

CAD MEN 虎門科技 | Ansys 精英合作夥伴

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

Mechanical

**Mesh Reinforcement Optimization
and HPC**

虎門技術團隊
講師: 王弘政

Mathias Wang (M.S.)



Ansys



Meshing

Ansys

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

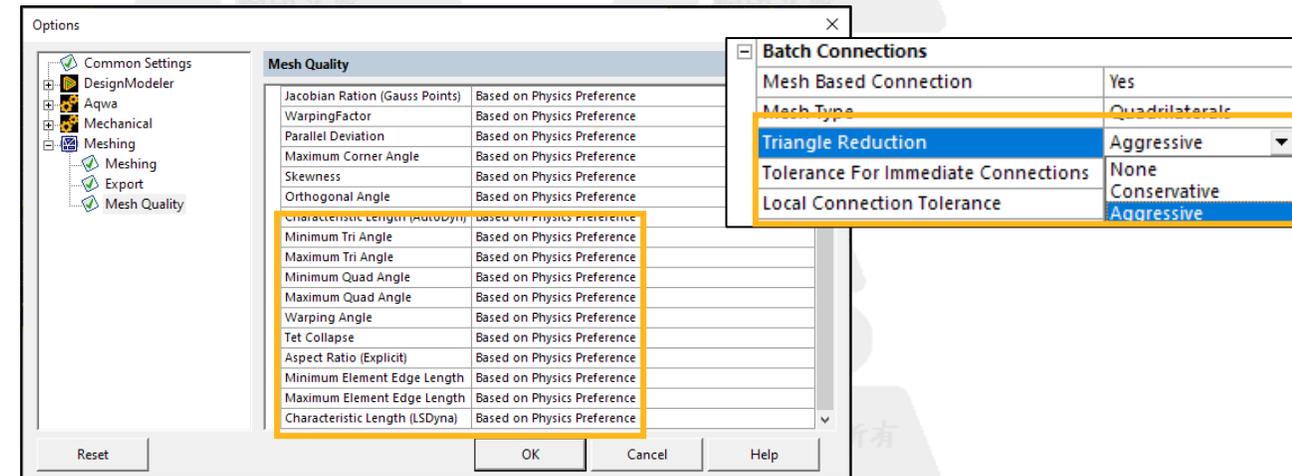
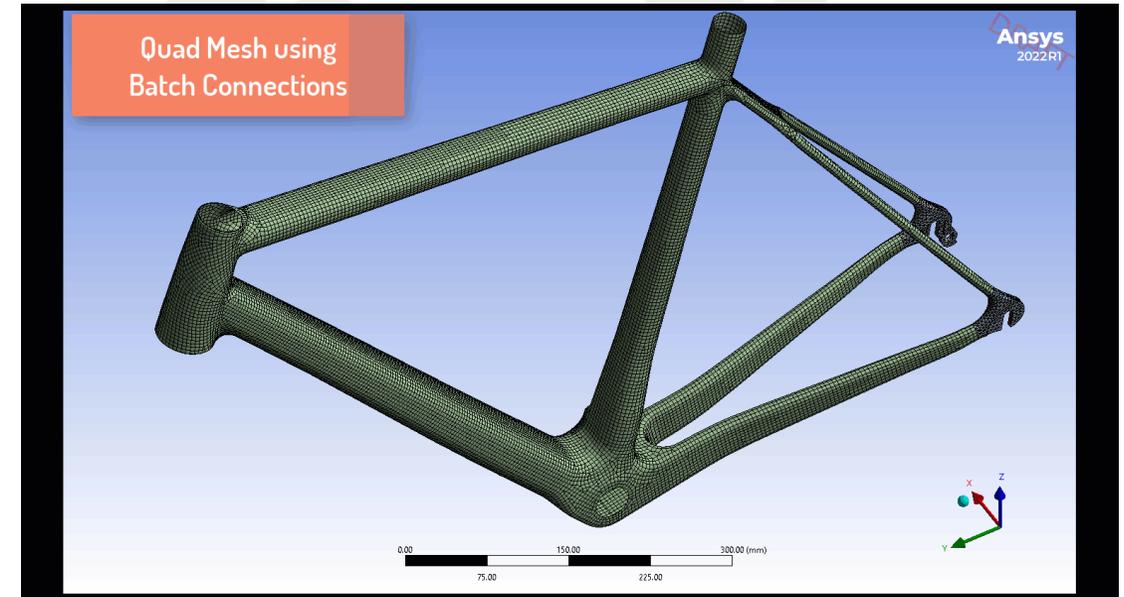
- Improved pave mesh in Multizone
- Improved default mesh with MultiZone for simple shapes e.g. cylinder, sphere, pipe, etc
- Split Angle for less decomposition
- Body-Fitted Cartesian
 - Support for Edge Sizing
- Beta Multizone Options:
 - CartSweep Decomposition (2.5D Geometry)
 - ThinSweep Decomposition (Thin Geometry)

• SpaceClaim Meshing

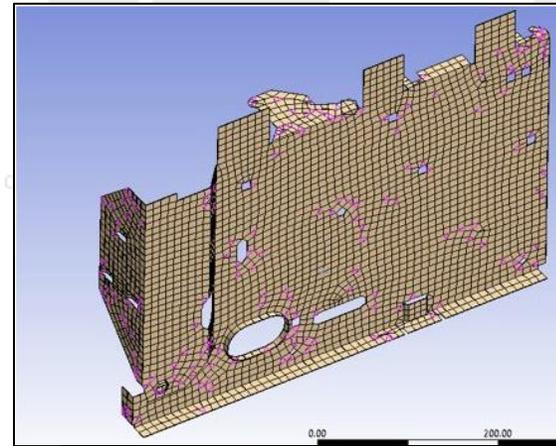
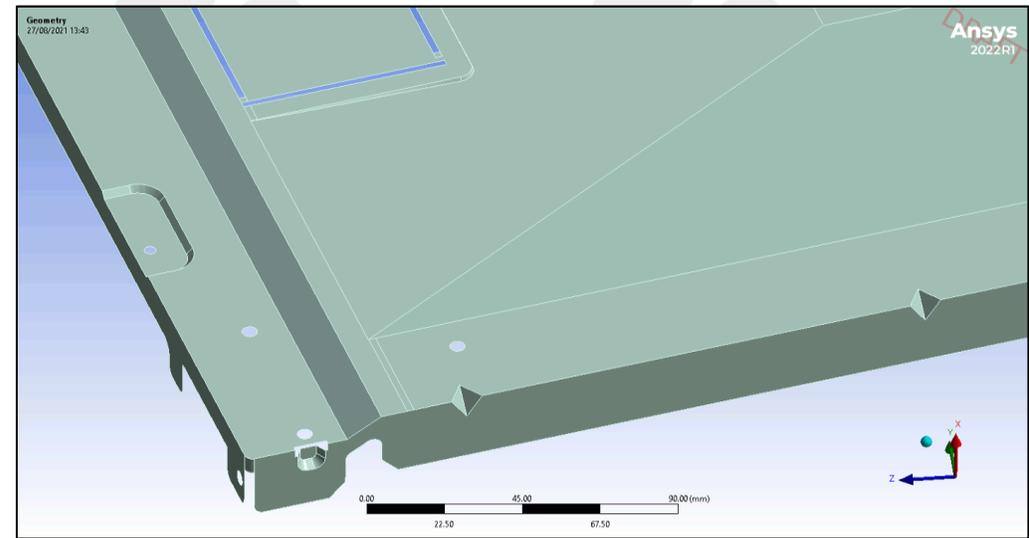
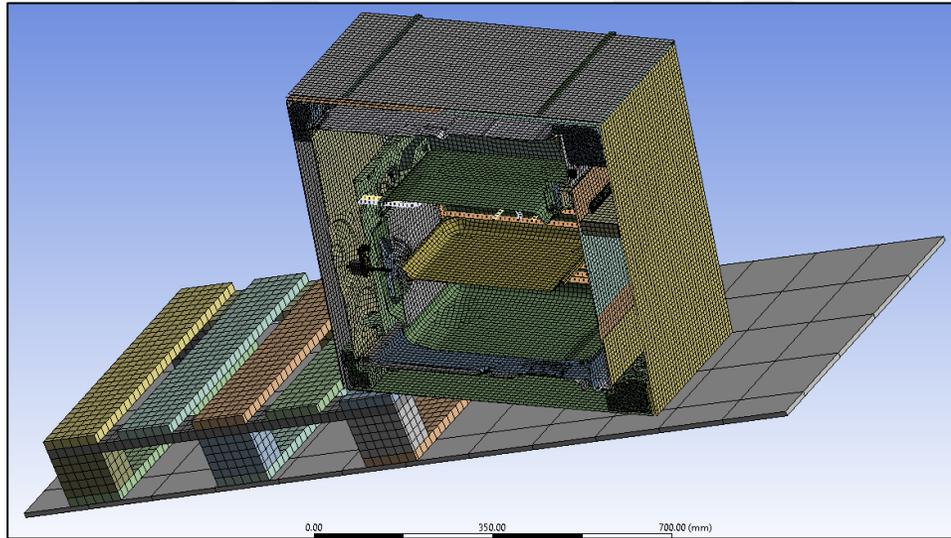
- Thin Body Meshing
- Robustness, Performance, Usability
- Meshing for Explicit Improvements

Shell Meshing: Triangle Reduction & Quality

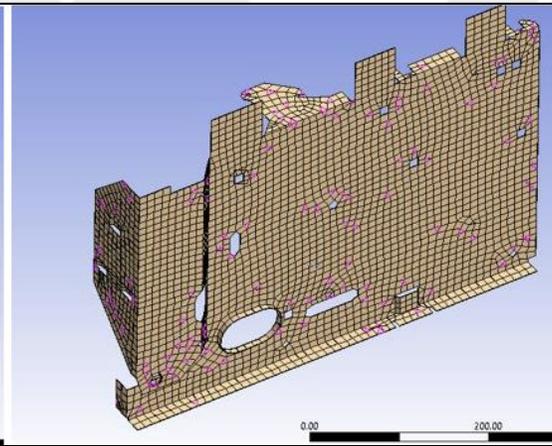
- Available with Batch Connections
- Option to control level of triangles in quad mesh
 - None
 - Remove triangles near shell edges
 - Conservative (default)
 - Aggressive
 - Remove as many triangles as possible sometimes at cost of quality
 - Seen to reduce tri count by up to 80% in some cases and below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



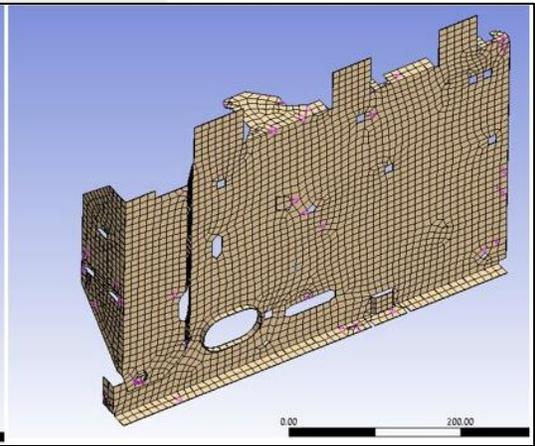
Steamer Shell Mesh: Tri Reduction



Tri Reduction = None
Number of tri = 184



Tri Reduction = Conservative
Number of tri = 126



Tri Reduction = Aggressive
Number of tri = 48

虎門科技版權所有
翻印必究

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.

✓	Step	Weld Curve	Edge Mesh Size (mm)	Weld Angle (°)	Offset Layer Height (mm)	Number Of Layers	Offset
<input checked="" type="checkbox"/>	1	LineBody1	75.	45.	75.	1	
<input checked="" type="checkbox"/>	2	LineBody2	75.	45.	75.	1	
<input checked="" type="checkbox"/>	3	LineBody3	75.	45.	75.	1	
<input checked="" type="checkbox"/>	4	LineBody4	75.	45.	75.	1	
<input checked="" type="checkbox"/>	5	LineBody5	75.	45.	75.	1	
<input checked="" type="checkbox"/>	6	LineBody6	75.	45.	75.	1	
<input checked="" type="checkbox"/>	7	LineBody7	75.	45.	75.	1	
<input checked="" type="checkbox"/>	8	LineBody8	75.	45.	75.	1	
<input checked="" type="checkbox"/>	9	LineBody9	75.	45.	75.	1	
<input checked="" type="checkbox"/>	10	LineBody10	75.	45.	75.	1	
<input checked="" type="checkbox"/>	11	LineBody11	75.	45.	75.	1	
<input checked="" type="checkbox"/>	12	LineBody12	75.	45.	75.	1	

Context Menu:

- Add
- Delete
- Activate all Selections
- Deactivate all Selections
- Go To Selected Items in Tree
- Promote to Weld Control

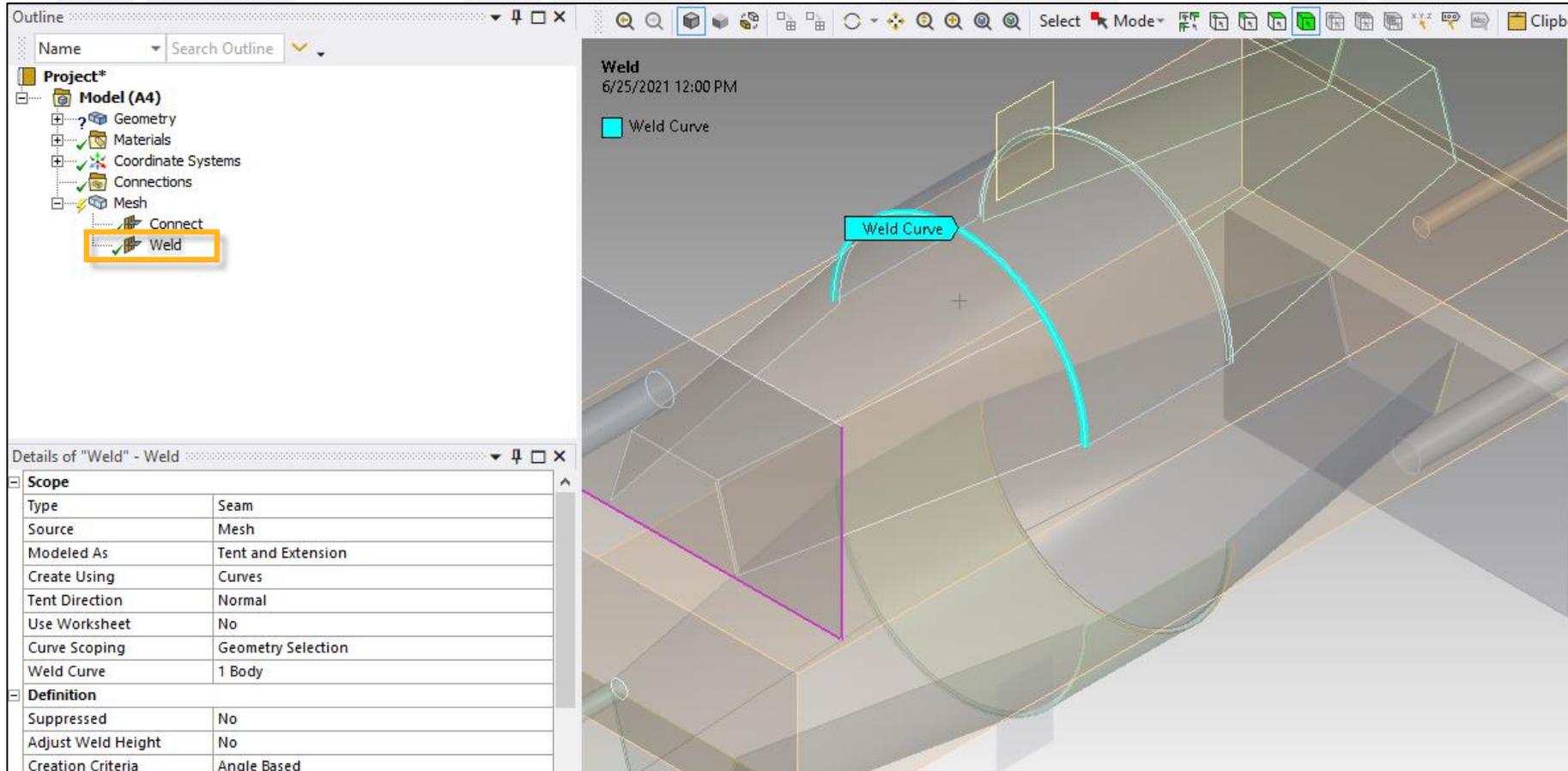
✓	Step	Weld Curve	Edge Mesh Size (mm)	Weld Angle (°)	Offset Layer Height (mm)	Number Of Layers	Offset
<input checked="" type="checkbox"/>	1	LineBody1	75.	45.	75.	1	
<input checked="" type="checkbox"/>	2	LineBody2	75.	45.	75.	1	
<input checked="" type="checkbox"/>	3	LineBody3	75.	45.	75.	1	
<input checked="" type="checkbox"/>	4	LineBody4	75.	45.	75.	1	
<input checked="" type="checkbox"/>	5	LineBody5	75.	45.	75.	1	
<input checked="" type="checkbox"/>	6	LineBody6	75.	45.	75.	1	
<input checked="" type="checkbox"/>	7	LineBody7	75.	45.	75.	1	
<input checked="" type="checkbox"/>	8	LineBody8	75.	45.	75.	1	
<input checked="" type="checkbox"/>	9	LineBody9	75.	45.	75.	1	
<input checked="" type="checkbox"/>	10	LineBody10	75.	45.	75.	1	
<input checked="" type="checkbox"/>	11	LineBody11	75.	45.	75.	1	
<input checked="" type="checkbox"/>	12	LineBody12	75.	45.	75.	1	

