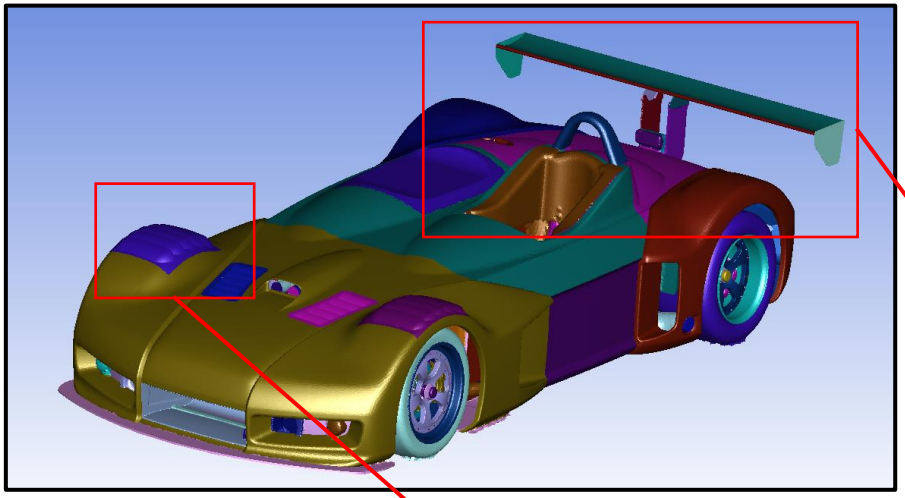
ANSYS高级整车级前处理建模工作流程

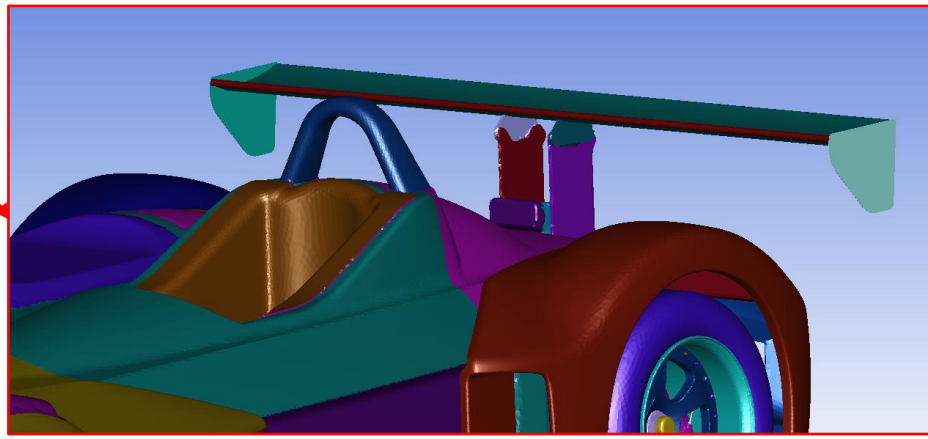
- 成功案例：World Car前处理建模
  - 采用脚本从包面到体网格生成全过程，体网格如图

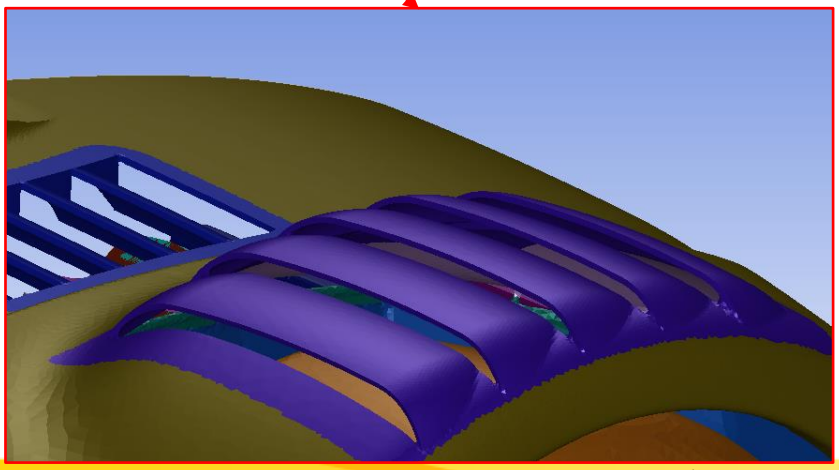


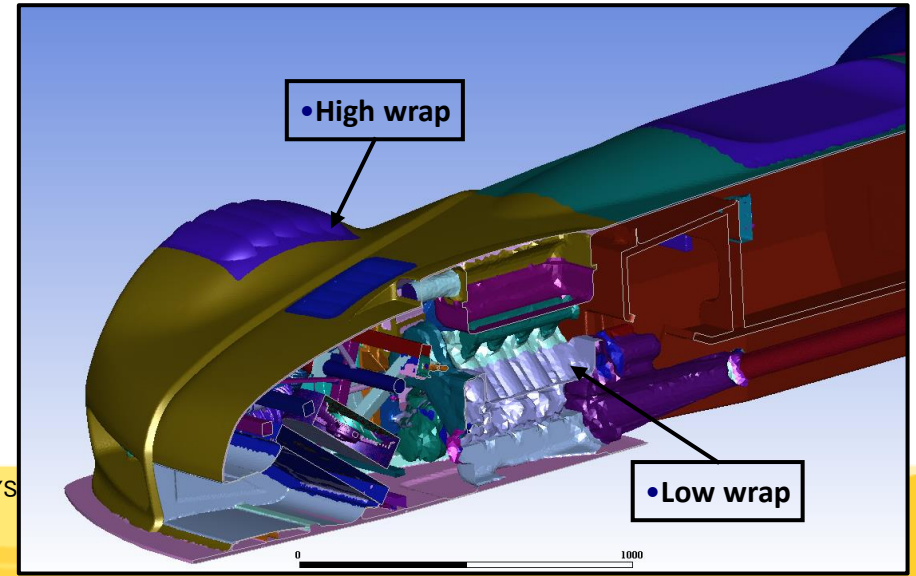
ANSYS高级整车级前处理建模工作流程

- 成功案例：World Car前处理建模
  - 采用脚本包面后的表面网格，特征保持的很好。









ANSYS高级整车级前处理建模工作流程

- 成功案例：World Car前处理建模
  - 采用脚本生成体网格
    - 总网格数：30million
    - 网格质量：0.9599（Orthoskew）

