CADMEN 虎門科技 | Ansys精英合作夥伴

Ansys
2022 R1
Engineering What's Ahead.
新技術線上研討會

Mechanical

Mesh Reinforcement Optimization and HPC

虎門技術團隊
講師:王弘政
Mathias Wang (M.S.)
TECHNISCHE UNIVERSITÄT DARMSTADT

Ansys

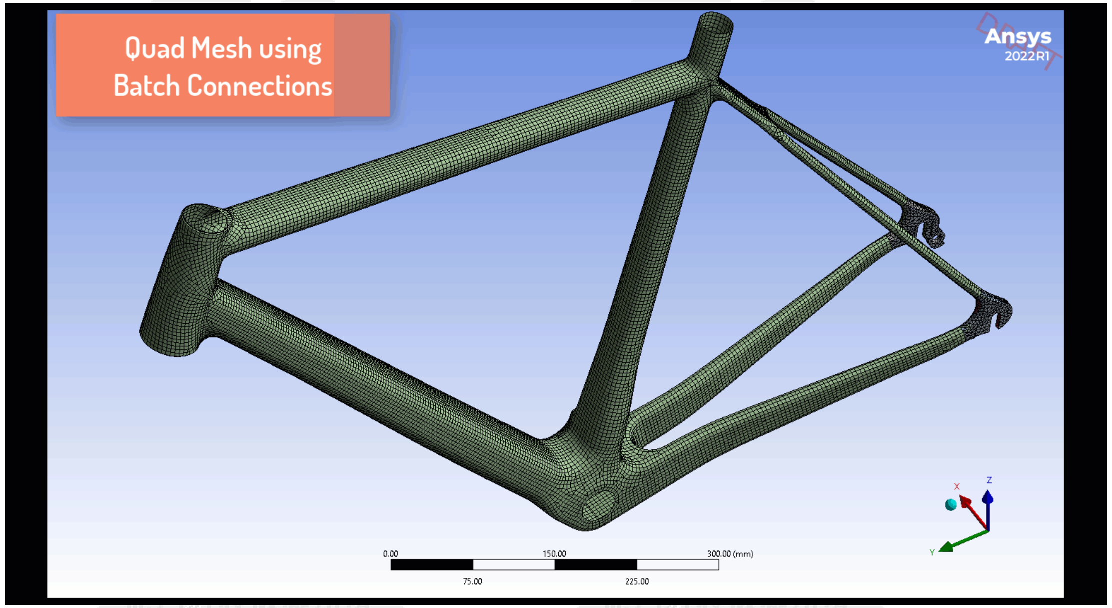
Meshing

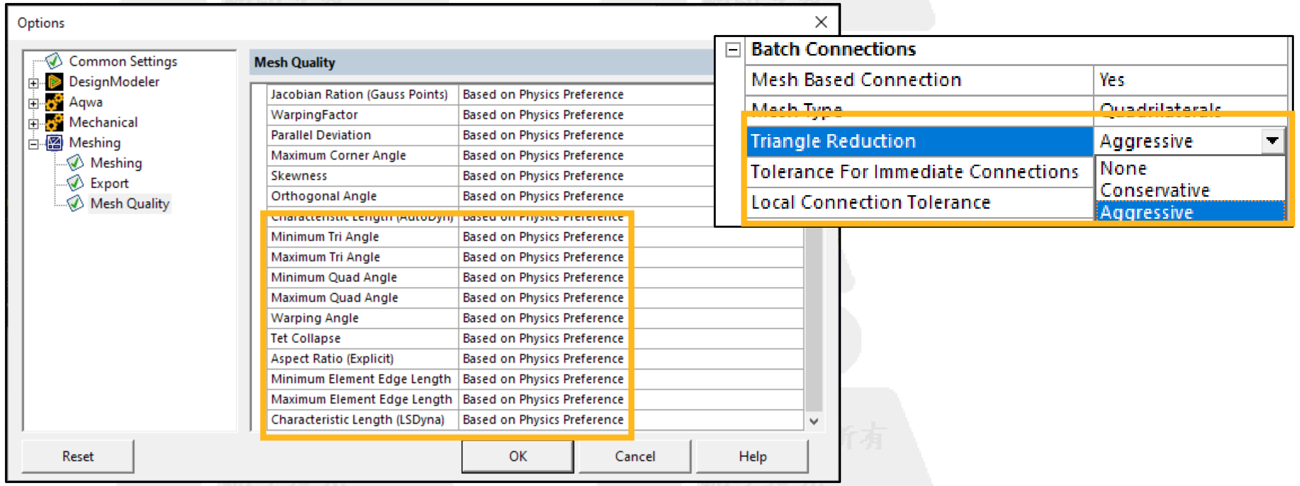
New Features

- **Shell Meshing**
  - Tri Reduction
  - Support for new metric visualization:
    - Warping angle
    - Min quad/tri angle
    - Max quad/tri angle
    - Min/Max Element Edge Length
- **Welds Meshing**
  - Enhancements to Weld Worksheet
  - Usability and error handling improvements
  - Visualisation improvements
  - HAZ layer Named Selections
- **Pull**
  - Line Coating
  - Quadratic mesh support
- **Explicit Physics Preference**
  - New defaults/behaviours for Tet meshing
  - New defaults/behaviours for Hex meshing
  - Quality targeting for Aspect Ratio (Explicit)
  - New default metric visibility per physics preference
  - Support for new metric visualization:
    - Characteristic Length (LS Dyna)
    - Aspect Ratio (Explicit)
    - Tet Collapse
- **General Tet Meshing**
  - Improved robustness of defeaturing
  - Improved Error/Warning messages
  - Feature Detection for solid holes and fillets
  - Proximity Gap Factor for AR control in coarse mesh
  - Diagnostics-based Named Selection Worksheet tools:
    - Intersecting surface mesh failures
    - Free edge mesh
    - Sharp angle
    - Body Interference
    - Defeatured Topology
  - Model Walk extension to mesh elements/element clusters
- **Hex Meshing**
  - Improved pave mesh in Multizone
  - Improved default mesh with MultiZone for simple shapes e.g. cylinder, sphere, pipe, etc
  - Split Angle for less decomposition
  - Body-Fitted Cartesian
    - Support for Edge Sizing
  - Beta Multizone Options:
    - CartSweep Decomposition (2.5D Geometry)
    - ThinSweep Decomposition (Thin Geometry)
- **SpaceClaim Meshing**
  - Thin Body Meshing
  - Robustness, Performance, Usability
  - Meshing for Explicit Improvements

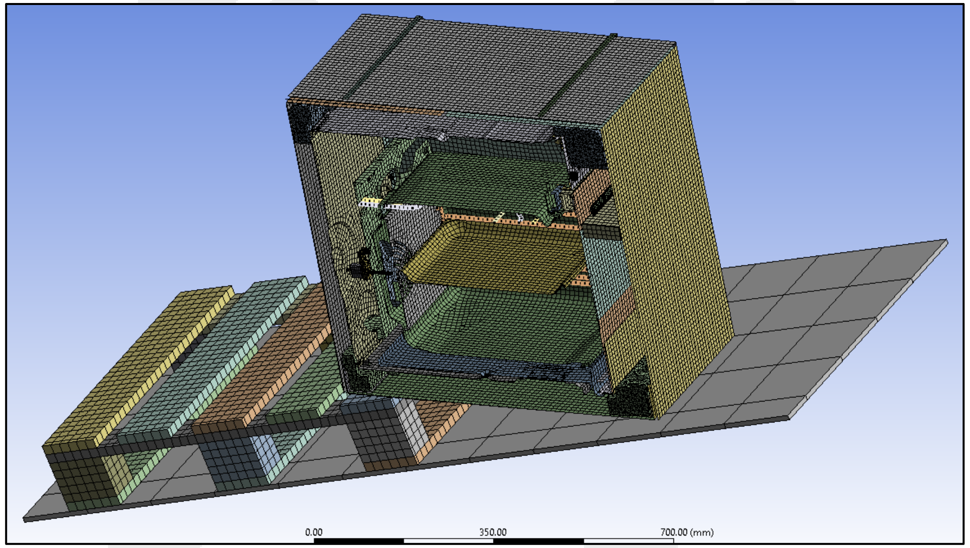
Shell Meshing: Triangle Reduction & Quality

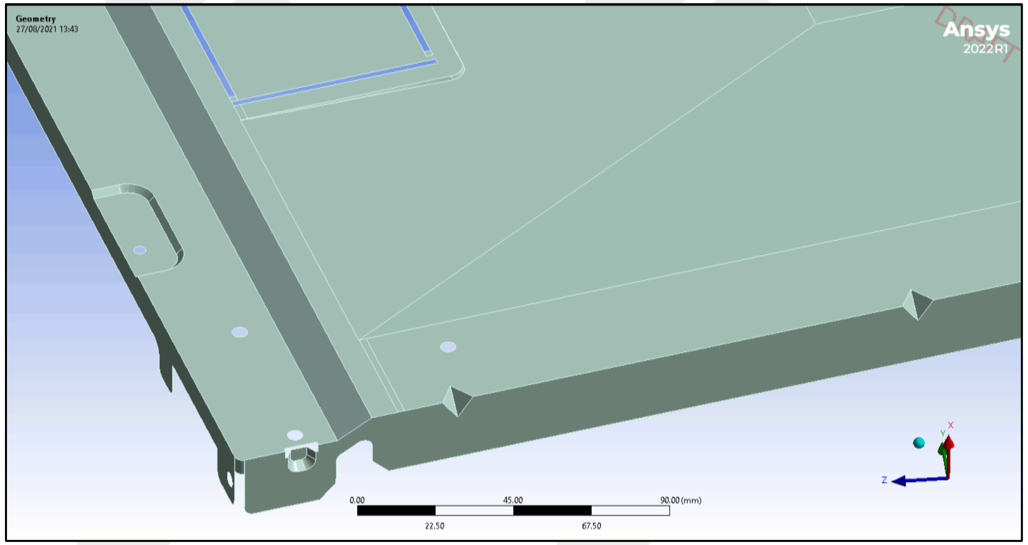
- Available with Batch Connections
- Option to control level of triangles in quad mesh
  - None
  - Conservative (default)
    - Remove triangles near shell edges
  - Aggressive
    - Remove as many triangles as possible sometimes at cost of quality
    - Seen to reduce tri count by up to 80% in some cases and below 1-3%
- New Quality Metrics
  - Hidden/Shown based on Physics Preference chosen
  - User can customise

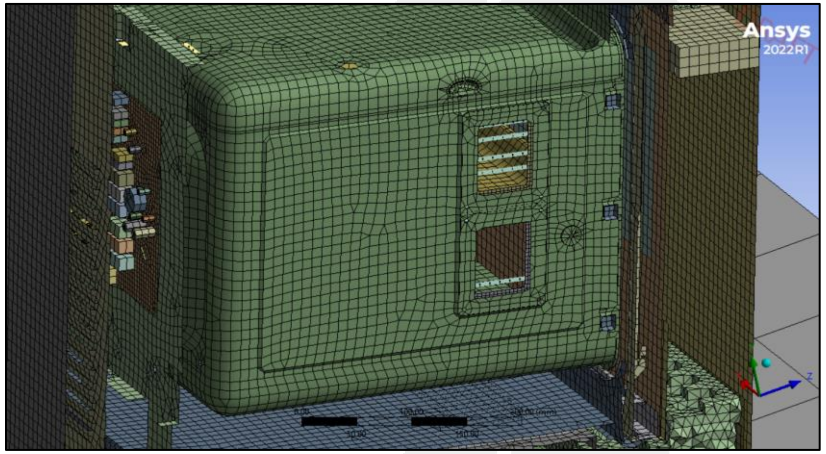


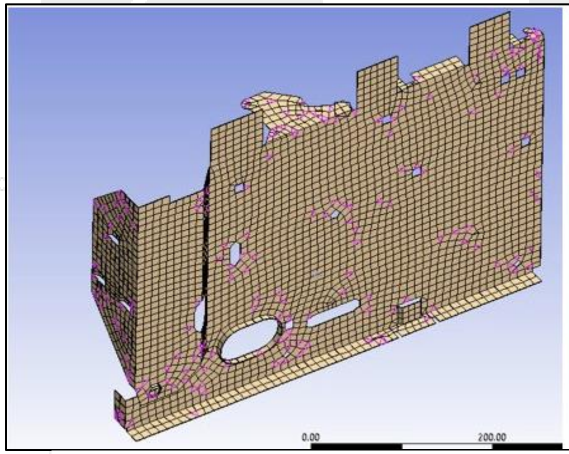


Steamer Shell Mesh: Tri Reduction

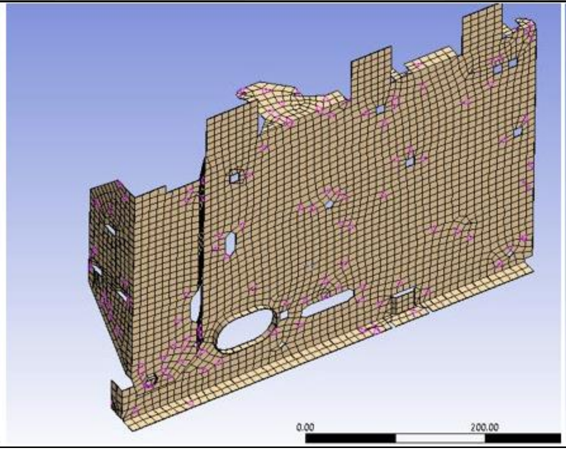




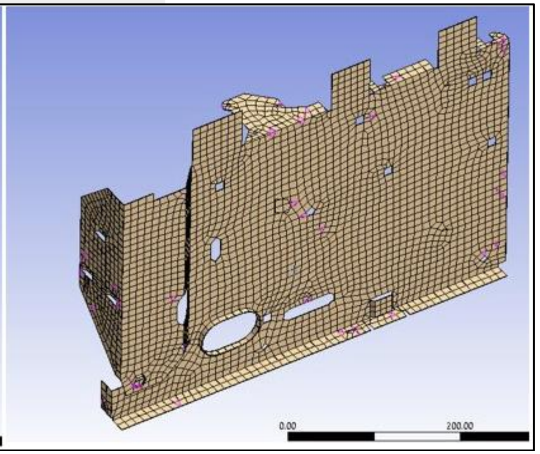




Tri Reduction = None
Number of tri = 184



Tri Reduction = Conservative
Number of tri = 126



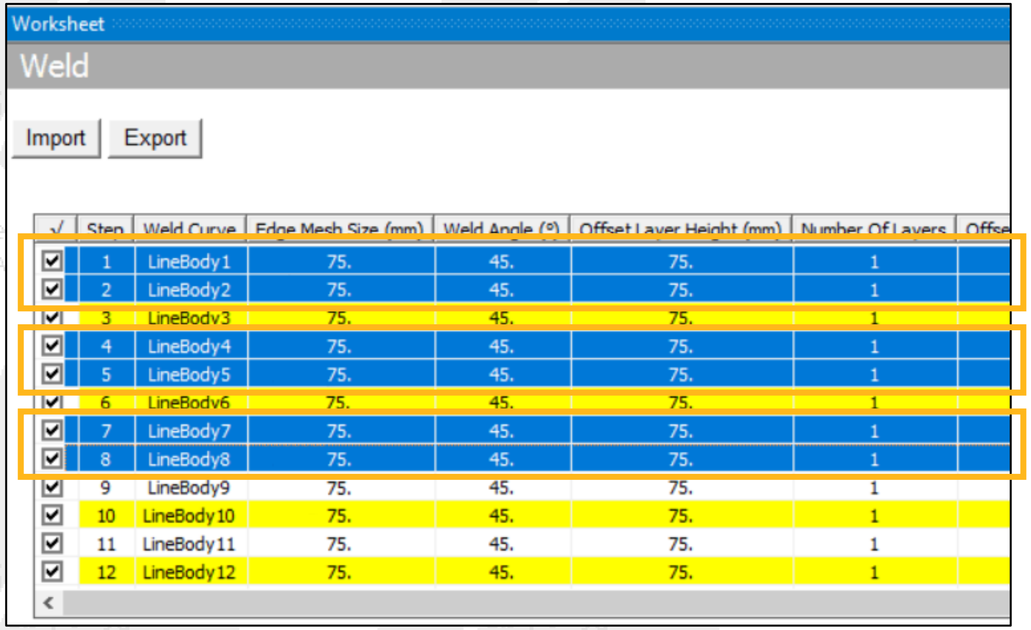
Tri Reduction = Aggressive
Number of tri = 48

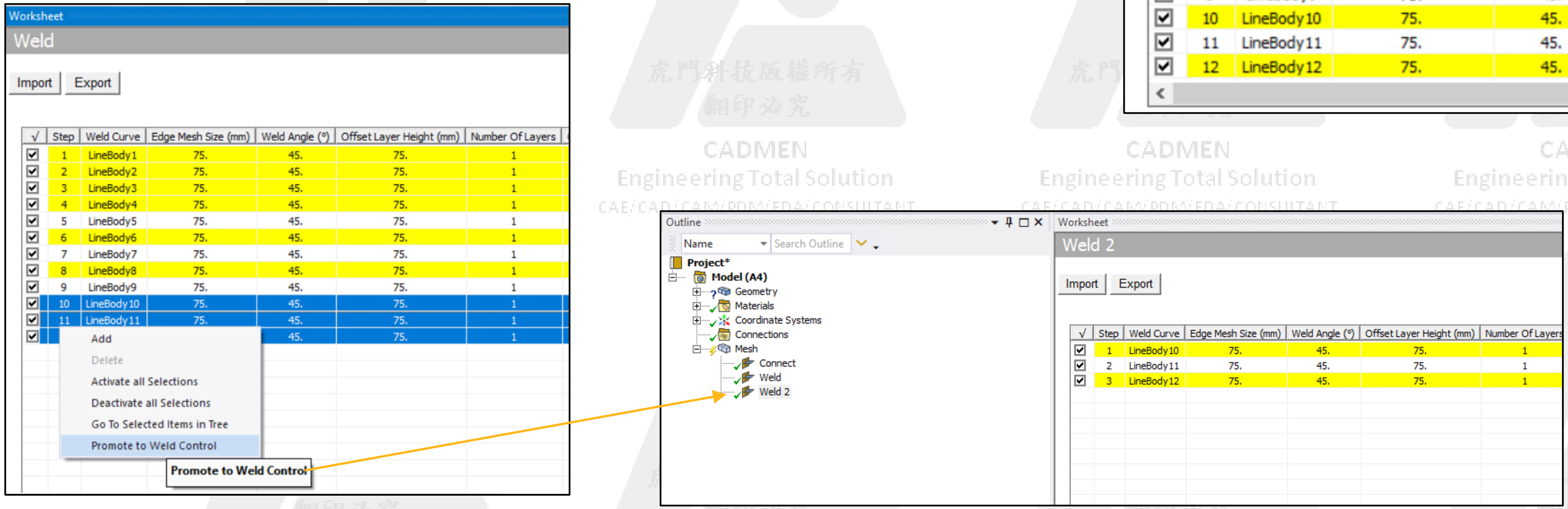
Terminology Changes to Align with Industry

Old Terminology	New Terminology
Tent	Angled
Extension	Normal
Tent and Extension	Normal and Angled
Seam	Continuous Seam
Skip	Intermittent Seam
Create Offset Layer	Create HAZ Layer
Number of Layer	Number of HAZ [1,2,3, No]
Offset Layer Growth Rate	HAZ Growth Rate
Offset Layer Height	HAZ Distance
Weld Width	Weld Width (Leg01)
Weld Height	Weld Height (Leg02)
Default Changes	
Weld Creation Criteria	Default change to Width Based instead of Angle Based for Creation Criteria

Weld Meshing: Worksheet Enhancements

- Work with multiple rows from weld worksheet
  - Selection, Activation, Deactivation etc…
  - Go to selected items in tree
- Promote to weld control
  - creates new weld control with worksheet by removing selected rows from original.

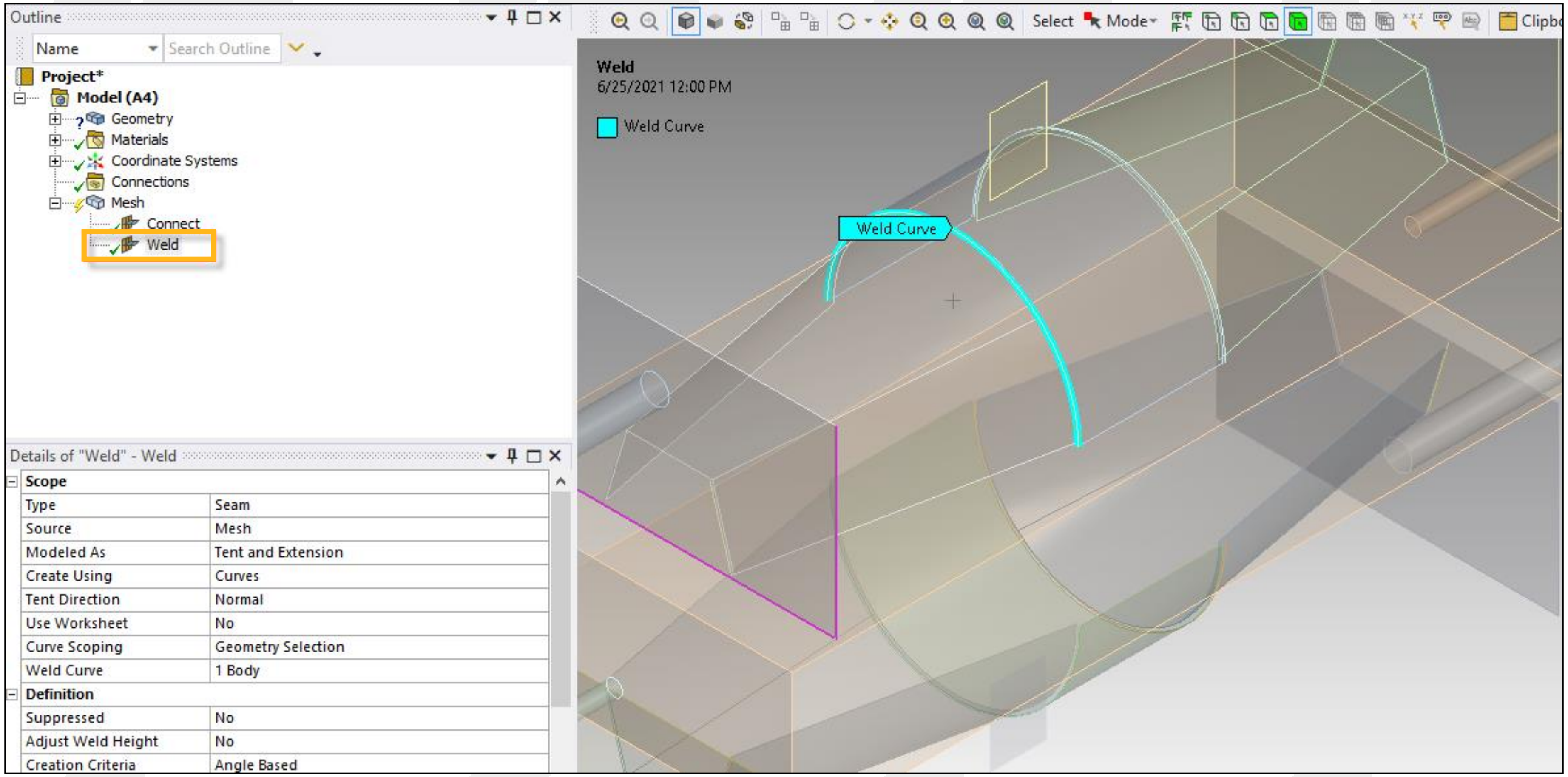




Ansys

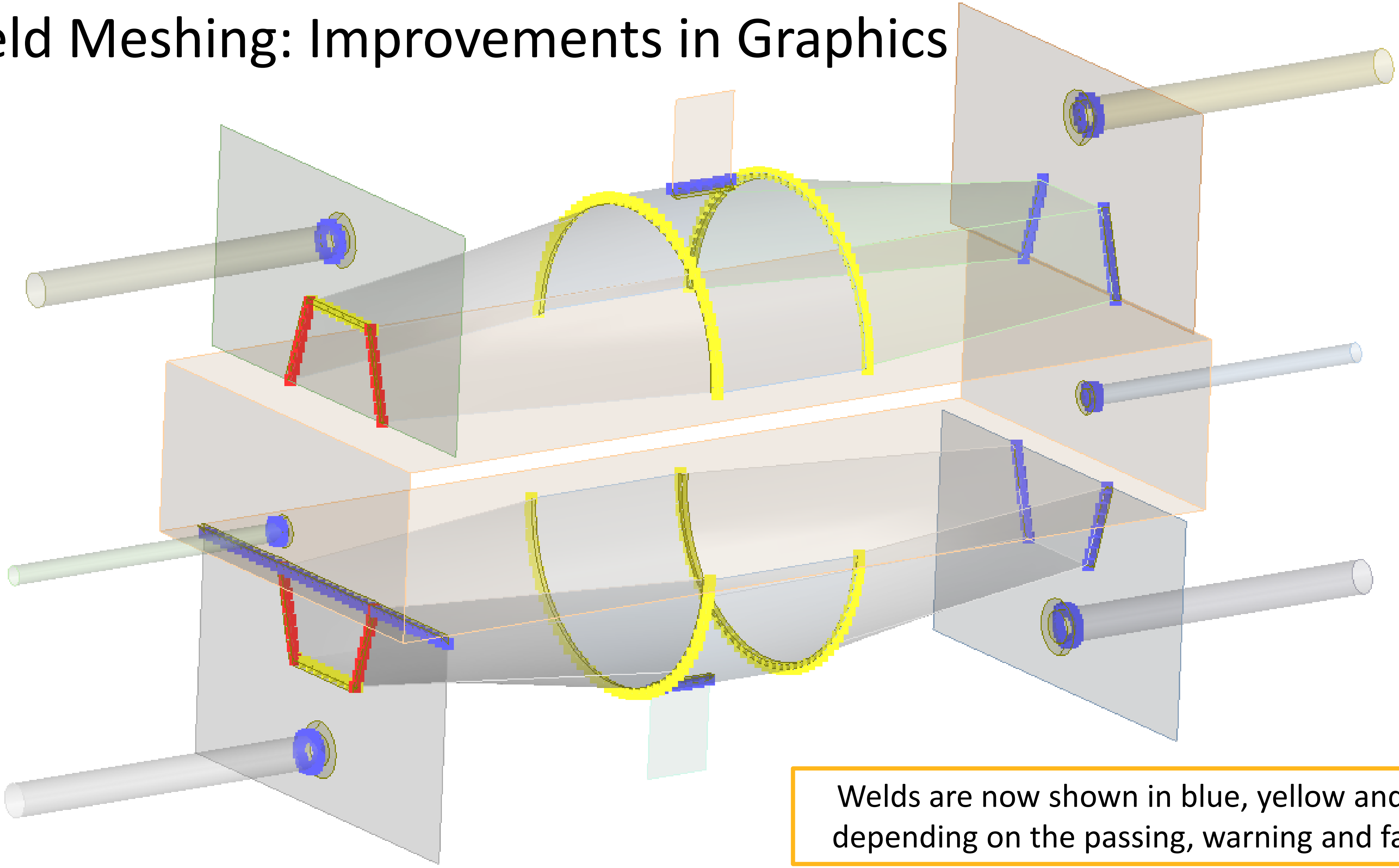
Highlighting Weld Curve with Annotations

- Select weld control object corresponding weld curve body is highlighted with annotation



Ansys

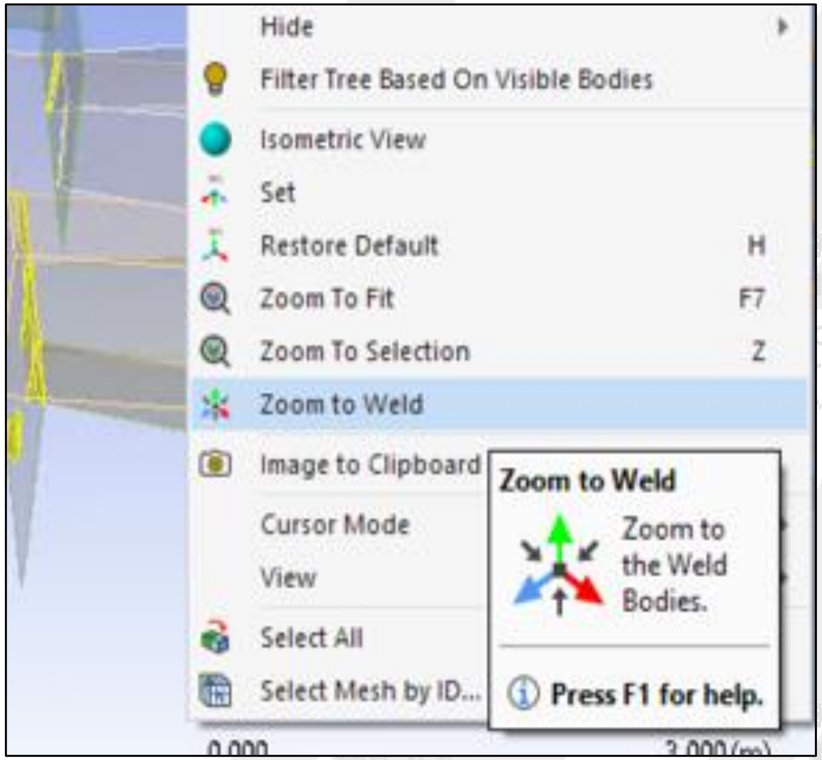
Weld Meshing: Improvements in Graphics

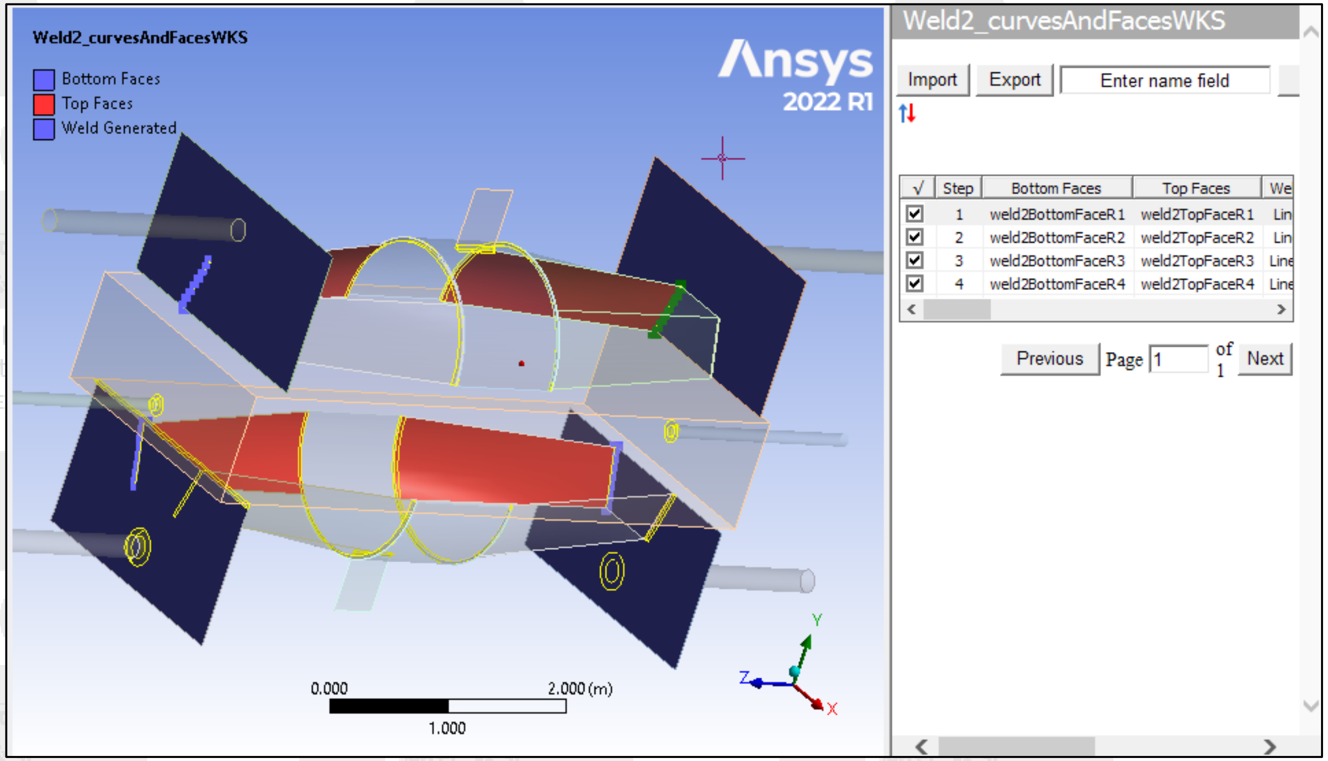


Welds are now shown in blue, yellow and red colors depending on the passing, warning and failing status