Outline:

- Project*
- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Mesh
 - Connect
 - Weld
 - Weld 2

Highlighting Weld Curve with Annotations

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



虎門科技版權所有
翻印必究

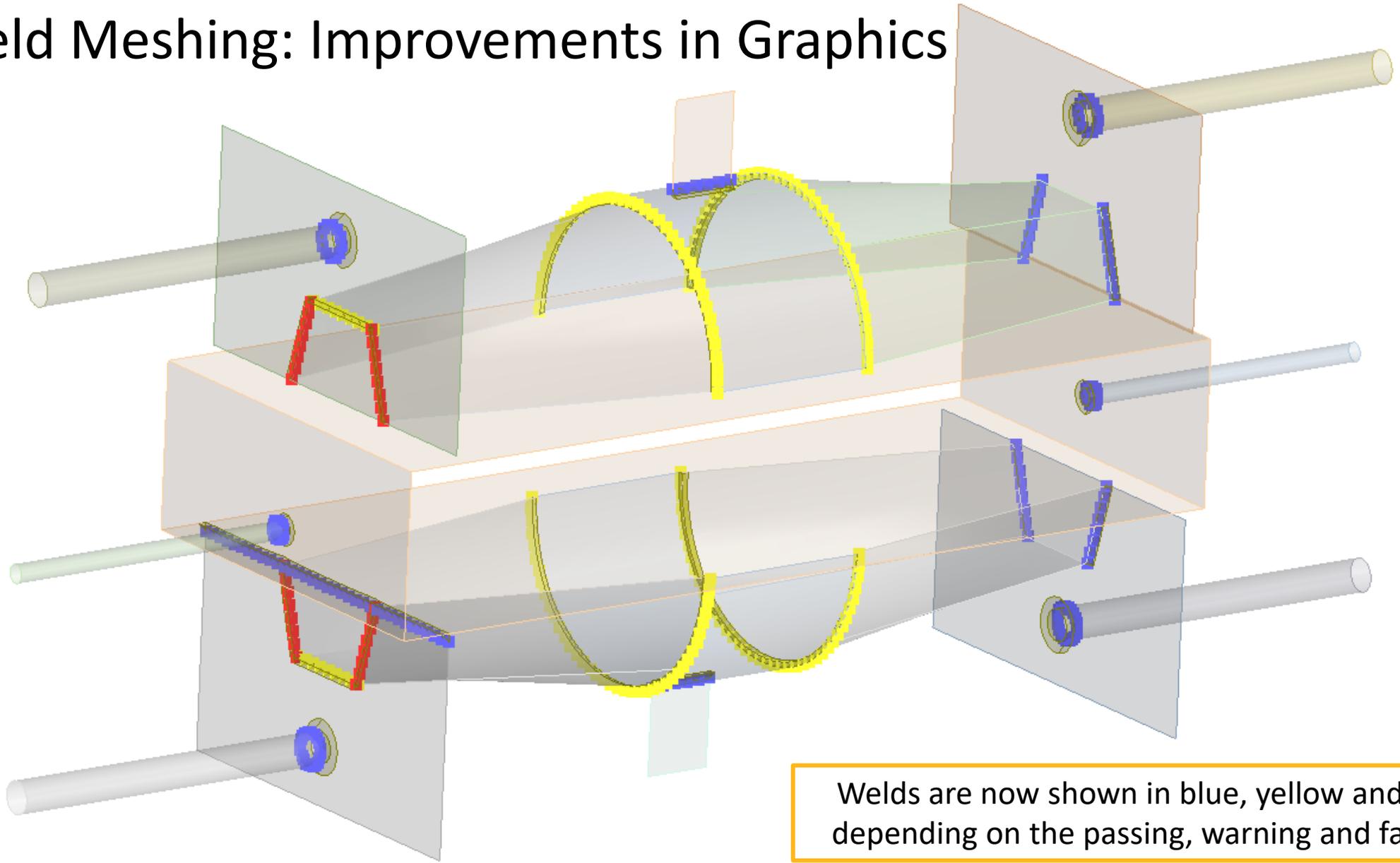
CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT

Ansys

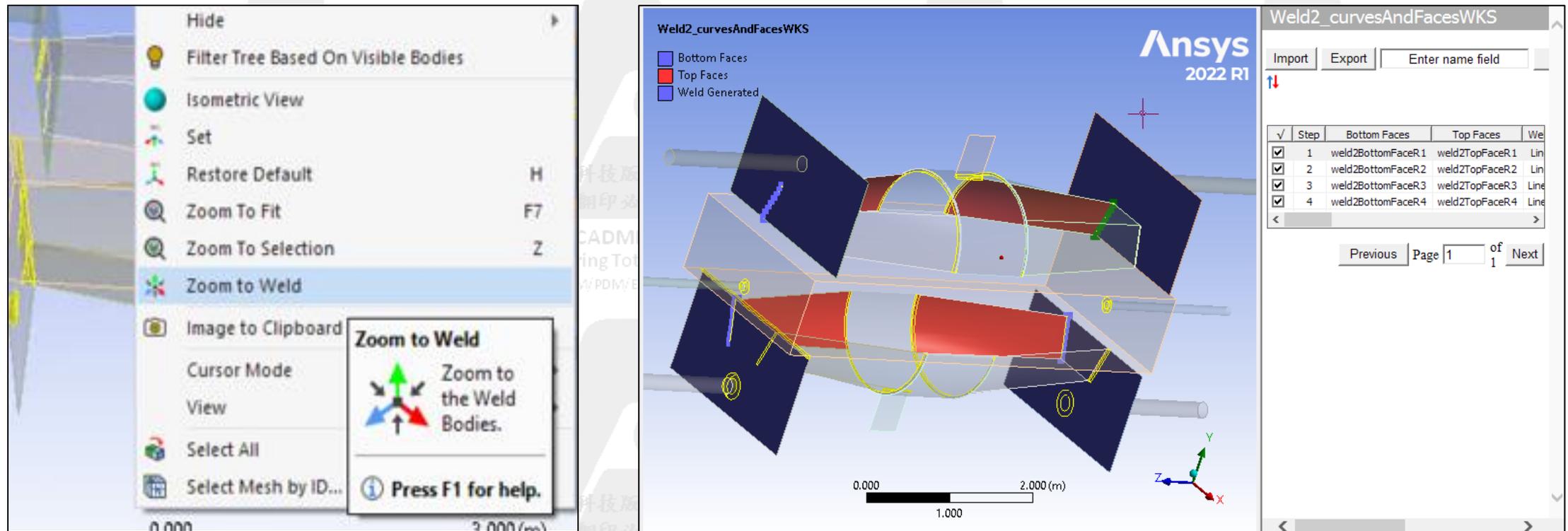
Weld Meshing: Improvements in Graphics



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

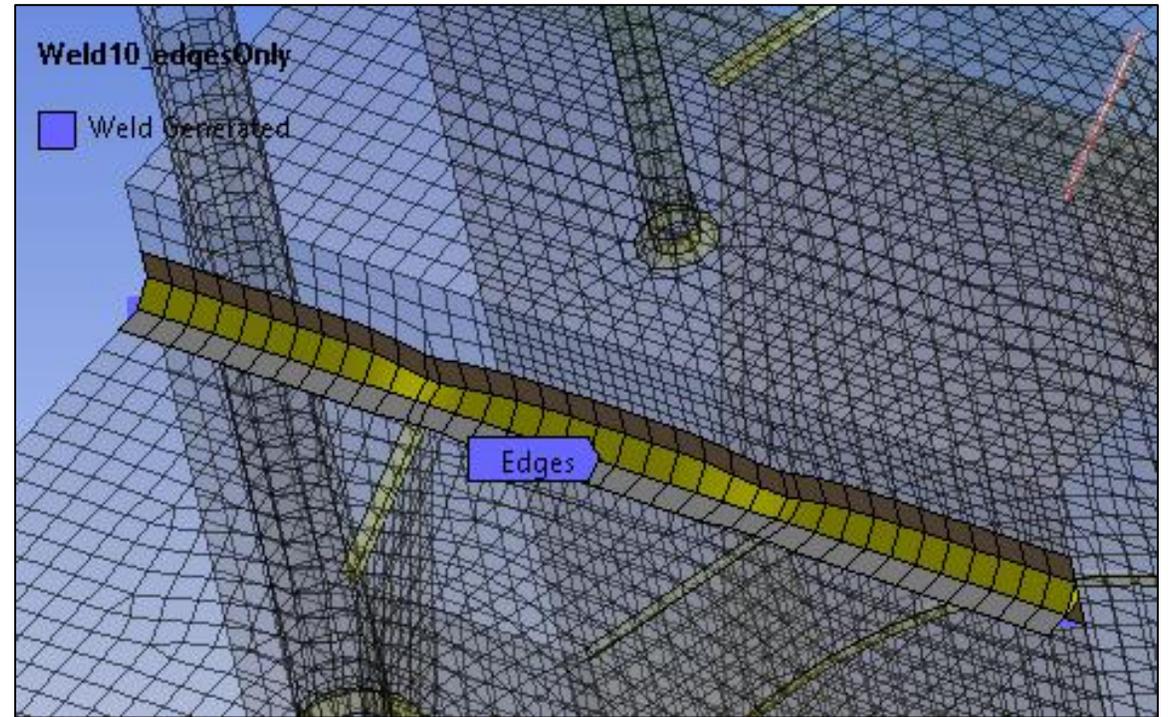
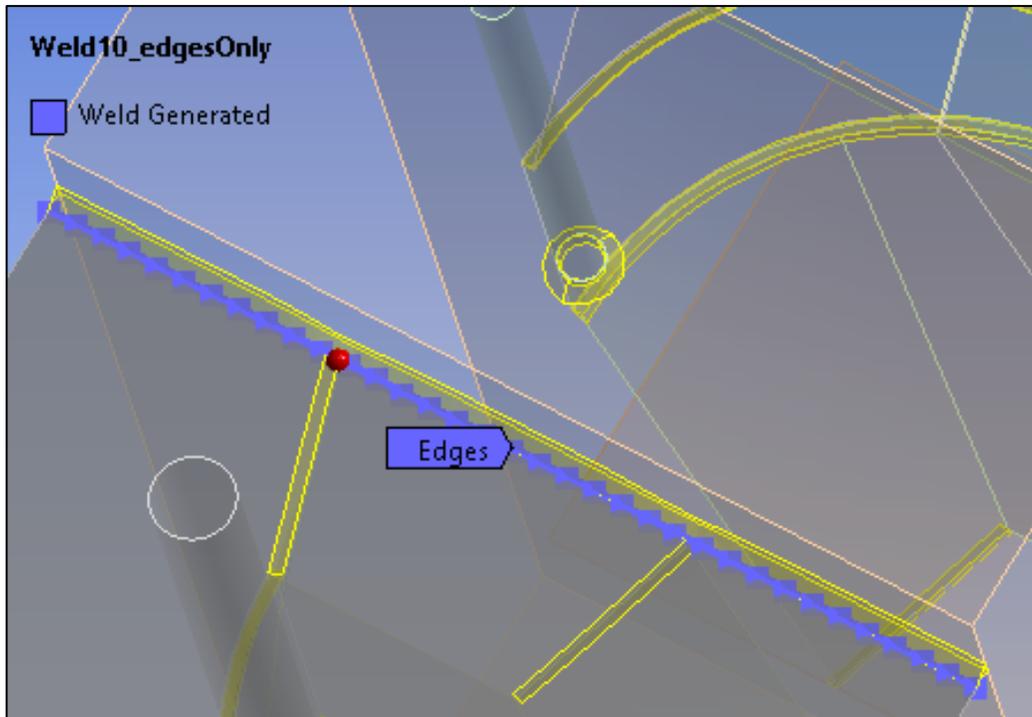
Weld Meshing: Improvements in Graphics