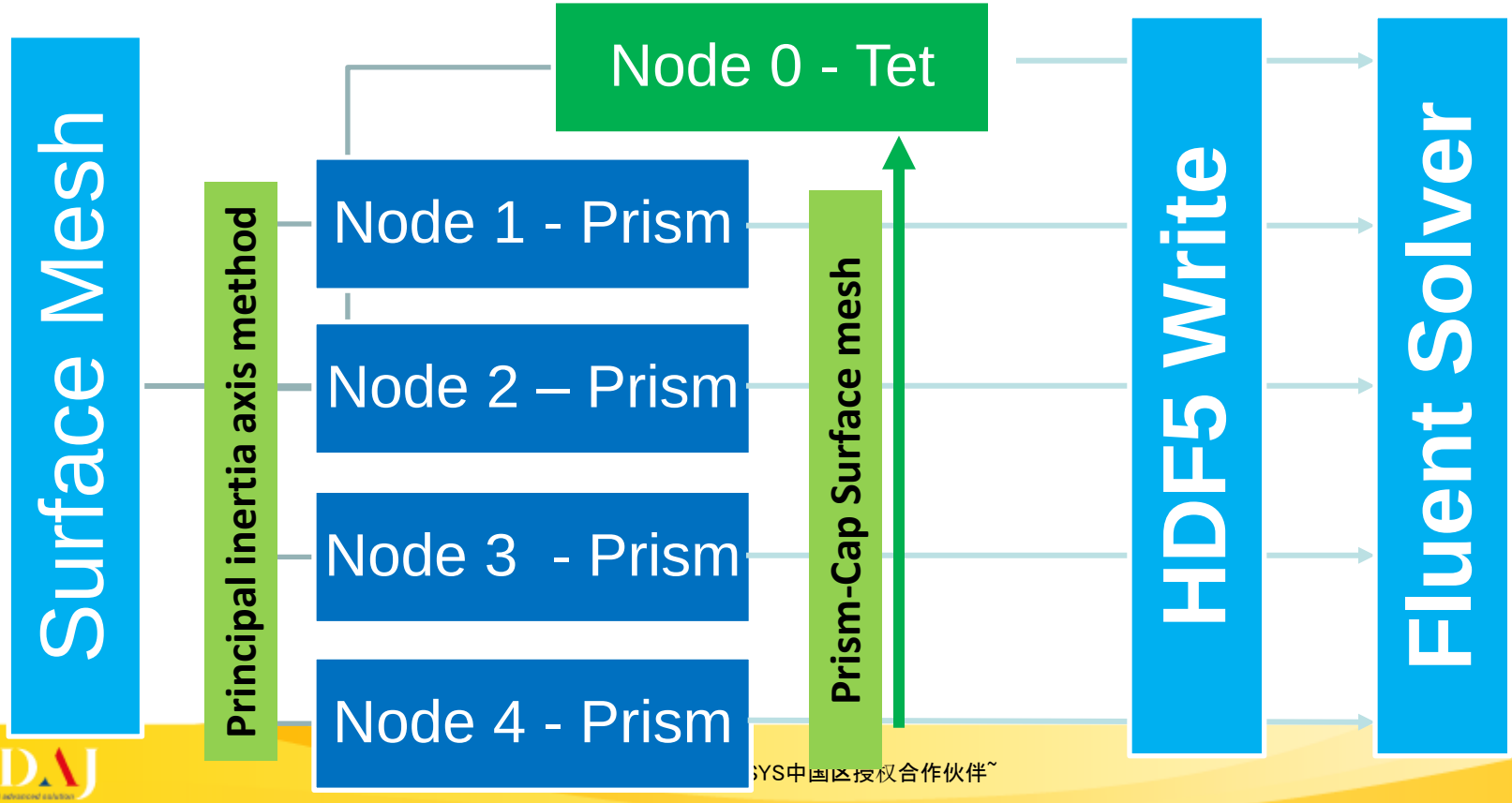
```
Wall-Clock Usage:
================================================================================
Computed SF in ---------------------------------23.25 minutes (0.3875 hours).
Merge zones in ---------------------------------9.75 minutes (0.1625 hours).
Wrapped all live regions in --------------------32.65 minutes (0.54416667 hours).
Merge in ---------------------------------------4.5333333 minutes (0.075555556 hours).
Imprint in -------------------------------------46.65 minutes (0.7775 hours).
Surface Remesh to target size in ---------------22.733333 minutes (0.37888889 hours).
Surface Improved (inflate) in ------------------5 minutes (0.083333333 hours).
Surface Improved (general) in ------------------13.516667 minutes (0.22527778 hours).
Surface Improved (resolve_int) in --------------5.05 minutes (0.084166667 hours).
PostTet Surface Improve in ---------------------21.15 minutes (0.3525 hours).
Connect Mesh Objects with Wrapper in -----------6.2166667 minutes (0.10361111 hours).
Compute Volume-Fill Regions in -----------------2.9166667 minutes (0.048611111 hours).
Prisms generated in ----------------------------35.1 minutes (0.585 hours).
Prisms Improved (Continuous) in ----------------43.533333 minutes (0.72555556 hours).
Rezoned in -------------------------------------33.383333 minutes (0.55638889 hours).
Volume-fill in ---------------------------------18.15 minutes (0.3025 hours).
================================================================================
T O T A L  M E S H I N G  T I M E: ------------342.53333 MINUTES (5.7088889 HOURS)
================================================================================
CPU-Time: 283.00843 min (4.7168072 hours)
--------------------------------------------------------------------------------
-------------------------------------------
        | Virtual Mem Usage (GB)|
ID      | Current     Peak      | Page Faults
-----------------------------------------------
serial  | 23.9008     25.6975   | 2.064e+08
-----------------------------------------------
Total   | 23.9008     25.6975   | 2.064e+08
-------------------------------------------
```