Weld Meshing: Improvements in Graphics

- “Zoom to Weld” option is added in the graphics context menu which zooms to fit the scoped references of the weld component. For the object using worksheet “Zoom to Weld” will be applied to the selected items in the worksheet
- For interaction with the worksheet, Picking and Highlight of the weld components can be done using the Imported Data Highlight button in the Graphics Toolbar

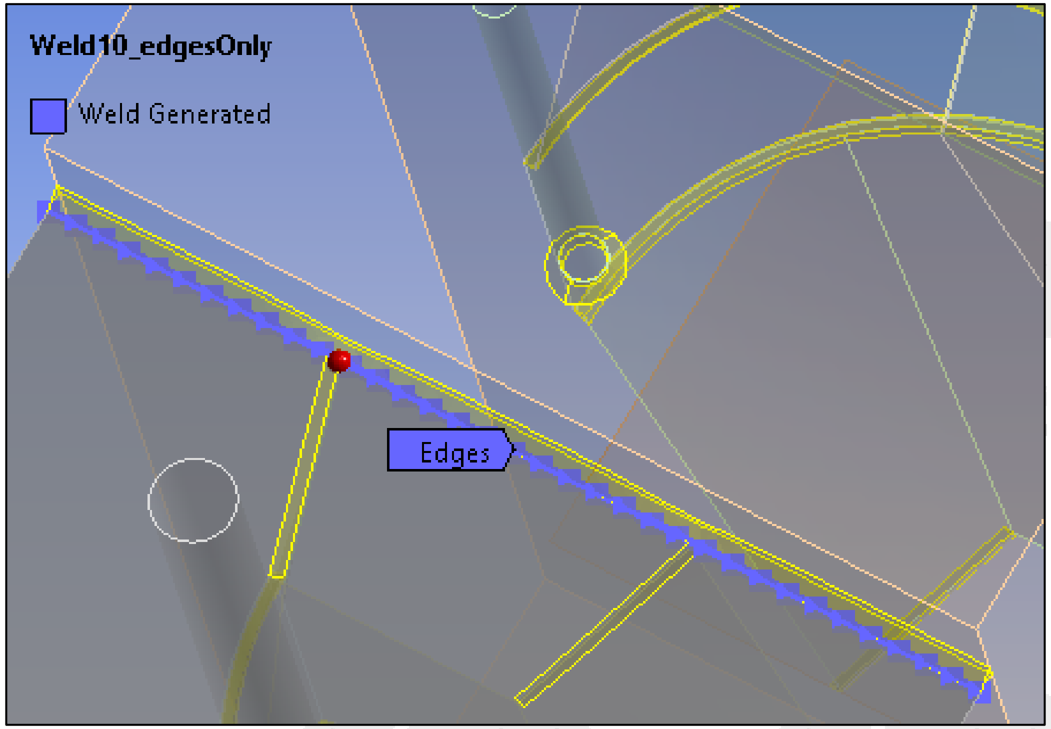


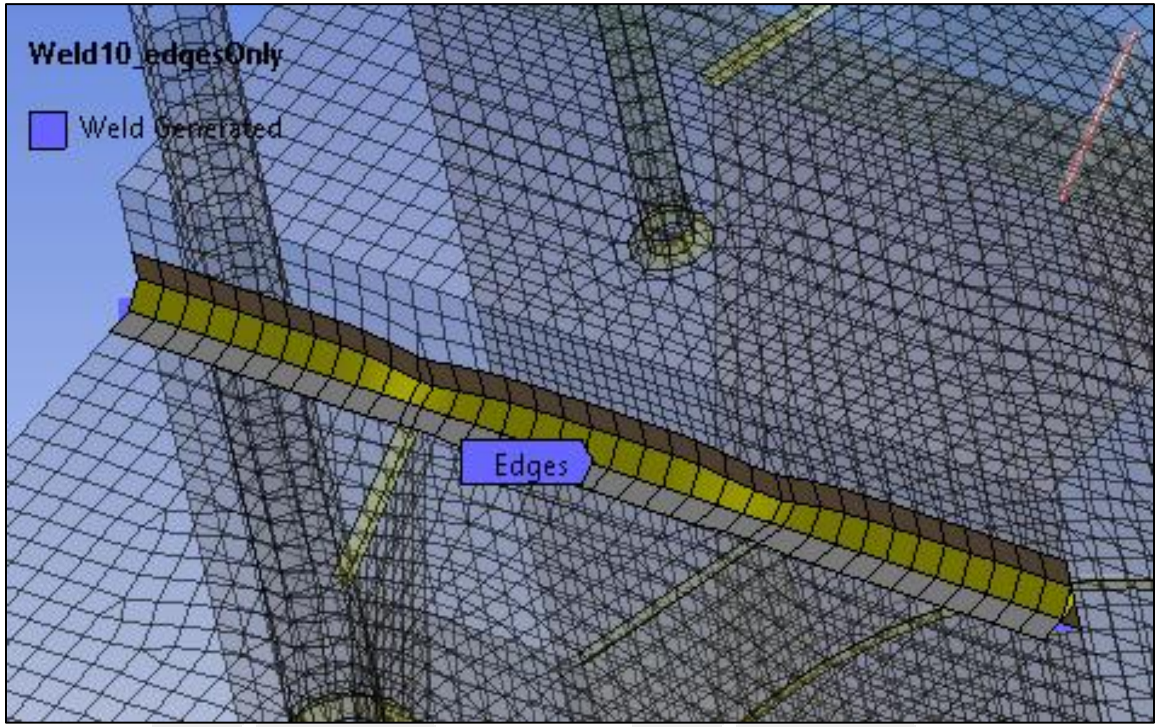


Ansys

Weld Meshing: Improvements in Graphics

- The geometry references for different weld types scoping are highlighted based on the Mesh States. Dots along the weld edges/curves is displayed which represent the edge mesh size
- Turn on “Show Mesh” displays the Weld and HAZ layer elements as solid and other elements as translucent

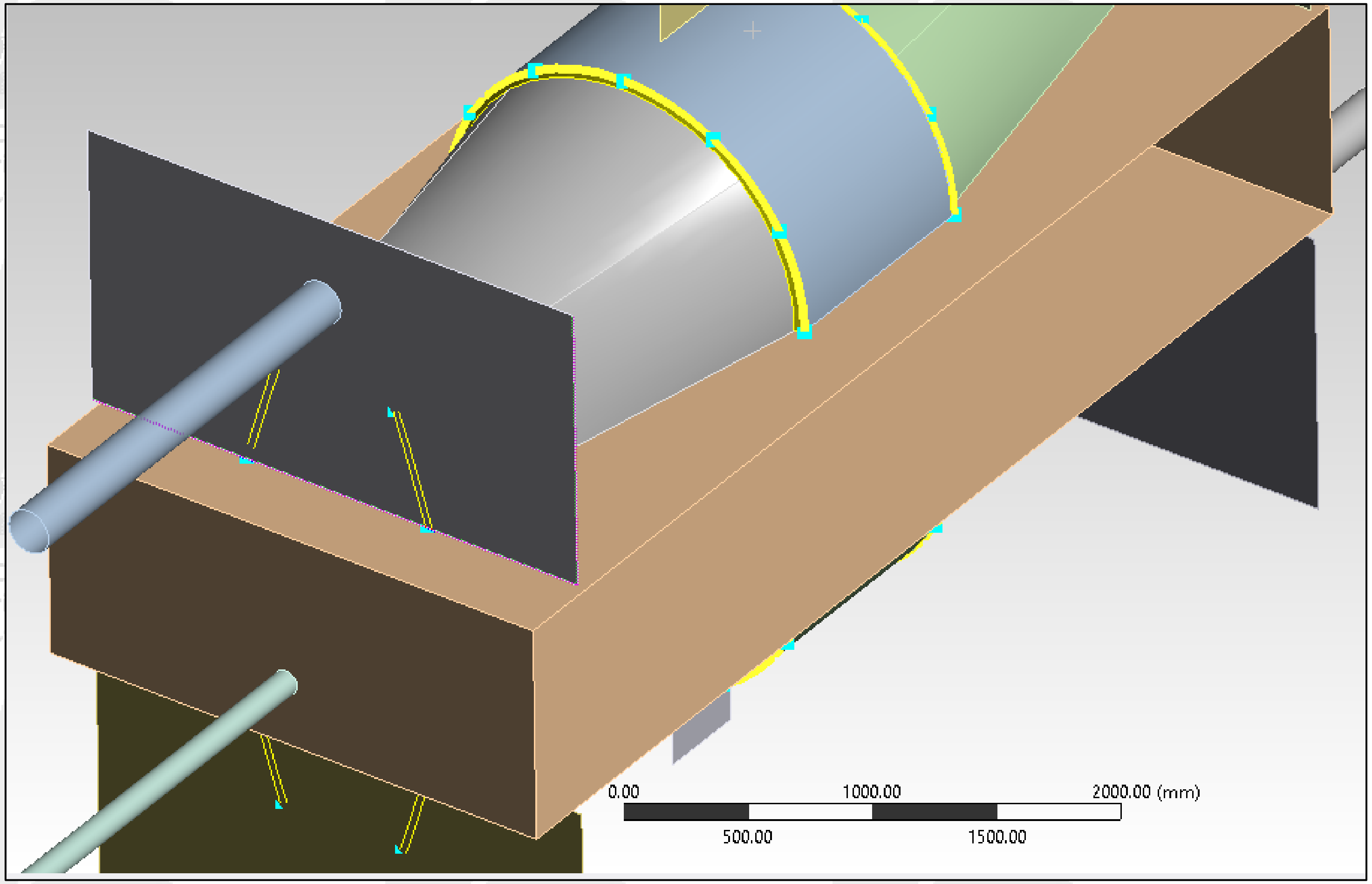




Ansys

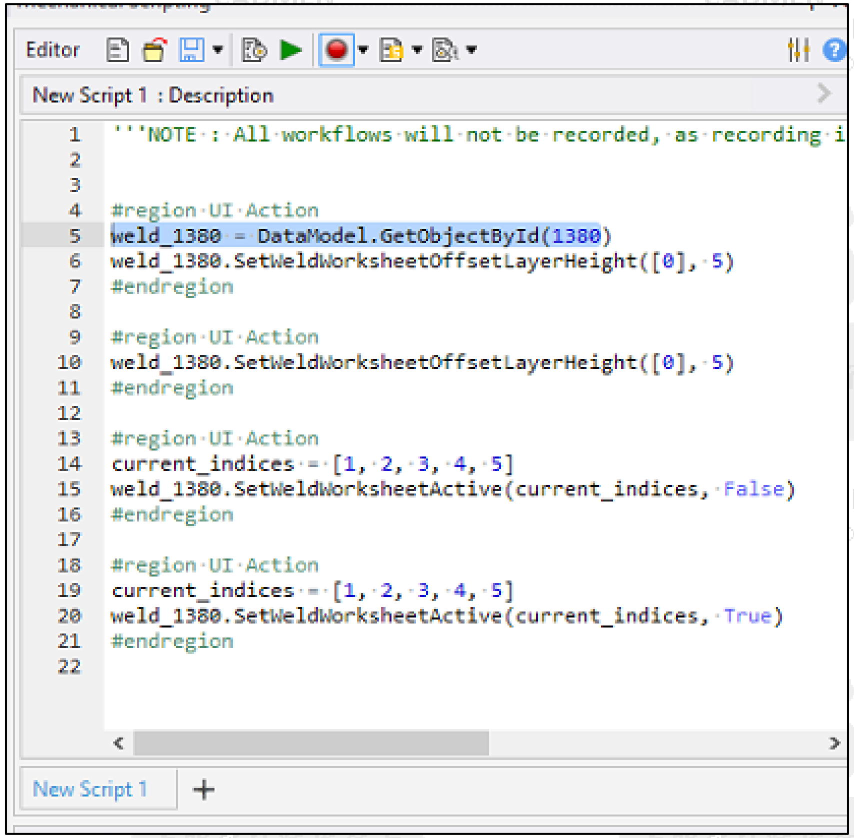
Preview Weld

- Weld faces are created without mesh
- Weld face failure or warning messages on preview is being implemented



Weld Meshing: Scripting and Recording with Worksheet Editing

- All functions related to worksheet editing can now be recorded for automation



Ansys

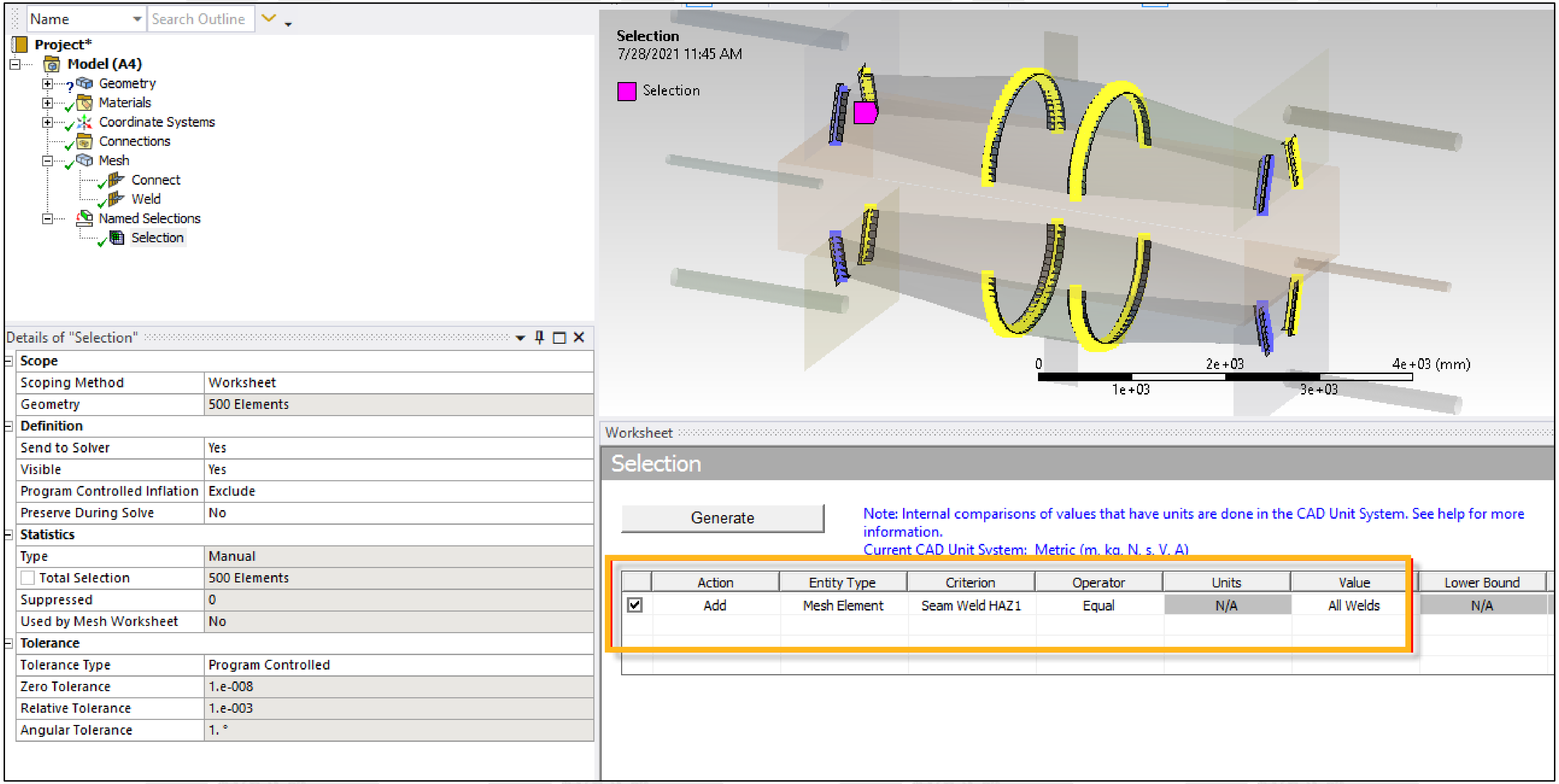
Weld Meshing: Editing of Multiple Cells for a Parameter Through Scripting

- Current worksheet has limitations in selecting/editing multiple cells in a column
- The new scripting capability can help us to overcome the limitation

Here is a code snippet to show how we can change offset layer height for row #2 to 6

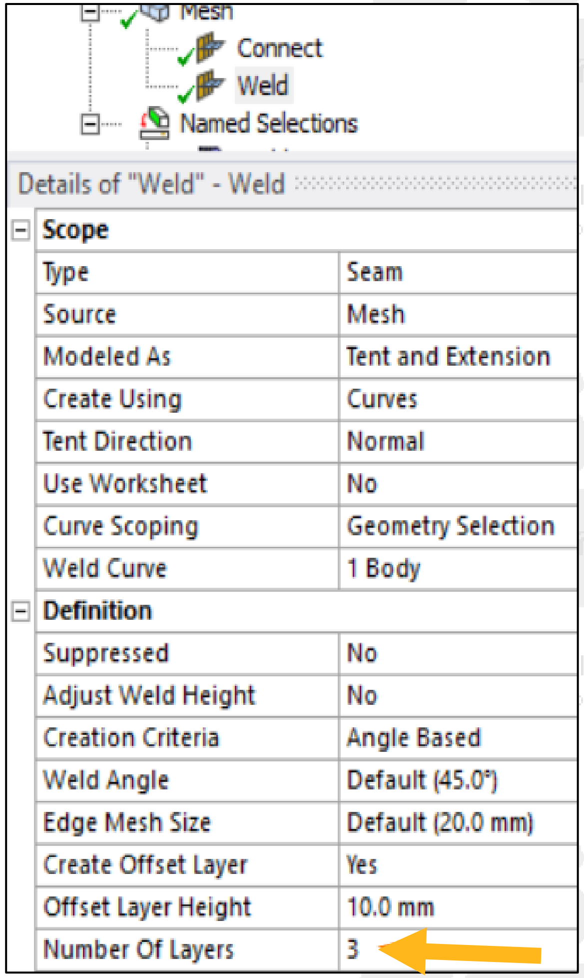
```
current_indices_test = [1, 2, 3, 4, 5]
weld_1380 = DataModel.GetObjectById(1380)
weld_1380.SetWeldWorksheetOffsetLayerHeight(current_indices_test, 4)
```

Weld Meshing: Criteria Based HAZ Named Selection

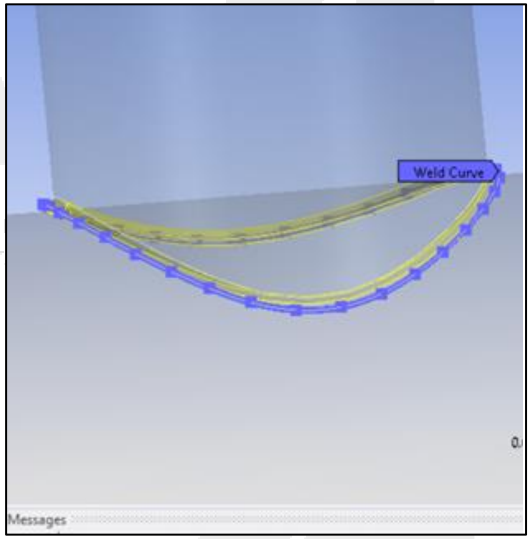


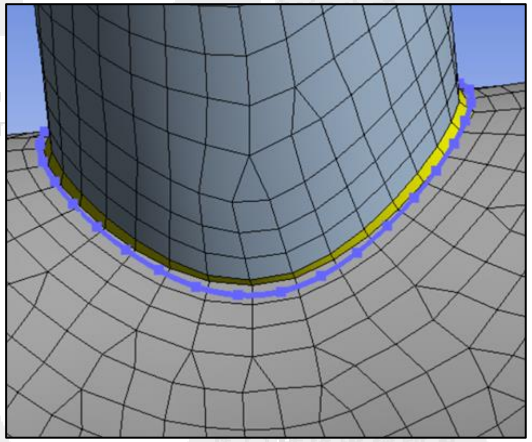
Ansys

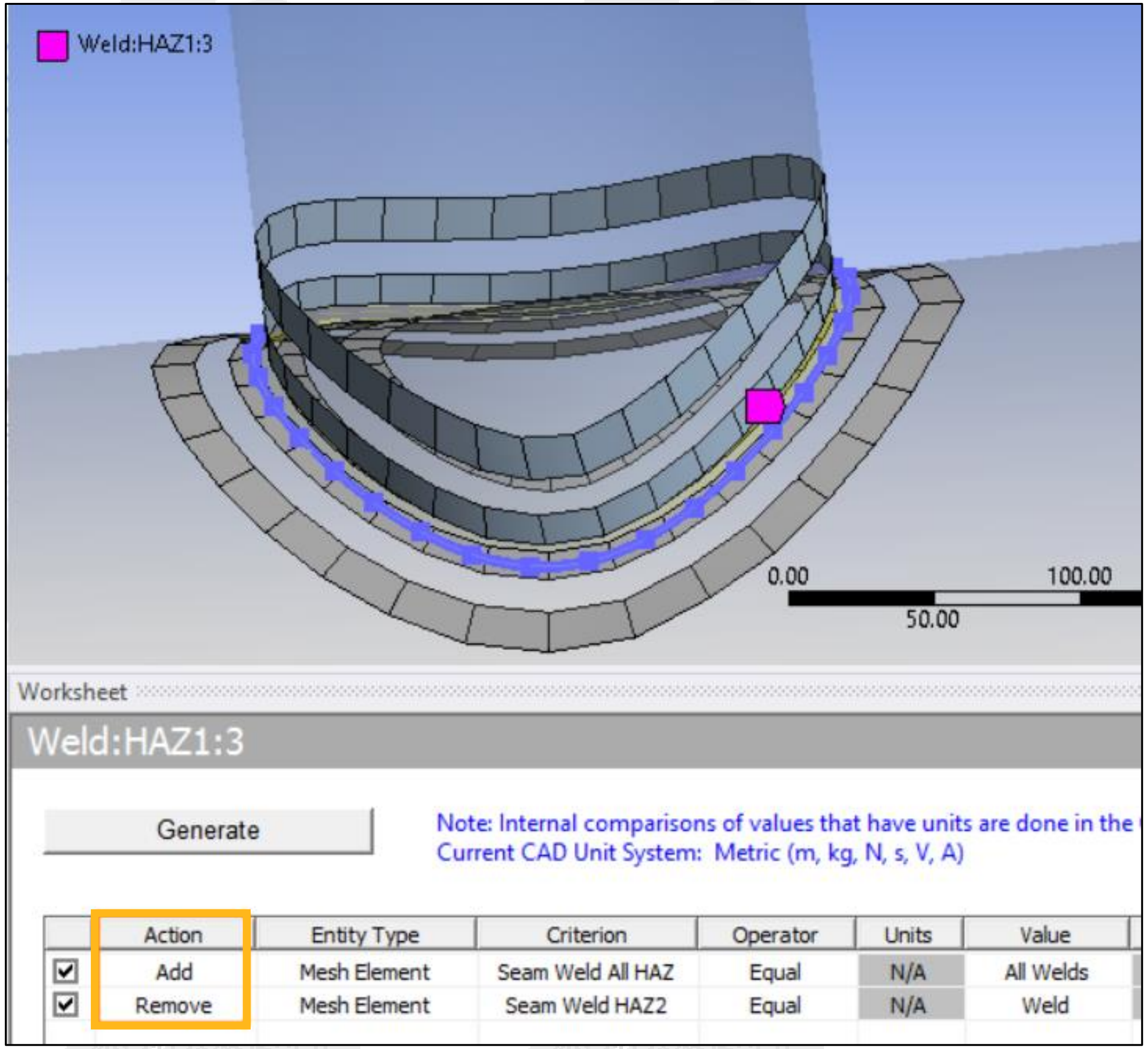
Weld Meshing: Criteria Based HAZ Named Selection



Scope	
Type	Seam
Source	Mesh
Modeled As	Tent and Extension
Create Using	Curves
Tent Direction	Normal
Use Worksheet	No
Curve Scoping	Geometry Selection
Weld Curve	1 Body
Definition	
Suppressed	No
Adjust Weld Height	No
Creation Criteria	Angle Based
Weld Angle	Default (45.0°)
Edge Mesh Size	Default (20.0 mm)
Create Offset Layer	Yes
Offset Layer Height	10.0 mm
Number Of Layers	3

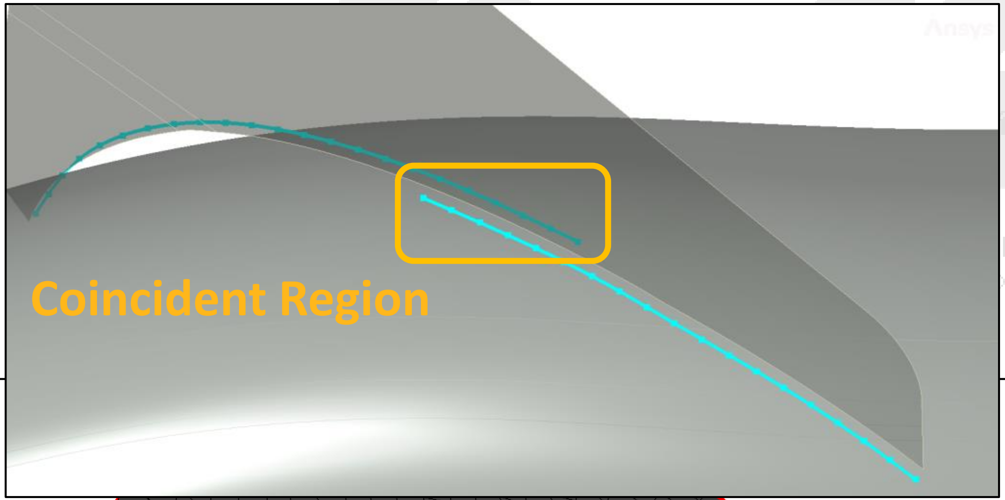


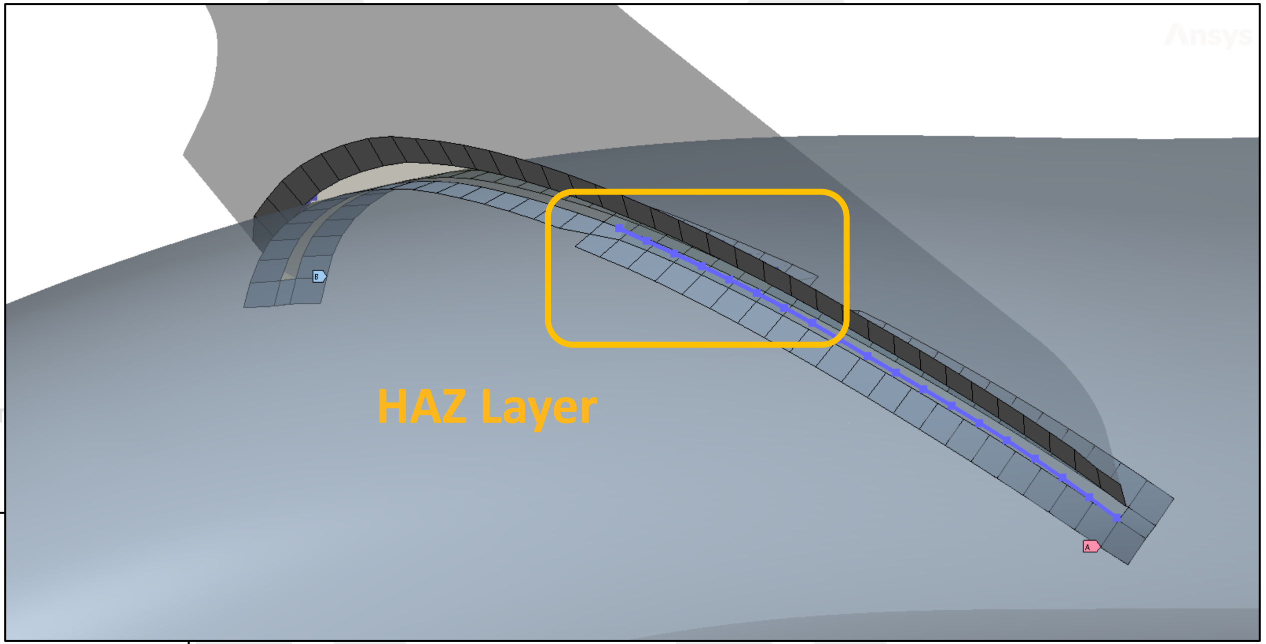


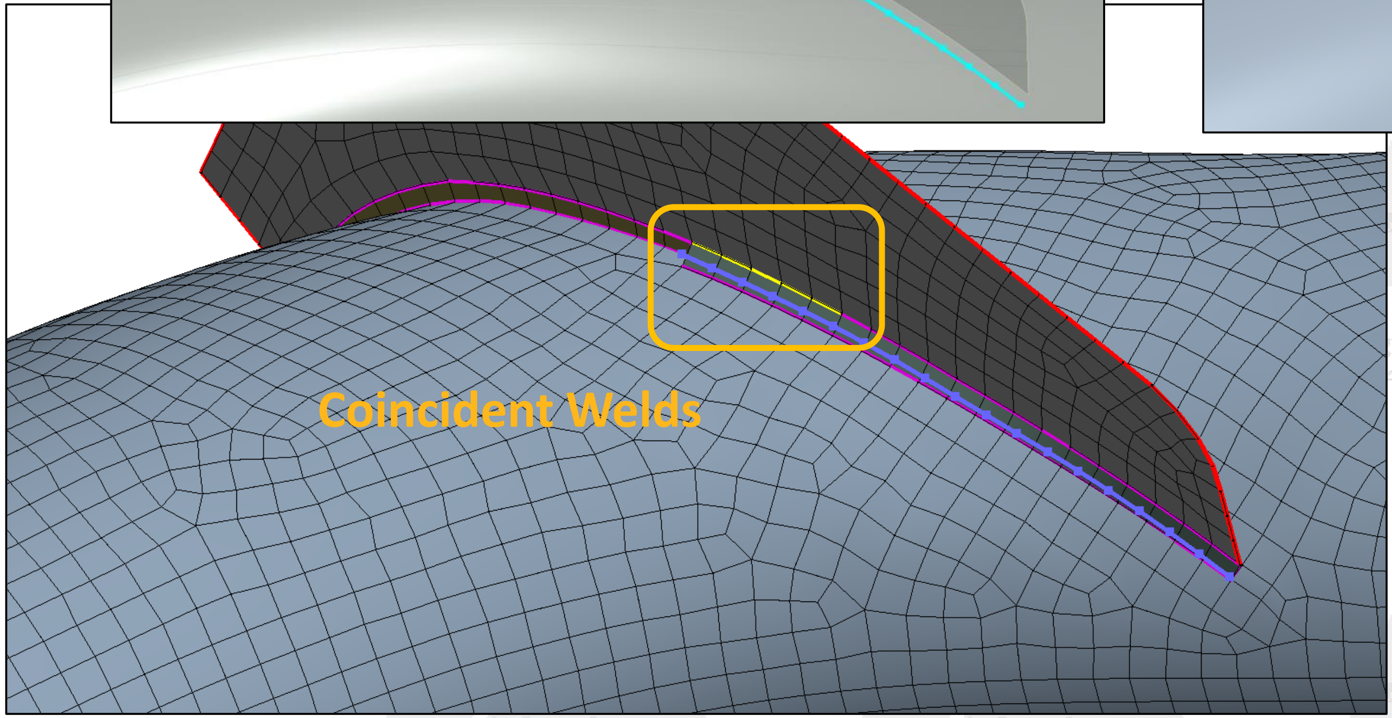


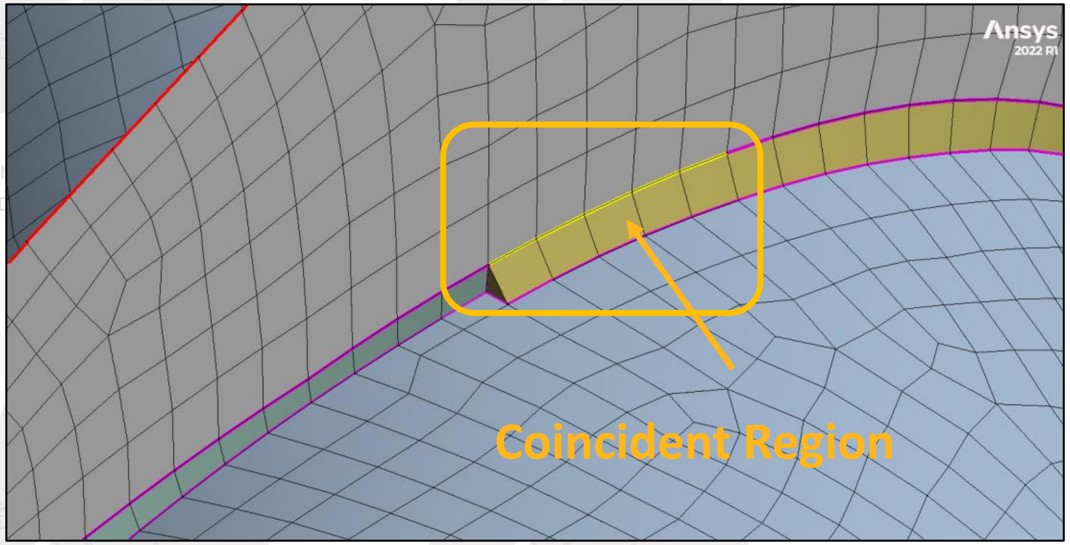
	Action	Entity Type	Criterion	Operator	Units	Value
☑	Add	Mesh Element	Seam Weld All HAZ	Equal	N/A	All Welds
☑	Remove	Mesh Element	Seam Weld HAZ2	Equal	N/A	Weld