- “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



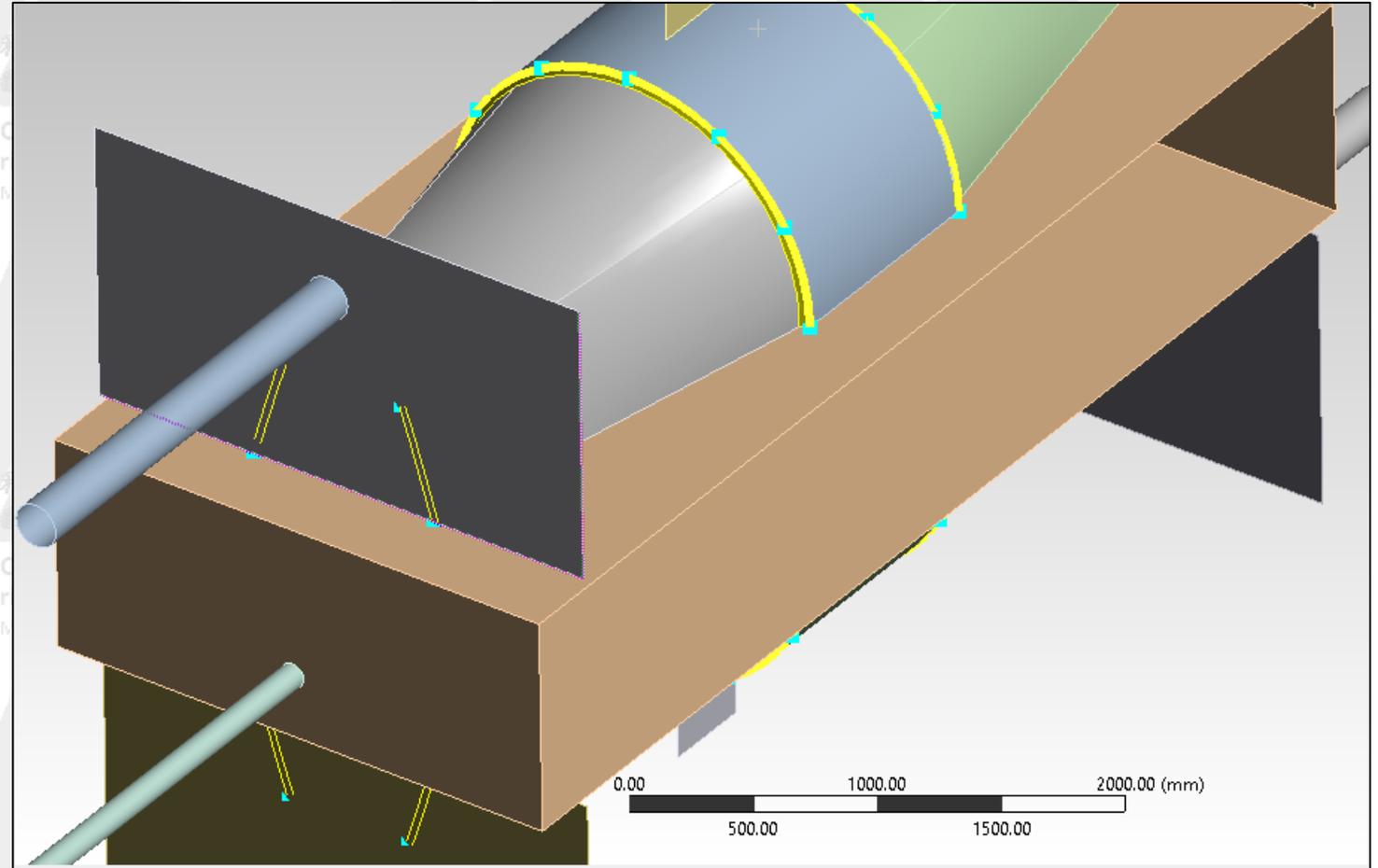
Weld Meshing: Improvements in Graphics

- 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



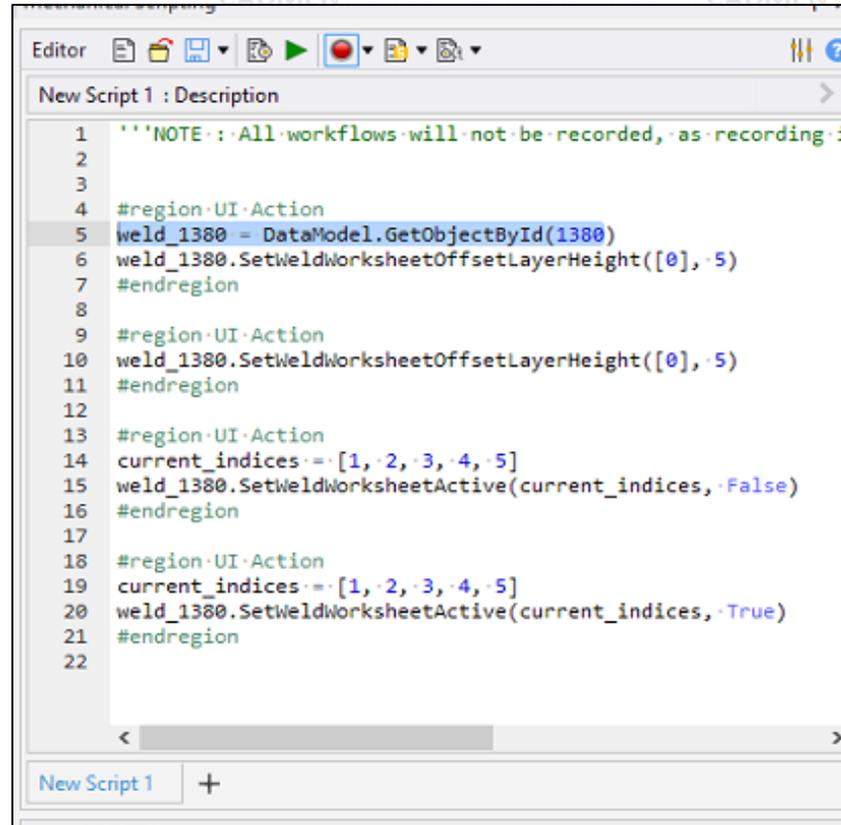
Preview Weld

- 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



```
1 '''NOTE: All workflows will not be recorded, as recording is disabled'''
2
3
4 #region UI Action
5 weld_1380 = DataModel.GetObjectById(1380)
6 weld_1380.SetWeldWorksheetOffsetLayerHeight([0], 5)
7 #endregion
8
9 #region UI Action
10 weld_1380.SetWeldWorksheetOffsetLayerHeight([0], 5)
11 #endregion
12
13 #region UI Action
14 current_indices = [1, 2, 3, 4, 5]
15 weld_1380.SetWeldWorksheetActive(current_indices, False)
16 #endregion
17
18 #region UI Action
19 current_indices = [1, 2, 3, 4, 5]
20 weld_1380.SetWeldWorksheetActive(current_indices, True)
21 #endregion
22
```

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

Project*

- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Mesh
 - Connect
 - Weld
 - Named Selections
 - Selection

Details of "Selection"

Scope

Scoping Method	Worksheet
Geometry	500 Elements

Definition

Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Preserve During Solve	No

Statistics

Type	Manual
<input type="checkbox"/> Total Selection	500 Elements
Suppressed	0
Used by Mesh Worksheet	No

Tolerance

Tolerance Type	Program Controlled
Zero Tolerance	1.e-008
Relative Tolerance	1.e-003
Angular Tolerance	1. °

Worksheet

Selection

Generate

Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, ka, N, s, V, A)

	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound
<input checked="" type="checkbox"/>	Add	Mesh Element	Seam Weld HAZ1	Equal	N/A	All Welds	N/A

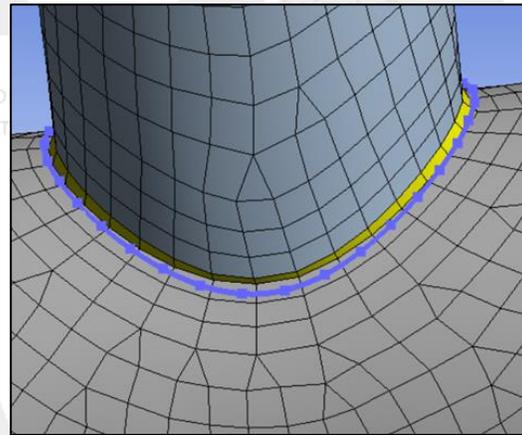
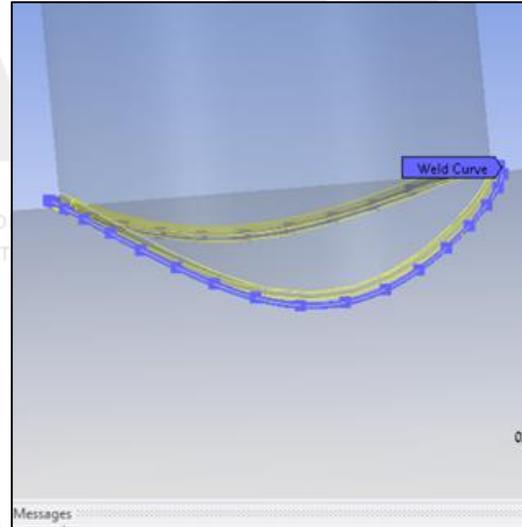
Weld Meshing: Criteria Based HAZ Named Selection

Mesh

- Connect
- Weld
- Named Selections

Details of "Weld" - Weld

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



Weld:HAZ1:3

0.00 50.00 100.00

Worksheet

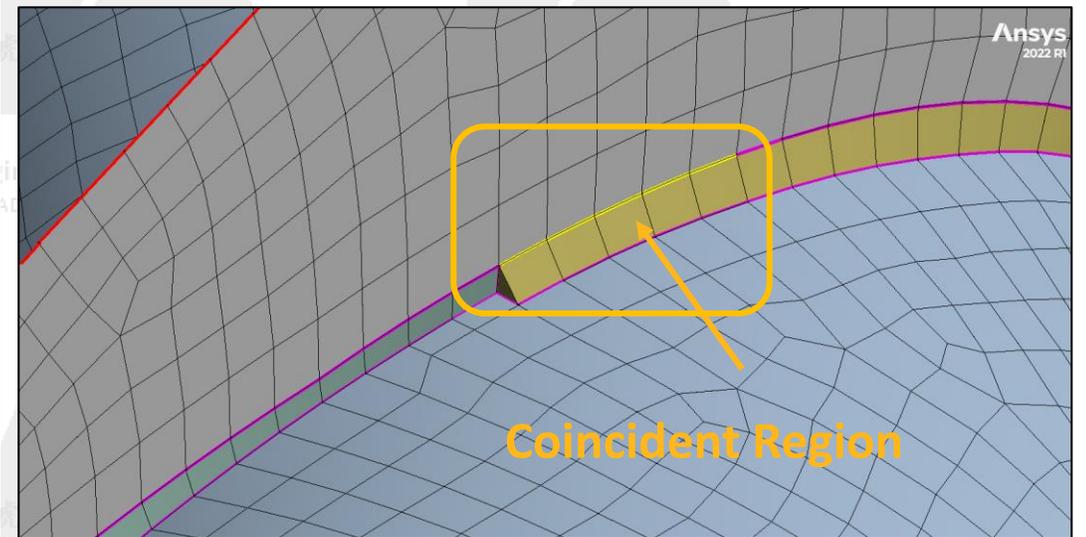
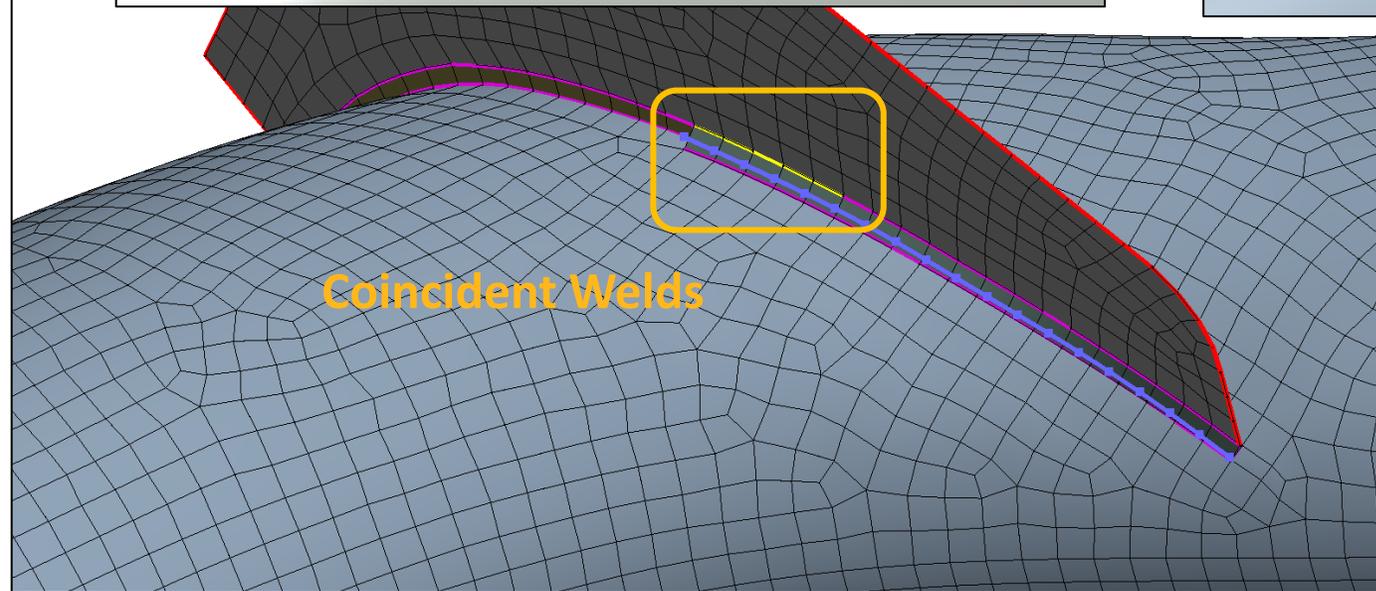
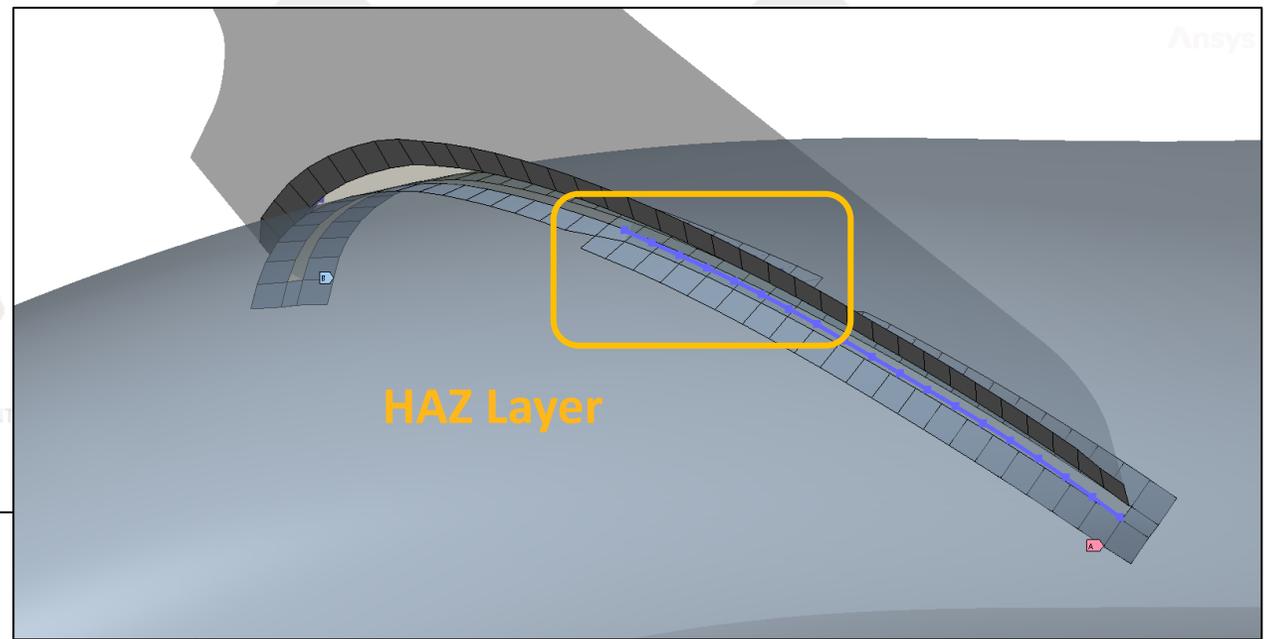
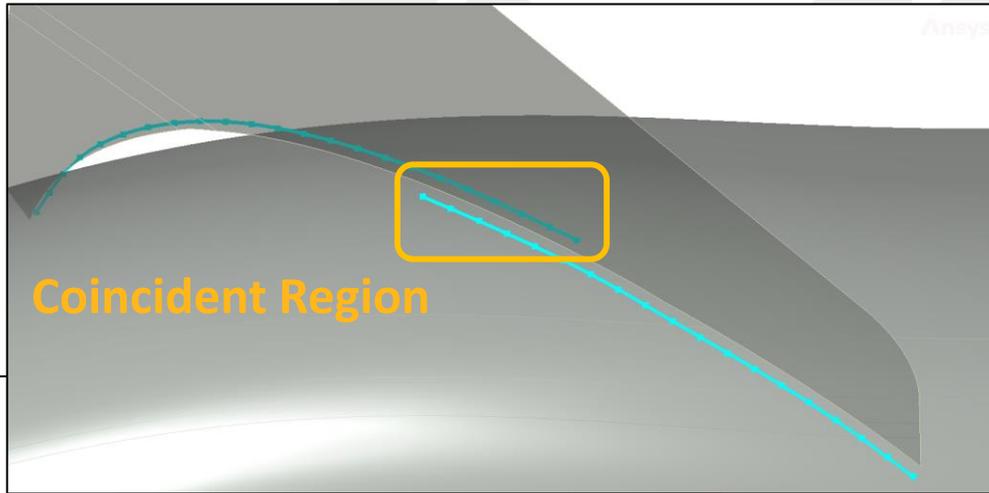
Weld:HAZ1:3

Generate

Note: Internal comparisons of values that have units are done in the Current CAD Unit System: Metric (m, kg, N, s, V, A)

	Action	Entity Type	Criterion	Operator	Units	Value
<input checked="" type="checkbox"/>	Add	Mesh Element	Seam Weld All HAZ	Equal	N/A	All Welds
<input checked="" type="checkbox"/>	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 Axisymmetric bodies. The Line Coating bodies share nodes with the underlying elements

Line Coating bodies sharing nodes with underlying elements

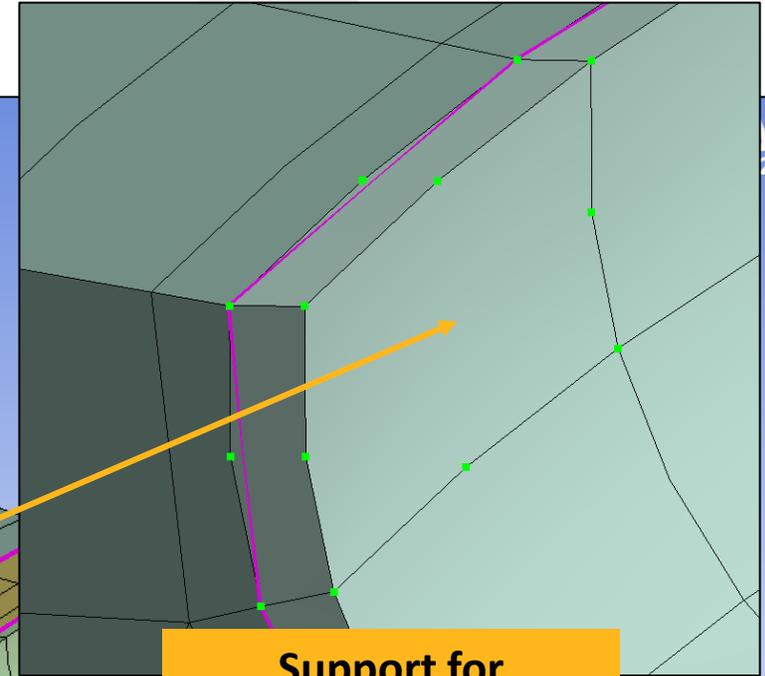
Details of "Pull (Surface Coating)"	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Method	Line Coating
Suppressed	No
Part Properties	
Material	Structural Steel
Stiffness Option	Stress Evaluation Only

Details of "Pull (Surface Coating)"	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Method	Line Coating
Suppressed	No
Part Properties	
Material	Structural Steel
Stiffness Option	Stress Evaluation Only