目录

- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

ANSYS高性能计算解决方案

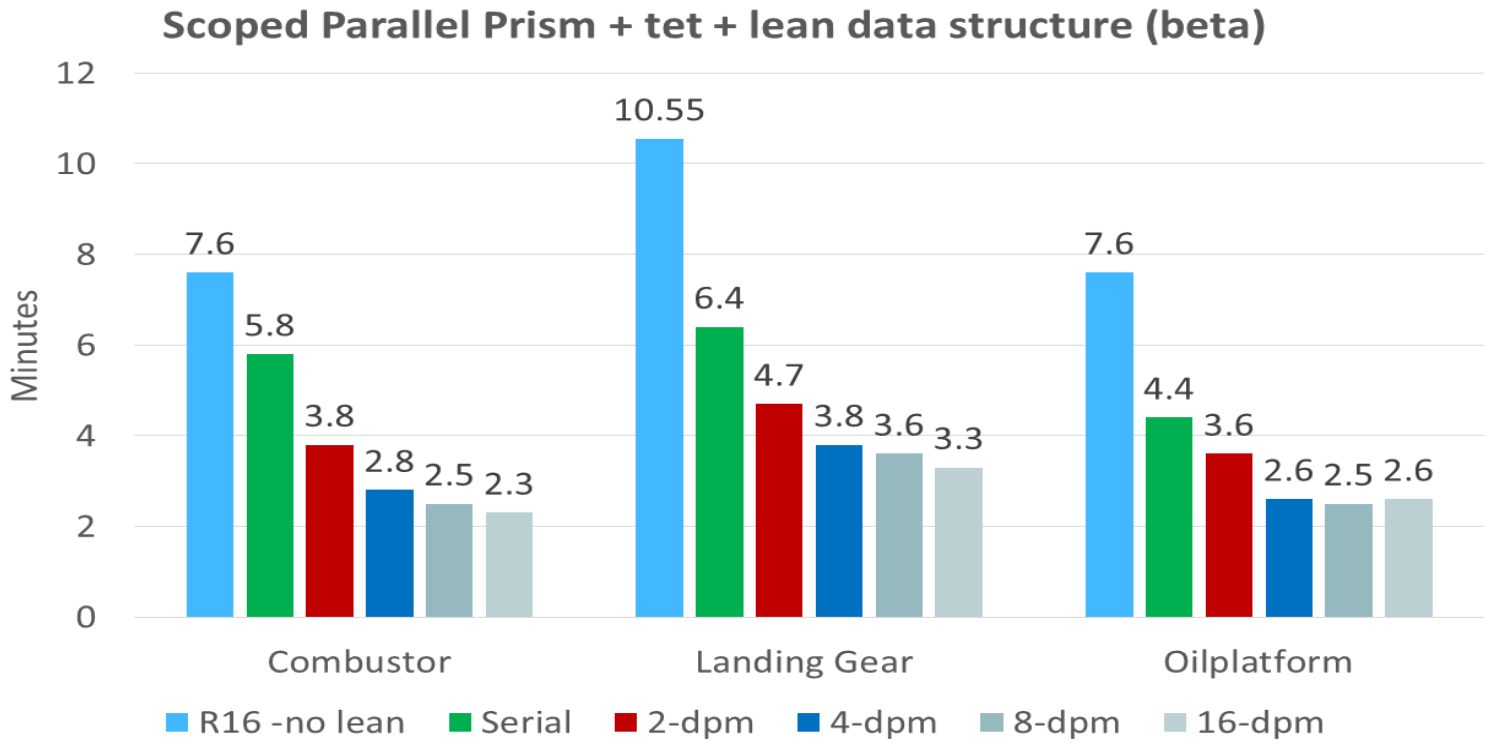
- 前处理网格方案
  - 采用并行网格生成方式
  - 在8节点下并行是串行网格生成的2-3倍



ANSYS高性能计算解决方案

前处理网格方案

- 采用并行边界层生成+串行四面体网格是生成效率



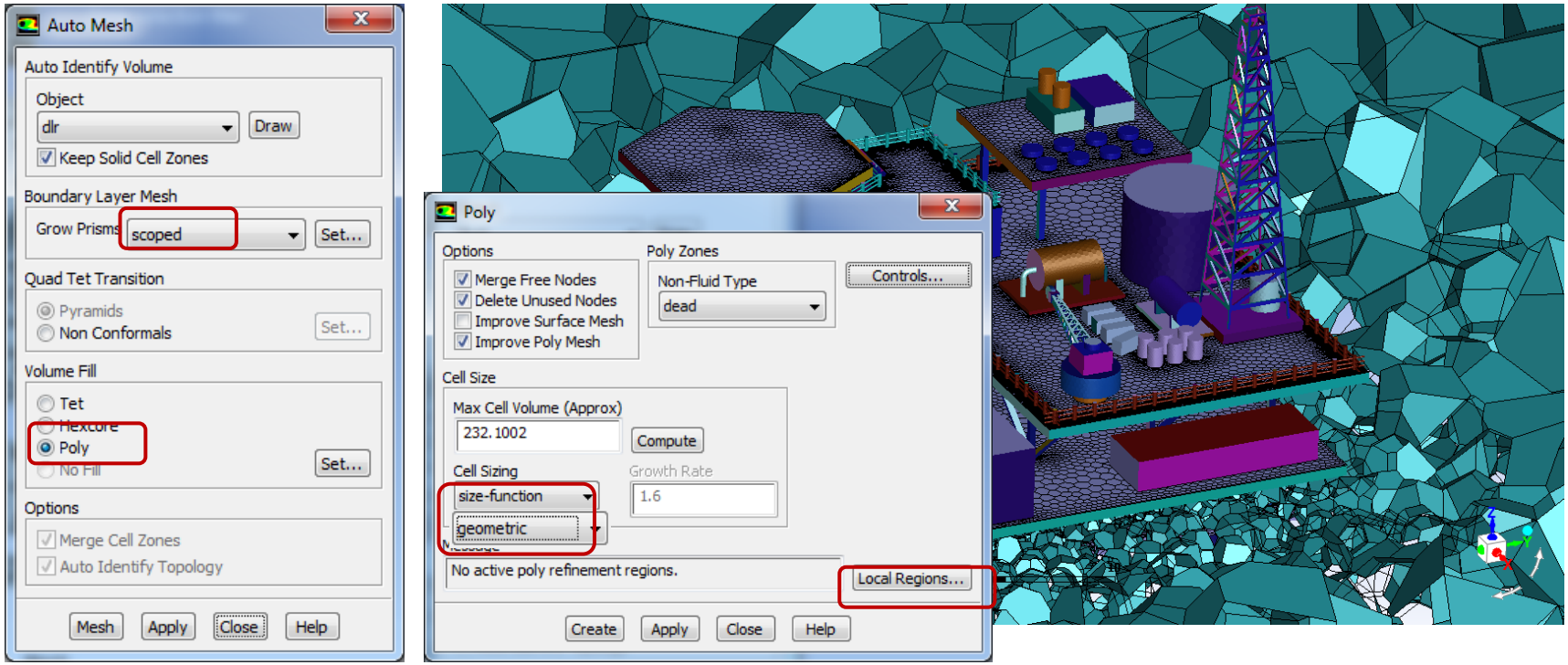
Combustor
4.7 M Prism
3.5 M Tet

Landing Gear
7.0 M Prism
5.7 M Tet

Oil platform
3.6 M Prism
5.7 M Tet

ANSYS高性能计算解决方案

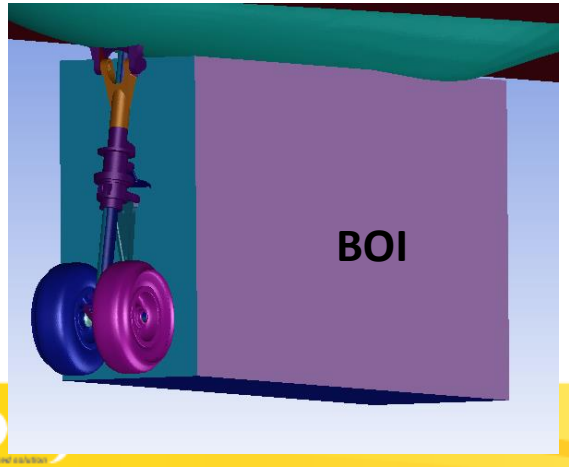
- 前处理网格方案
  - 直接生成高质量的多面体网格
  - 串行直接生成多面体比串行转化多面体节省2-3倍内存，同时加速2-3倍