Coincident Welds



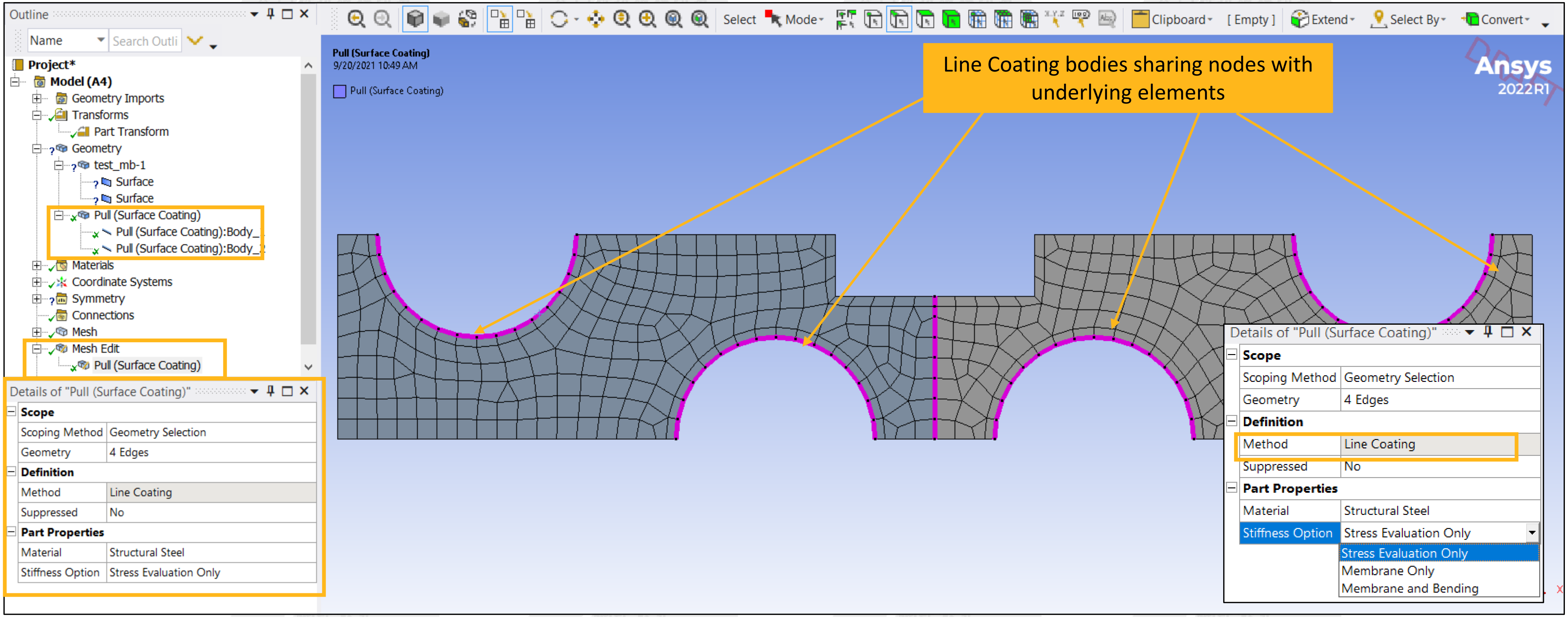




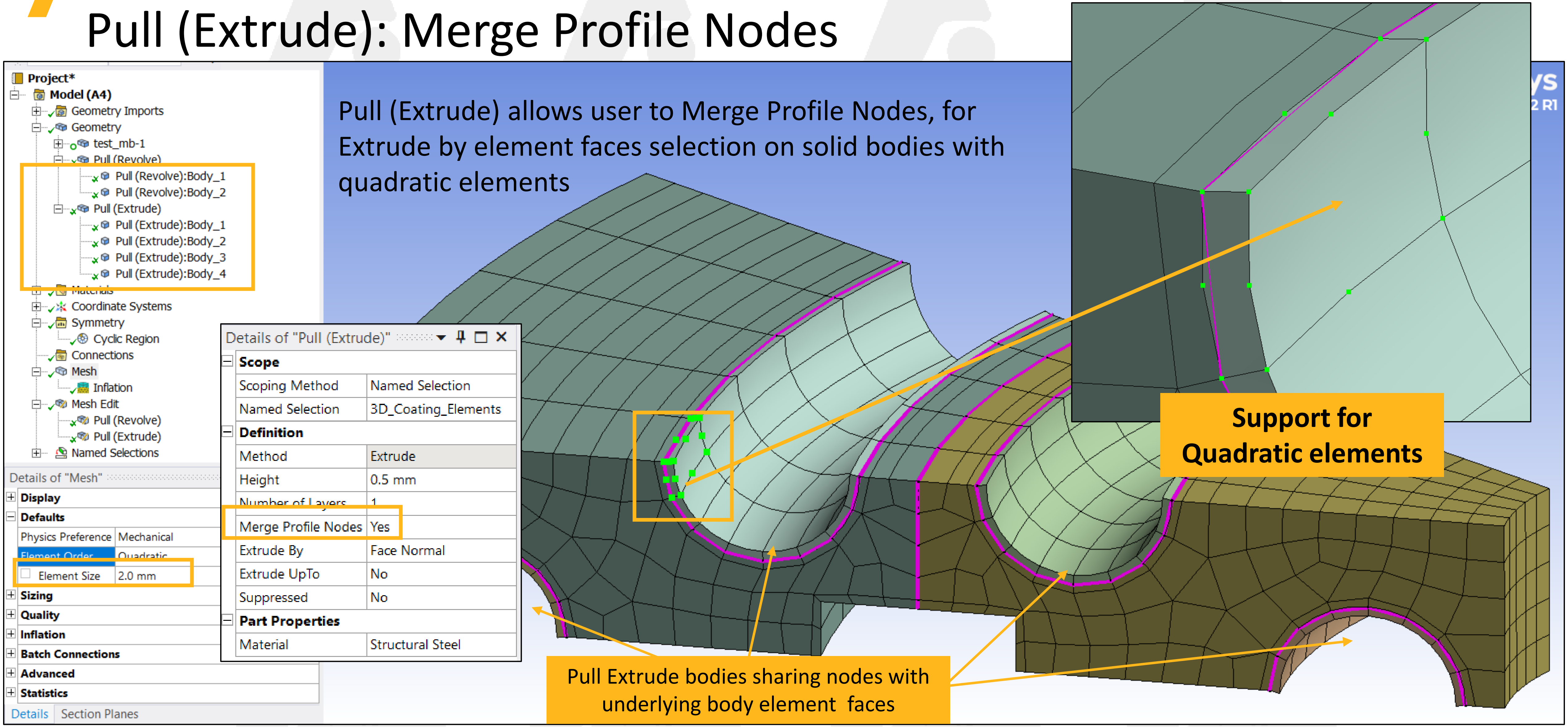


Pull (Line Coating)

- Pull (Line Coating) allows user to create a "Line Coating" Line bodies using the Boundary Edges of 2D Axysymmetric bodies. The Line Coating bodies share nodes with the underlying elements

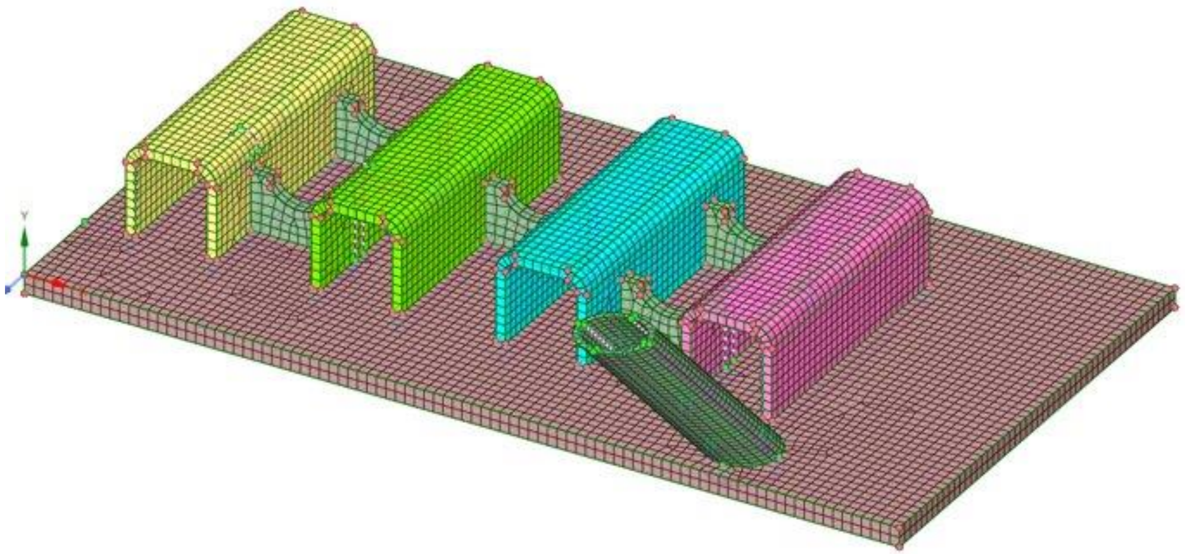


Pull: Support for Quadratic Elements (Curvilinear Mid-Nodes)
Pull (Extrude): Merge Profile Nodes



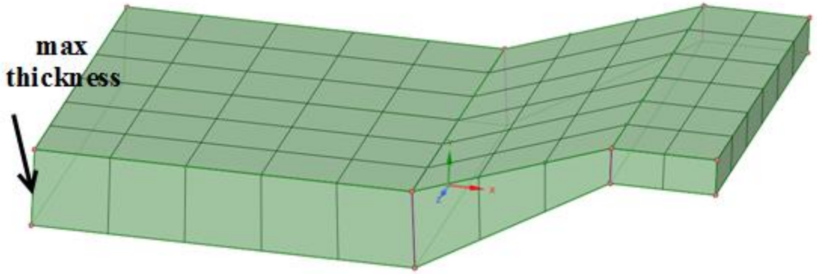
Spaceclaim Meshing: Overview

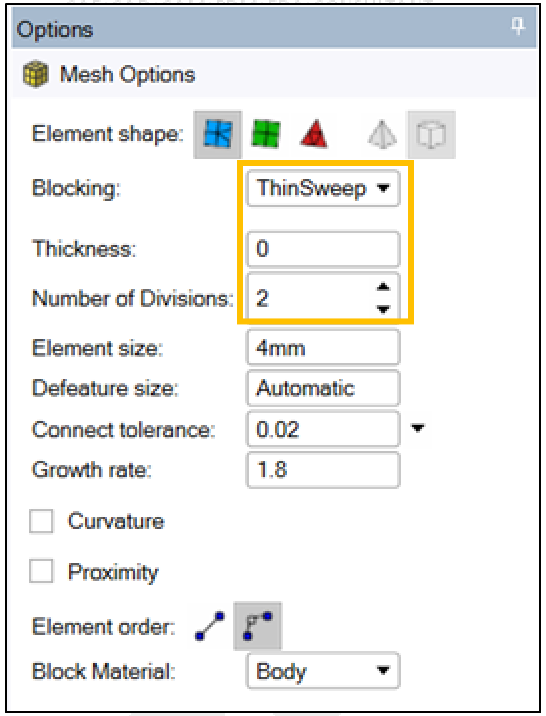
- Thin Body Meshing
  - Full Release of ThinSweep Block Decomposition
- Robustness, Performance, Usability
  - Pull improvements
  - New Mapping Options at block controls level
  - Improved quality for All Quadrilateral method
  - Improved performance
    - Faster "activate" for meshing in SpaceClaim
    - Faster surface meshing of circuit board type models
    - Faster model transfer to Ansys Mechanical
  - Improved robustness for external documents
  - Better support for *.cdb/*.inp files
- Meshing for Explicit Improvements
  - New Explicit Physics option (Beta)
  - Better uniformity of mesh
  - Characteristic Length calculation consistent with LS-DYNA
  - Better support for LS-DYNA *.k files



Spaceclaim Meshing: ThinSweep Blocking

- Full release of ThinSweep option which was previously Beta
- Set number of divisions across thickness and (optionally) thickness of material
- For models with varying thickness, specify a Thickness value slightly more than the maximum thickness of the model.



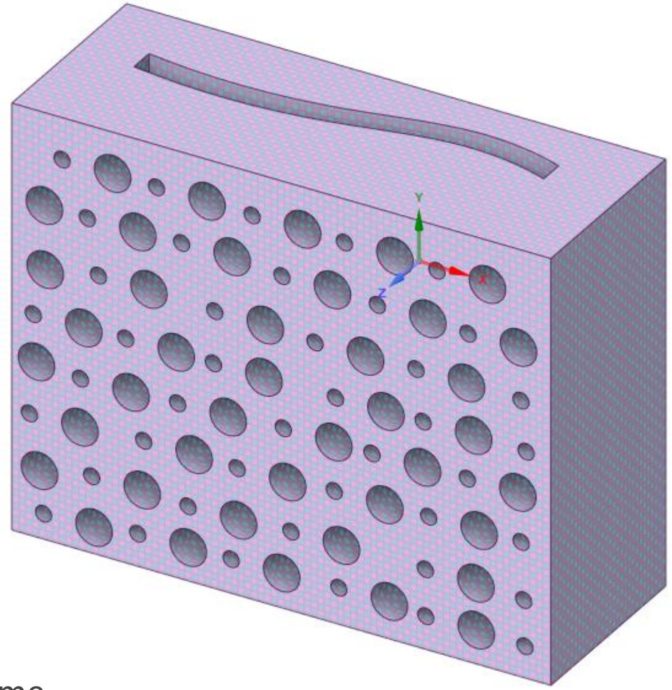


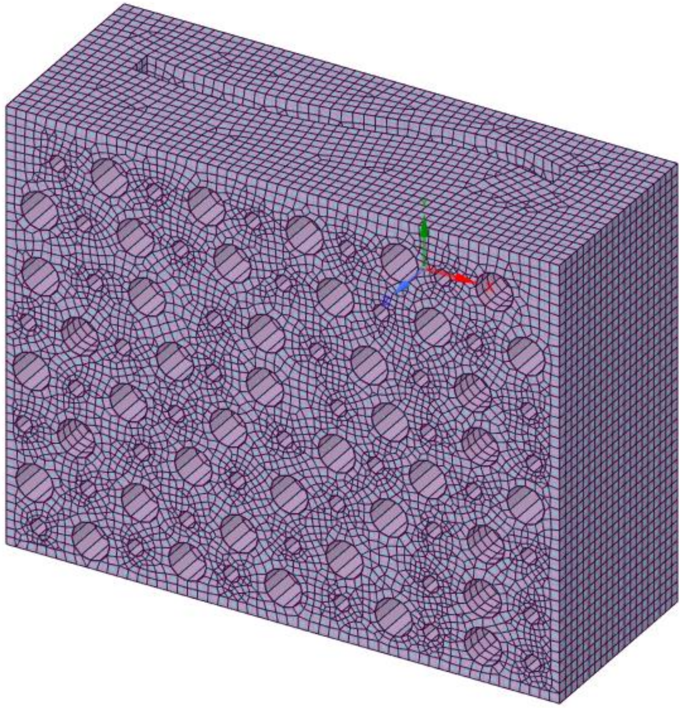
Ansys

Spaceclaim Meshing: Surface Meshing Performance

New option: Prime surface meshing at local control level

Apply faster Prime Surface meshing by Toggling ON the “Use Prime” option





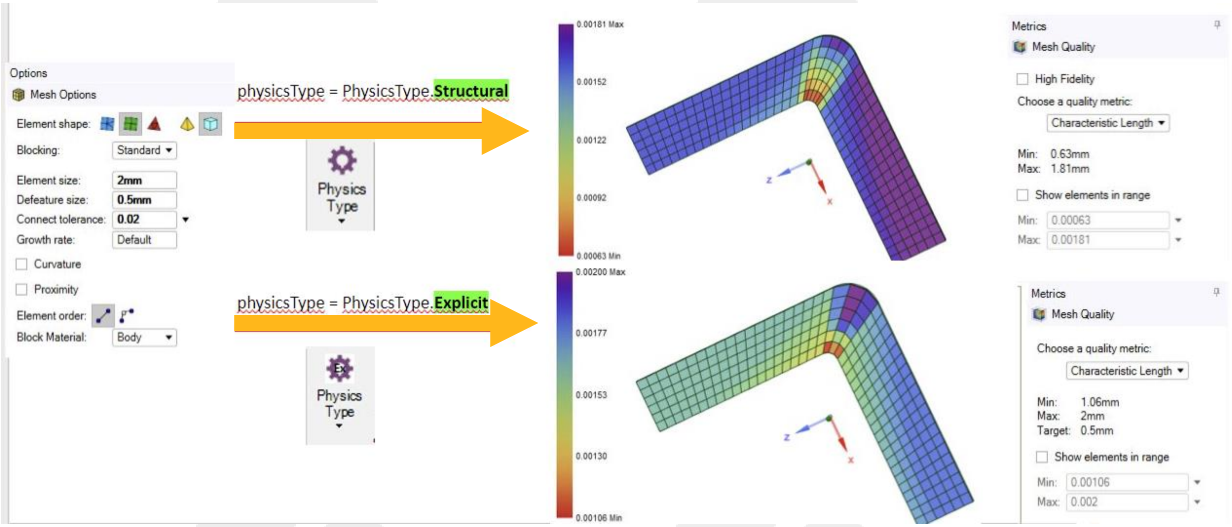
Geometry Surfaces
options.Method = MapMeshMethod.QuadsPrime

Block Faces:
options.ConvertType = ConvertBlockType.ToFreeQuadDom
options.ElementShape = ElementShapeType.PrimeQuadDominant

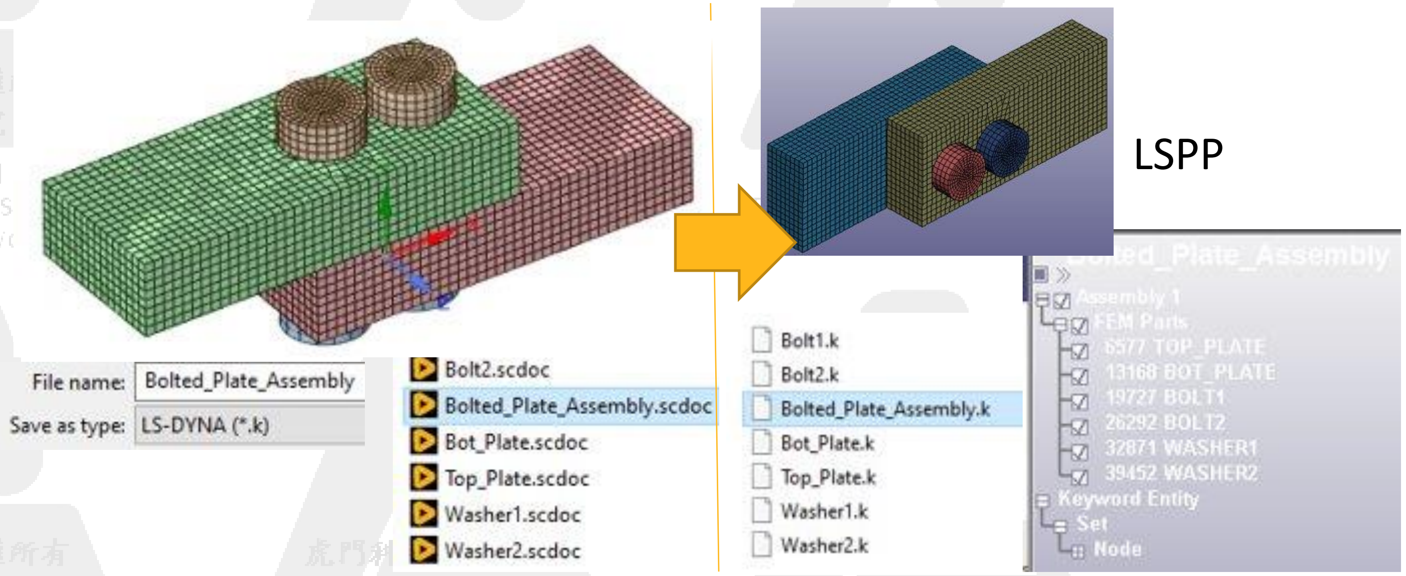
Ansys

Spaceclaim Meshing: Explicit

- New Explicit Physics preference in dropdown
  - Heavier default defeaturing tolerance
  - Target characteristic length and optimisation for better uniformity



- Better support for /input *include files
  - Export .k assembly and include file



Ansys

Explicit Meshing

Tetrahedral Meshing

Mesh Diagnostics

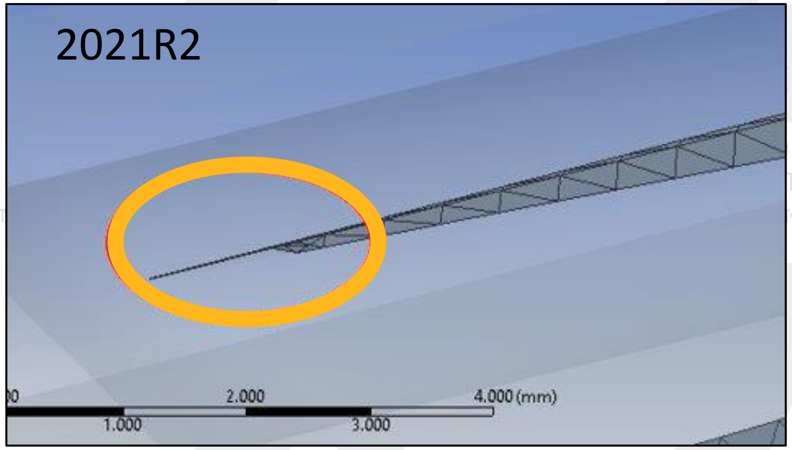
Feature Based Meshing

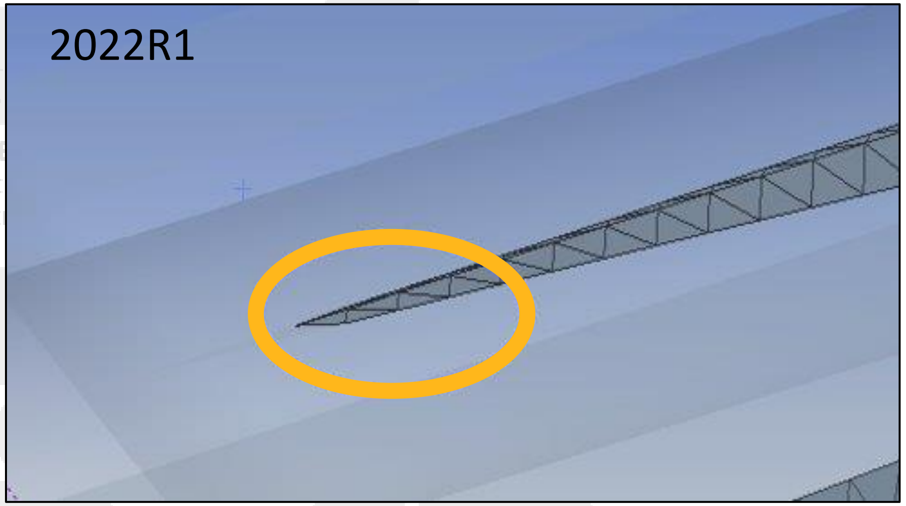
Ansys

Improved Robustness for Patch Conforming Tetrahedra

- Increasing the defeaturing tolerance avoids small elements but also causes more instability in the meshing algorithms

- In 2022 R1, many improvements to robustness have been made for heavy defeaturing required by Explicit users