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

Pull (Extrude): Merge Profile Nodes

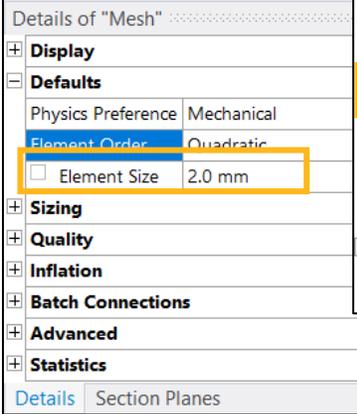
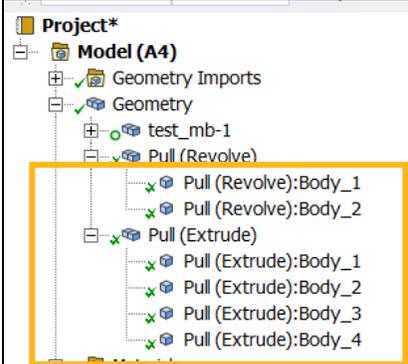
Pull (Extrude) allows user to Merge Profile Nodes, for Extrude by element faces selection on solid bodies with quadratic elements



Support for Quadratic elements

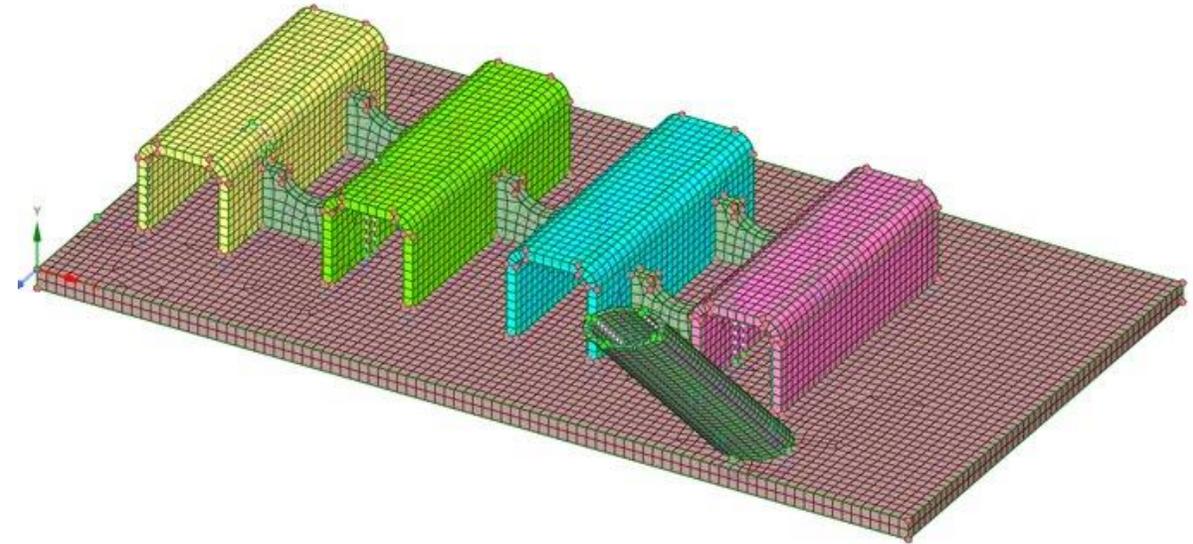
Details of "Pull (Extrude)"	
Scope	
Scoping Method	Named Selection
Named Selection	3D_Coating_Elements
Definition	
Method	Extrude
Height	0.5 mm
Number of Layers	1
Merge Profile Nodes	Yes
Extrude By	Face Normal
Extrude UpTo	No
Suppressed	No
Part Properties	
Material	Structural Steel

Pull Extrude bodies sharing nodes with underlying body element faces



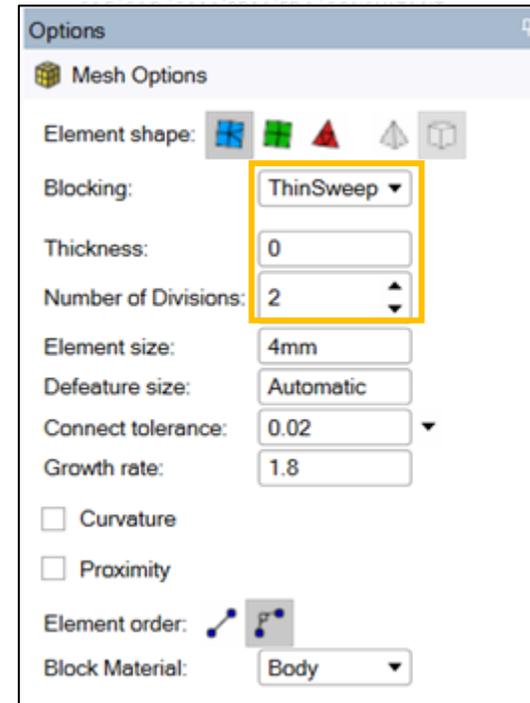
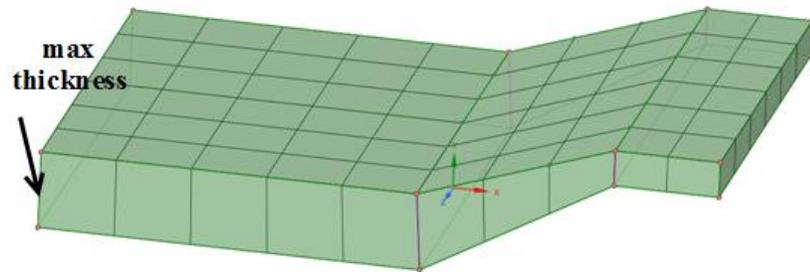
Spaceclaim Meshing: Overview

- 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



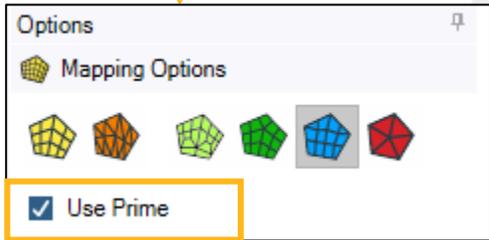
Spaceclaim Meshing: ThinSweep Blocking

- 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.

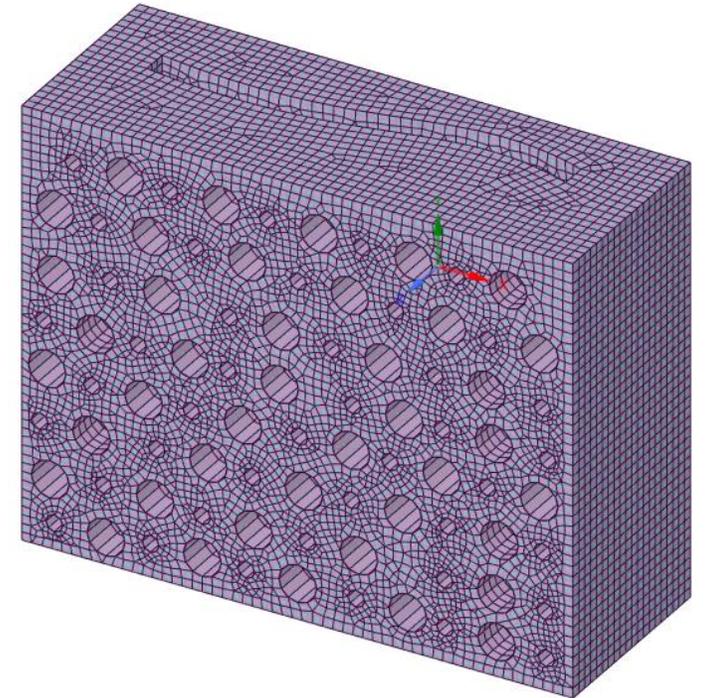
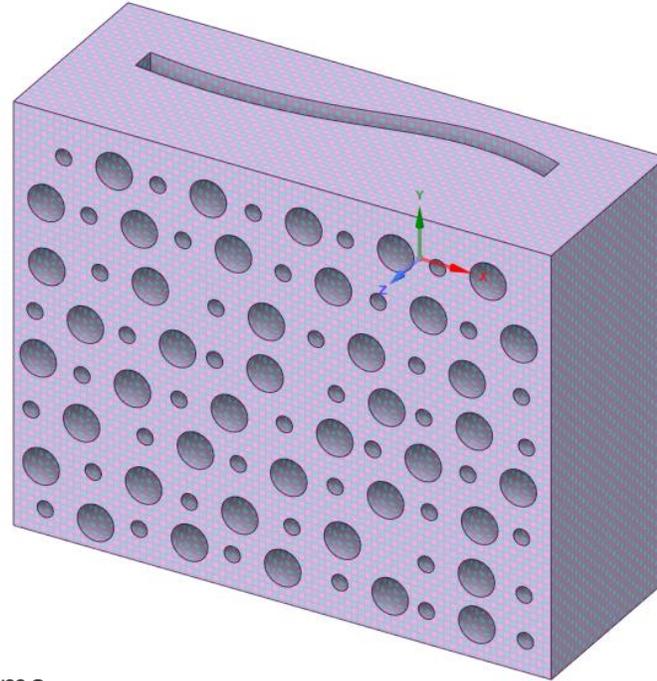


Spaceclaim Meshing: Surface Meshing Performance

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

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

The image shows two screenshots of the meshing process for a bent pipe. The top screenshot shows the 'Structural' physics type selected in the 'Physics Type' dropdown. The resulting mesh is coarser, with a maximum element size of 1.81mm. The bottom screenshot shows the 'Explicit' physics type selected. The resulting mesh is significantly finer and more uniform, with a maximum element size of 2mm and a target characteristic length of 0.5mm. The 'Mesh Options' panel on the left shows settings like Element size (2mm), Defeature size (0.5mm), and Connect tolerance (0.02).

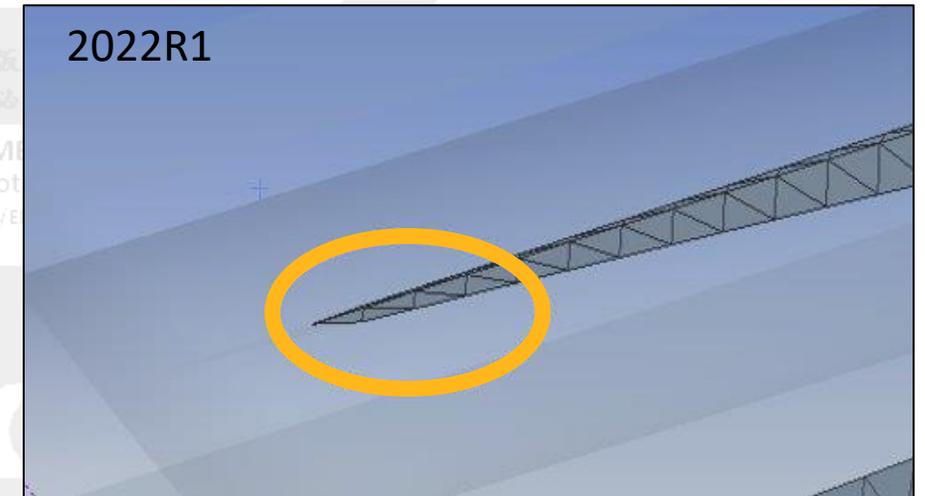
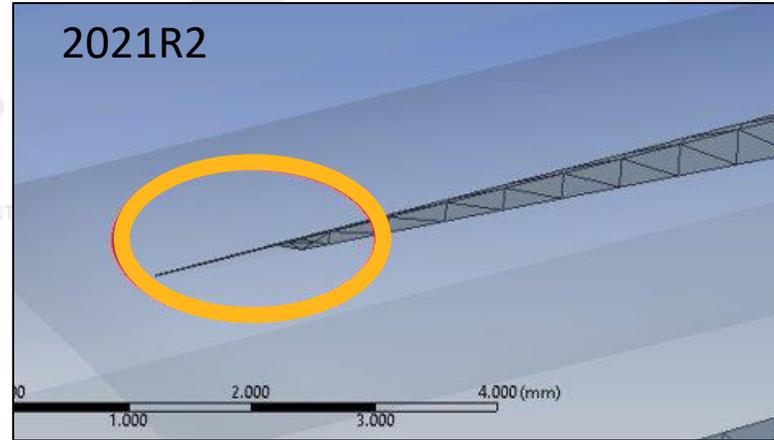
This section illustrates the export of an assembly mesh. On the left, a 3D model of a bolted plate assembly is shown with a mesh. A file dialog box is open, showing the file name 'Bolted_Plate_Assembly' and the save type 'LS-DYNA (*.k)'. In the center, a list of files is shown, including 'Bolt2.scdoc', 'Bolted_Plate_Assembly.scdoc', 'Bot_Plate.scdoc', 'Top_Plate.scdoc', 'Washer1.scdoc', and 'Washer2.scdoc'. On the right, the 'LSP' (Load Step) panel is visible, showing a tree view of the assembly with 'Bolted_Plate_Assembly.k' selected. The tree view includes 'Assembly 1', 'FEM Parts', and various parts like '6577 TOP_PLATE', '13168 BOT_PLATE', '19727 BOLT1', '26292 BOLT2', '32871 WASHER1', and '39452 WASHER2'.

Explicit Meshing
Tetrahedral Meshing
Mesh Diagnostics
Feature Based Meshing



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

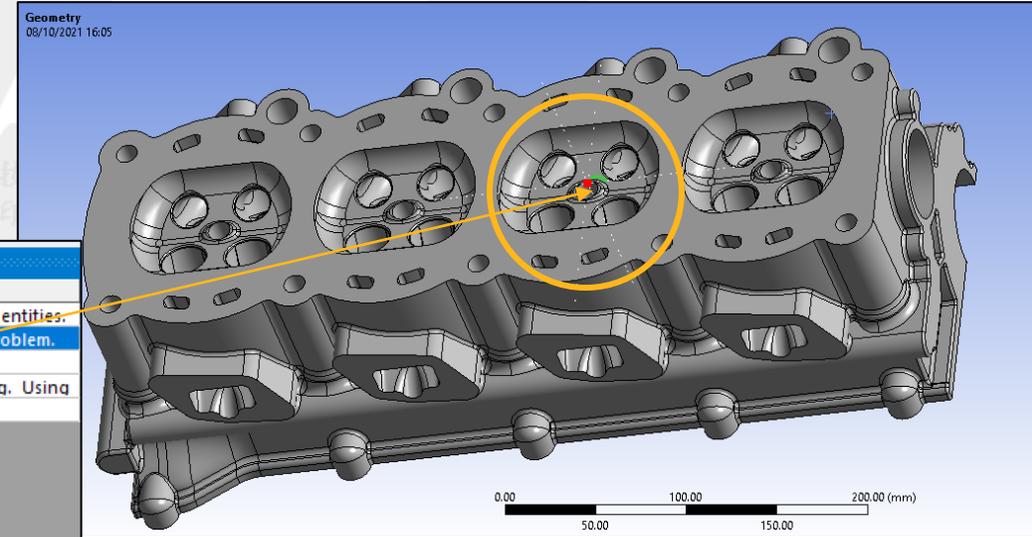


Improved Error Handling

- Intersecting Surface Mesh RMB Options

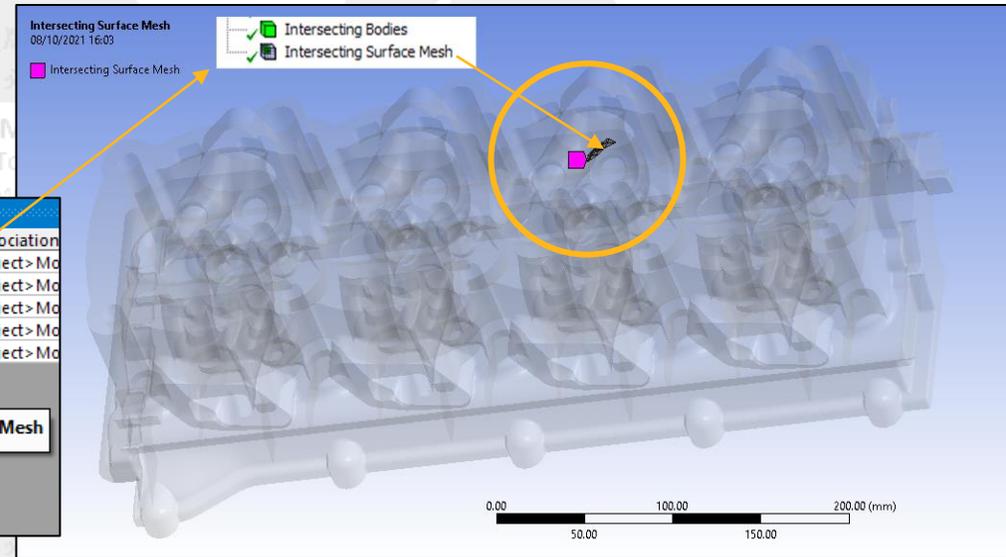
Messages	
	Text
Error	One or more entities failed to mesh. The mesh of the bodies containing these entities may not be up-to-date. However, meshing might be successful on the other entities.
Error	The surface mesh is intersecting or close to intersecting, making it difficult to create a volume mesh. Please adjust the mesh size or adjust the geometry to fix the problem.
Warning	Some boundaries of protected topologies have been defeated. Right click on this message and select "Go To Object" to see the affected edges.
Warning	Quad map meshing failed on one or more surfaces. A surface could be narrow or have been improperly paired up for meshing. Using
Error	A mesh could not be generated using the current meshing options and settings.