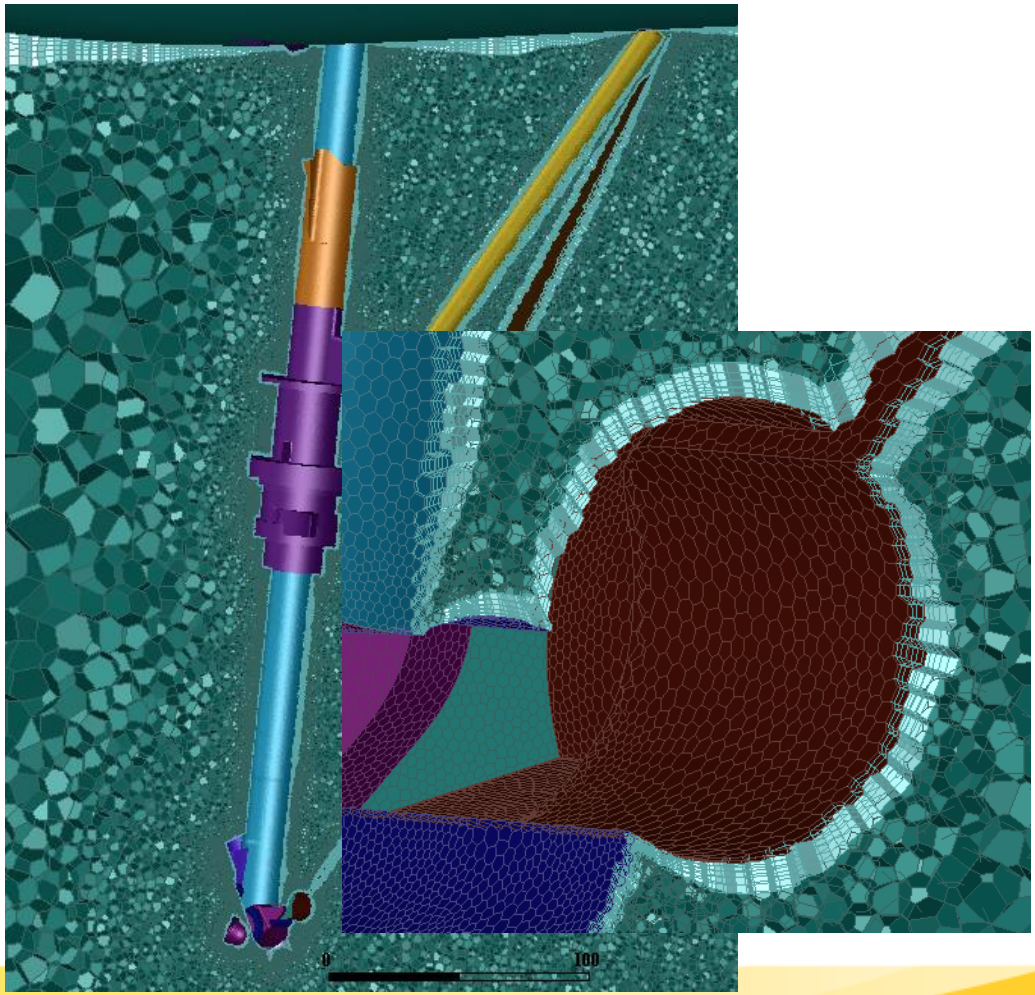


ANSYS高性能计算解决方案

- 前处理网格方案
  - 对比直接生成多面体网格质量及效率

采用BOI创建起落架网格
- 8.1 million poly cells
- R16.1
  - Worst initial OS = 0.99
    - 3 cells over 0.9 fixed by ANM
  - Time = 32 min
  - Peak memory 12 GB
- R17.0
  - Worst OS = 0.9
  - Time = 20 min
  - Peak memory 12 GB





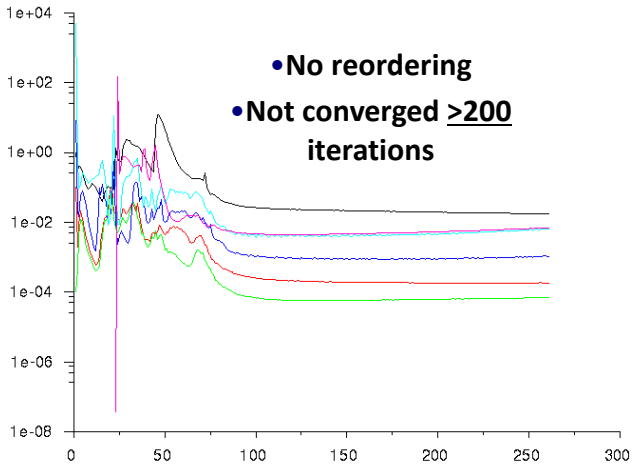
目录

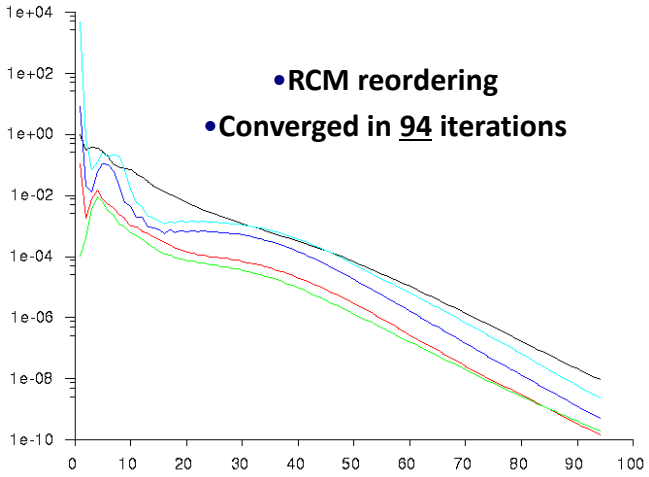
- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

ANSYS高性能计算解决方案

求解器解决方案

- Coupled Solver默认采用Conservative Coarsening算法
  - ✓ 有效改善收敛特性
  - ✓ 特别是对于那些采用多面体网格、网格质量较差的算例
- 代数多重网格加速求解器自动对线性矩阵进行Reorder
  - ✓ 大幅改善收敛特性
- FLUENT在此方面一直较竞争对手有明显优势，R17进一步扩大这种优势