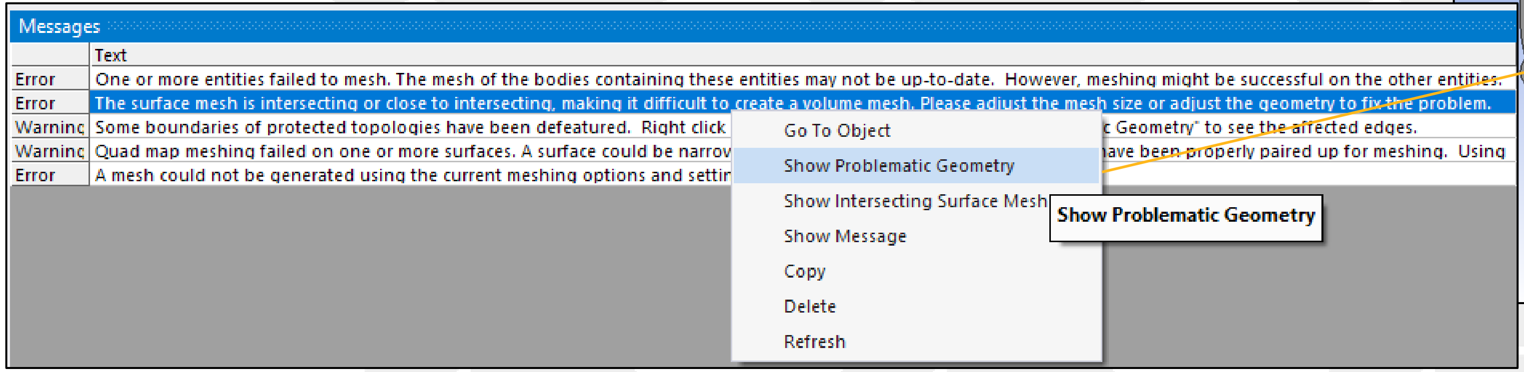


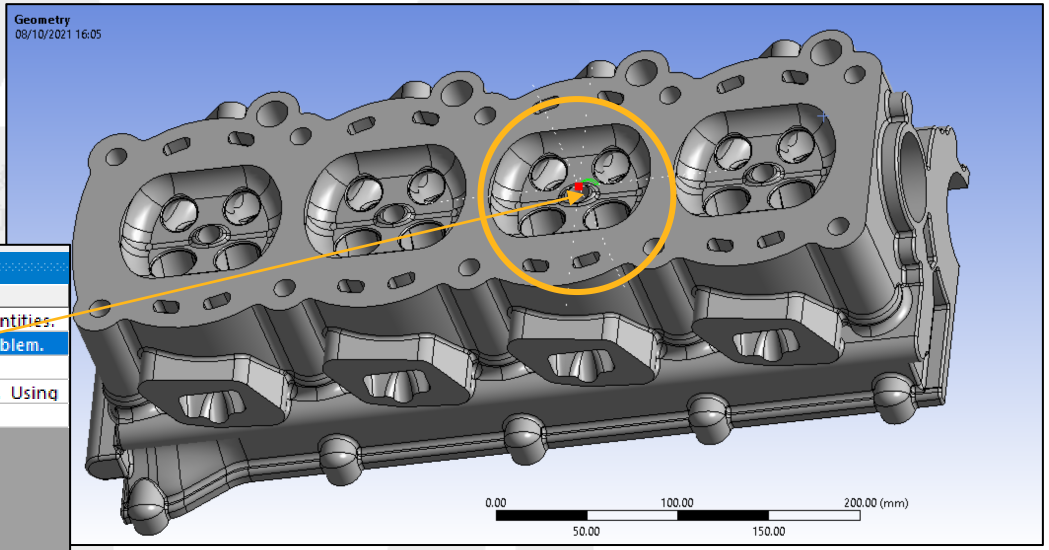


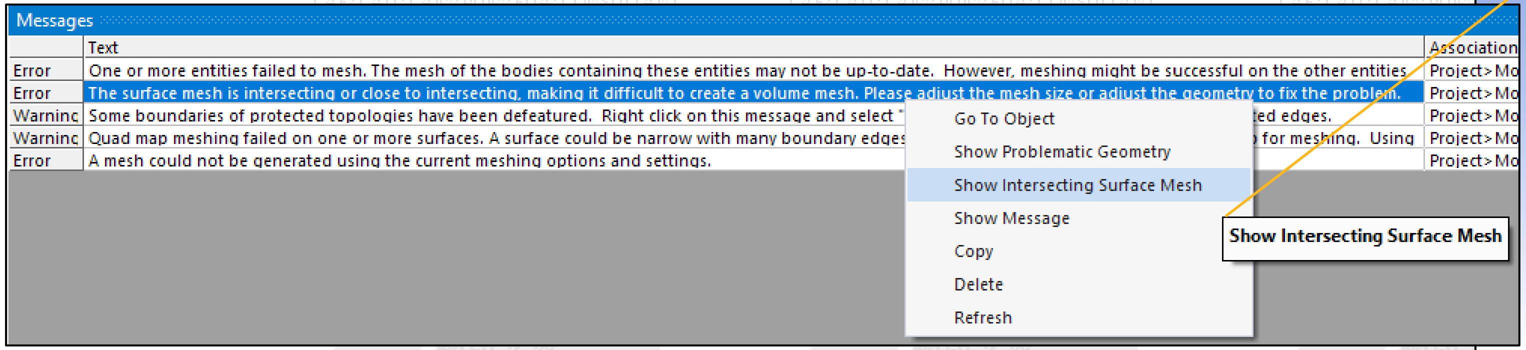
Ansys

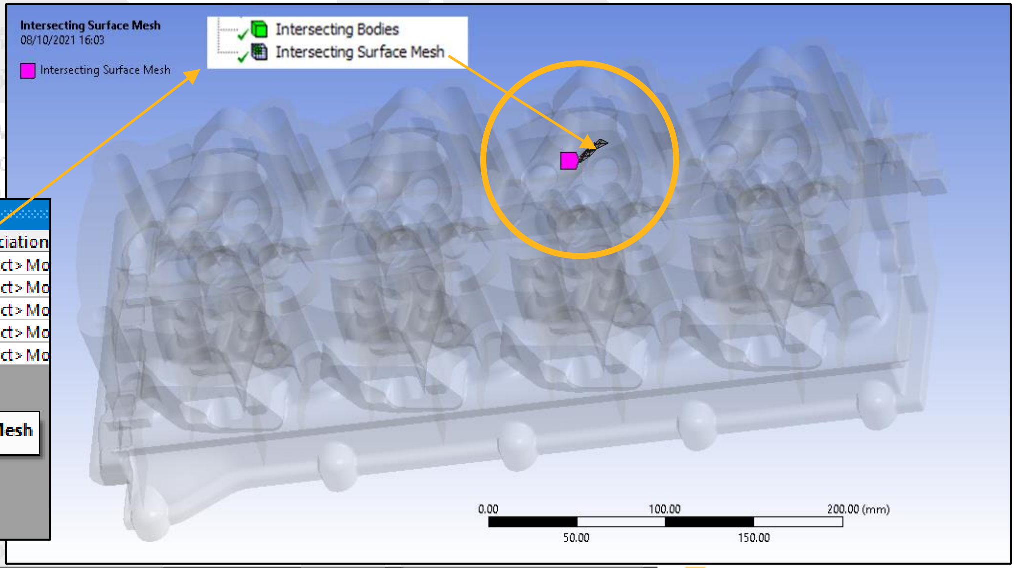
Improved Error Handling

- Intersecting Surface Mesh RMB Options





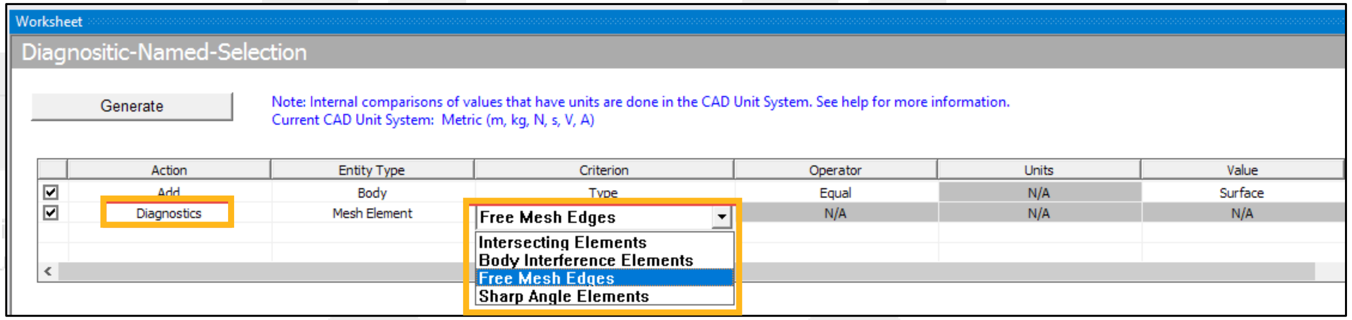


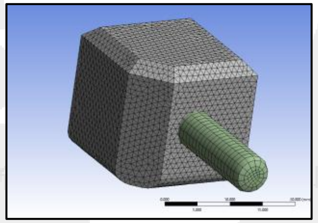


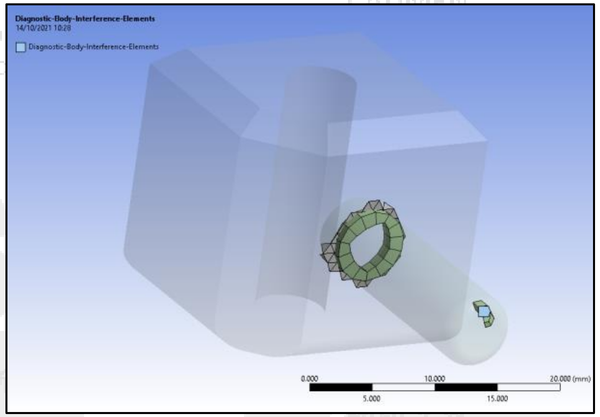
Ansys

Diagnostics Tools

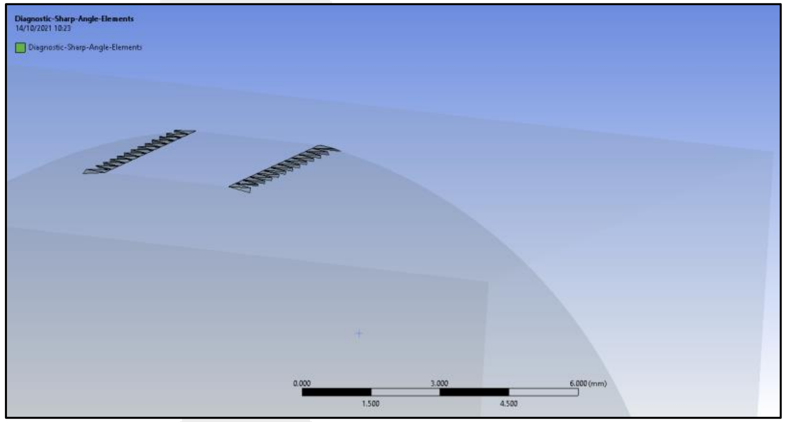
- Use worksheet Named Selection to select bodies and then run Diagnostics for visualisation of the problem
- New tools to find issues and fix via additional settings or return to geometry tool for modifications
- Options available at 2022 R1:
  - Mesh Element:
    - **Intersecting surface mesh failures**
    - **Free edge mesh**
    - **Sharp angle**
    - **Body Interference**
  - Topology
    - **Defeatured Faces**







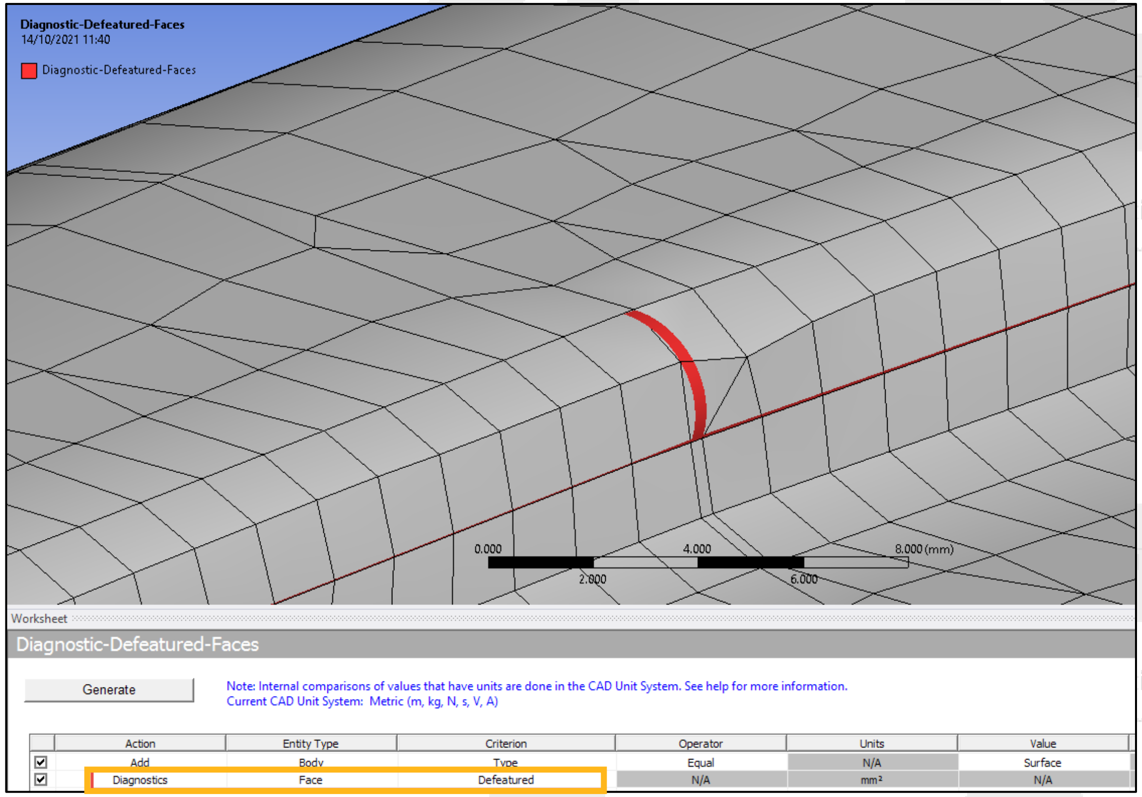
Body Interference

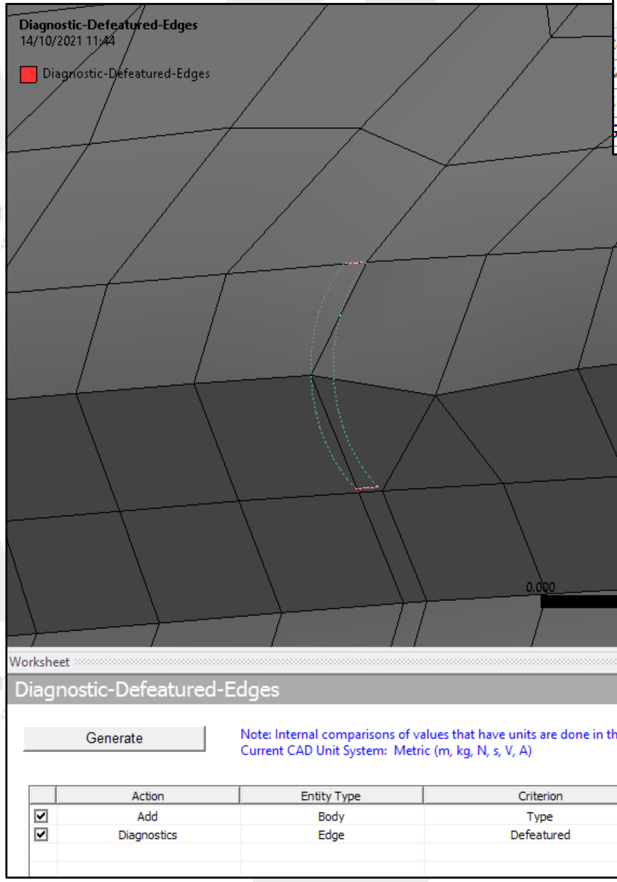


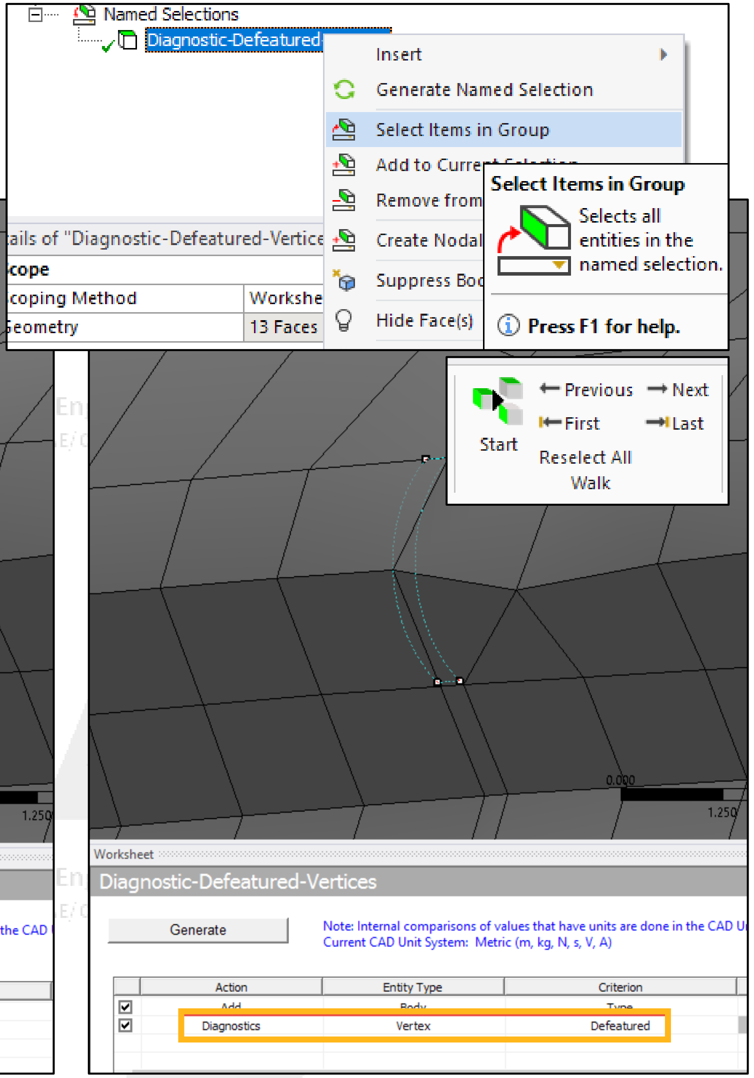
Sharp Angle Elements

Ansys

Mesh Diagnostics Tools: Defeaturing Checks





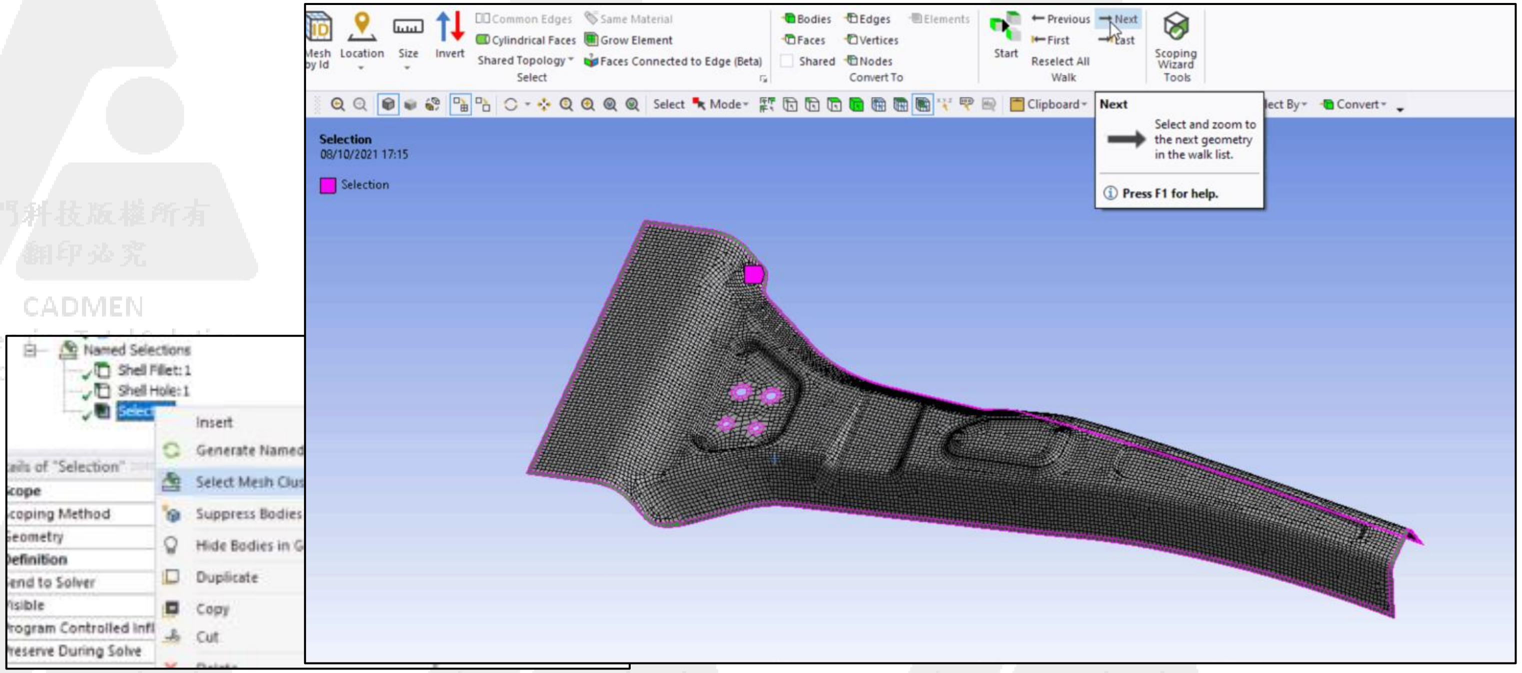


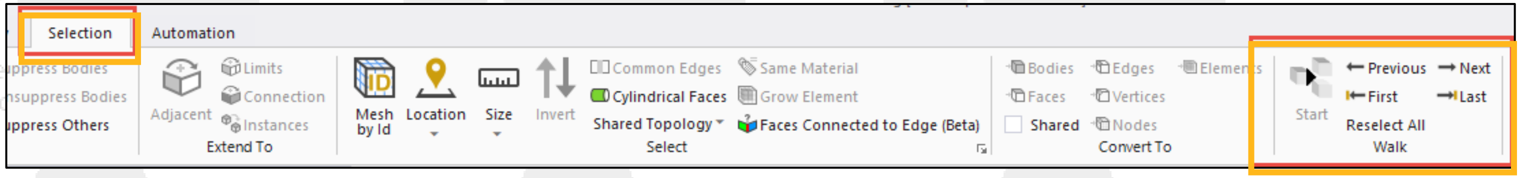
- Detect and examine where defeaturing has occurred after meshing
- Use Selection Walk to traverse defeatured faces

Ansys

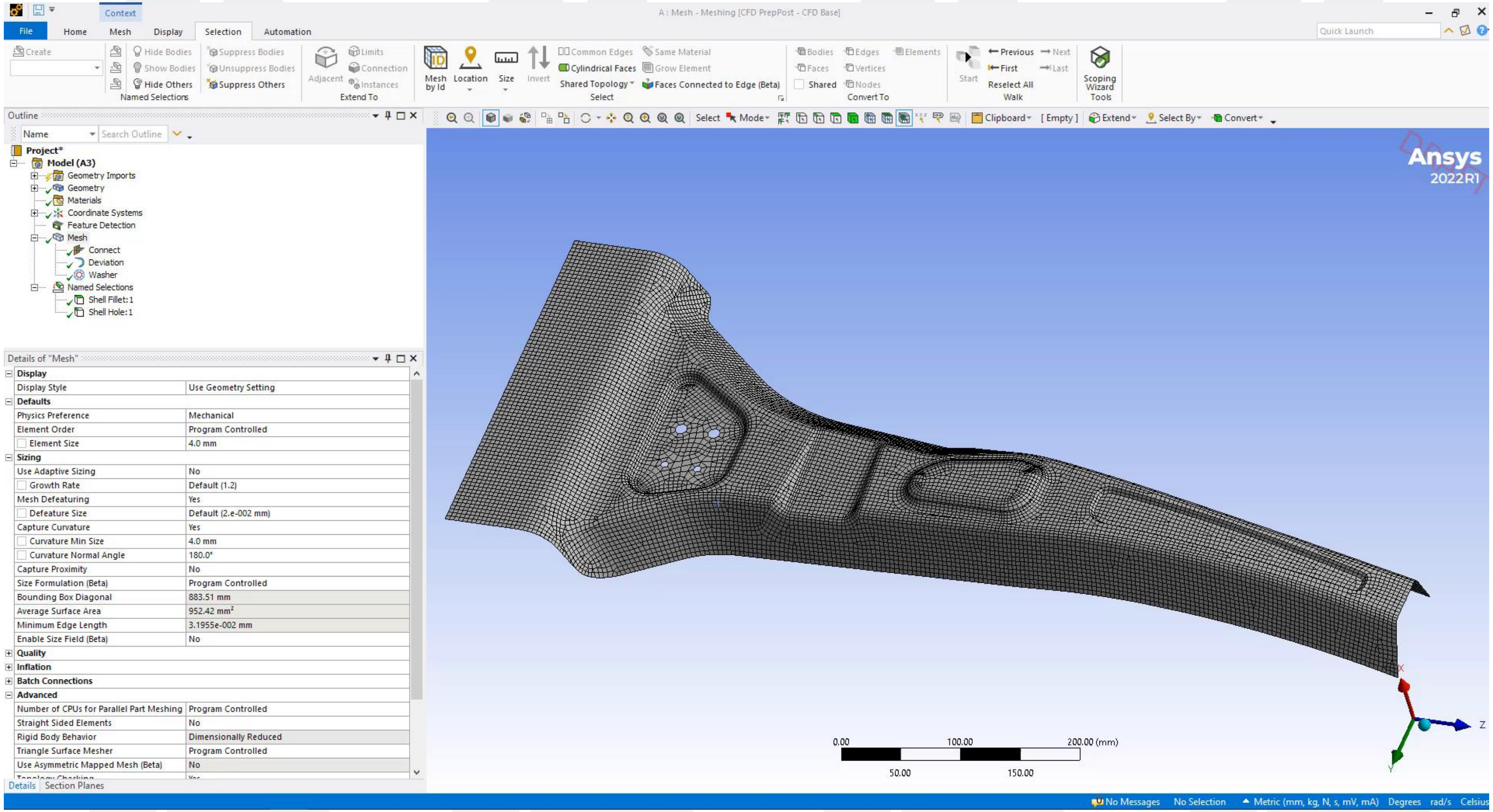
Mesh Cluster Walk

- Model Walk has been enhanced to walk through
  - Mesh Elements
  - Mesh Element Clusters
- Useful for new diagnostics tools to traverse multiple issues
- New RMB option on Element based NS from Diagnostics
  - Select Mesh Clusters in Group





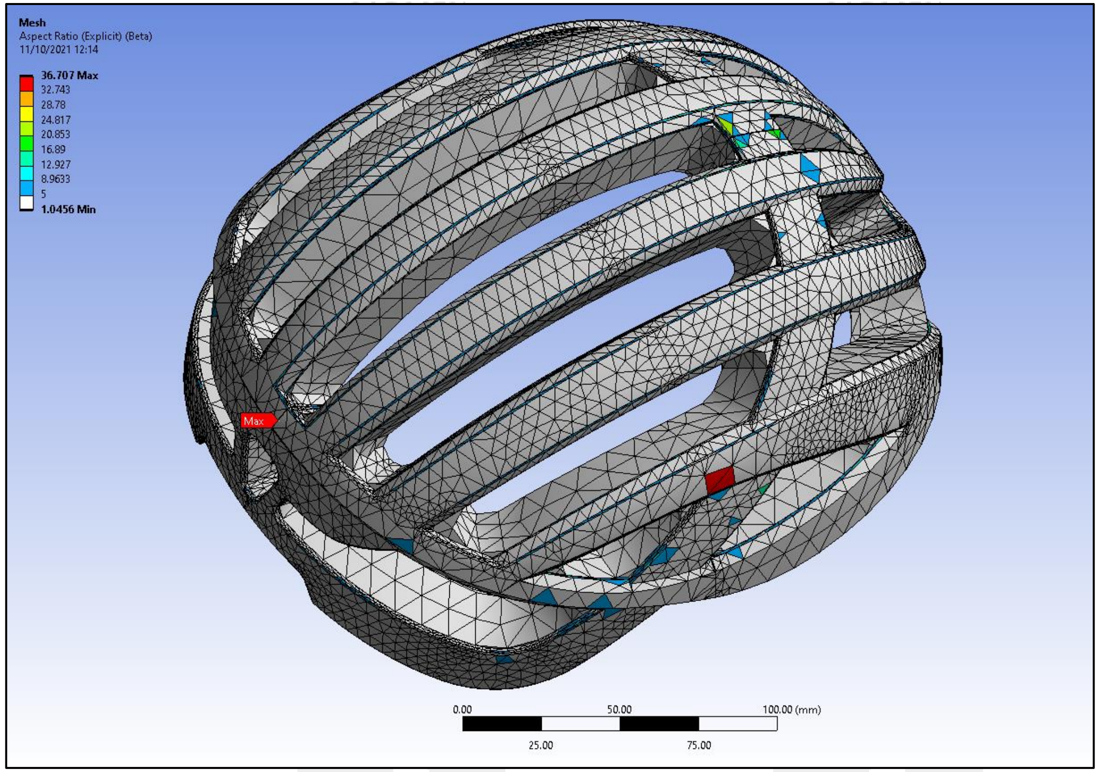
Mesh Cluster Walk: Short Demo with Free Face Diagnostics



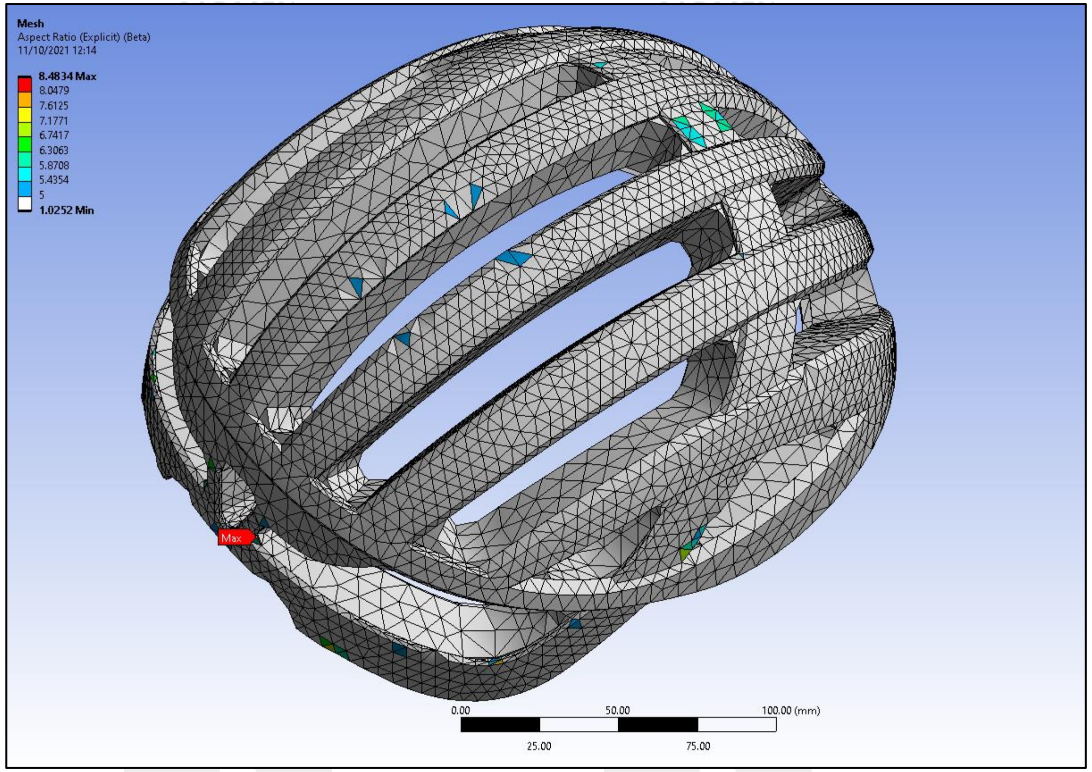
Default Surface Mesher Explicit Physics Pref. = Adv. Front

Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	
Rigid Body Behavior	Full Mesh
Triangle Surface Mesher	Advancing Front

- **More uniform and smooth triangular surface mesh**



Previous Default Surface Mesh



New Default Surface Mesh

Ansys

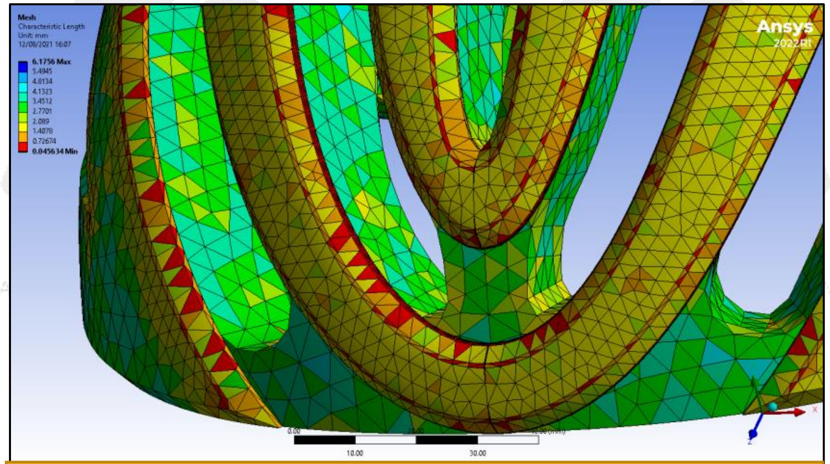
Tet Meshing Aspect Ratio Targeting (Explicit Physics Pref.)

- Aspect Ratio based meshing criteria drastically reduces the max. Aspect Ratio (AR)
- This help improve the Characteristic Length (CL)significantly which has a big impact of the Explicit CFL Time-Step $(\Delta t = \frac{Characteristic\ Length}{Speed\ of\ Sound})$
- Help run the analysis without much mass-scaling

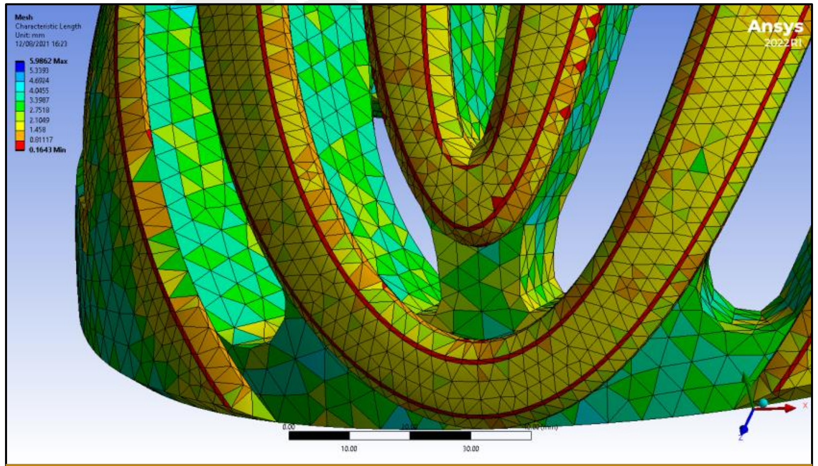
⊟ Quality	
Check Mesh Quality	Yes, Errors and Warnings
☐ Target Element Quality	Default (0.200000)
☐ Target Characteristic Length (LSDyna)	Default (0.5 mm)
☐ Target Aspect Ratio (Explicit)	5.