- Go To Object
- Show Problematic Geometry
- Show Intersecting Surface Mesh
- Show Message
- Copy
- Delete
- Refresh



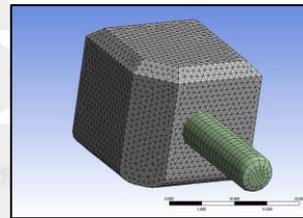
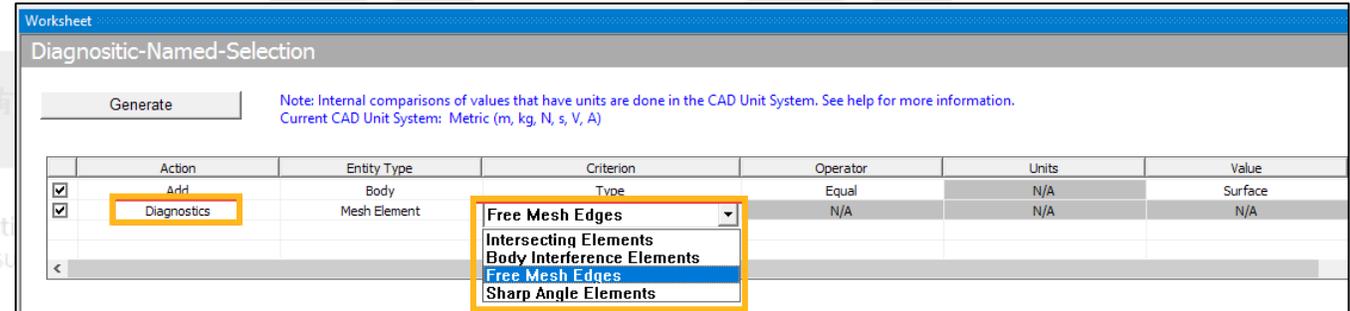
Messages	
	Text
Error	One or more entities failed to mesh. The mesh of the bodies containing these entities may not be up-to-date. However, meshing might be successful on the other entities.
Error	The surface mesh is intersecting or close to intersecting, making it difficult to create a volume mesh. Please adjust the mesh size or adjust the geometry to fix the problem.
Warning	Some boundaries of protected topologies have been defeated. Right click on this message and select "Go To Object" to see the affected edges.
Warning	Quad map meshing failed on one or more surfaces. A surface could be narrow with many boundary edges for meshing. Using
Error	A mesh could not be generated using the current meshing options and settings.

- Go To Object
- Show Problematic Geometry
- Show Intersecting Surface Mesh
- Show Message
- Copy
- Delete
- Refresh

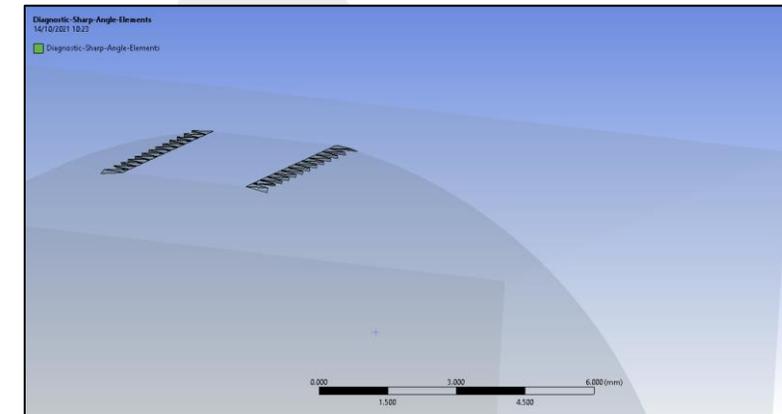
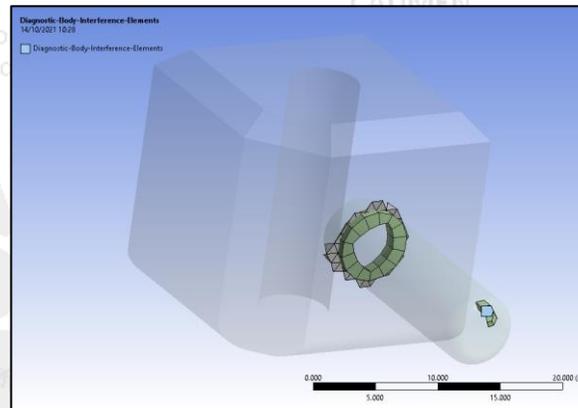


Diagnostics Tools

- 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

Mesh Diagnostics Tools: Defeating Checks

Diagnostic-Defeated-Faces
14/10/2021 11:40

Diagnostic-Defeated-Faces

Worksheet: Diagnostic-Defeated-Faces

Generate

Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

Action	Entity Type	Criterion	Operator	Units	Value
<input checked="" type="checkbox"/>	Body	Type	Equal	N/A	Surface
<input checked="" type="checkbox"/>	Diagonistics	Face	N/A	mm ²	N/A

Diagnostic-Defeated-Edges
14/10/2021 11:44

Diagnostic-Defeated-Edges

Worksheet: Diagnostic-Defeated-Edges

Generate

Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

Action	Entity Type	Criterion
<input checked="" type="checkbox"/>	Body	Type
<input checked="" type="checkbox"/>	Diagonistics	Edge

Diagnostic-Defeated-Vertices

Worksheet: Diagnostic-Defeated-Vertices

Generate

Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

Action	Entity Type	Criterion
<input checked="" type="checkbox"/>	Body	Type
<input checked="" type="checkbox"/>	Diagonistics	Vertex

Named Selections

- Insert
- Generate Named Selection
- Select Items in Group
- Add to Current Selection
- Remove from Current Selection
- Create Nodal Selection
- Suppress Body
- Hide Face(s)

Select Items in Group
Selects all entities in the named selection.

Press F1 for help.

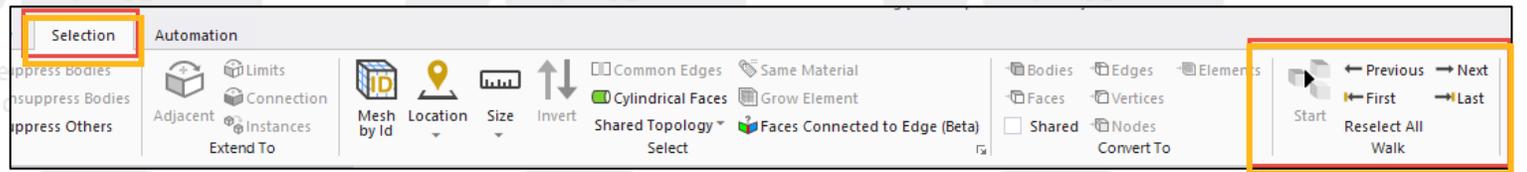
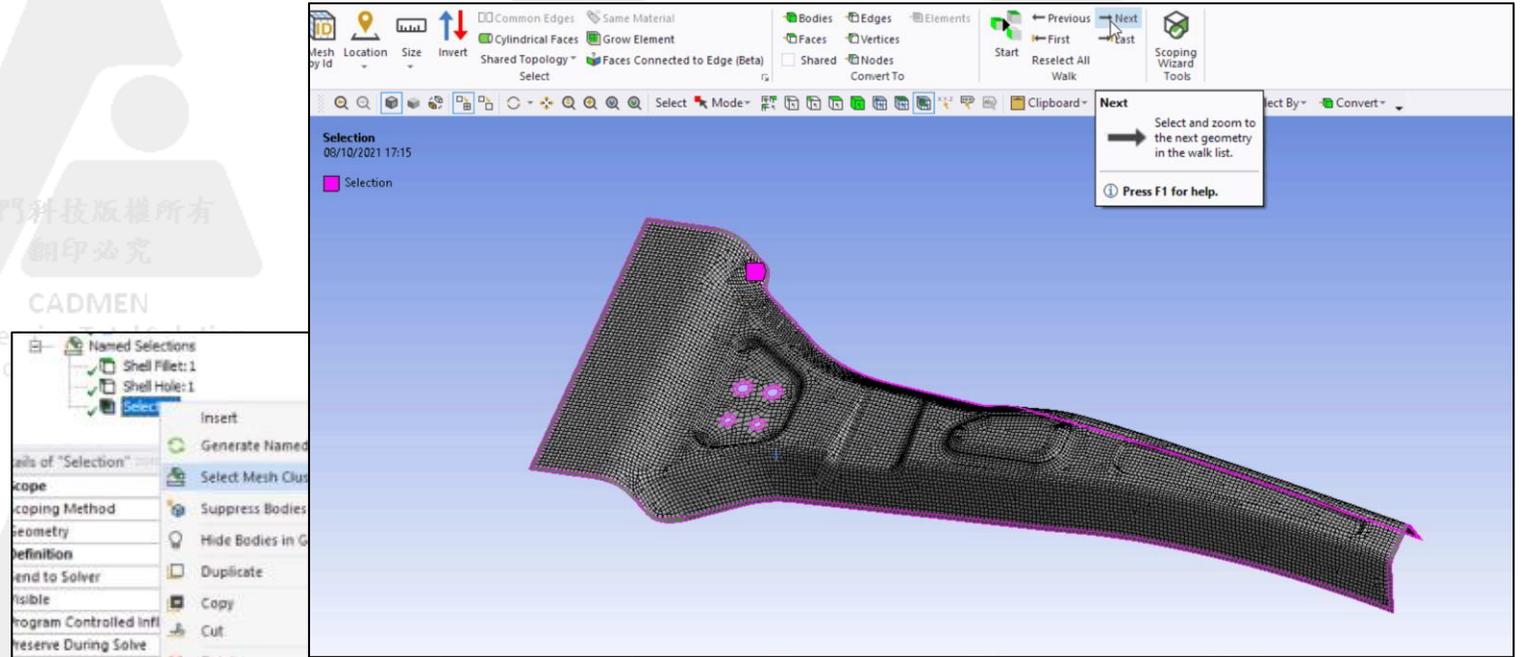
Selection Walk

- ← Previous → Next
- ↶ First ↷ Last
- Start
- Reselect All
- Walk

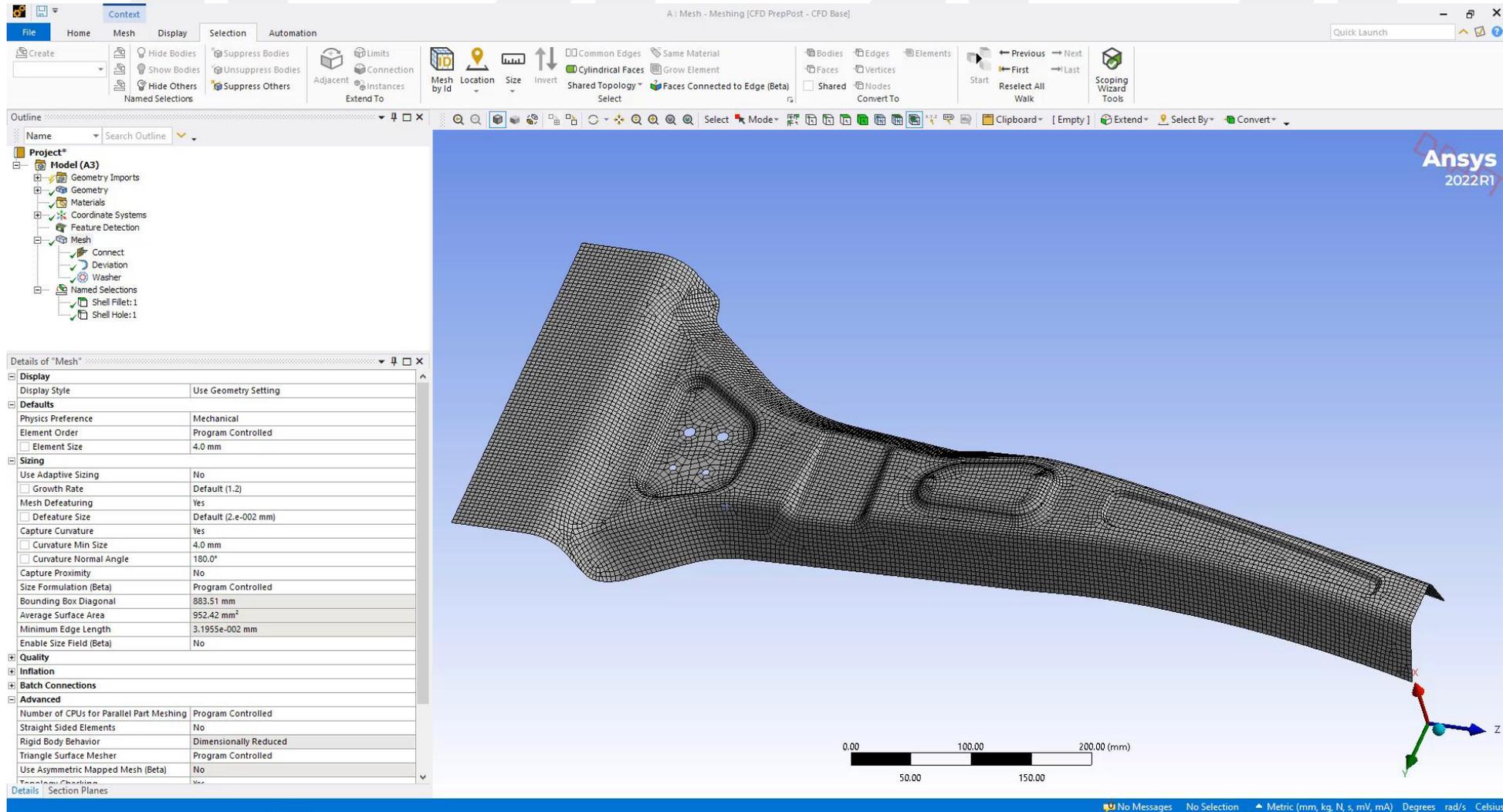
- Detect and examine where defeating has occurred after meshing
- Use Selection Walk to traverse defeated faces

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



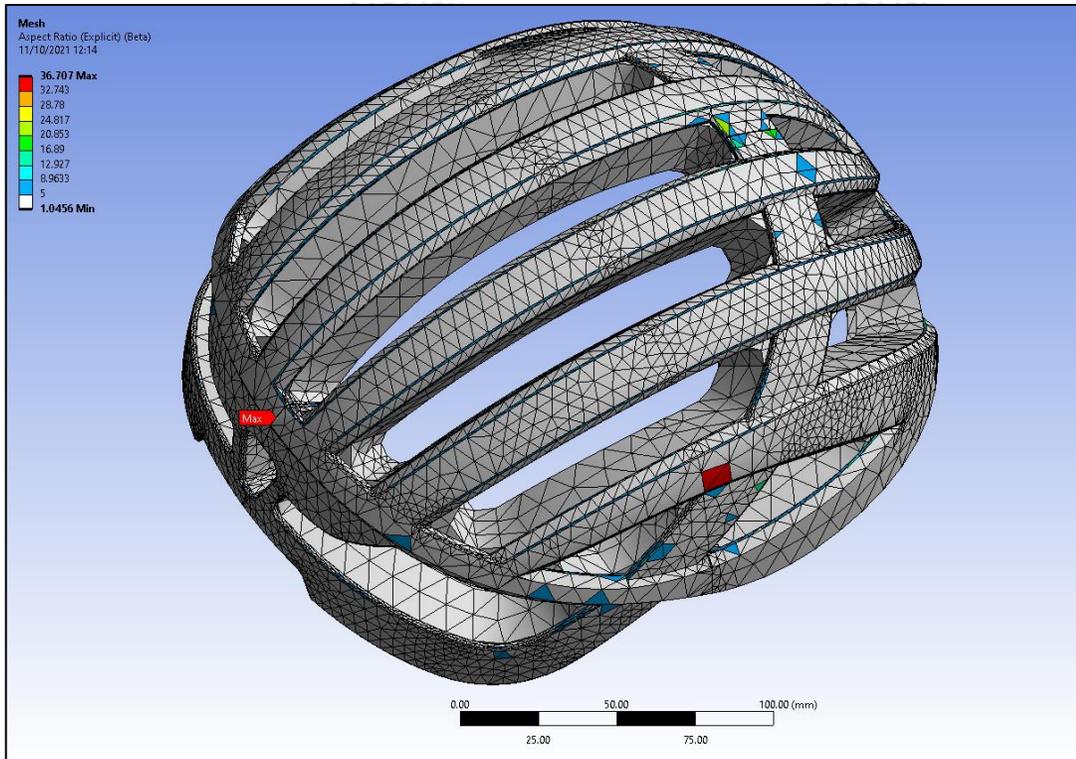
Mesh Cluster Walk: Short Demo with Free Face Diagnostics