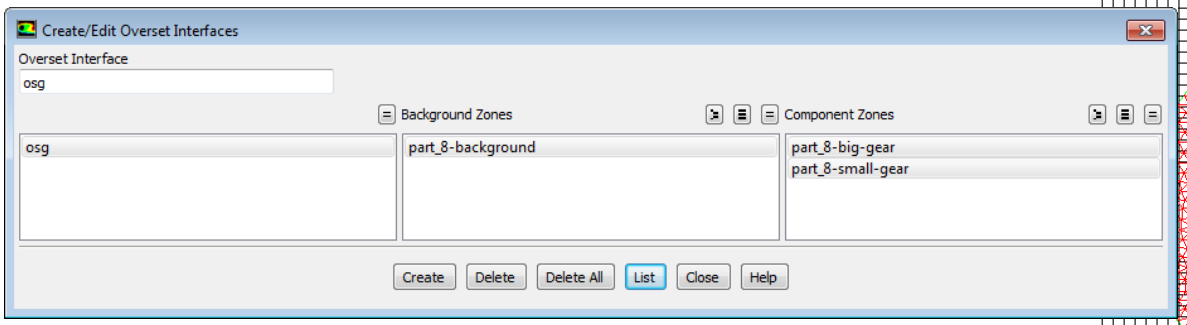


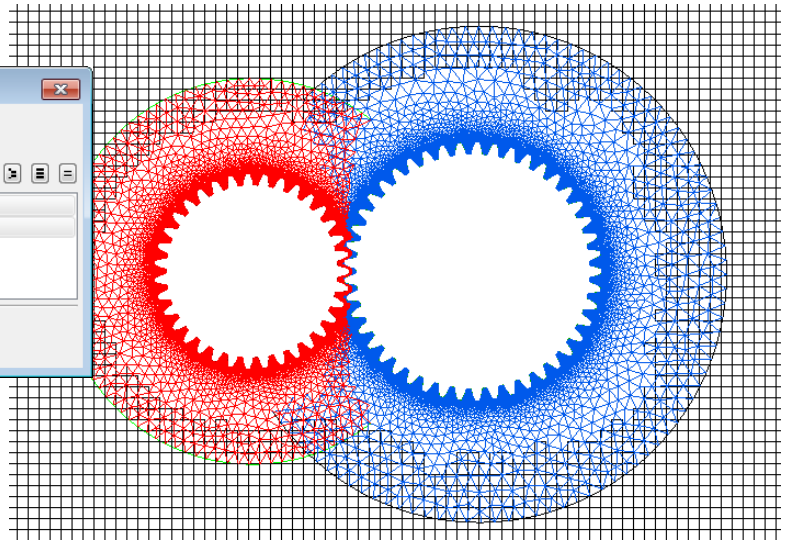


ANSYS高性能计算解决方案

求解器解决方案

- 相较传统的方法，Overset方法的初始网格生成简单，用户使用起来更简单:
  - ✓ Steady 和 transient (fixed mesh), 3D 和 2D 平面
  - ✓ 基于压力的耦合算法
  - ✓ 不可压密度模型
  - ✓ 单相或 VOF 多相流模型
  - ✓ 传热模型
  - ✓ k-epsilon 和 SST k-ω 湍动模型
  - ✓ **BETA**: 动网格、可压流、带表面张力的VOF模型、 pressure-far-field 边界、Workbench 支持、分离式算法支持





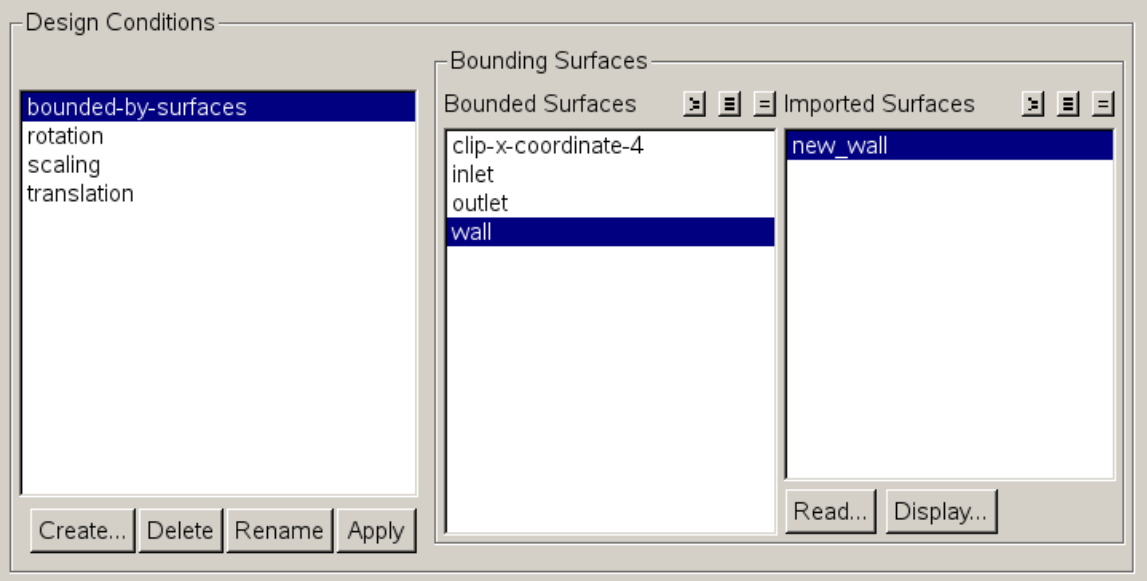
- **限制:**
- 不支持DBNS
- 不支持Node-based gradients