Challenges:

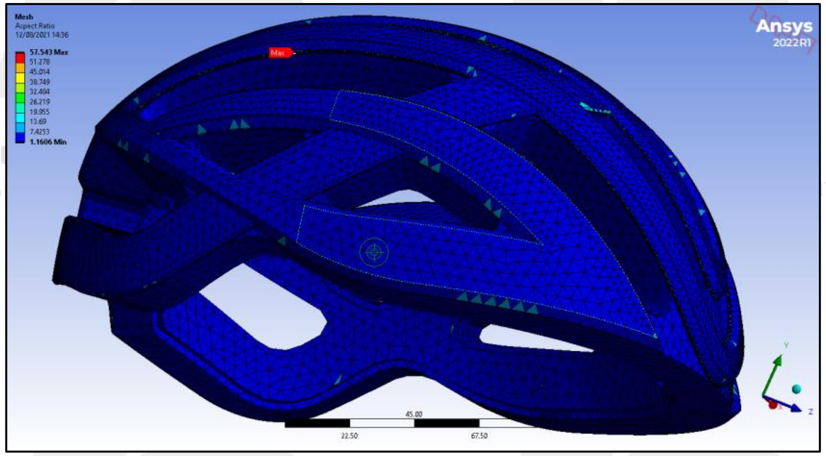
- Bike Helmet Example: A complex geometry with several intricate features

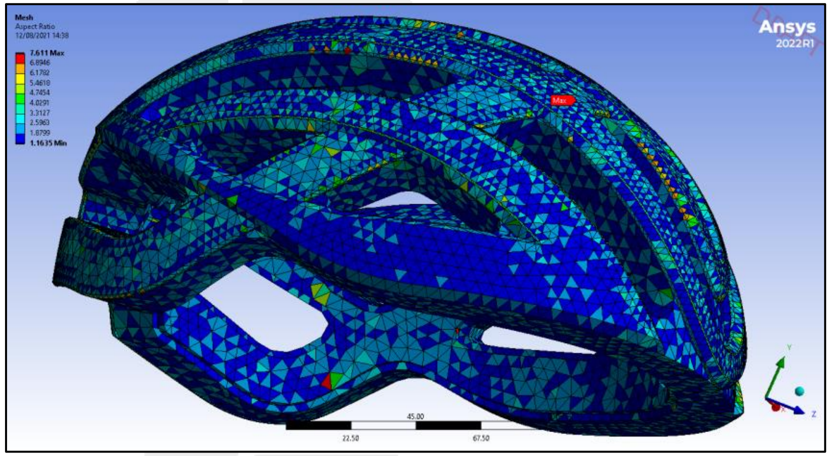


2021 R2 – Max AR=53, Min CL=0.05mm



2022 R1 – Max AR=7.6, Min CL=0.16mm



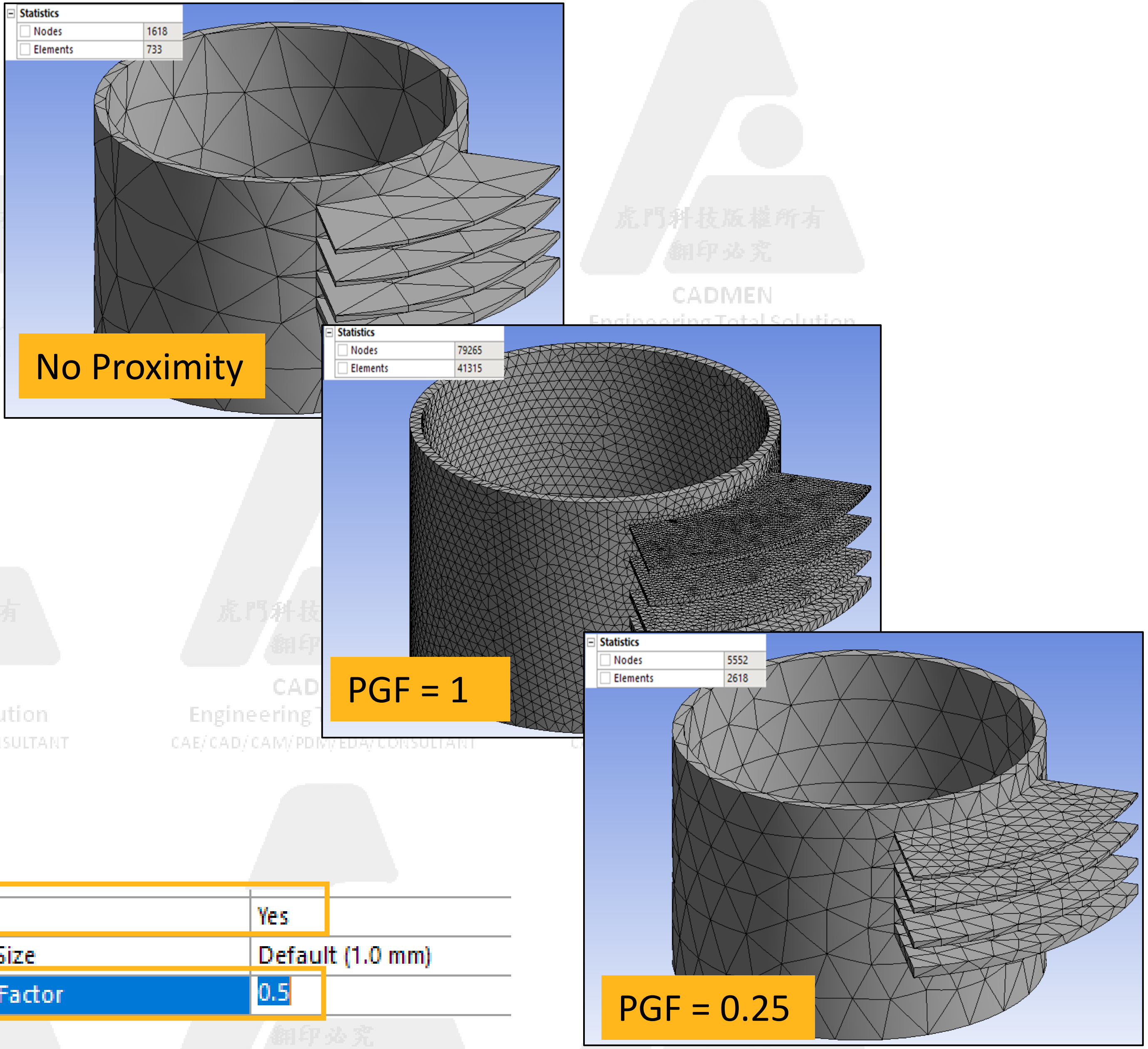


BEFORE **AFTER**

Ansys

Thin Solids

- Thin solids offer specific challenges:
  - How to avoid very high AR elements without increasing cell count considerably?
- Proximity Gap Factor now allows user to control mesh size across thin regions without requirement for full isotropic element (Non integer is allowed)
- e.g. Use Proximity Gap Factor = 0.25 to aim for aspect ratio of ~4 in thin regions



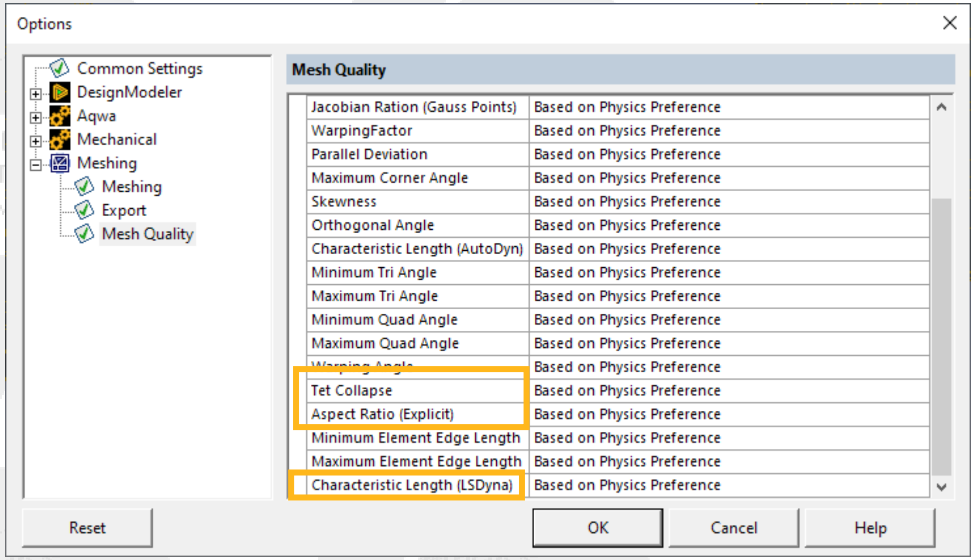
Capture Proximity	Yes
Proximity Min Size	Default (1.0 mm)
Proximity Gap Factor	0.5

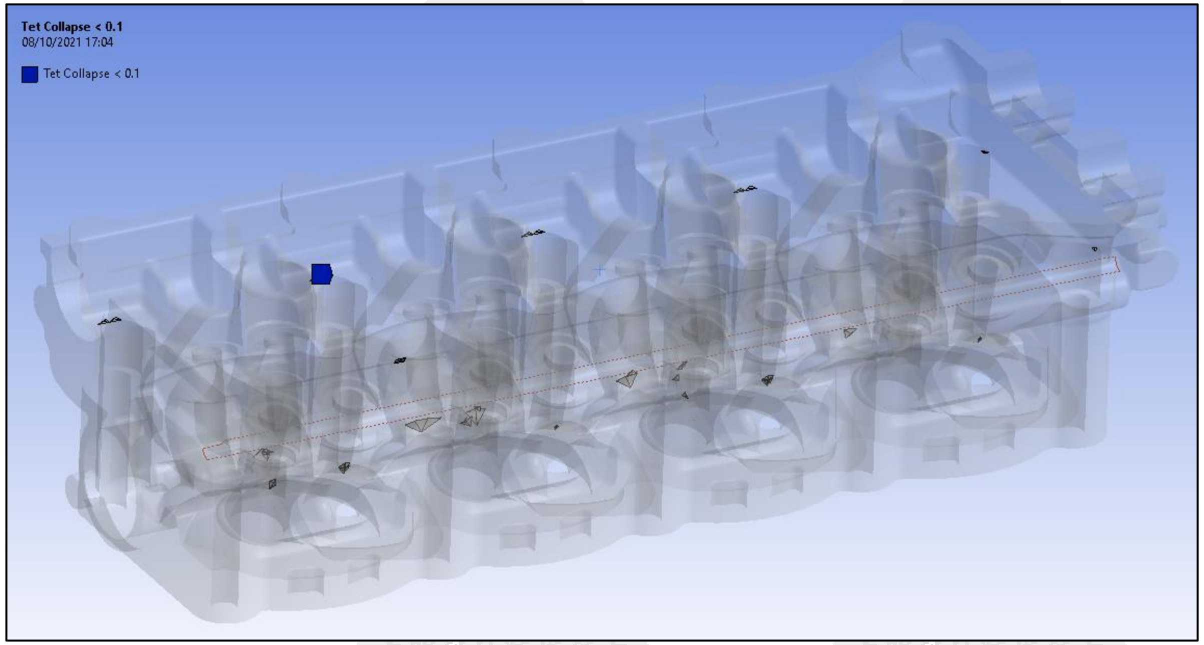
Ansys

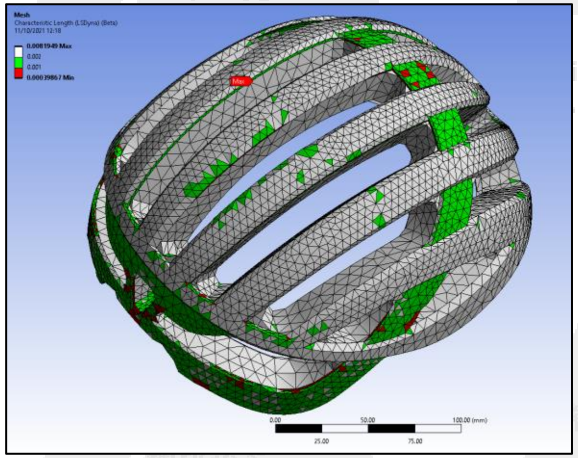
Quality

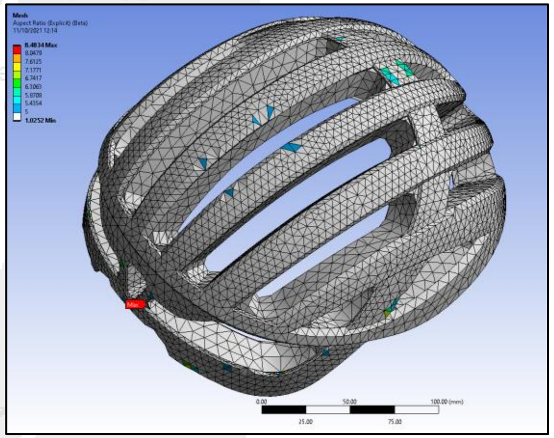
- Exposure of many more metrics as mentioned
  - Visibility of metrics is based on physics preference
  - New include LS Dyna Characteristic Length and Explicit Aspect Ratio

Quality	
Check Mesh Quality	Yes, Errors and Warnings
Target Element Quality	Default (0.200000)
Target Characteristic Length (LSDyna)	Default (0.5 mm)
Target Aspect Ratio (Explicit)	5.









Ansys

Feature Detection and Treatment

- Now available for **Solid** bodies
  - **3D hole** detection with option for mapped mesh treatment
  - **3D fillet** detection with option for mapped mesh treatment

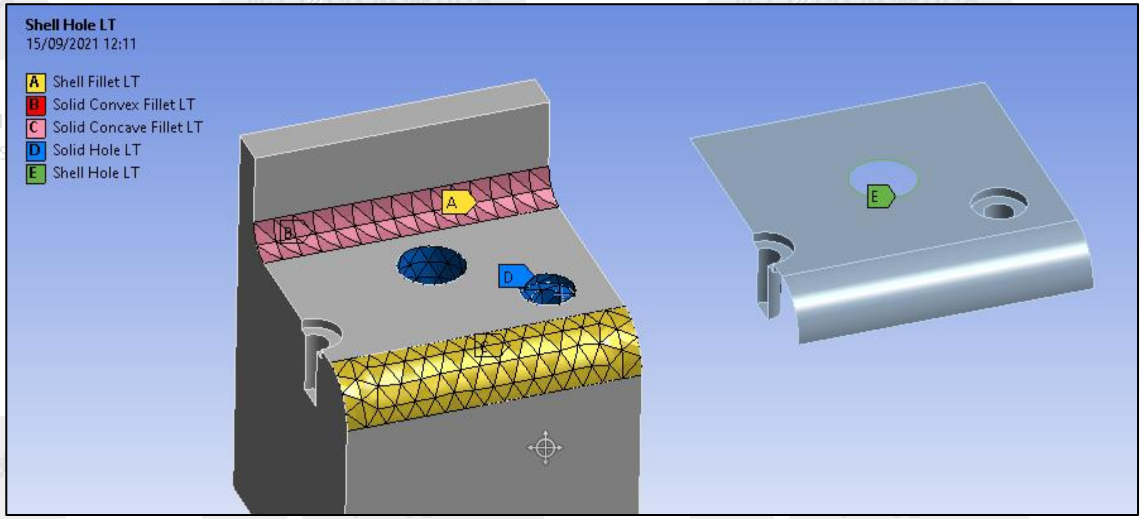
Worksheet

Feature Detection

*Right click on the grid to add/delete a row.

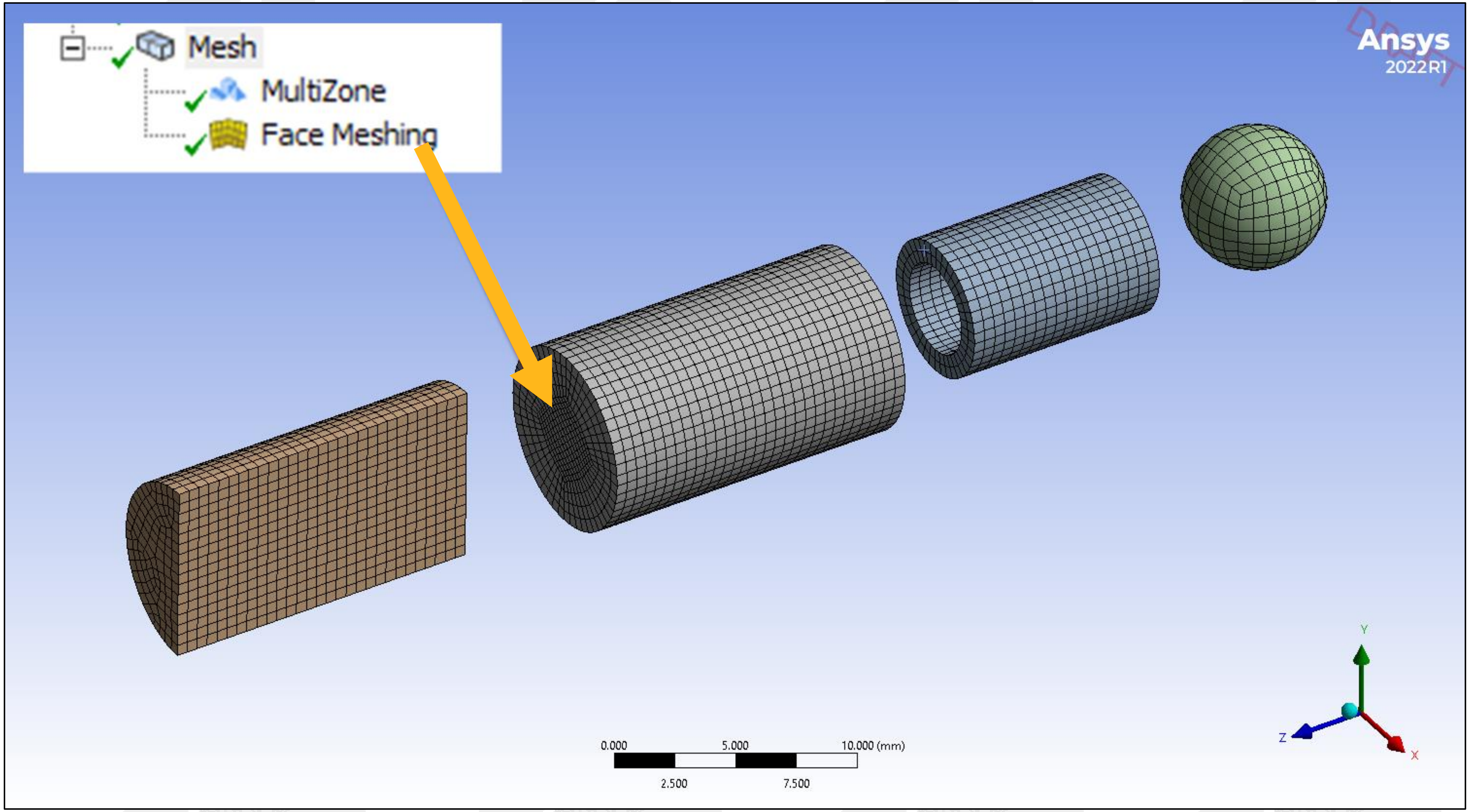
	Name	Type	Criteria	Operator	Value	Angle	Min Bound	Max Bound	Mesh Treatment
1	Shell Fillet LT 5mm	Shell Fillet	Radius	Less Than	5	0	0	0	Deviation Control
2	Solid Convex Fillet LT 5mm	Solid Convex Fillet	Radius	Less Than	5	0	0	0	Mapped Meshing
3	Solid Concave Fillet LT 5mm	Solid Concave Fillet	Radius	Less Than	5	0	0	0	Mapped Meshing
4	Solid Hole LT 10mm	Solid Holes	Radius	Less Than	10	0	0	0	Mapped Meshing
5	Shell Hole LT 10mm	Shell Holes	Radius	Less Than	10	0	0	0	Washer Control

Add Feature Detection



Ansys

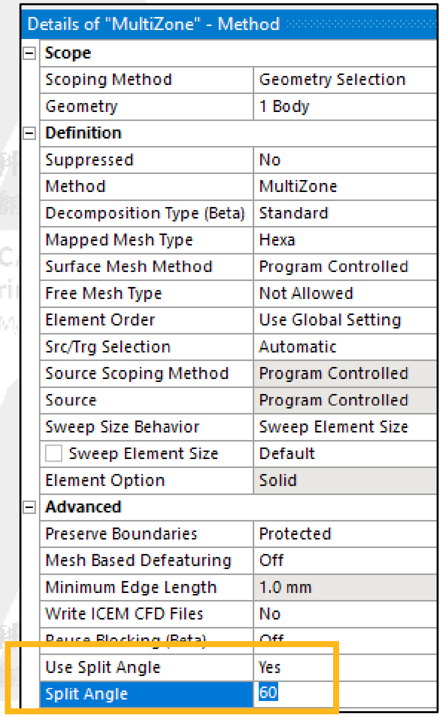
Hex Meshing: Less Decomposition in Mechanical MultiZone



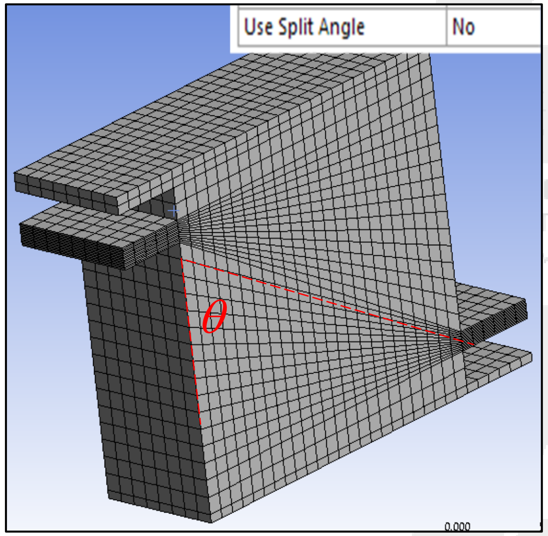
Ansys

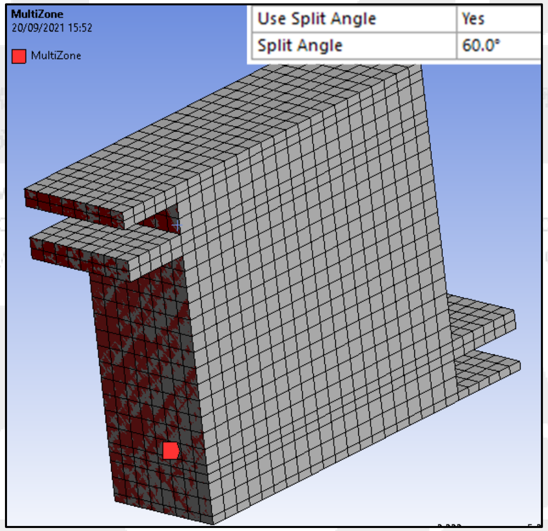
Hex Meshing: Split Angle

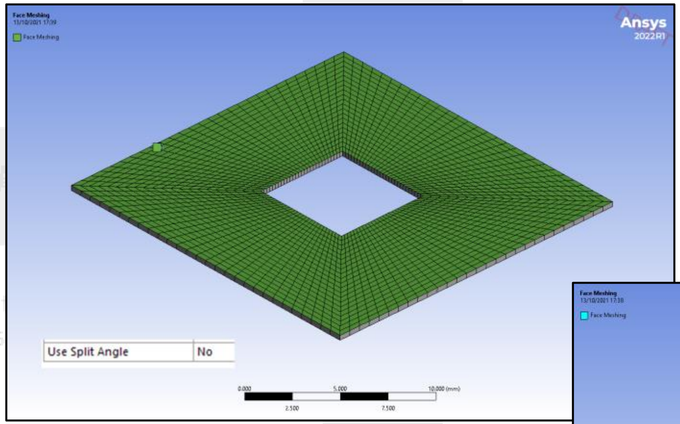
- Detect “skewed” blocks with bad angles and automatically cut them to yield better orthogonal meshes in a more automated way
  - Reduce need for decomposition in geometry tool
  - **Yes** by default for MZ controls created after Explicit Physics Preference is enabled

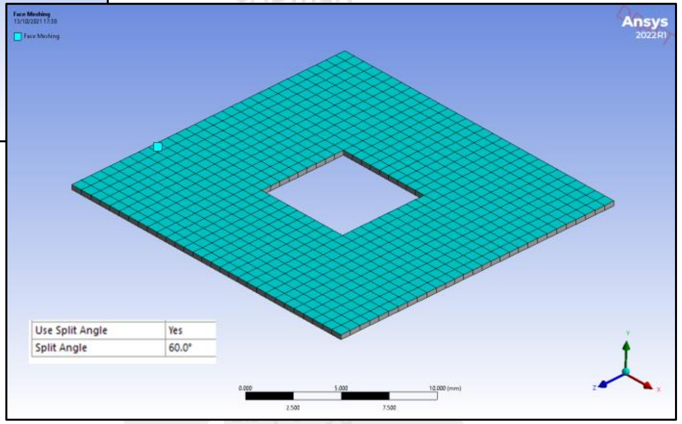


Details of "MultiZone" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Decomposition Type (Beta)	Standard
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Element Option	Solid
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	1.0 mm
Write ICEM CFD Files	No
Pause Blocking (Beta)	Off
Use Split Angle	Yes
Split Angle	60

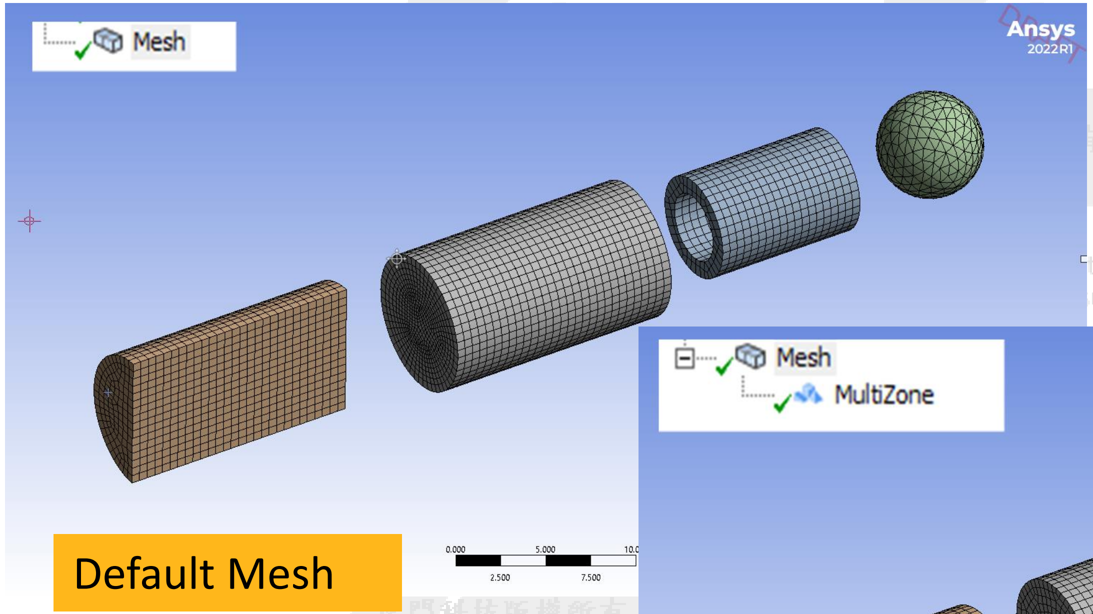




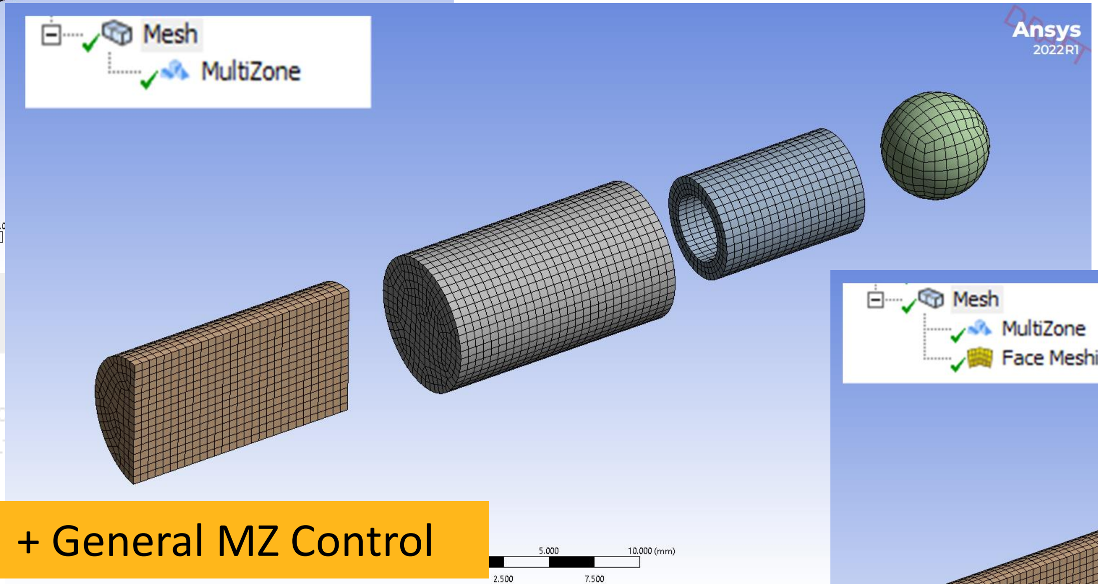




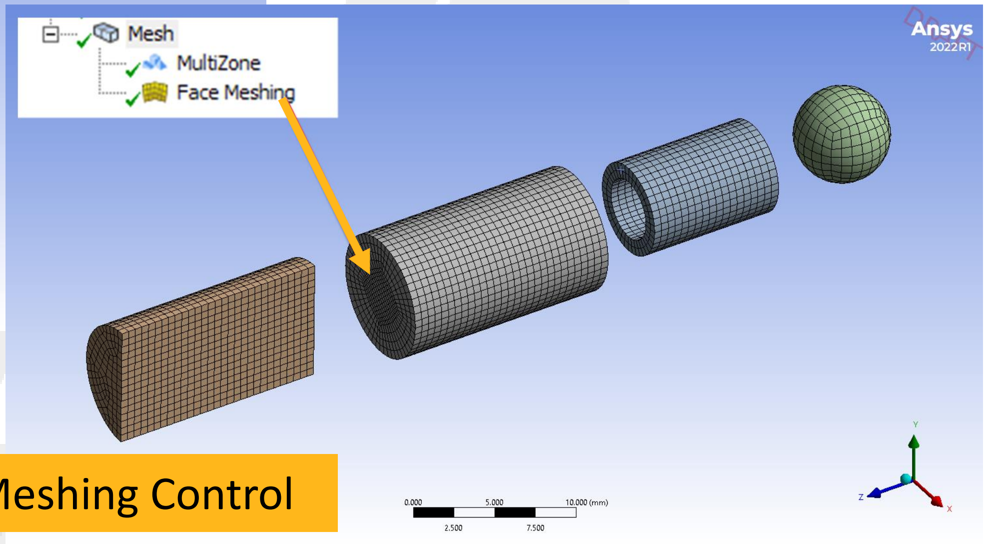
Hex Meshing: Better Orthogonality and Default Meshing Explicit Preferences



Default Mesh



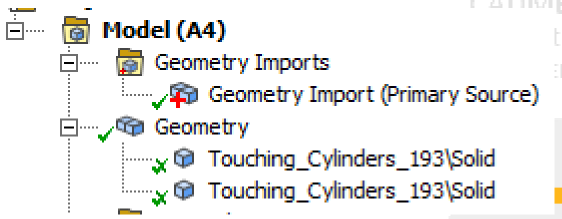
+ General MZ Control

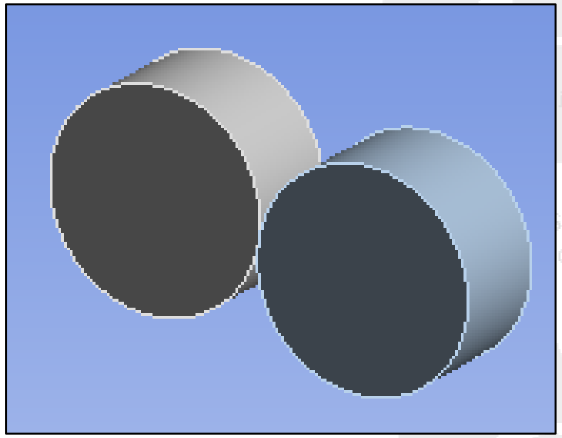


+ Face Meshing Control

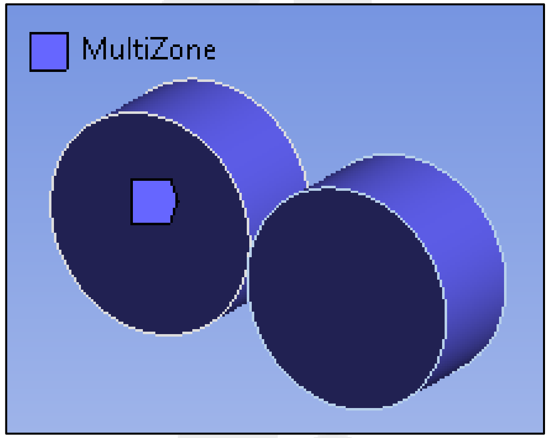
Hex Meshing: Better Default Mesh for Cylinders, Circles, and Spheres