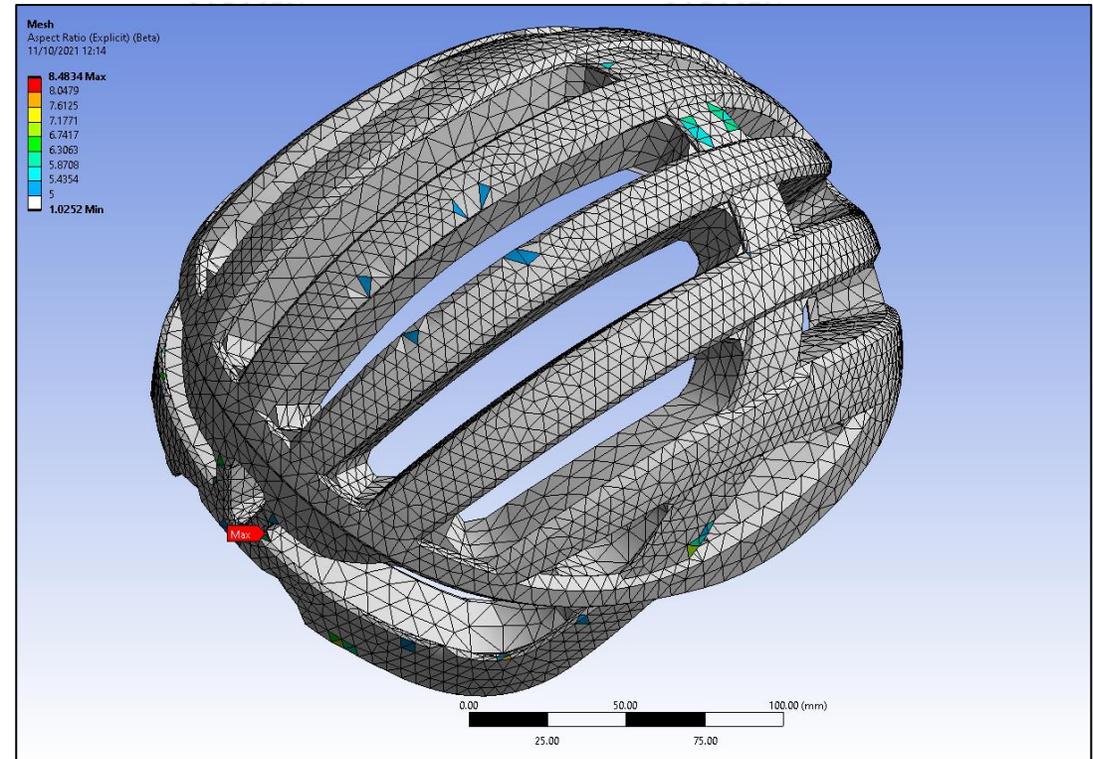
Default Surface Mesher Explicit Physics Pref. = Adv. Front

- More uniform and smooth triangular surface mesh

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



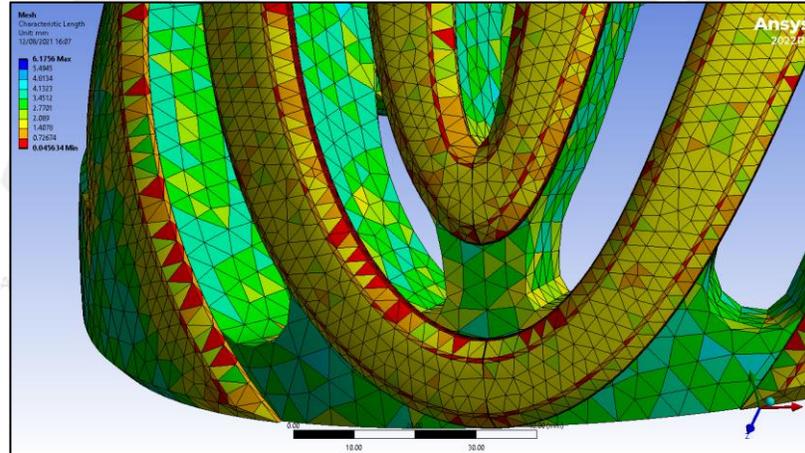
Previous Default Surface Mesh



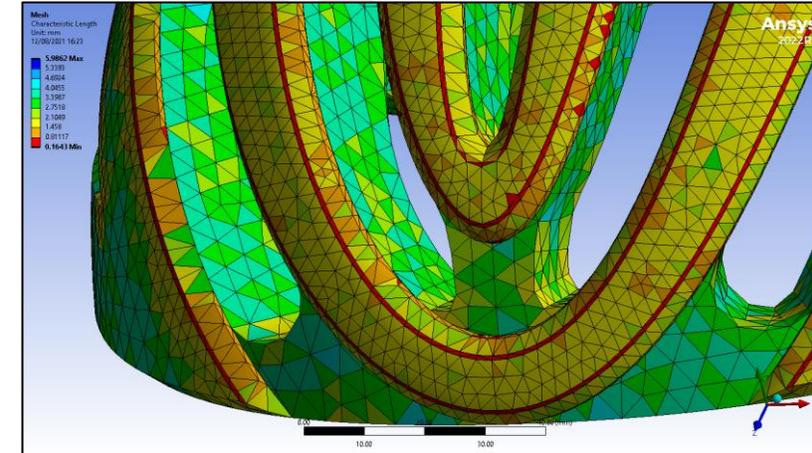
New Default Surface Mesh

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 ($\Delta t = \frac{\text{Characteristic Length}}{\text{Speed of Sound}}$)
- Help run the analysis without much mass-scaling



2021 R2 – Max AR=53, Min CL=0.05mm

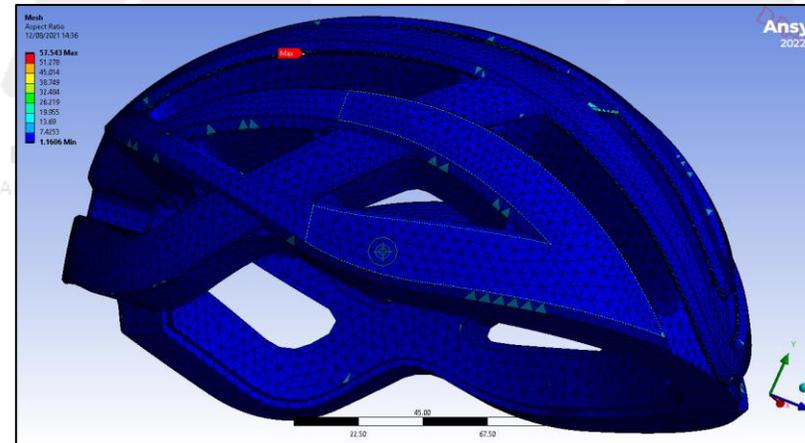


2022 R1 – Max AR=7.6, Min CL=0.16mm

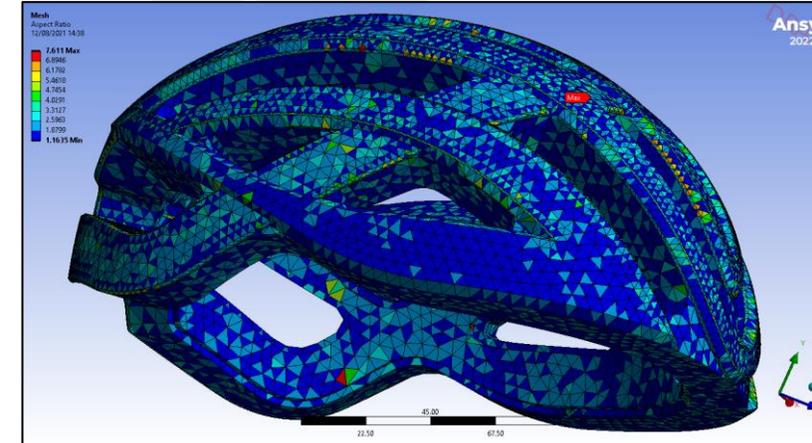
Quality	
Check Mesh Quality	Yes, Errors and Warnings
<input type="checkbox"/> Target Element Quality	Default (0.200000)
<input type="checkbox"/> Target Characteristic Length (LSDyna)	Default (0.5 mm)
<input checked="" type="checkbox"/> Target Aspect Ratio (Explicit)	5.

Challenges:

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



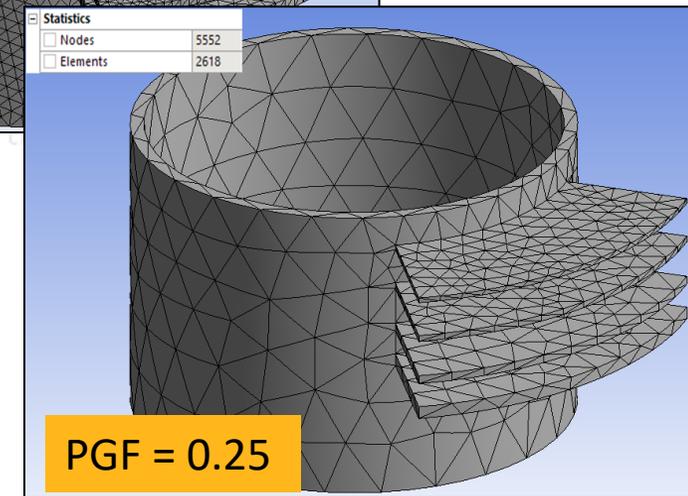
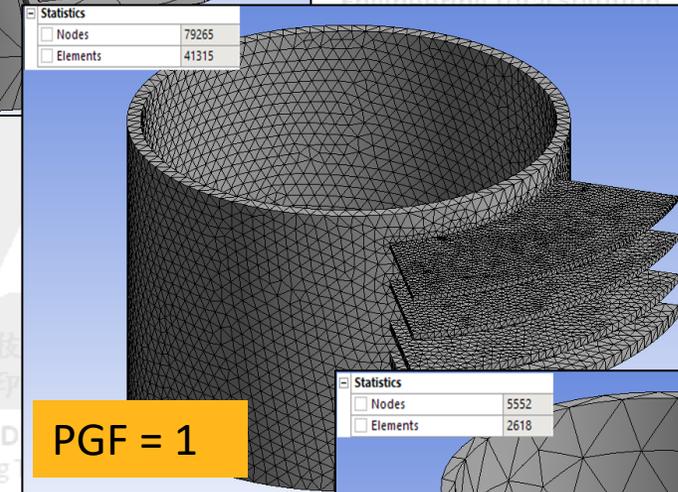
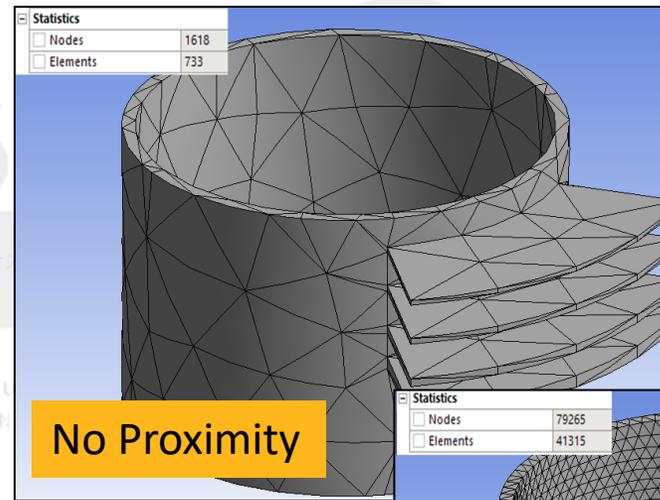
BEFORE



AFTER

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



<input checked="" type="checkbox"/> Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (1.0 mm)
<input checked="" type="checkbox"/> Proximity Gap Factor	0.5

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
<input type="checkbox"/> Target Element Quality	Default (0.200000)
<input type="checkbox"/> Target Characteristic Length (LSDyna)	Default (0.5 mm)
<input type="checkbox"/> Target Aspect Ratio (Explicit)	5.

Options

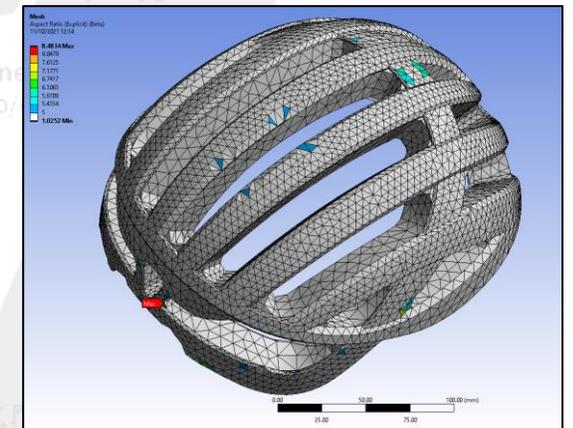
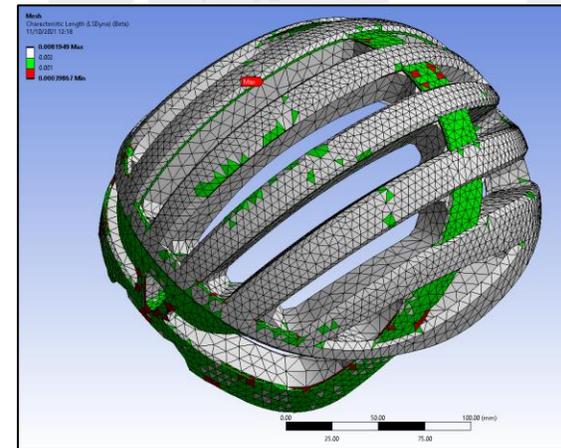
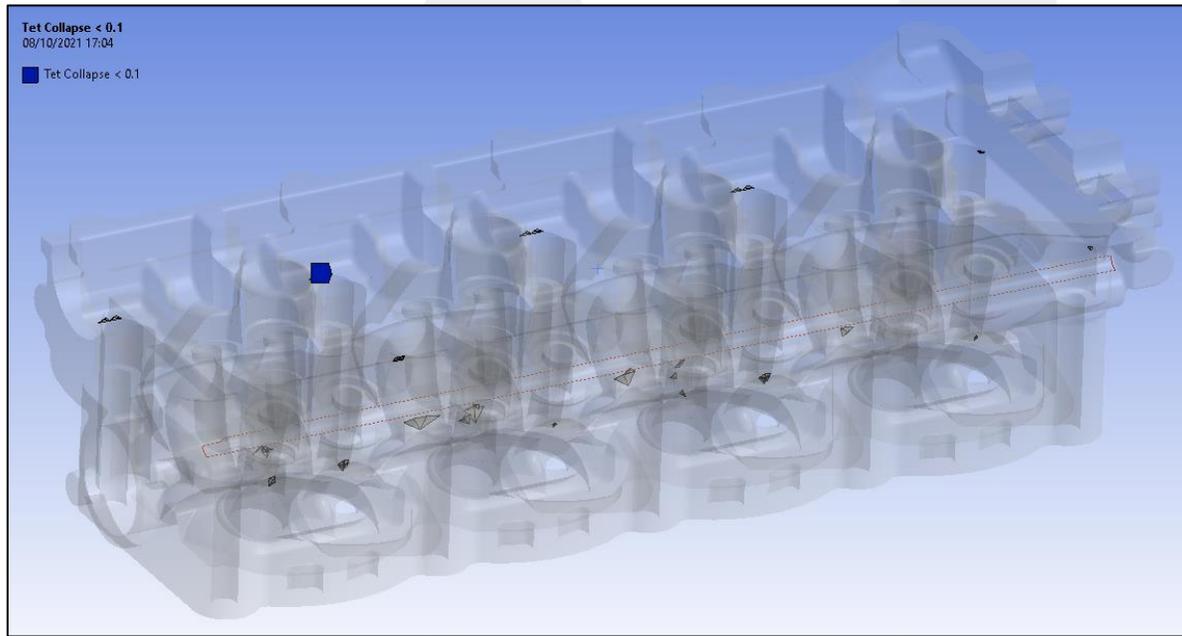
Common Settings

- DesignModeler
- Aqwa
- Mechanical
- Meshing
 - Meshing
 - Export
 - Mesh Quality

Mesh Quality

Jacobian Ratio (Gauss Points)	Based on Physics Preference
WarpingFactor	Based on Physics Preference
Parallel Deviation	Based on Physics Preference
Maximum Corner Angle	Based on Physics Preference
Skewness	Based on Physics Preference
Orthogonal Angle	Based on Physics Preference
Characteristic Length (AutoDyn)	Based on Physics Preference
Minimum Tri Angle	Based on Physics Preference
Maximum Tri Angle	Based on Physics Preference
Minimum Quad Angle	Based on Physics Preference
Maximum Quad Angle	Based on Physics Preference
Warping Angle	Based on Physics Preference
Tet Collapse	Based on Physics Preference
Aspect Ratio (Explicit)	Based on Physics Preference
Minimum Element Edge Length	Based on Physics Preference
Maximum Element Edge Length	Based on Physics Preference
Characteristic Length (LSDyna)	Based on Physics Preference