ANSYS高性能计算解决方案

求解器解决方案

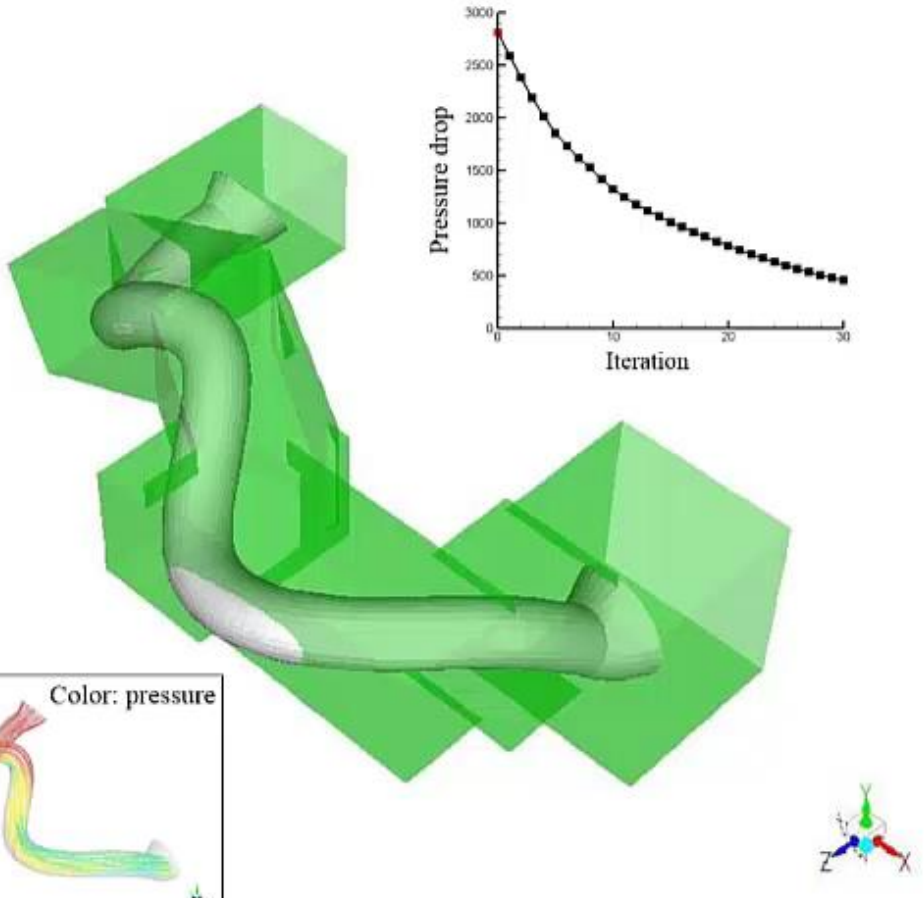
Adjoint Solver & Design Tool

- ✓ Adjoint Solver支持可压流计算
- ✓ Design tool 改进
  - 一般变形限制面的支持
  - 预定运动(rotation, translation, scaling)
  - 改善易用性- 约束检查
  - STL 输出



ANSYS高性能计算解决方案

- 求解器解决方案
  - 管道优化



•通过优化管子几何，通过管子的全压降是最小的。

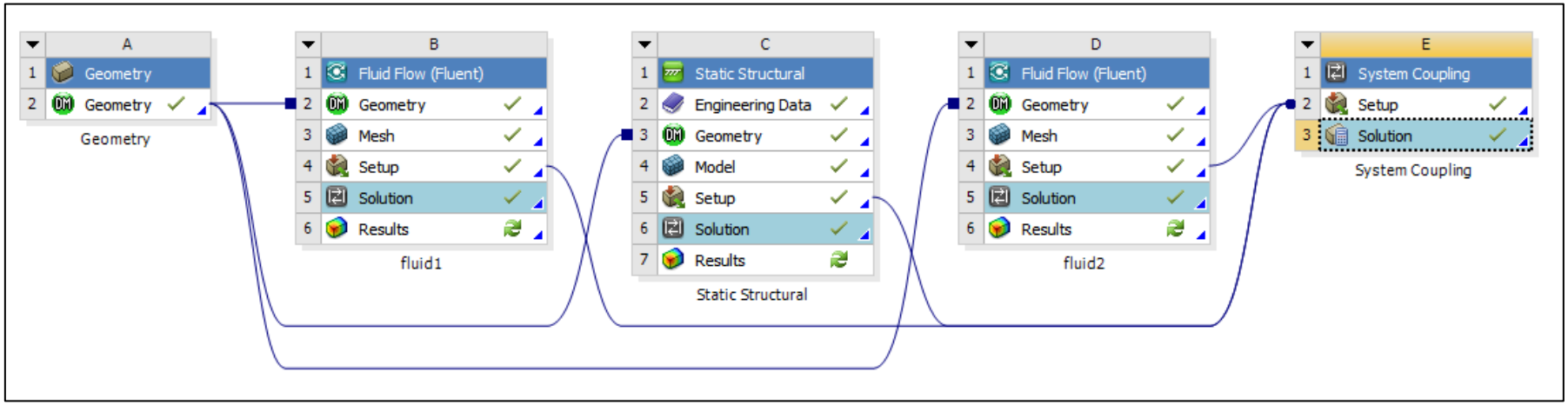
•当入口和出口被固定，管子被保持在一个复杂的边界面上，以此定义输入网格。

ANSYS高性能计算解决方案

求解器解决方案

多物理场耦合：

- ✓ 允许任意数目的CFX, Fluent and Mechanical分析项目通过System Coupling耦合
- ✓ 允许Fluent和Fluent通过System Coupling耦合