Touching Cylinders

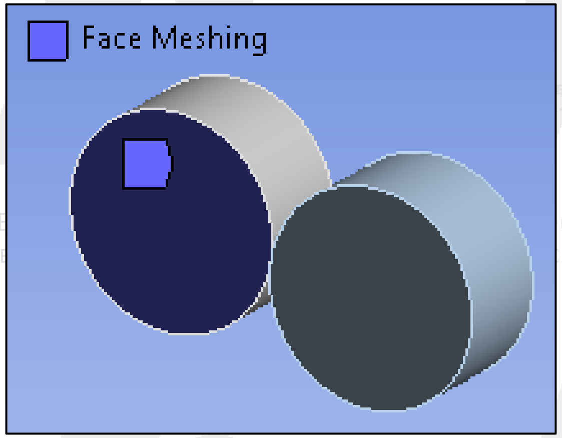


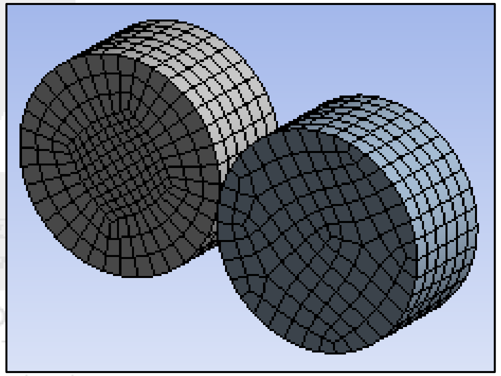


MultiZone Applied with no special inputs/selections – Automatic Hex



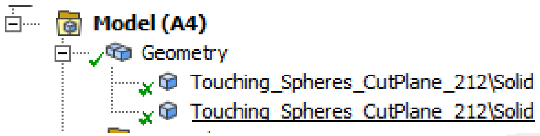
If Face Meshing is Applied O-Grid is automatic with no decomposition needed

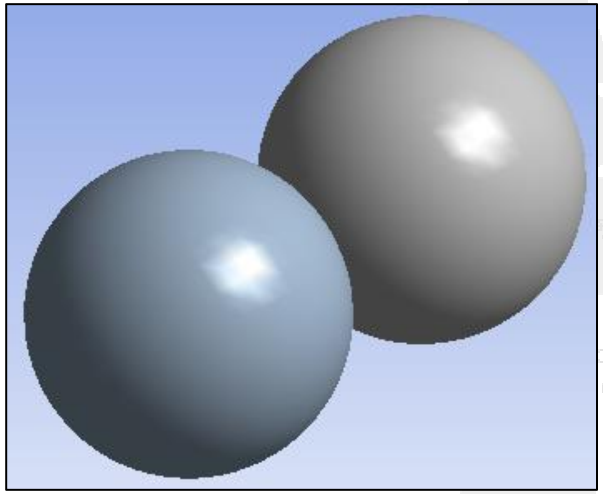




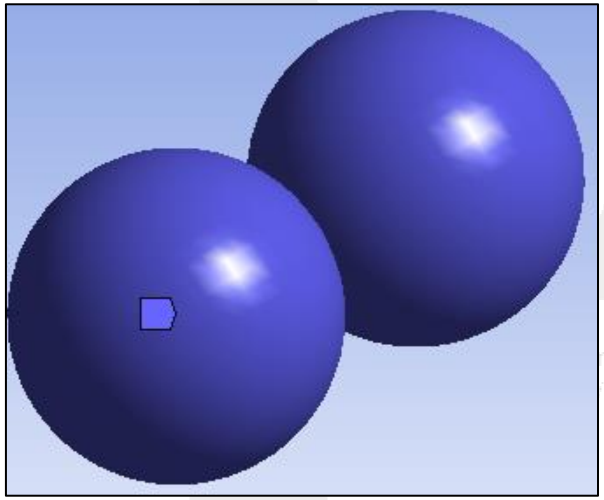
Hex Meshing: Better Default Mesh for Cylinders, Circles, and Spheres

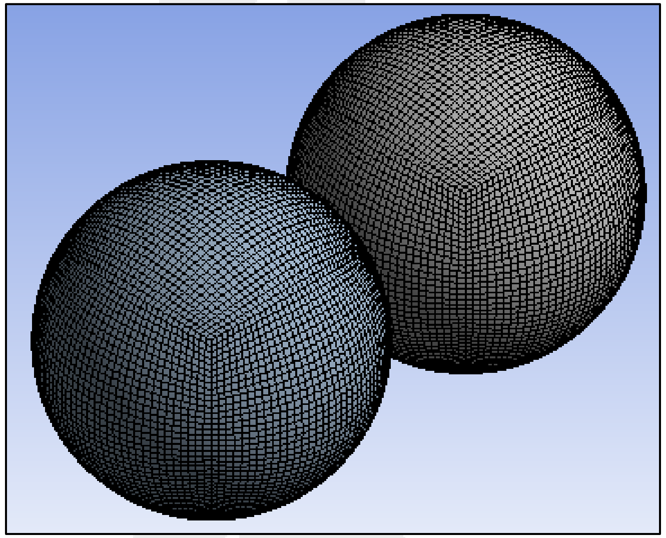
Touching Spheres

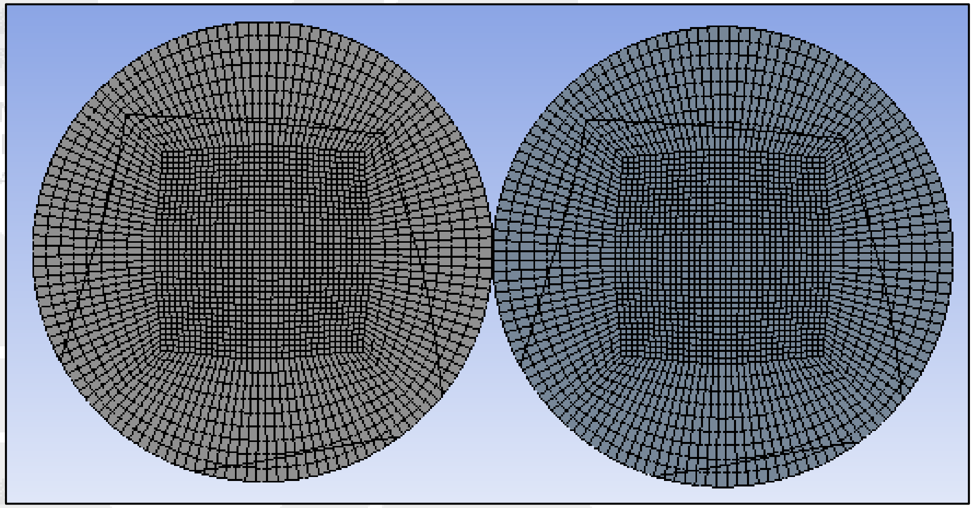




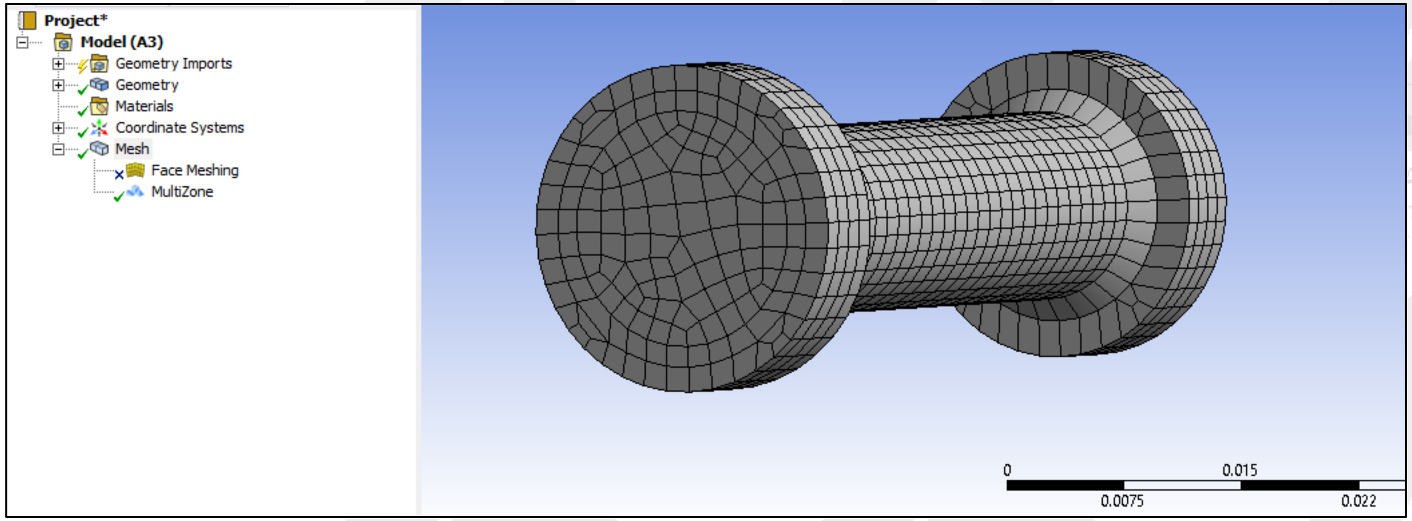
MultiZone Applied with no special inputs/selections – Automatic Hex



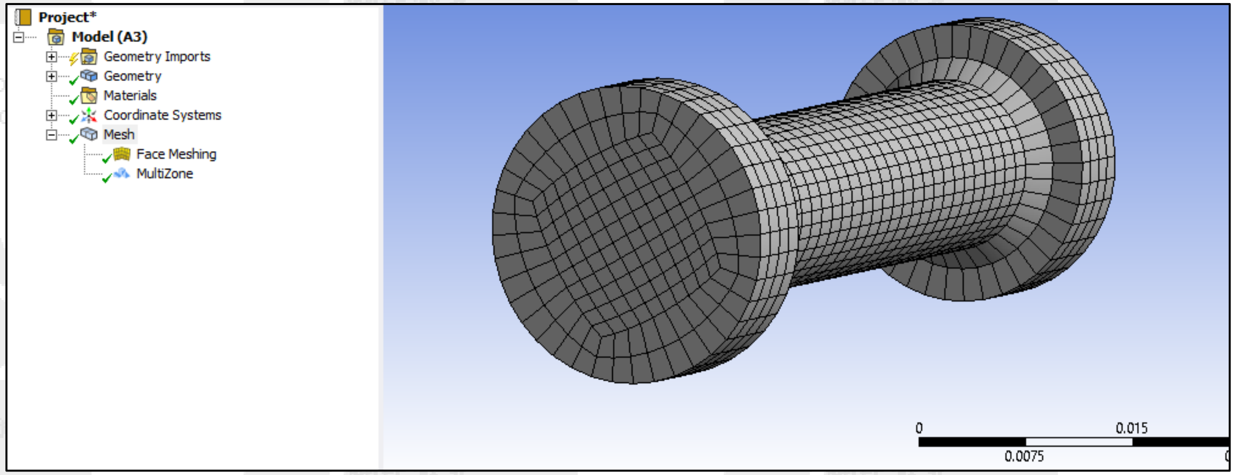




Hex Meshing: Improved Hex Mesh for Simple Shapes

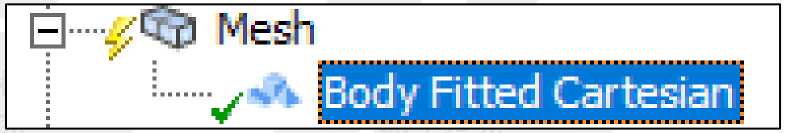


O-Grid results are more smoothed for Explicit Phys. Pref.



Hex Meshing: Body Fitted Cartesian - Edge Sizing Support

- BF Cart now supports “Edge Sizing” control

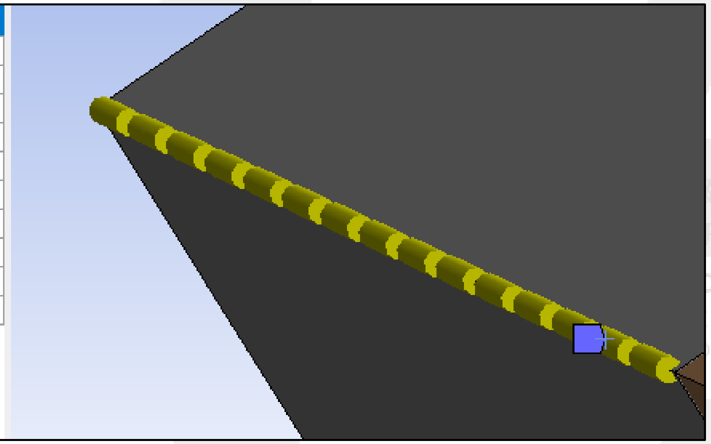


Details of "Body Fitted Cartesian" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1215 Bodies
Definition	
Suppressed	No
Method	Cartesian
Element Order	Use Global Setting
Type	Element Size
Element Size	7.5e-002 mm
Spacing Option	Default
Advanced	
Projection Factor	0.98
Project in constant Z-Plane	No
Stretch Factor in X	1.0
Stretch Factor in Y	1.0
Stretch Factor in Z	1.0
Coordinate System	Global Coordinate System
Write ICEM CFD Files	No

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	15
Advanced	
Behavior	Soft
Bias Type	No Bias

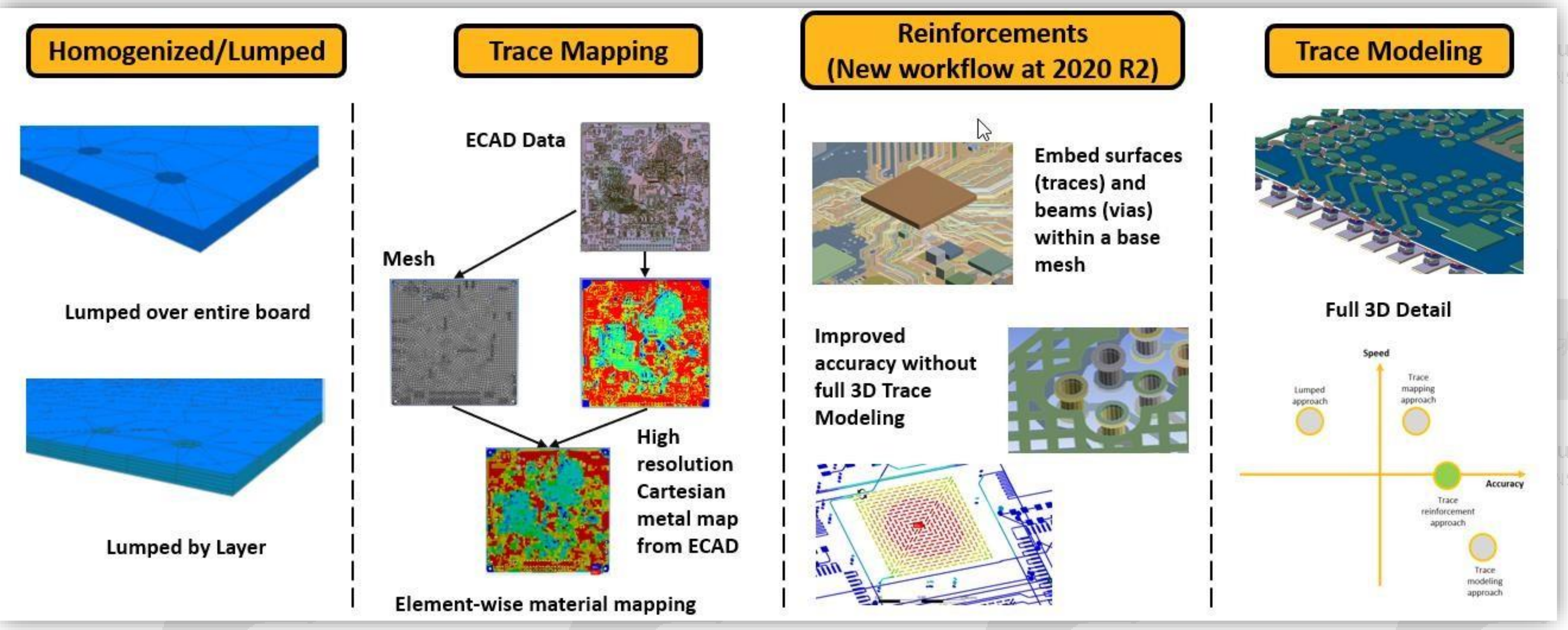


Reinforcement

Ansys

1 Flex Modeling - Introduction

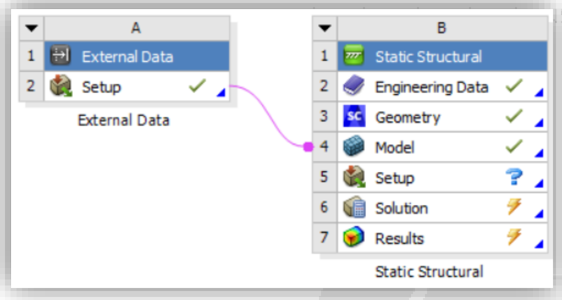
- ANSYS provides leading solutions for the modeling of flexes and electronic components. Multiple levels of fidelity supported
- Start by importing ECAD files directly into ANSYS tools, such as Sherlock, SpaceClaim and Mechanical



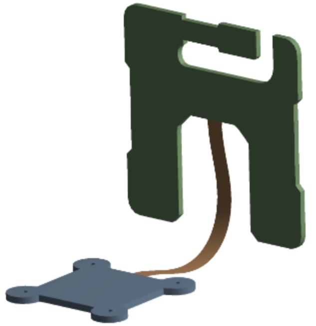
Flexes modeling techniques in Ansys

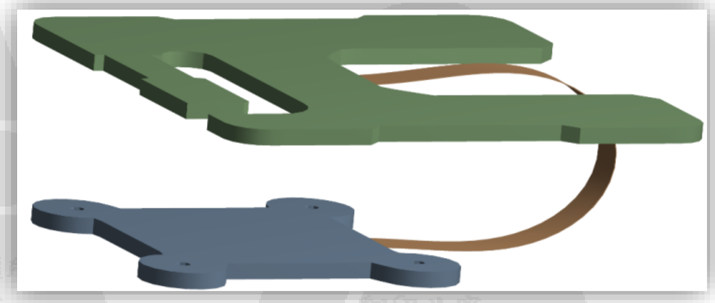
FCB Modeling in ANSYS- Trace Mapping

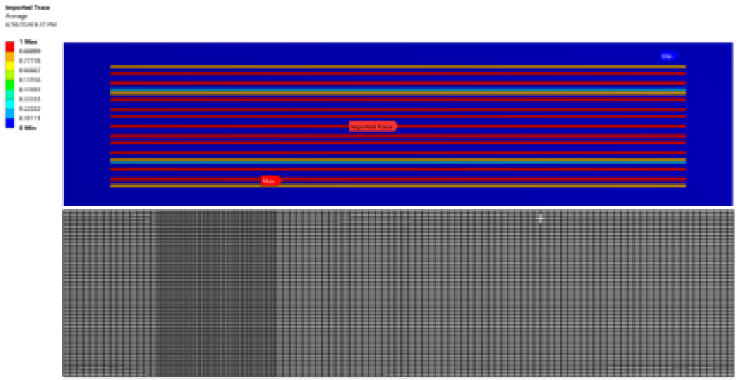
- This study focuses on modeling two different types of FCBs i.e. Rigid flex PCB and FCB cable
- <u>Rigid Flex PCB:</u> Modeling of Operating phase (Cyclic Loading)



Rigid Flex PCB

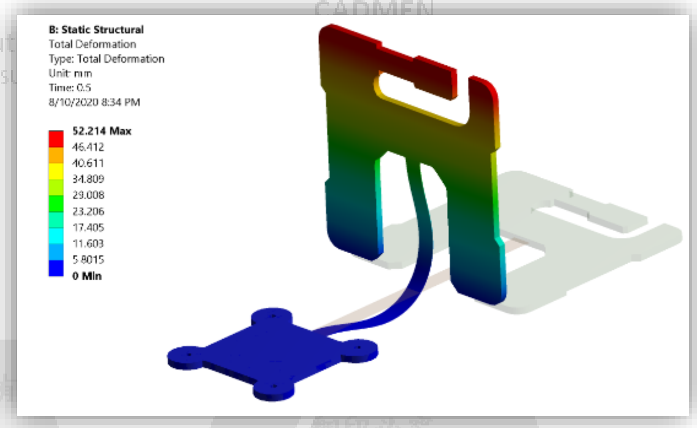






Traces

Mesh Density
(Fine along the width)



Ansys

Flex Bending – Trace Reinforcement Approach

- <u>Goal:</u>

- To demonstrate the Trace-Reinforcement approach
- To understand the preprocessing in Sherlock for thismethod
- To compare results with 3 different base elements:

1. **Solid Base Element with Reinforcements**
2. **Solid-Shell (SOLSH)Base Element with Reinforcements**
3. **Shell Base Element with Reinforcements**

Reaction Force Vector during Bending

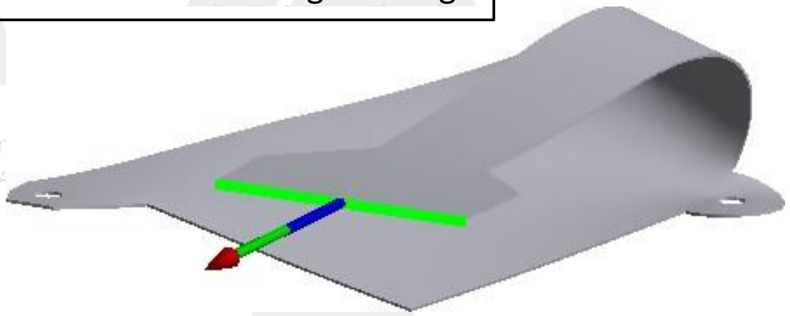
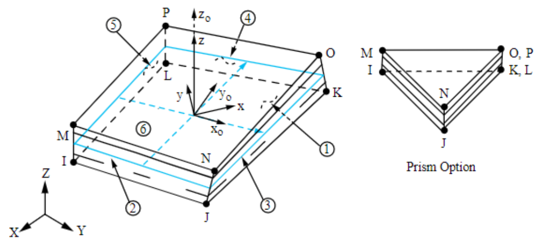
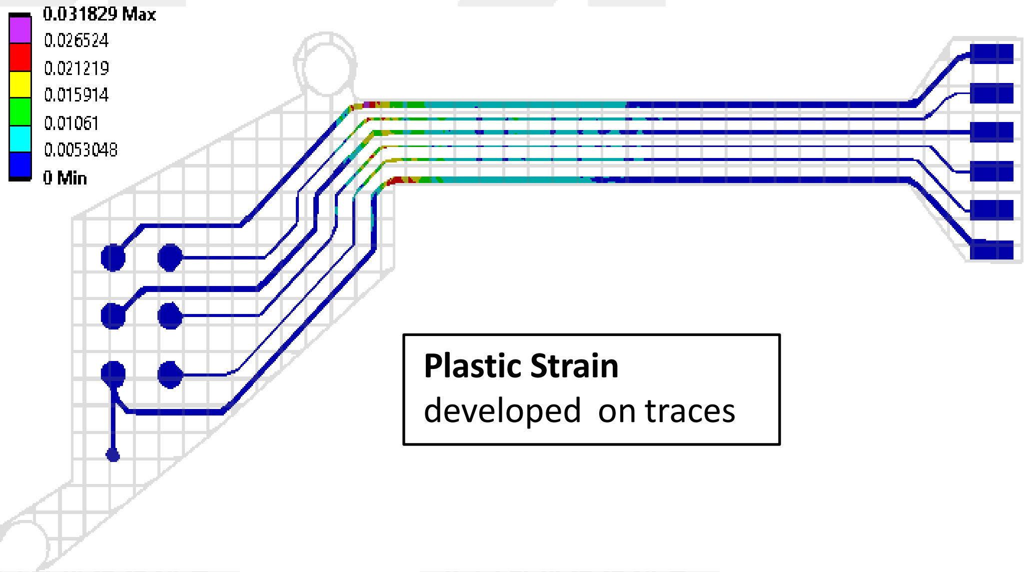


Figure 190.1: SOLSH190 Geometry





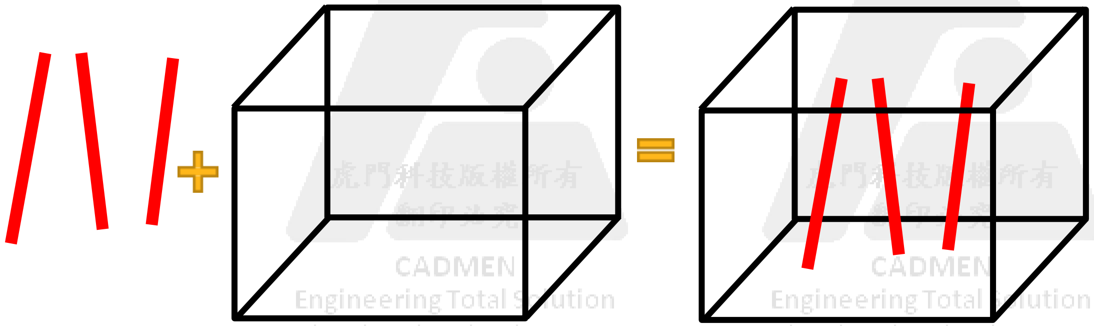
Plastic Strain
developed on traces

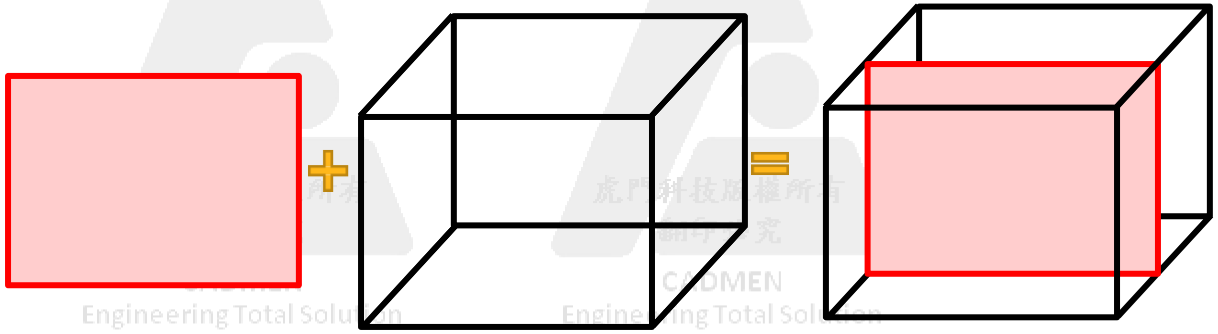
- <u>Model Description:</u>

- The model is 3 layered Pine phone flex
- Traces are represented with reinforcement elements
- Model is bent 180° as in the actual application (*refer to <u>page 5</u> for flex model description*)

Reinforcement Element Technology

- Reinforced materials are used extensively in civil construction, aircraft structures, automobiles, advanced sports equipment, and medical devices. Reinforcing commonly appears in fiber or cable forms, such as steel rebar in reinforced concrete, nylon strands in tires, and carbon fibers in various composite materials
- Ansys can model the reinforcing fibers with specialized reinforcing elements. The reinforcing elements interact with standard structural elements, referred to as the base elements, via the common nodes
- Two approaches are available:





Discrete Reinforcement:

In this approach, each fiber is modeled separately as a spar having only uniaxial stiffness. It could have nonuniform materials, cross-section areas, or arbitrary orientations

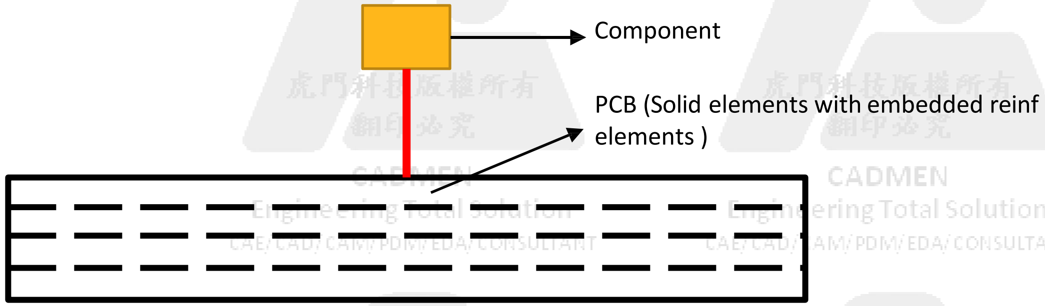
Smear Reinforcement:

In this approach, one layer of fiber with identical material, orientation, and cross-section area is treated as a homogeneous membrane having unidirectional stiffness or plane-stress state for homogeneous reinforcing membranes

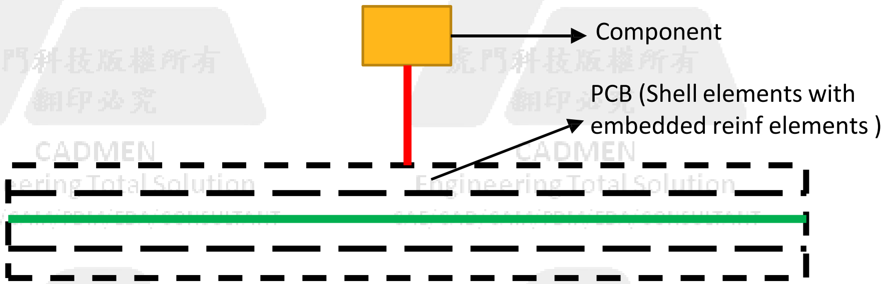
Reinforcement Element Technology

<u>Reinforcement element with Solid and Shell base elements</u>

- To represent trace metal layers within the PCB boards, smear reinforcement approach with Plane Stress option was used
- The base element could either be Solid element or Shell bodies
- For the solid option, each layer of the board is explicitly modeled since the dielectric material could be different in each layer
- For the shell option, multilayer section was defined to represent each layer's dielectric material



Reinforcement Approach (Solid base elements)



Reinforcement Approach (Shell base elements)

- *Refer to Ansys Help to read more on Reinforcement Elements: [REINF263](), [REINF264]() and [REINF265]()*

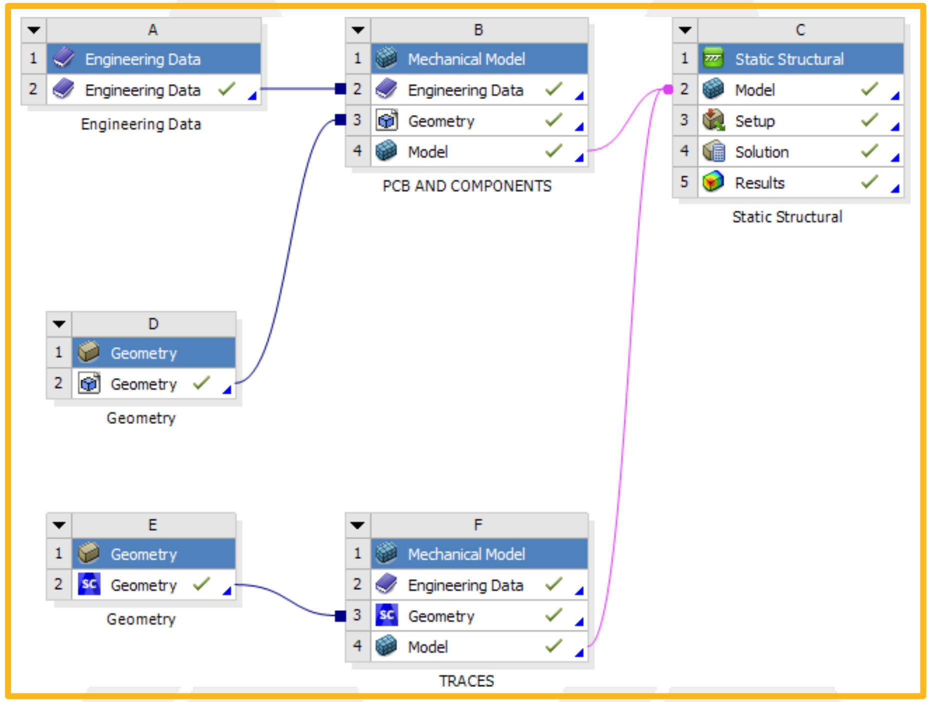
4. Workflow Overview

Sherlock:
- Import ECAD file and packages
- Create Solid PCB CAD (step file)
- Create traces as surface bodies (step file)
- Create wbjn and py file

Workbench:
- Read the wbjn file.
- The WBJN file calls the PY file and the STEP file to create the basic workflow

Space Claim:
- Open 'Geometry' of the PCB and components.
- Solid Approach: Merge all the layers into a single solid body.
- Shell Approach: Create shell body at the center of the boards

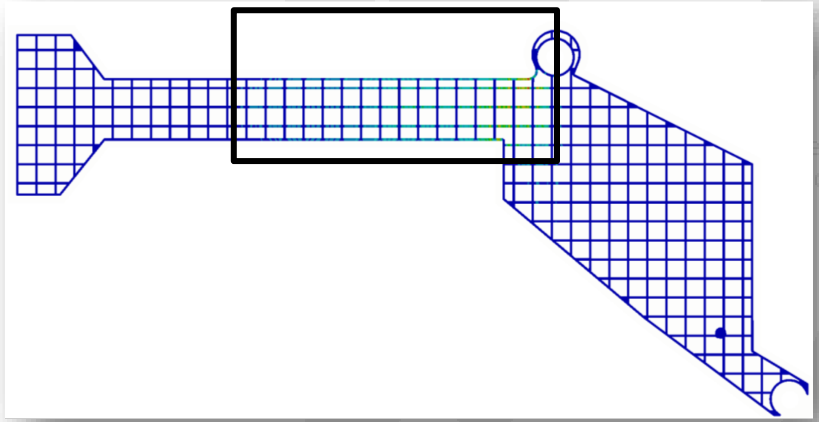
Mechanical:
- Assign materials
- Contact between the components and PCB
- Boundary Conditions
- Loading condition
- Solve and post process.