Reset OK Cancel Help



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

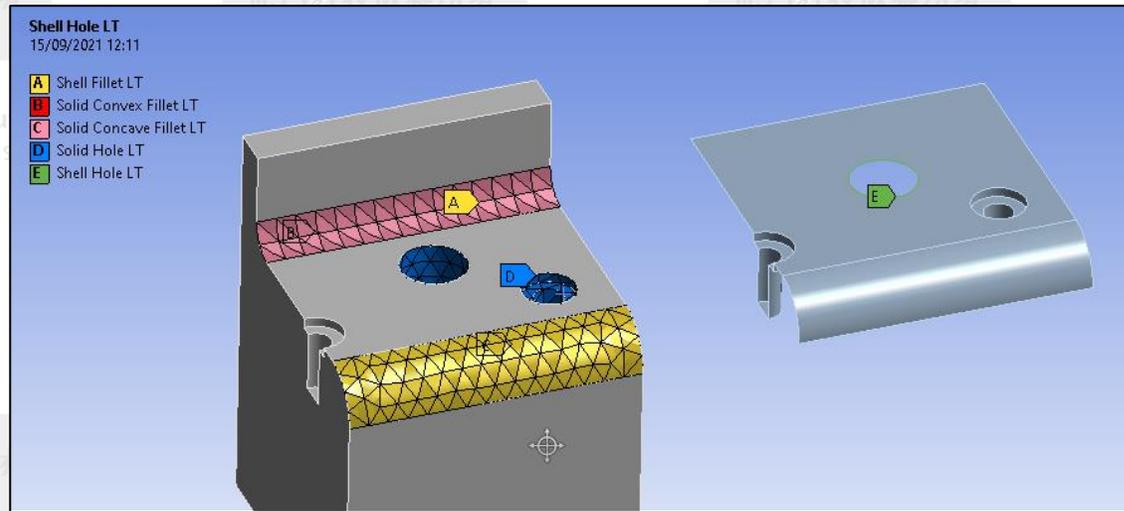
Worksheet

Feature Detection

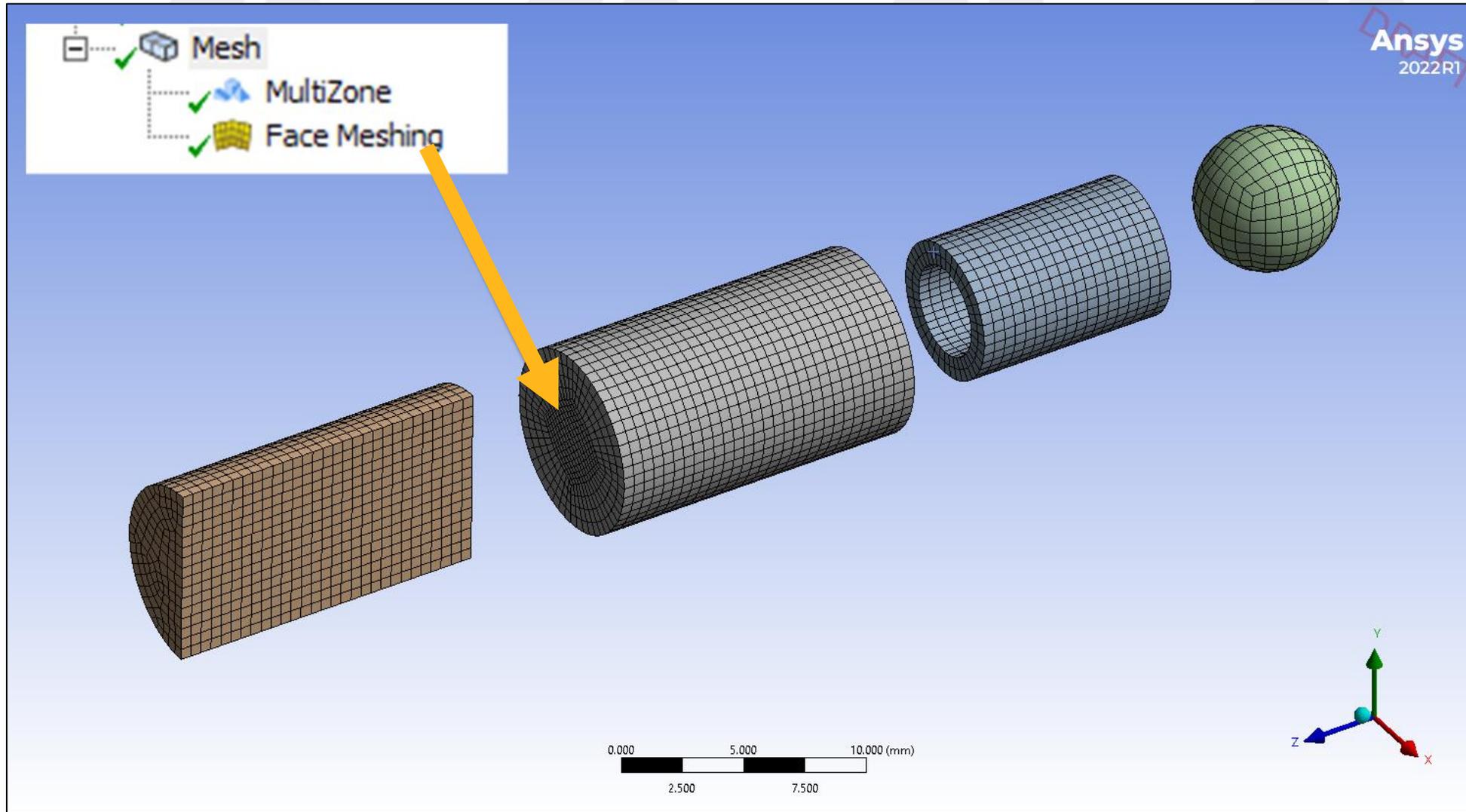
*Right click on the grid to add/delete a row.

	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



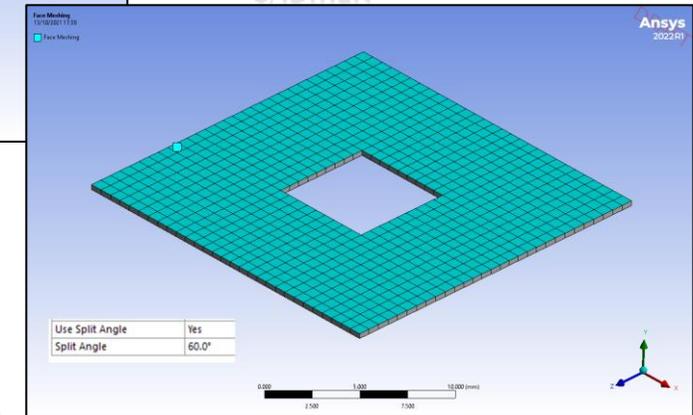
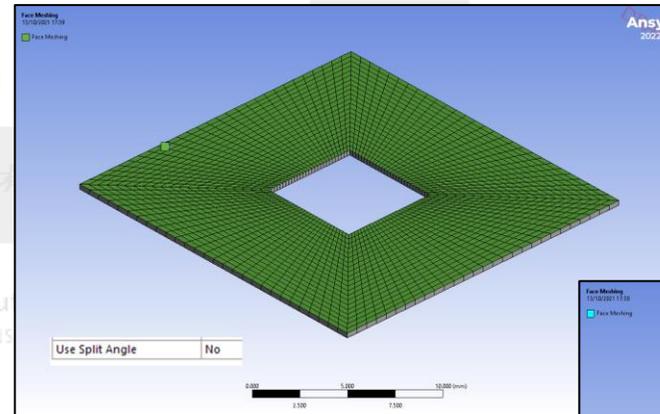
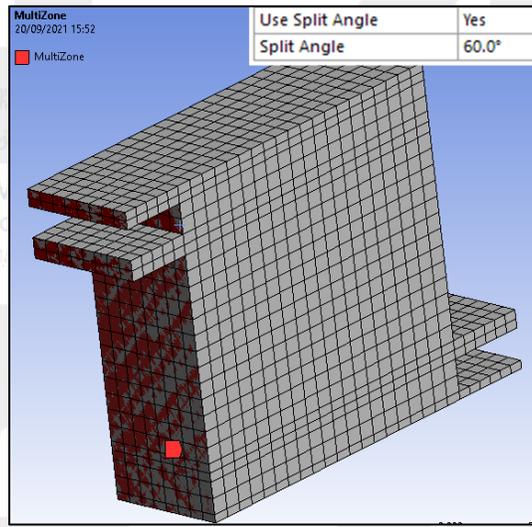
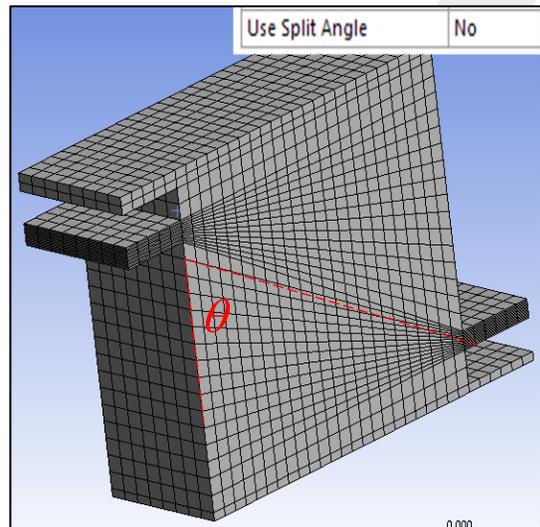
Hex Meshing: Less Decomposition in Mechanical MultiZone



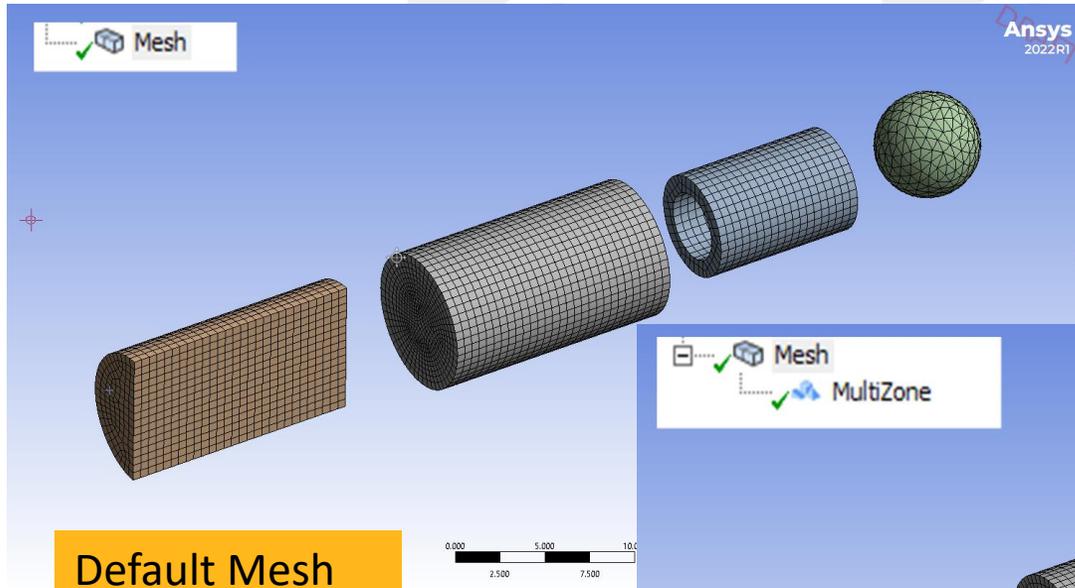
Hex Meshing: Split Angle

- 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

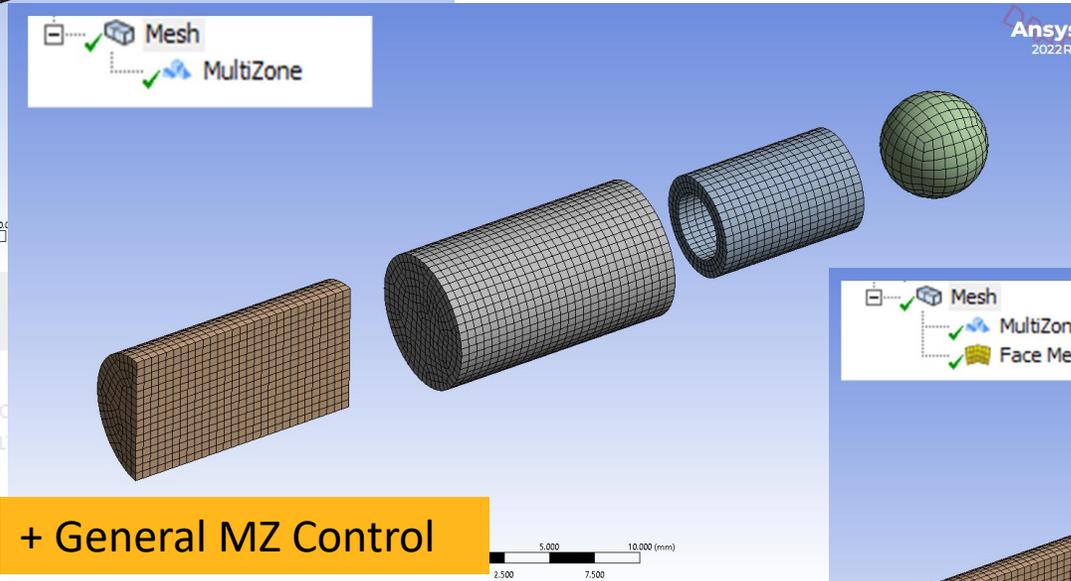
Details of "MultiZone" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Decomposition Type (Beta)	Standard
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
<input type="checkbox"/> 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
Reuse Blocking (Beta)	Off
Use Split Angle	Yes
Split Angle	60



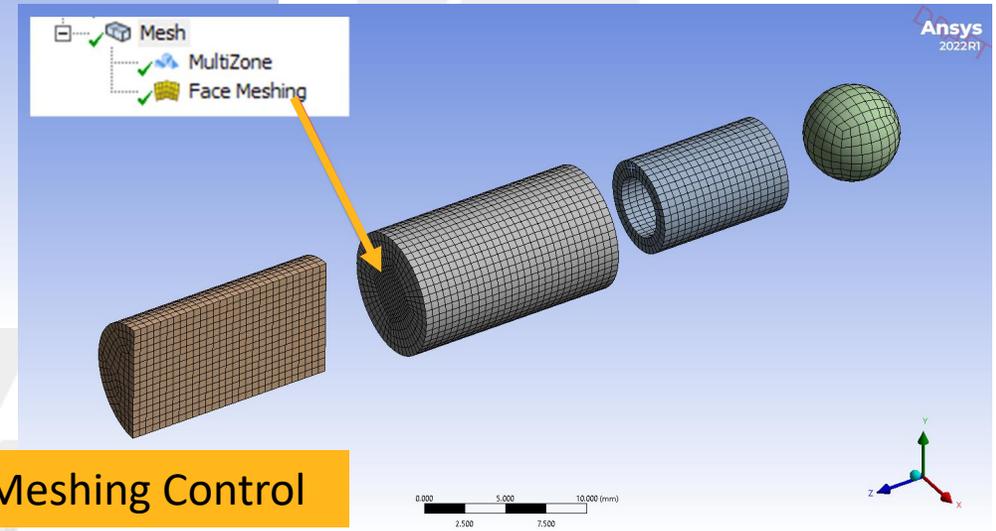
Hex Meshing: Better Orthogonality and Default Meshing Explicit Preferences