Two Fluent and one Mechanical system connected via System Coupling

- ✓ 可以求解更为复杂的流动问题和多物理场耦合问题

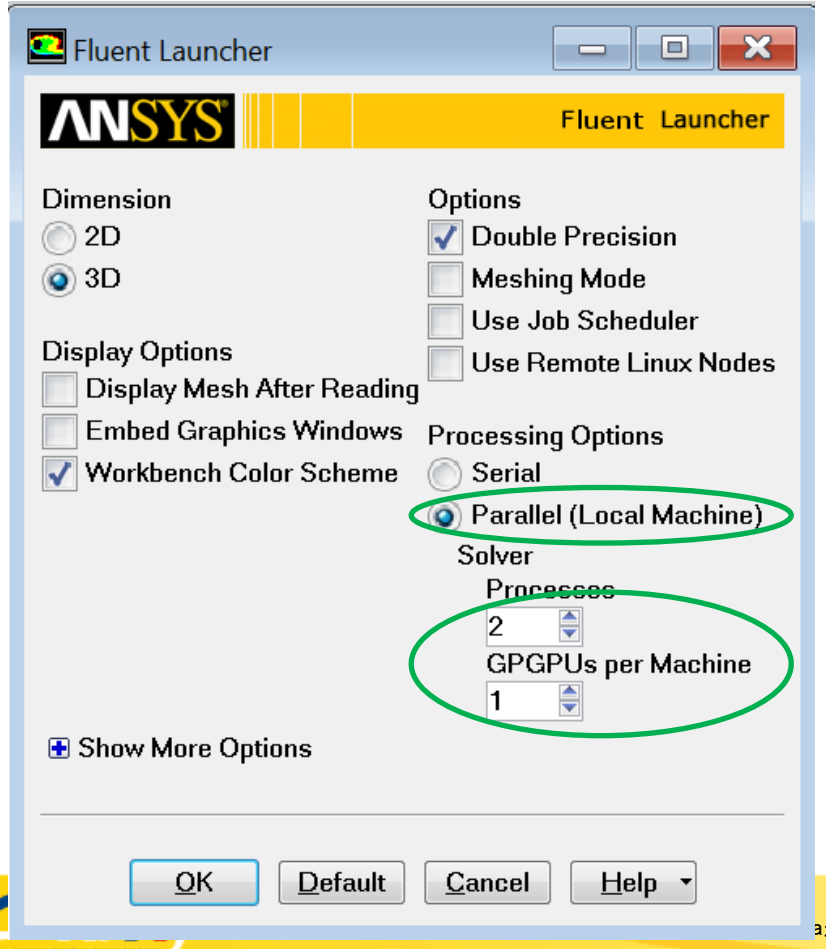
目录

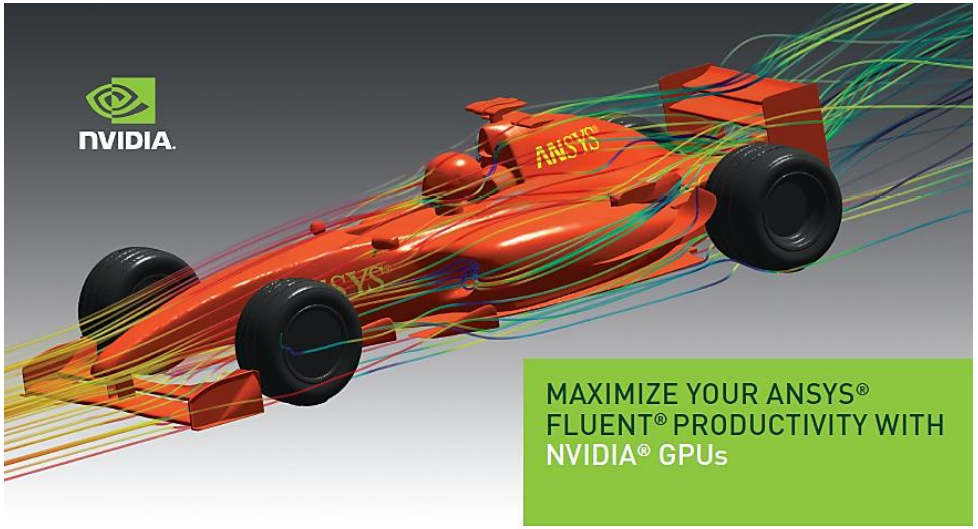
- CAE前处理及高性能并行计算的重要性：
- ANSYS 前处理解决方案：
- ANSYS高级整车级前处理建模工作流程（含案例）
  - 建模流程
  - 成功案例：World Car前处理建模
- ANSYS高性能计算解决方案
  - 前处理网格解决方案
  - 求解器方案
  - GPU加速