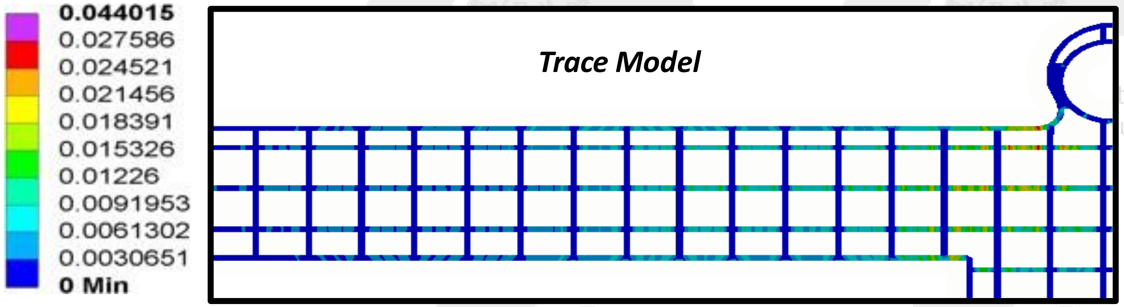


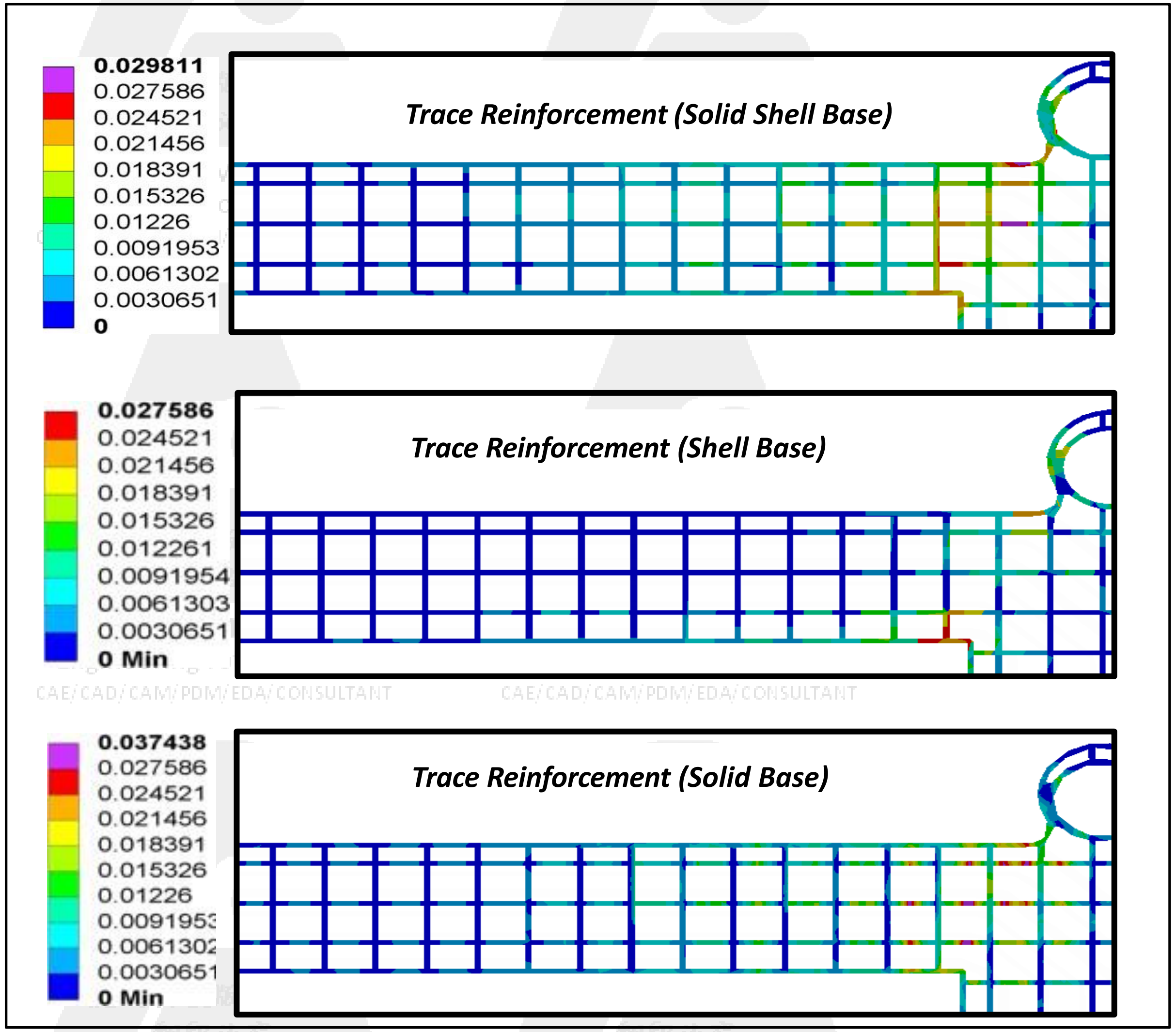
Workbench project Schematic

Comparative Analysis: Solid Vs Shell Vs Solid-Shell Base Element

Plastic Strain (Layer 1)



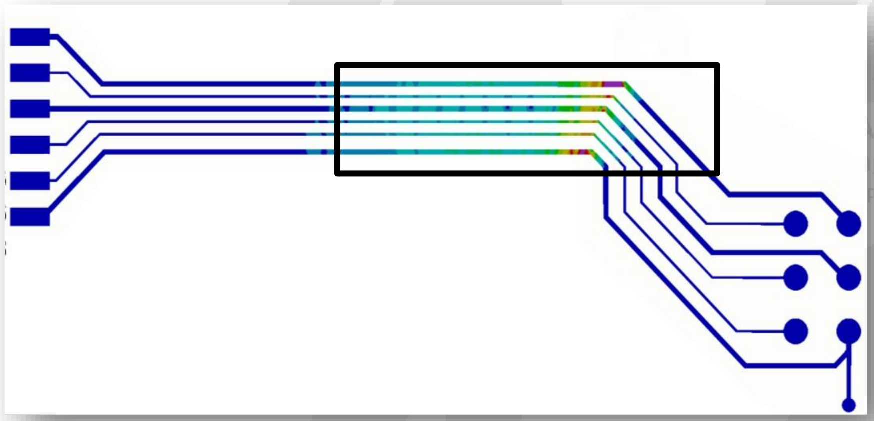


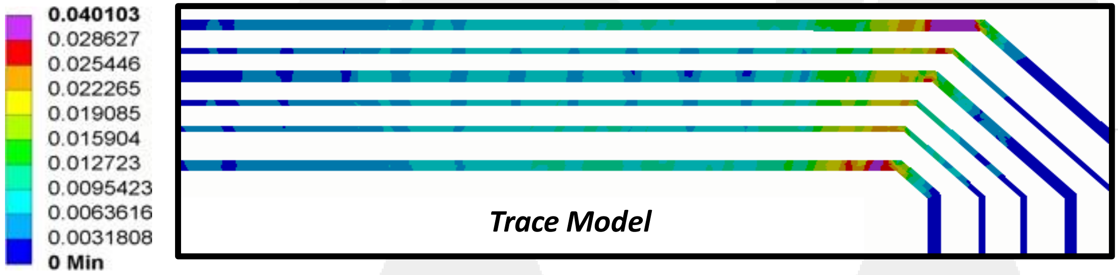


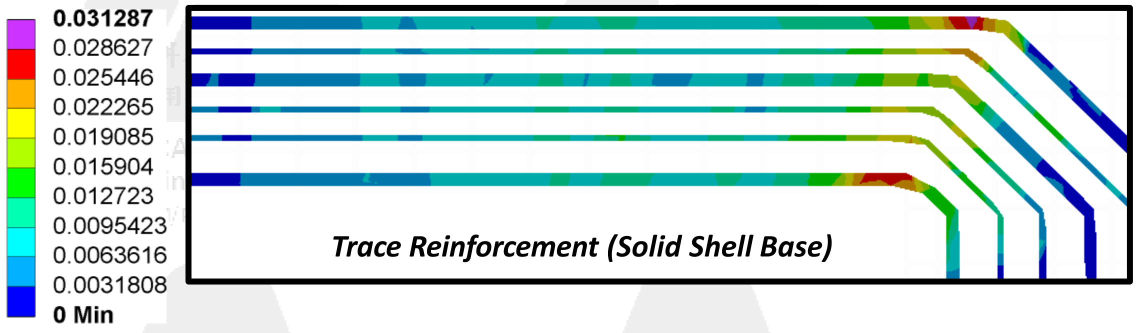
Trace Model approach is considered as benchmark and Trace-reinforcement approach with different base elements is compared against it

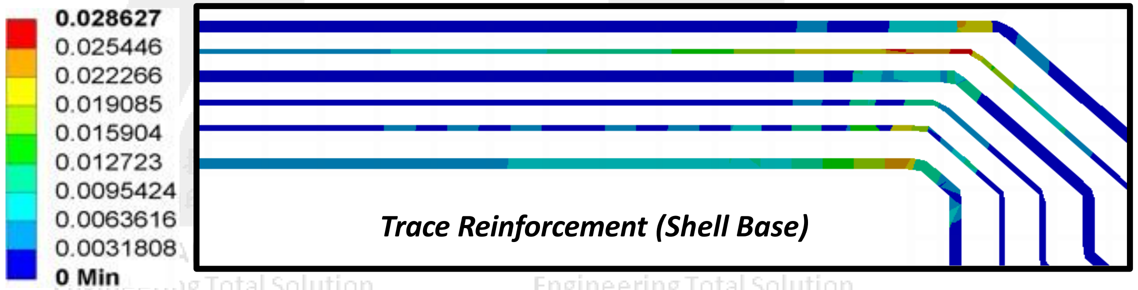
Comparative Analysis: Solid Vs Shell Vs Solid-Shell Base Element

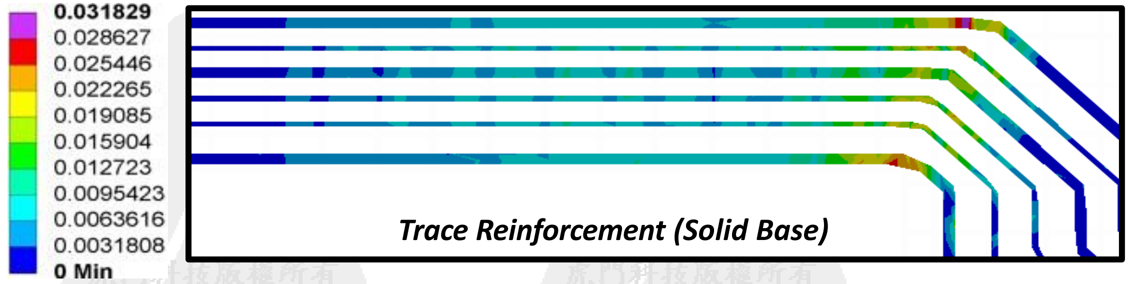
Plastic Strain (Layer 2)











Trace Model approach is considered as benchmark and Trace-reinforcement approach with different base elements is compared against it

Structural Optimization

Ansys

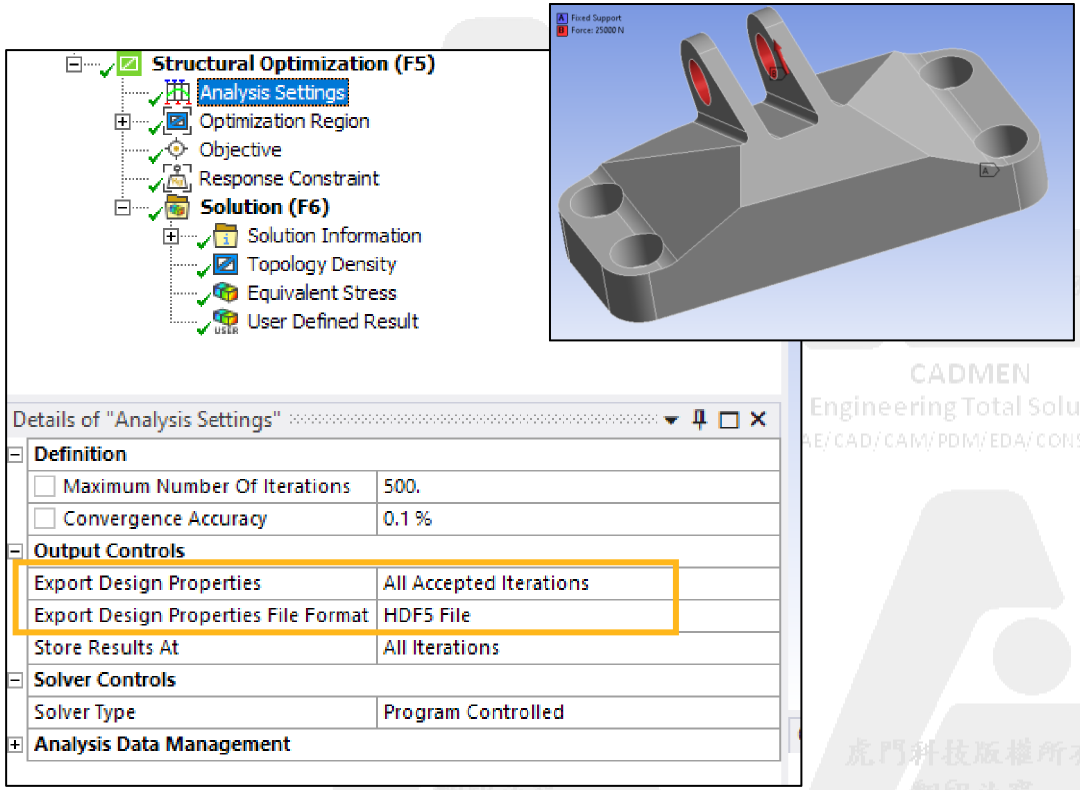
Export Design Properties

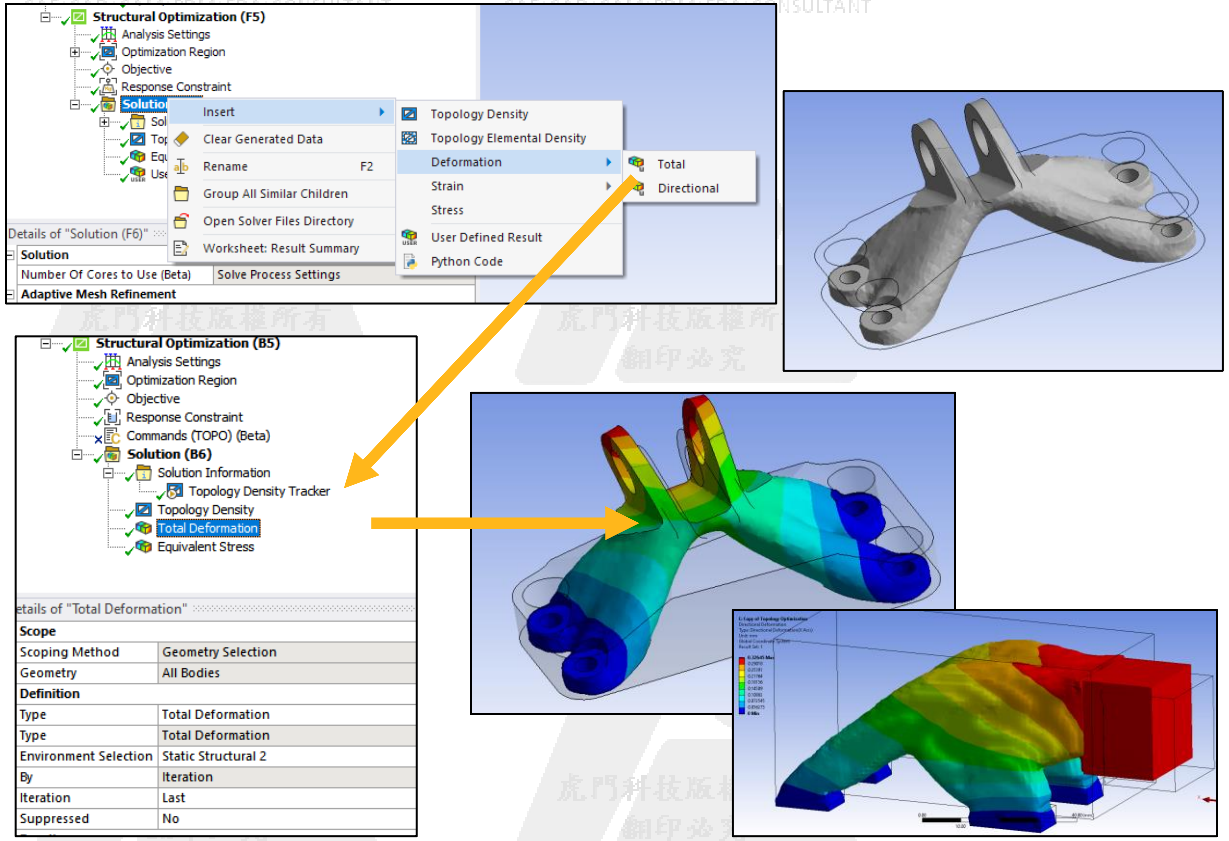
This new feature permits users to visualize the results **(deformation, stress, eigen-mode, etc)** on the final design, giving them the opportunity to quickly examine and validate the mechanical behavior.

(1) Optimization set-up:
Activate the option

(2) Run the optimization

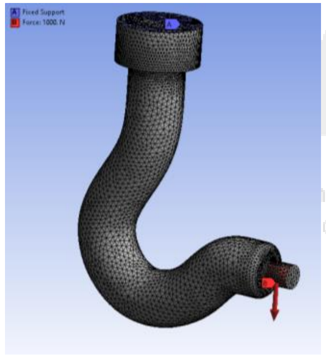
(3) Post:
Define your display

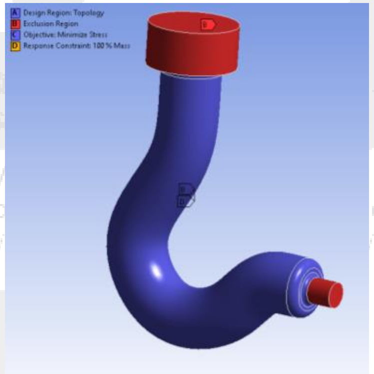


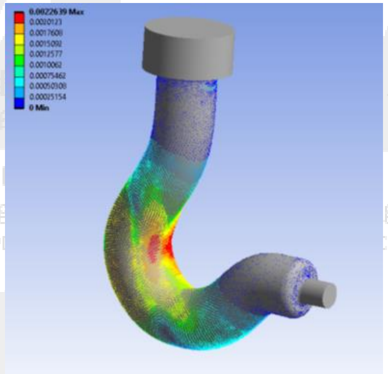


Maximum Principal Stress

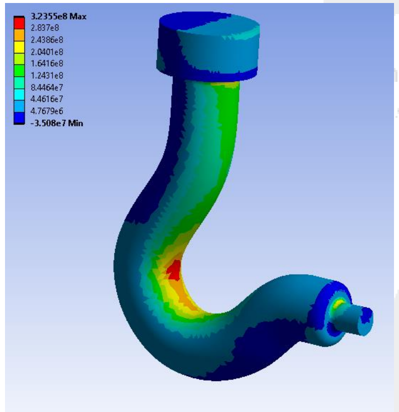
- ✓ **MPS is available alongside Von Mises eq. norm**
- ✓ Available for Topology (**SIMP or Level-Set**), **Shape and Lattice Optimization**
- ✓ Can be consumed as **an objective or as a constraint**



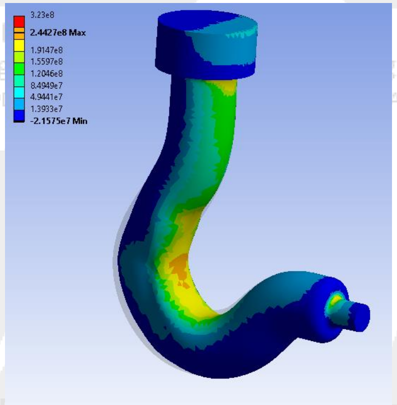


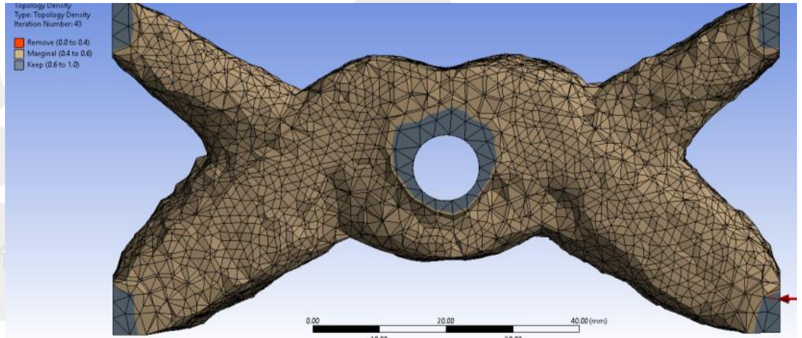


Mesh & Structural Set-up → Shape Optimization Set-up → Results & direct validation

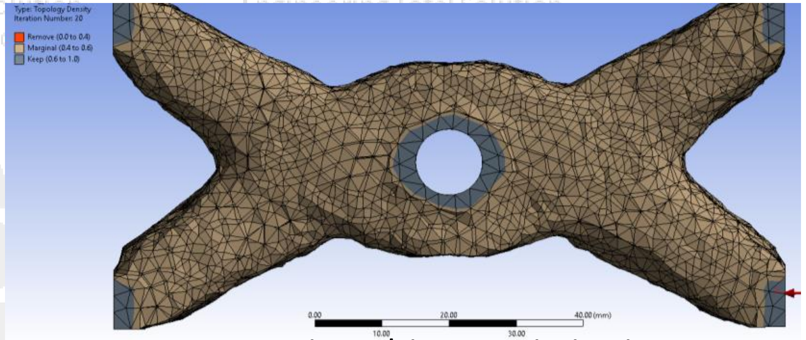


MPS – 25 %





MPS-driven optimization



VonMises-driven optimization

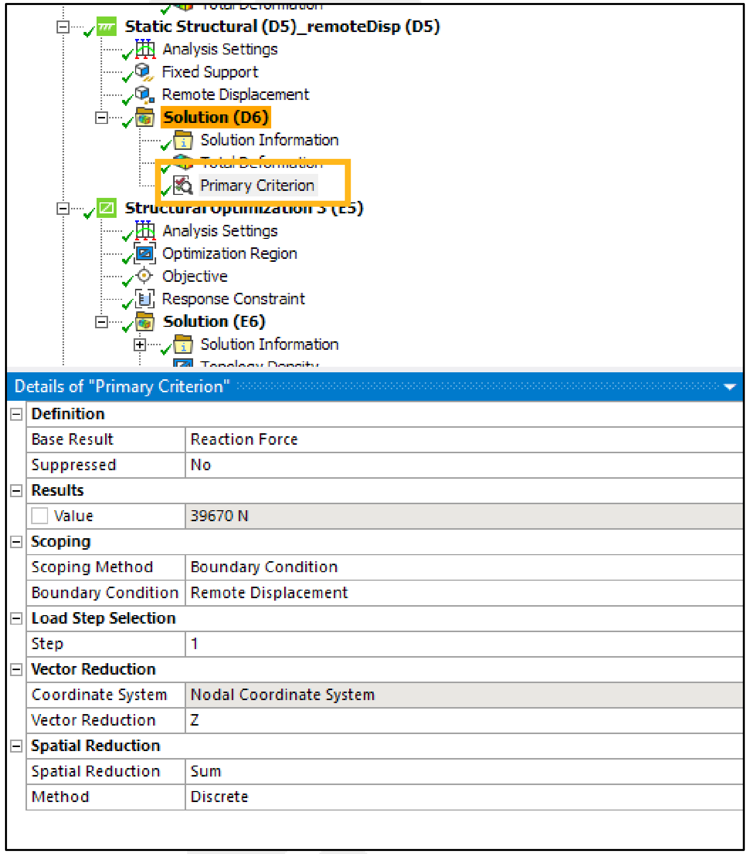
Ansys

User Defined Criterion

Since 2021 R1, a new capability has been introduced in a bid to create user-defined criterion in the upstream Static Linear Analysis system. This criterion (primary/composite) can then be consumed as objective or constraint in the downstream Structural Optimization

This feature has been extended:

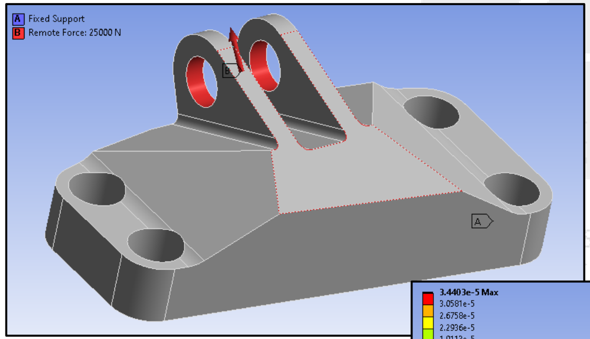
- **New base result: displacement, rotation, reaction-force or reaction-moment**
- **New scoping: remote point or support of boundary condition (eg remote-force, moment, remote-displacement)**

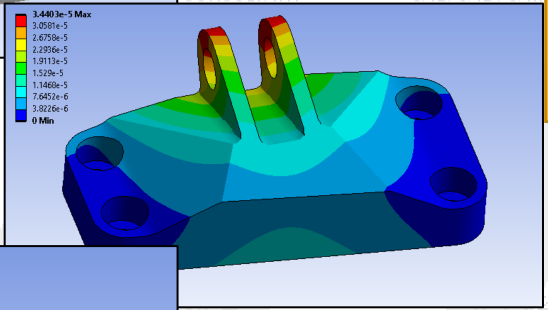


UDC has also been introduced in **Modal-Analysis**:

- **Single Frequency** criterion that aims to control the i-th eigenfrequency of interest
- **Robust Frequency** criterion that aims to control the i-th eigenfrequency of interest while handling efficiently mode-crossing

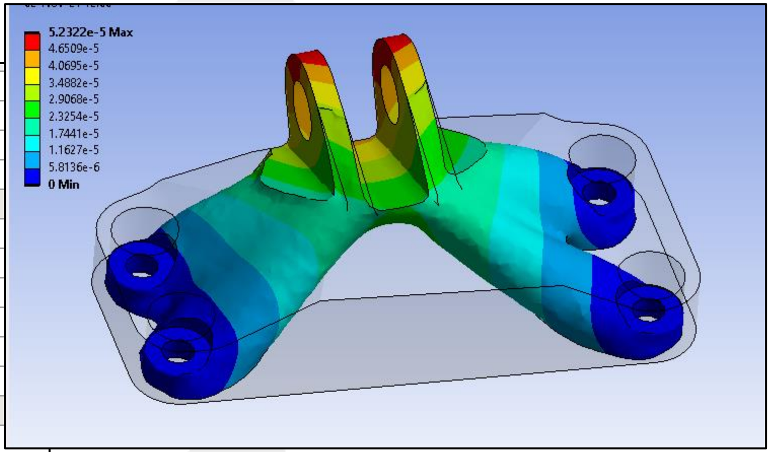
Example #1-2



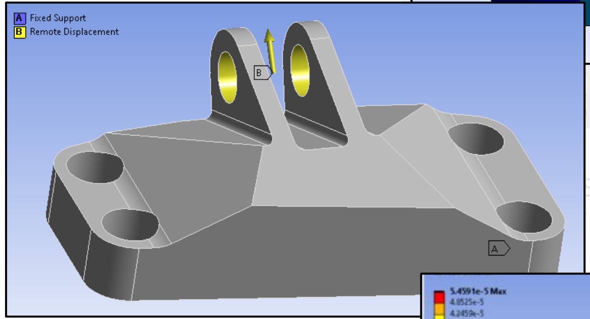


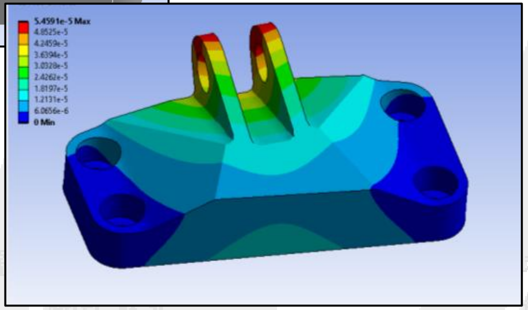
$$\begin{cases} \min_{\Omega}\left(primary\ criterion \right) \\ vol < 40\% \end{cases}$$

Definition	
Base Result	Displacement
Suppressed	No
Results	
Value	2.1427e-005 m
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Force
Load Step Selection	
Step	1
…Reduction	
…System	Nodal Coordinate System
…e	Constant
…ce Value	0.
…ector Reduction	Z
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete



z-disp of the remote-force support

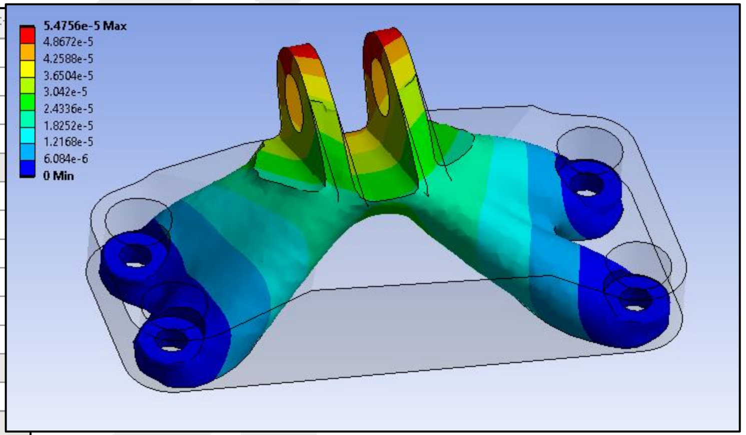




$$\begin{cases} \max_{\Omega}\left(primary\ criterion \right) \\ vol < 40\% \end{cases}$$

…etails of "Primary Criterion"