ANSYS高性能计算解决方案

GPU加速

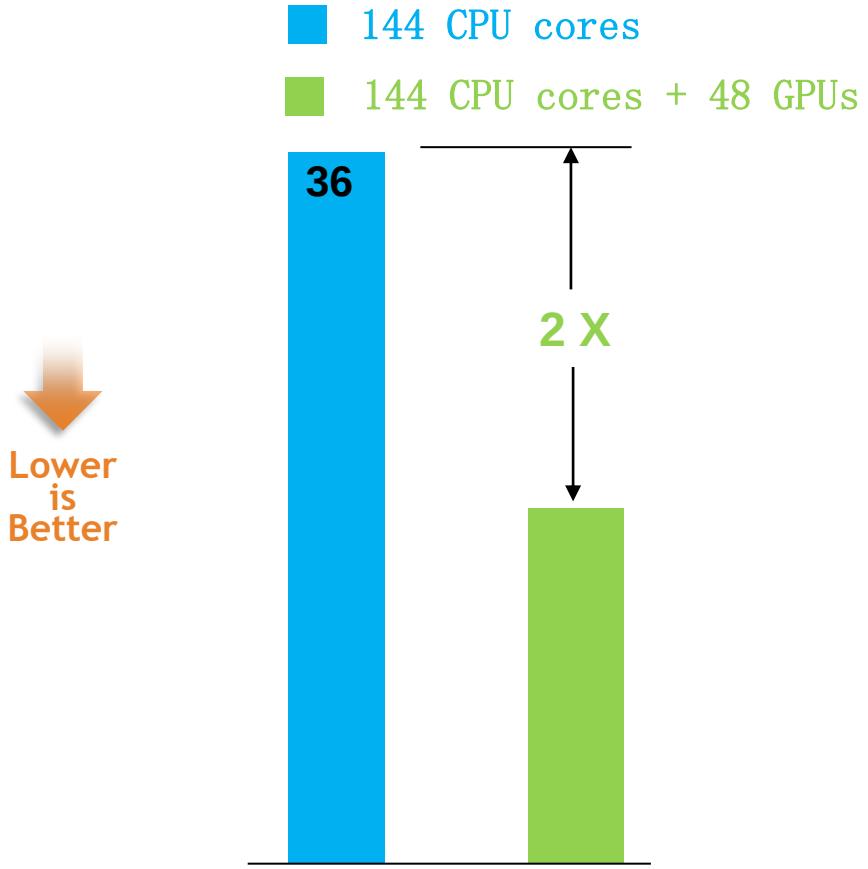
GPU启动

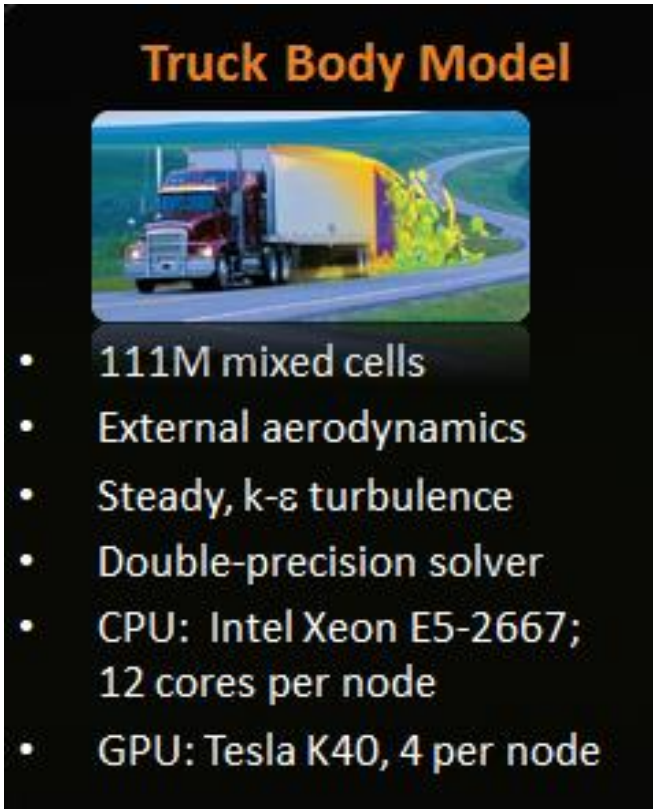




ANSYS高性能计算解决方案

- GPU加速
  - GPU降低了111million空气动力学问题

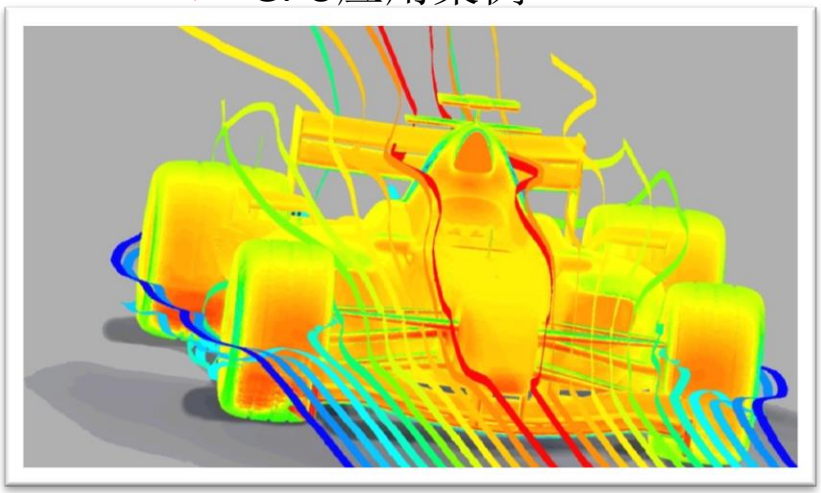




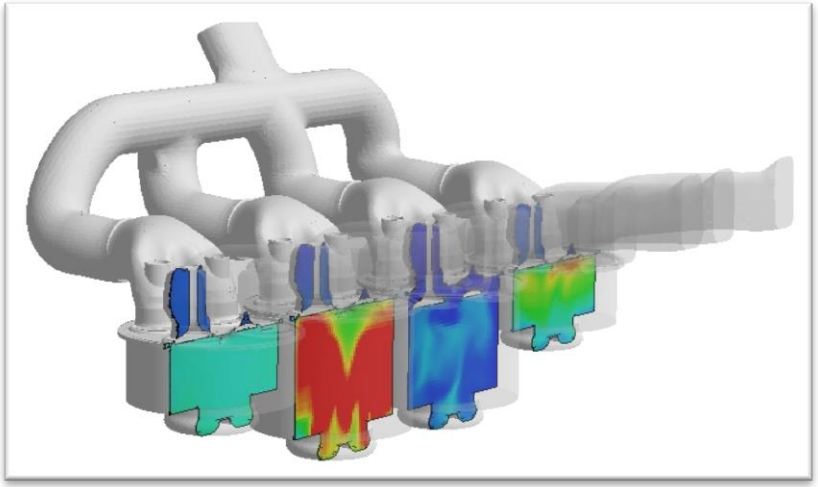
•Fluent solution time
• per iteration (secs)

ANSYS高性能计算解决方案

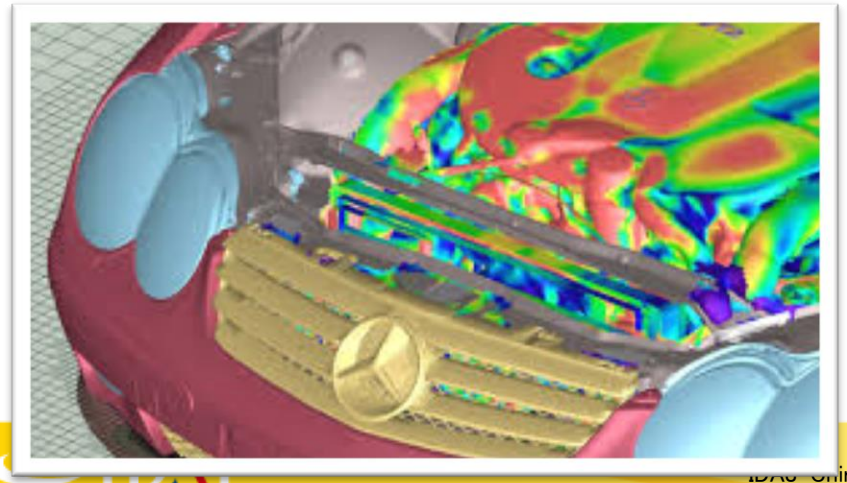
- GPU加速
  - GPU应用案例



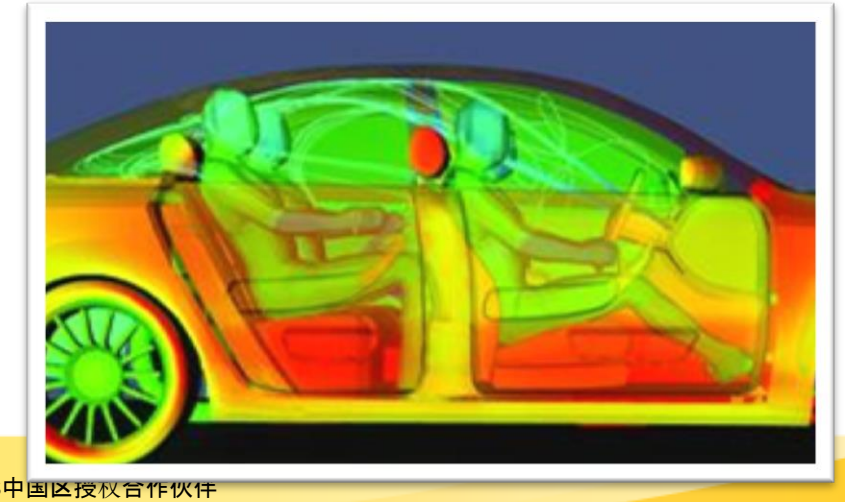
•External aerodynamics



•In-cylinder flow analysis



•Underhood thermal analysis



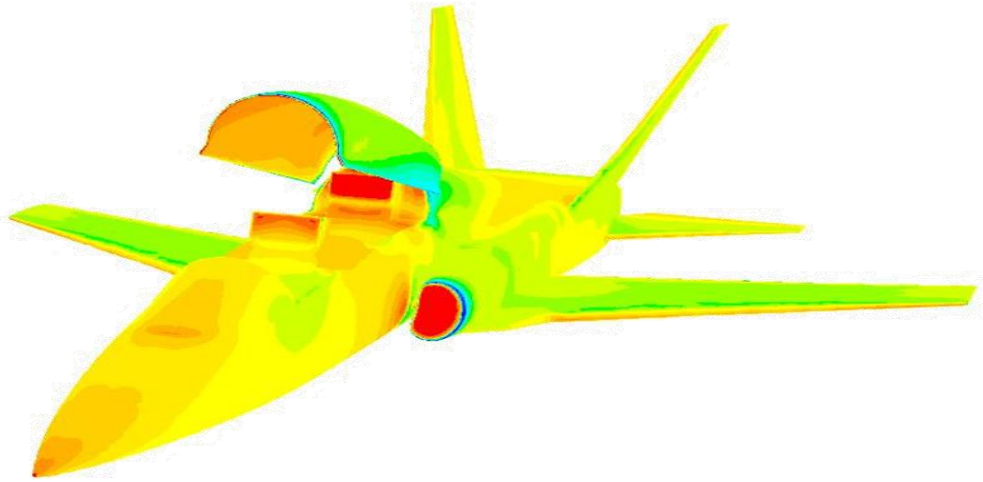
•Passenger comfort analysis

ANSYS高性能计算解决方案

GPU加速

GPU在航空的案例

- **ANSYS 17.0**
- **1.8M cells**
- **8 Cores (Xeon E5-2687)**
- **64 GB Ram**
- **NVIDIA K40 GPU**
- **Solution Time:**
  - **GPU : ~ 1 hr**
  - **No GPU: ~ 1.75 hrs**



- **GPU Produces a 43% Reduction in Solution Time**