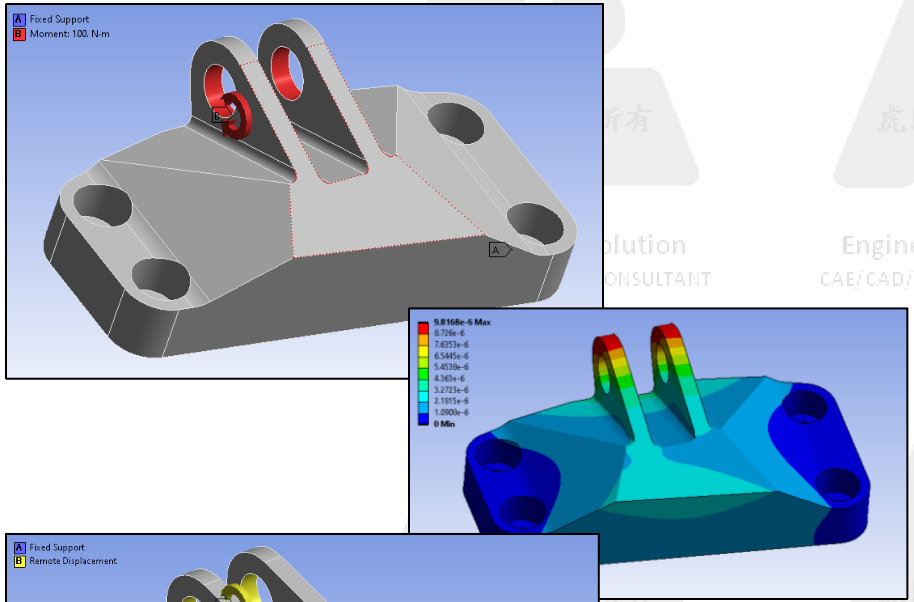
Definition	
Base Result	Reaction Force
Suppressed	No
Results	
Value	39670 N
Scoping	
Scoping Method	Boundary Condition
…dary Condition	Remote Displacement
… Selection	
	1
…duction	
…ordinate System	Nodal Coordinate System
Reference Type	Constant
Reference Value	0.
Vector Reduction	Z
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete



z-reaction-force
at the remote-displacement support

Ansys

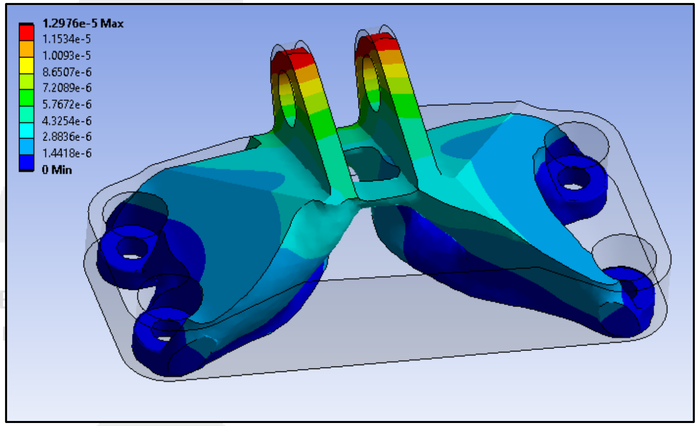
Example #3-4



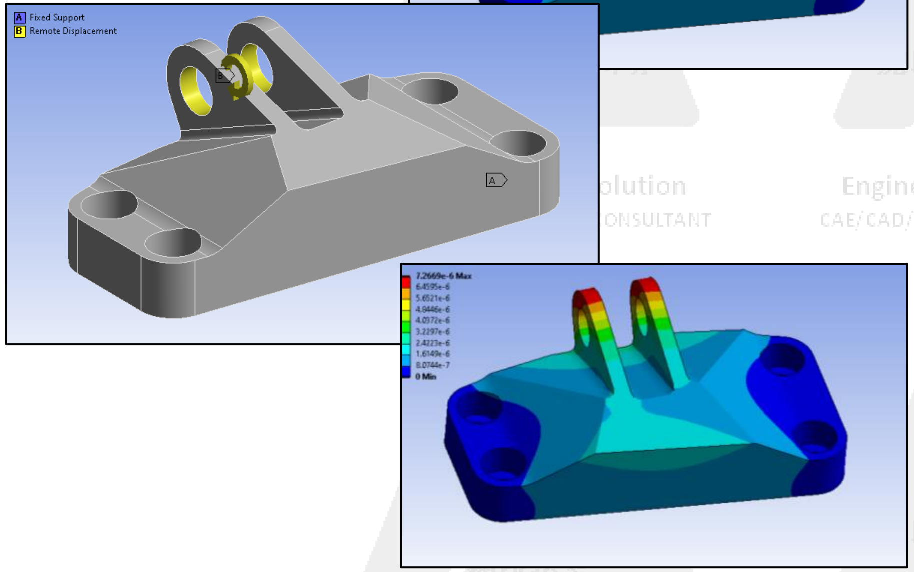
$$\max_{\Omega}\left(primary\ criterion\right)$$
$$vol < 40\%$$

Details of "Primary Criterion"

Definition	
Base Result	Rotation
Suppressed	No
Results	
Value	-1.3509e-002 °
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Moment
Load Step Selection	
	1
Reduction	
oordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete



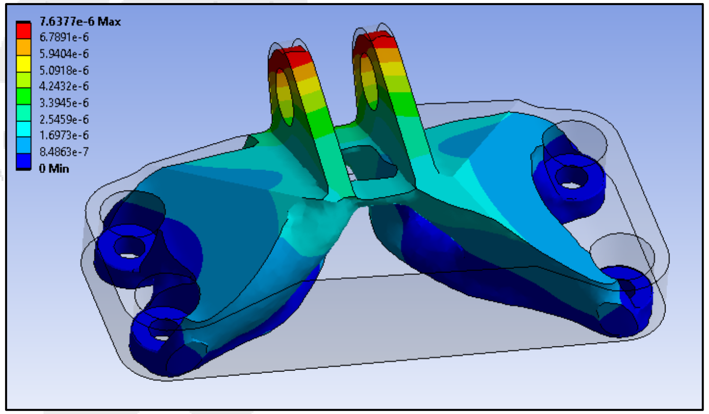
x-rotation of the remote-moment support



$$\min_{\Omega}\left(primary\ criterion\right)$$
$$vol < 40\%$$

Details of "Primary Criterion"

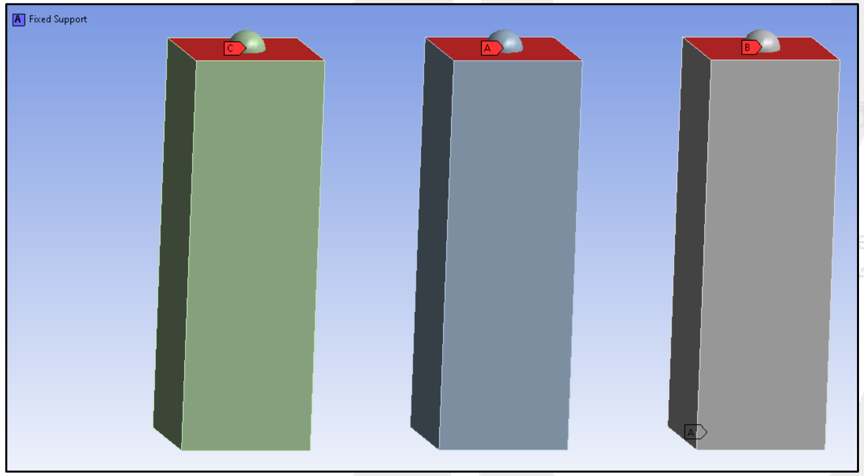
Definition	
Base Result	Reaction Moment
Suppressed	No
Results	
Value	-74.025 N·m
Scoping	
ping Method	Boundary Condition
Condit on	Remote Displacement
p Selection	
ep	1
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete



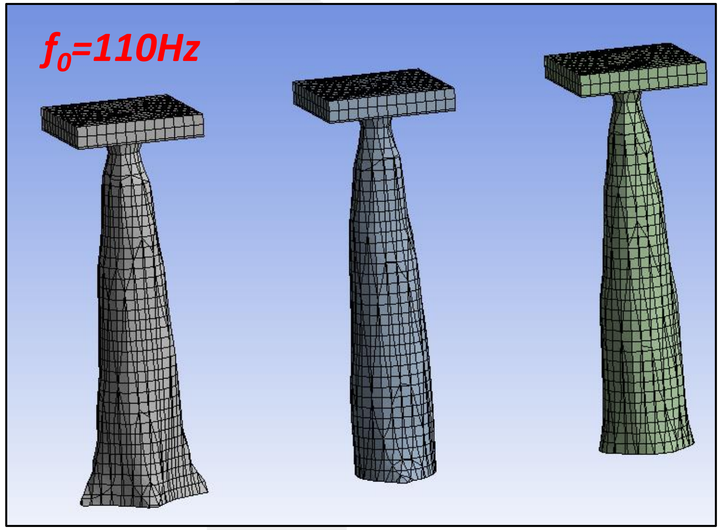
x-reaction-moment
at the remote-displacement support

Ansys

UDC Modal: Robust Frequency Example



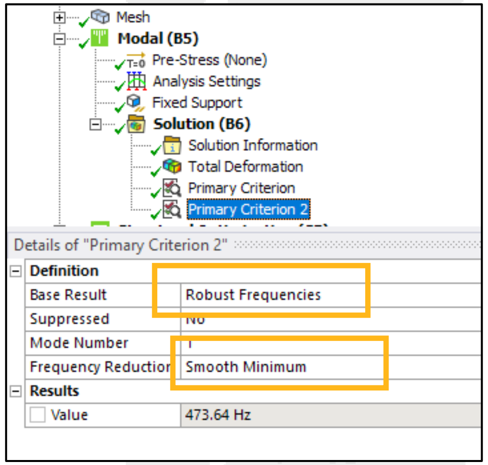
$$\textit{optim}\#1 \quad \begin{cases} \max_{\Omega} f_0 & [\text{single freq}] \\ \textit{mass} \le 25\% \end{cases}$$

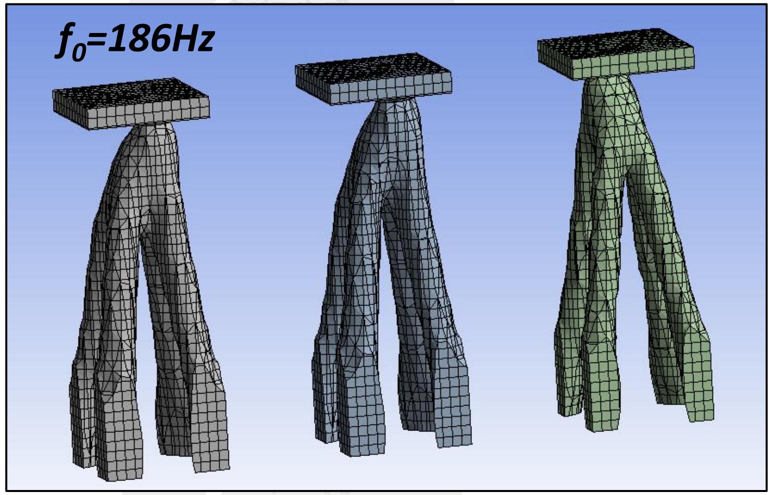


+70% better

Better performance thanks to the robust-frequency criterion

$$\textit{optim}\#2 \quad \begin{cases} \max_{\Omega} \tilde{f}_0 & [\text{robust freq}] \\ \textit{mass} \le 25\% \end{cases}$$





Context of this example

- 3 bodies, fixed at the bottom, remote mass at the top
- 6 first modes share the same eigenfrequency, ie 465Hz
- The purpose is to maximize the first eigenfrequency

Notes

- The optimizer may converge prematurely due to the "mode-crossing" phenomenon
- Roughly speaking, the modes order changes during the optimization and confuses the optimizer
- The "smooth minimum" option manages this context

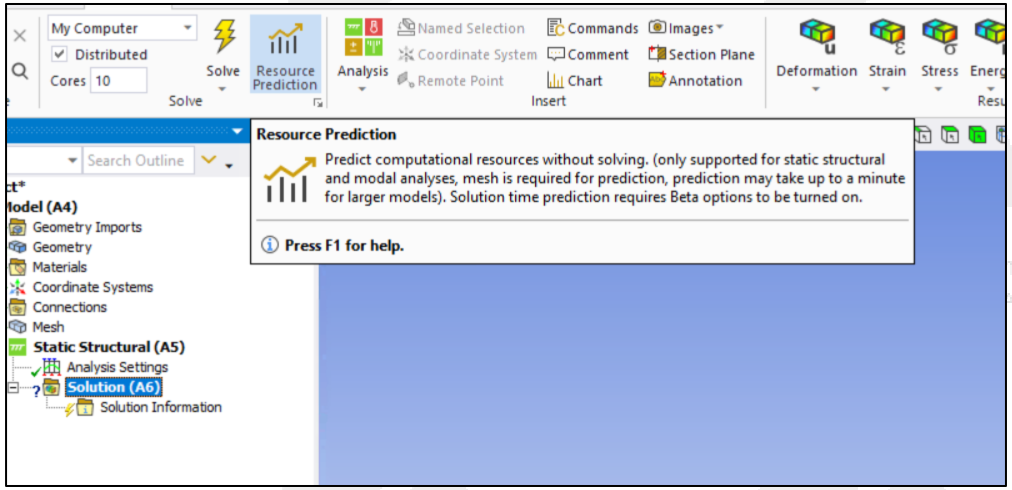
Ansys

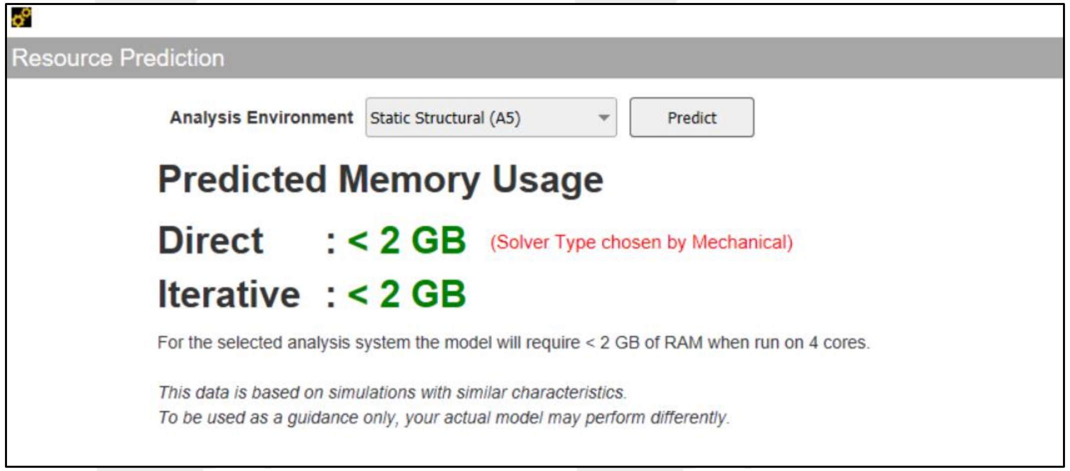
MAPDL Solver/HPC

Ansys

Resource Prediction Enhancements

- Switched to new neural network algorithm by default
  - Exposure of resource prediction is only via Mechanical GUI
  - Improved accuracy for memory requirement predictions
  - Reduced installation size





Ansys

Distributed Ansys Enhancements

- Introduction of new form of parallelism - Hybrid
  - Combines DMP and SMP (distributed + shared memory parallelism)
  - Activated via a new command line argument → -nt <#>

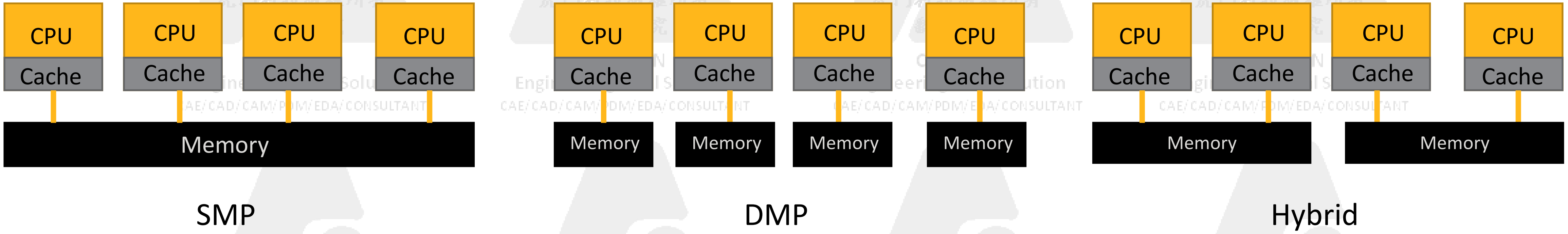
- **SMP** → -smp & -np N to specify using "N" OpenMP threads
- **DMP** → -np N to specify using "N" MPI processes
- **Hybrid** → *-np N* to specify using "N" MPI processes
  -nt M to specify using "M" OpenMP threads per process during SOLVE command

Total core count = N X M = P cores

ansys221 -b -np 8 -nt 2 <test.dat> out will use 8 processes with 2 threads per process for a total of 16 CPU cores

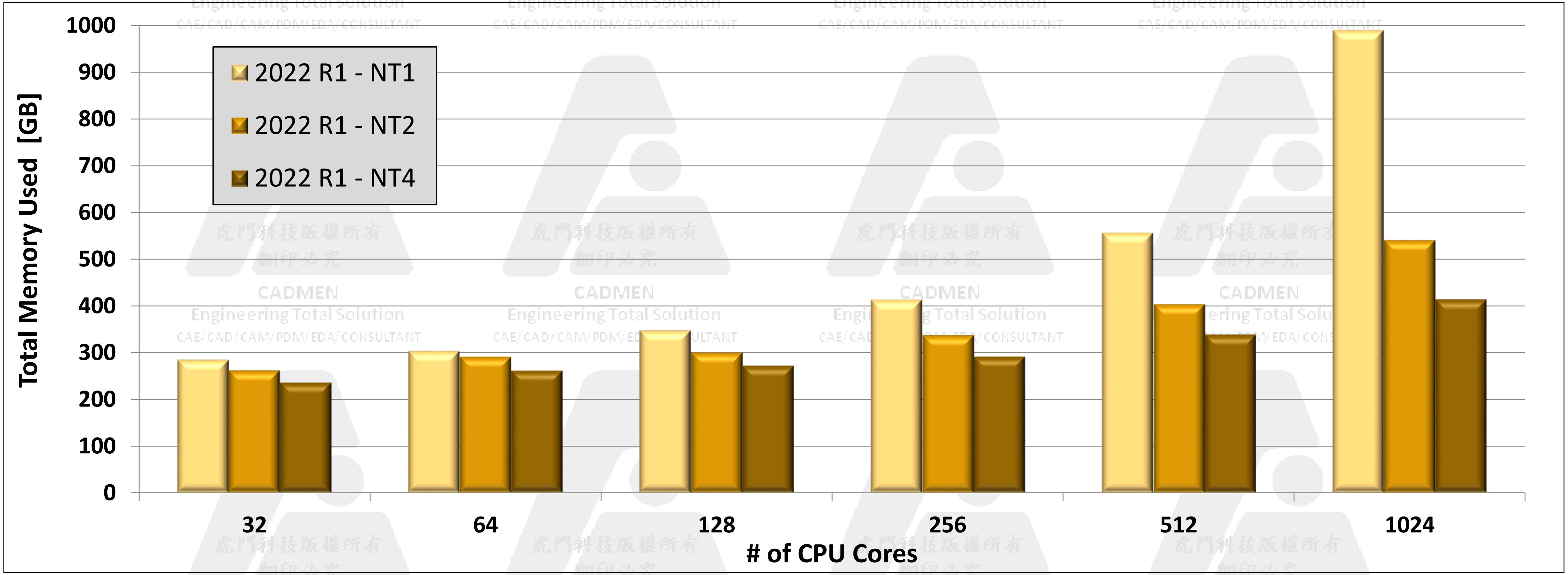
Distributed Ansys Enhancements

- Hybrid parallel
  - Wide applicability → Works for all features supported in DMP mode
  - Supported with all platforms and MPI libraries
  - Reduces memory requirements → Run larger jobs on your cluster
  - Improves scalability → Utilize more cores in your compute nodes



Distributed Ansys Enhancements

- Hybrid parallel reduces total memory requirements (Engine benchmark)

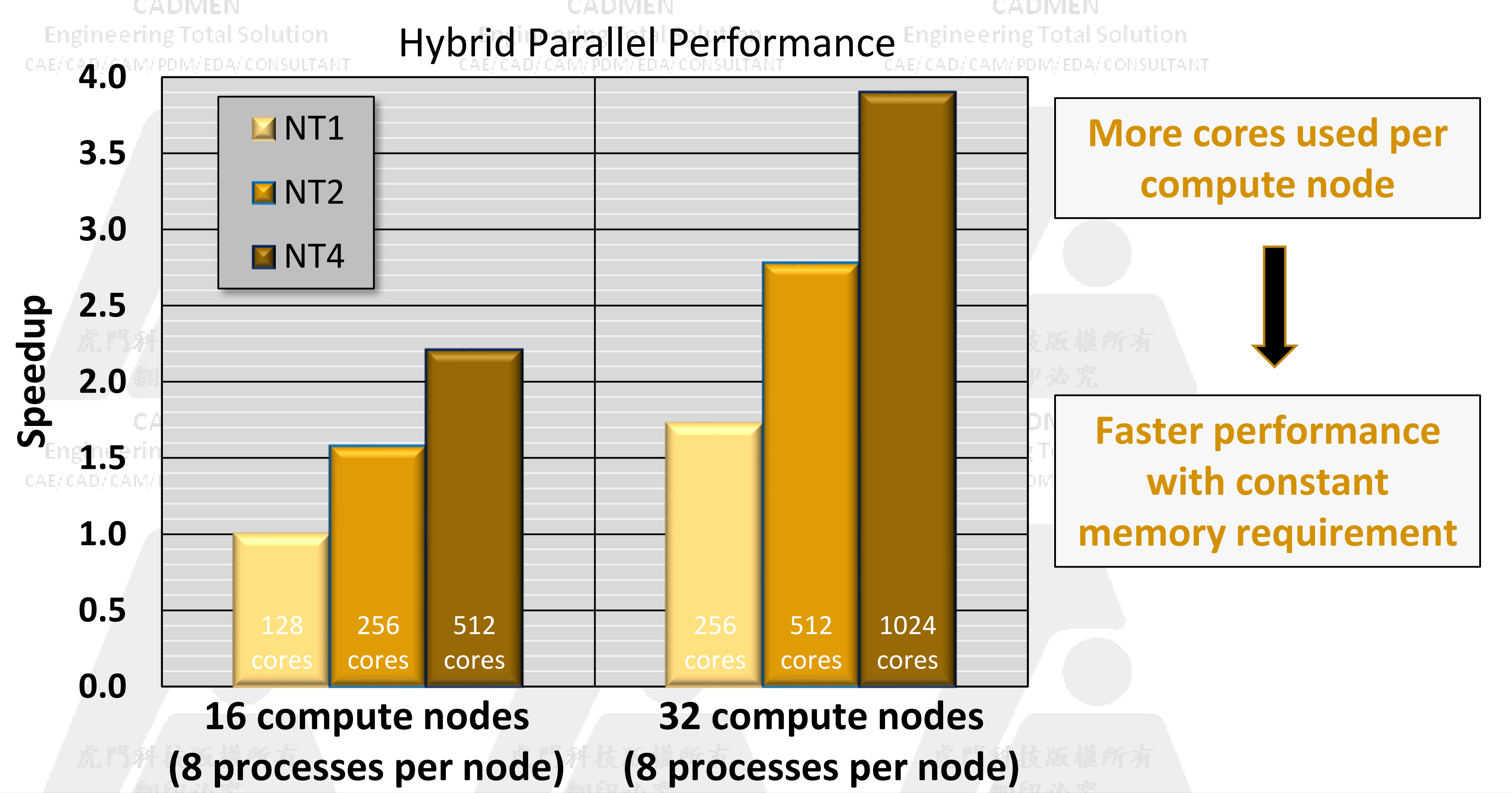


Ansys

Distributed Ansys Enhancements

- Hybrid parallel → use more cores per compute node (equal RAM use)

- 88 million DOF; sparse solver
- Nonlinear static analysis involving contact, large deflections, 126 Newton-Raphson iterations to converge
- Linux cluster; each compute node contains 2 Intel Xeon Platinum 8168 processors (44 cores), 346GB RAM, SSD, CentOS 7.9



More cores used per compute node

↓

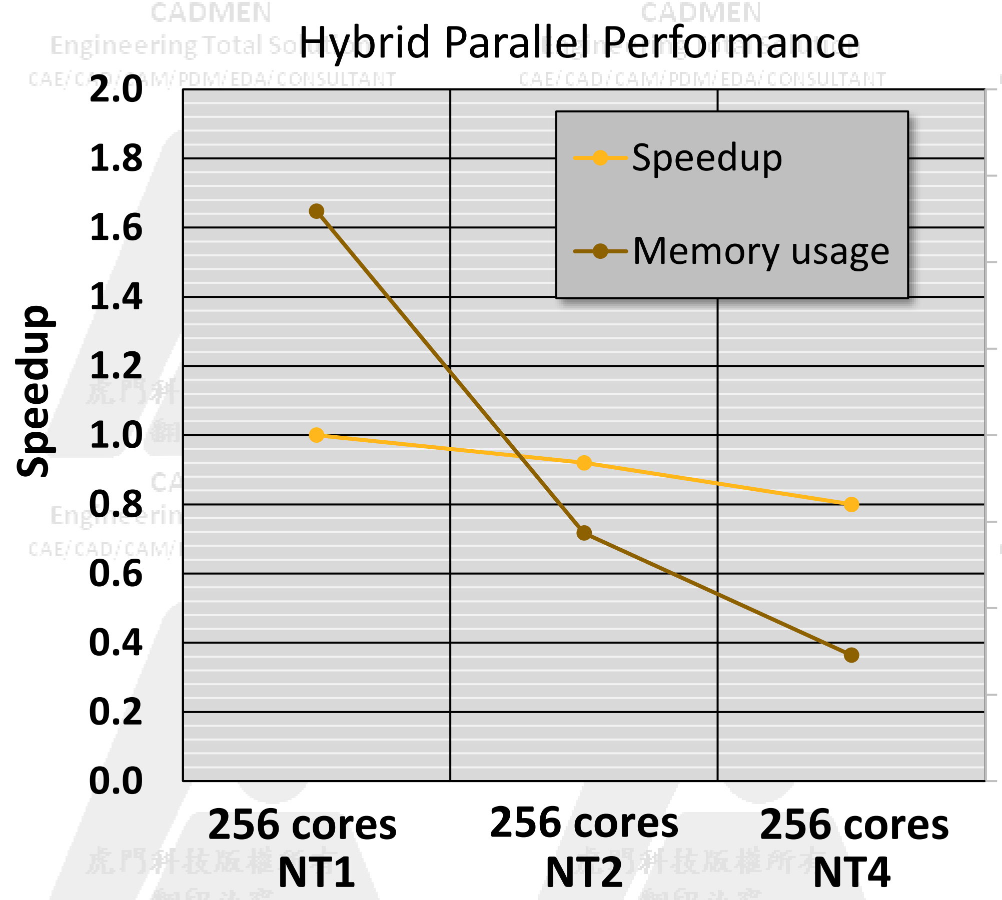
Faster performance with constant memory requirement

Ansys

Distributed Ansys Enhancements

- Hybrid parallel → reduce memory and hardware required (same core count)

- 88 million DOF; sparse solver
- Nonlinear static analysis involving contact, large deflections, 126 Newton-Raphson iterations to converge
- Linux cluster; each compute node contains 2 Intel Xeon Platinum 8168 processors (44 cores), 346GB RAM, SSD, CentOS 7.9



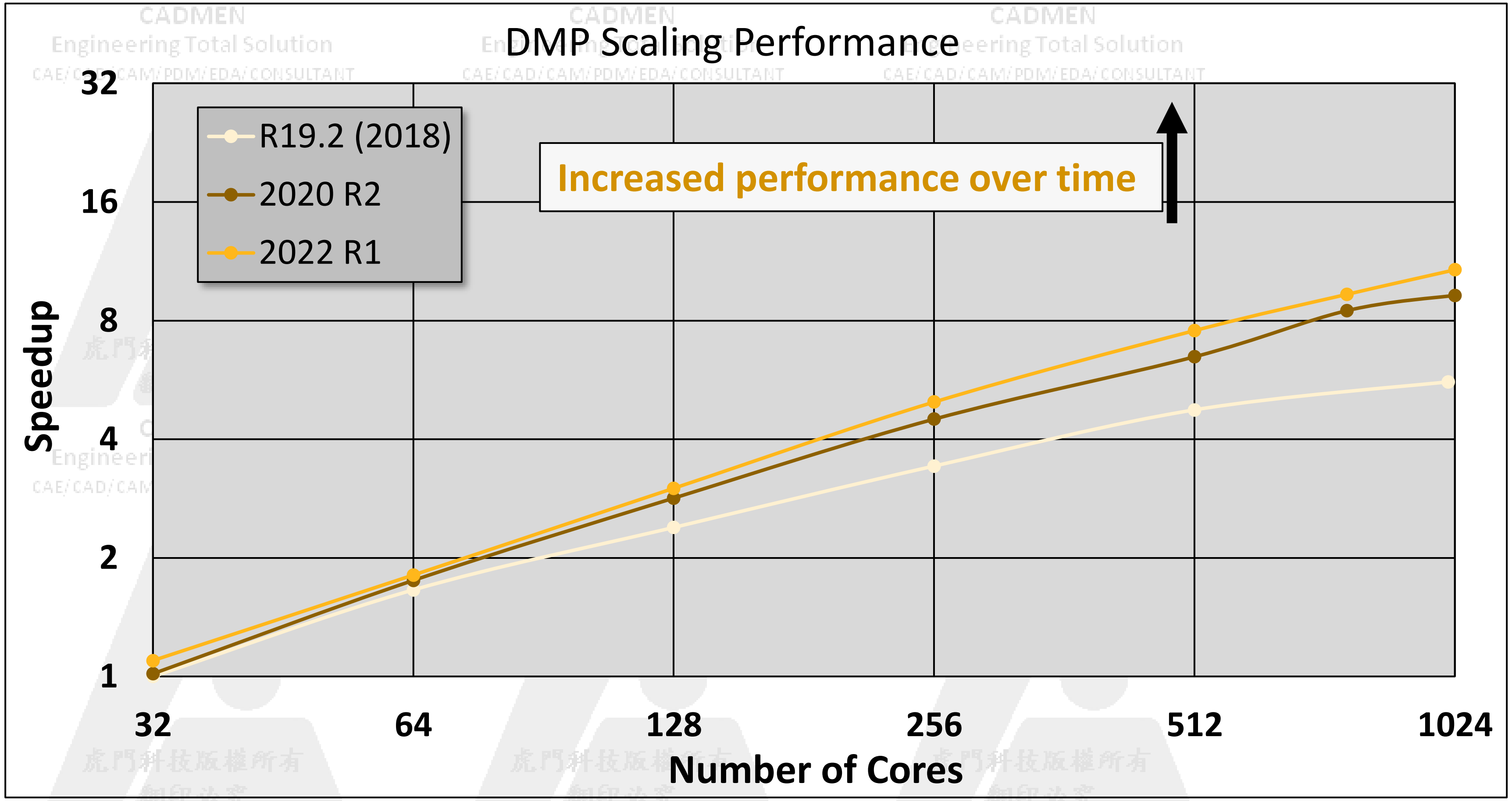
Less MPI processes with more threads per process

Slightly reduced performance but significantly less memory usage and fewer compute nodes required

Distributed Ansys Enhancements

- Improved scaling at higher core counts

- 5.6 million DOF; sparse solver
- Nonlinear static analysis involving contact, constraint equations, unsymmetric matrices
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6



Ansys

Distributed Ansys Enhancements

- Reduced memory usage at higher core counts

- 5.6 million DOF; sparse solver
- Nonlinear static analysis involving contact, constraint equations, unsymmetric matrices
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6