Default Mesh



+ General MZ Control



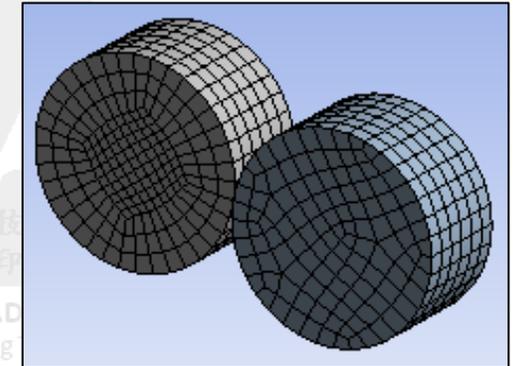
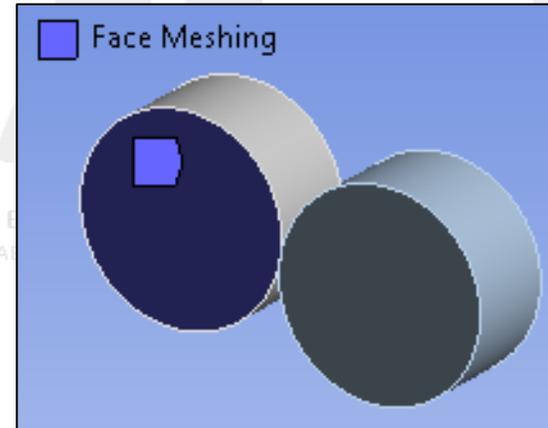
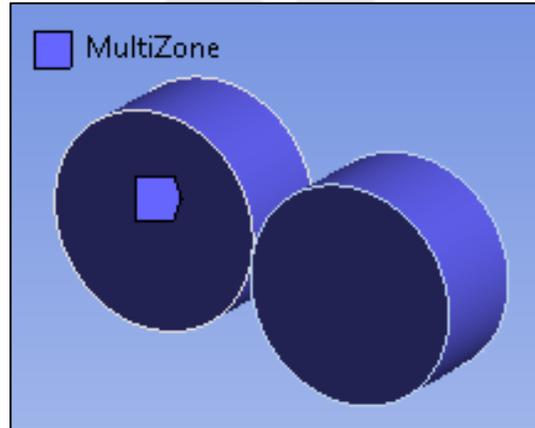
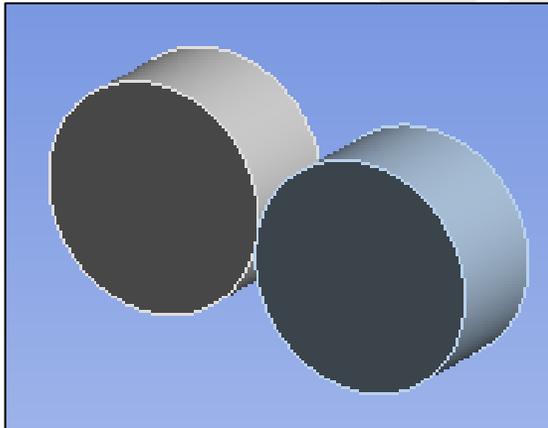
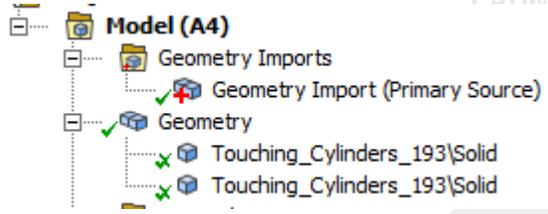
+ Face Meshing Control

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

Touching Cylinders

MultiZone Applied with no special inputs/selections – Automatic Hex

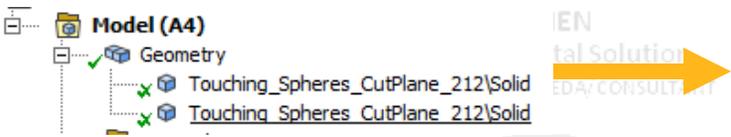
If Face Meshing is Applied O-Grid is automatic with no decomposition needed



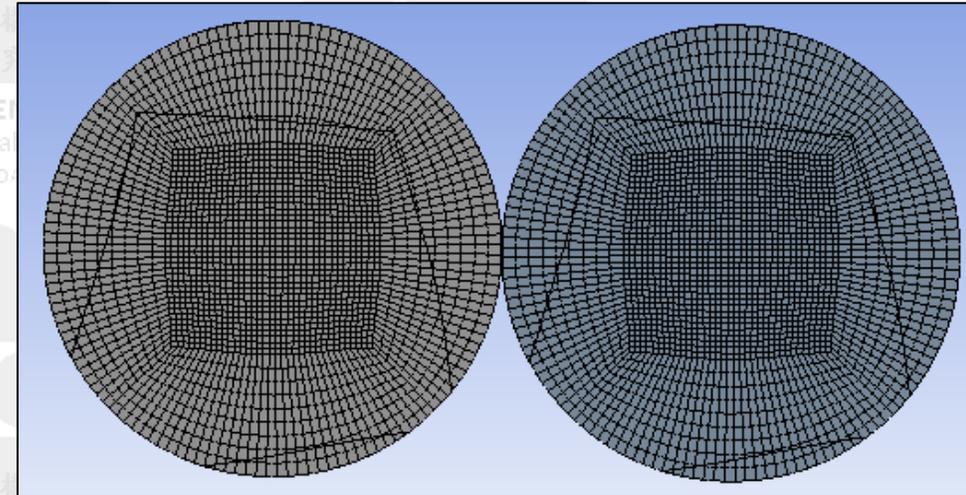
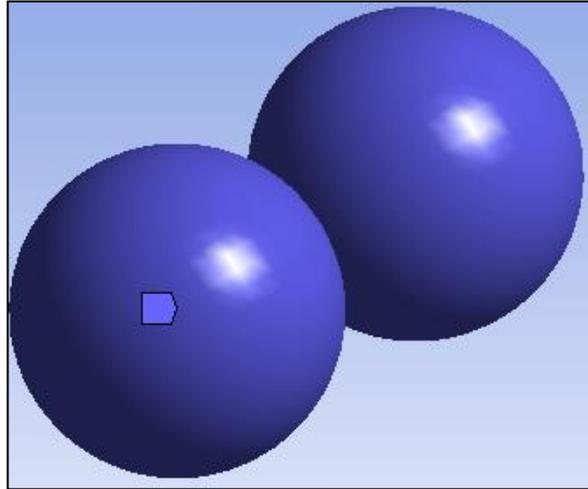
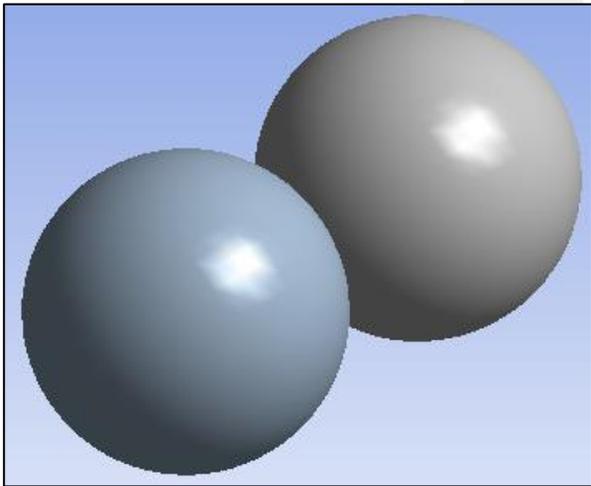
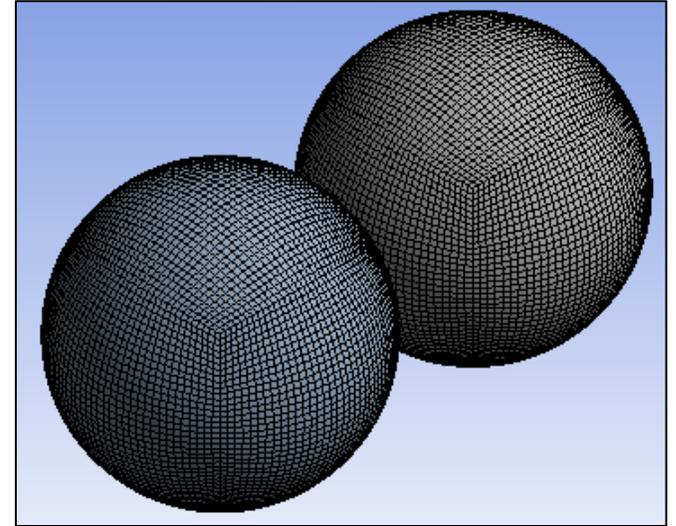
Hex Meshing: Better Default Mesh for Cylinders, Circles, and Spheres

Touching Spheres

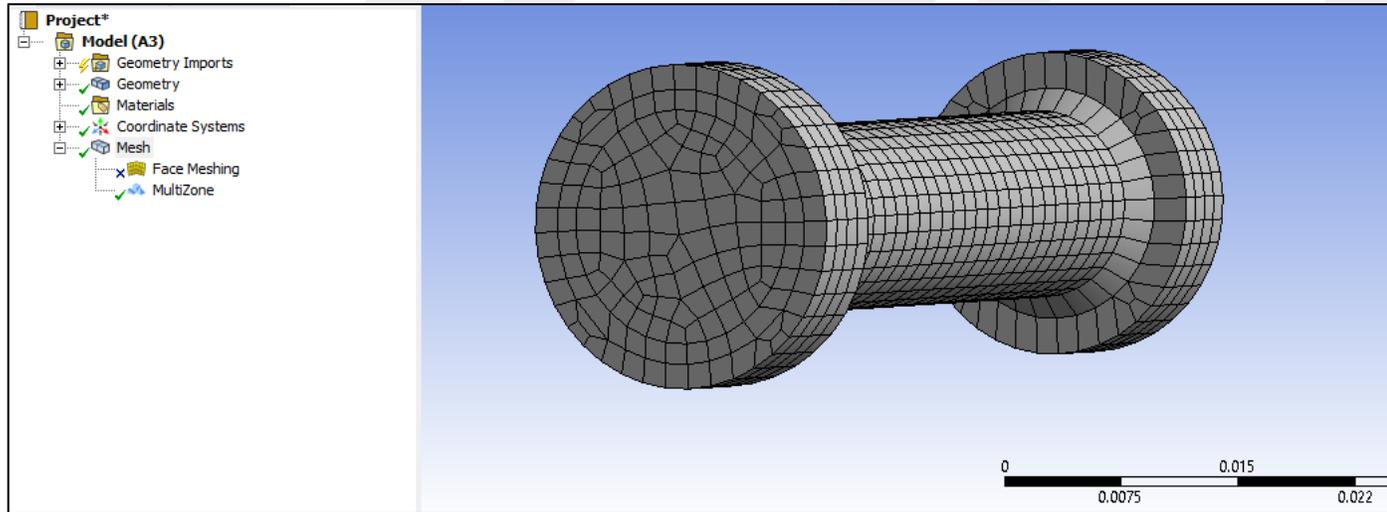
MultiZone Applied with no special inputs/selections – Automatic Hex



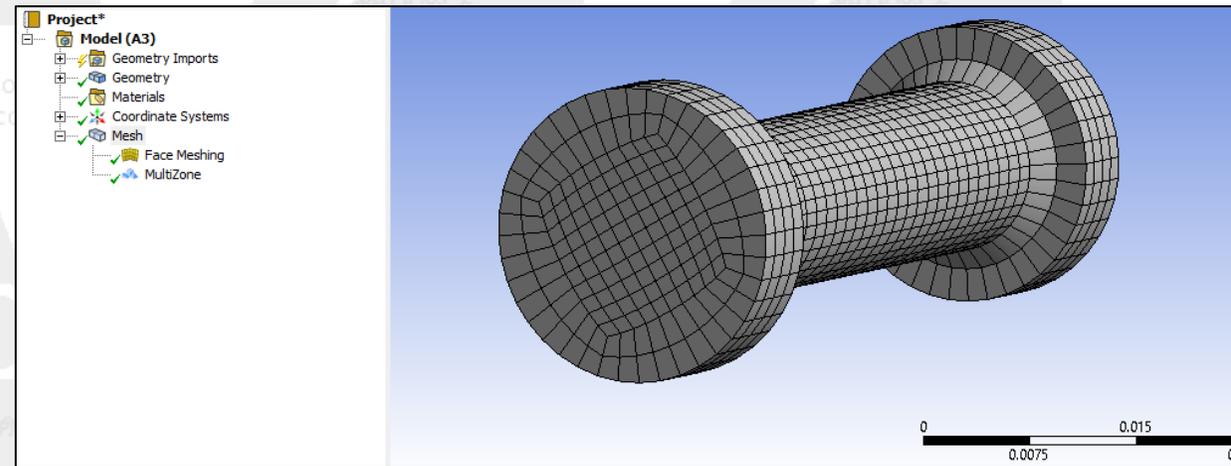
CADMEN
Engineering Total Solution
CAE/CAD/CAM/PDM/EDA/CONSULTANT



Hex Meshing: Improved Hex Mesh for Simple Shapes



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

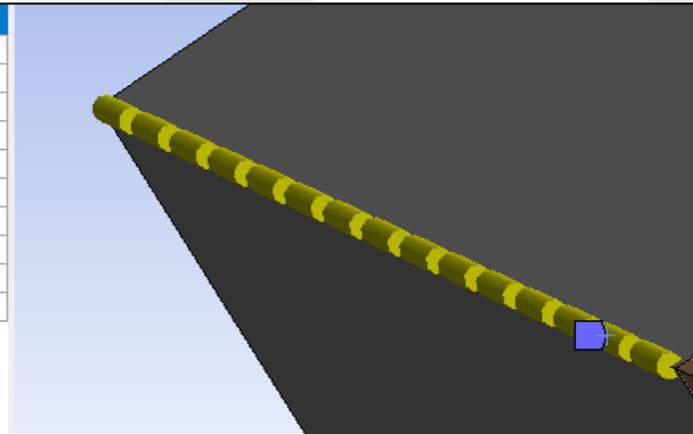


Hex Meshing: Body Fitted Cartesian - Edge Sizing Support

- BF Cart now supports “Edge Sizing” control



Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
<input checked="" type="checkbox"/> Number of Divisions	15
Advanced	
Behavior	Soft
Bias Type	No Bias



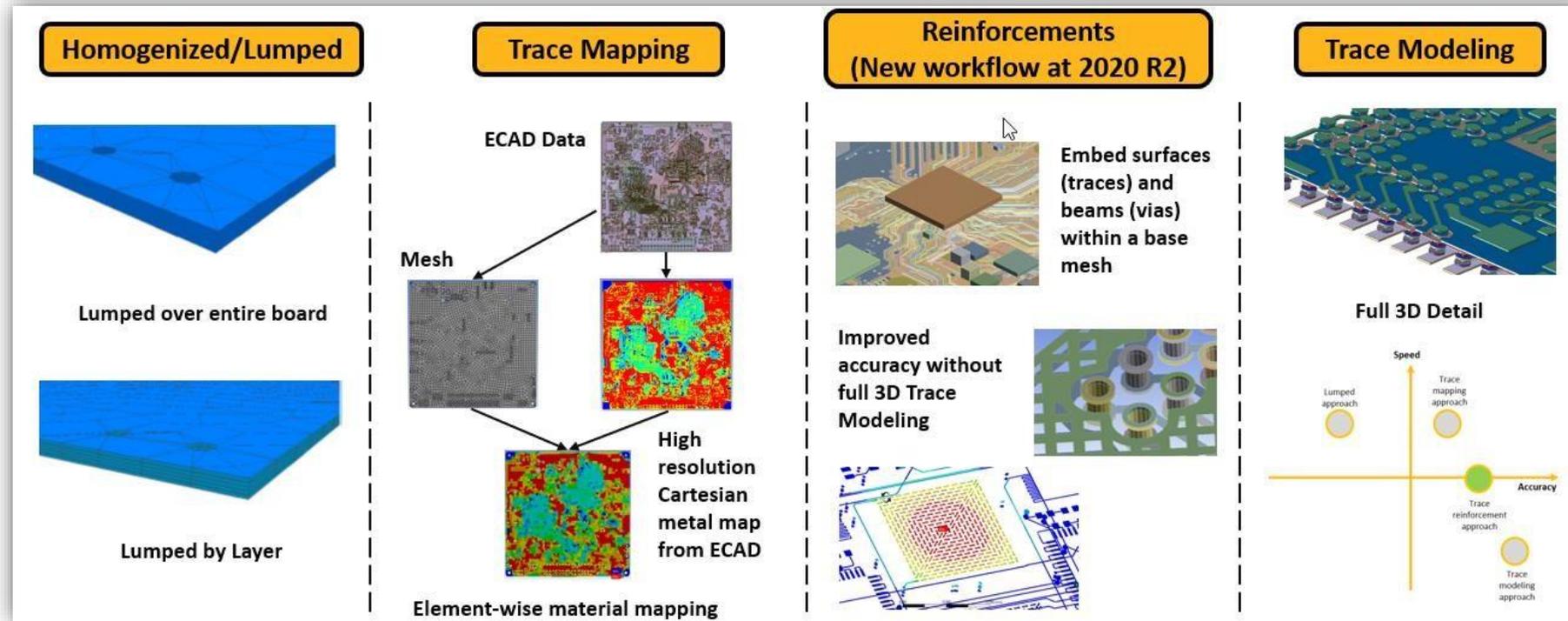
Details of "Body Fitted Cartesian" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1215 Bodies
Definition	
Suppressed	No
Method	Cartesian
Element Order	Use Global Setting
Type	Element Size
<input type="checkbox"/> Element Size	7.5e-002 mm
Spacing Option	Default
Advanced	
Projection Factor	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 ICFM CFD Files	No

Reinforcement

Ansys

1 Flex Modeling - Introduction

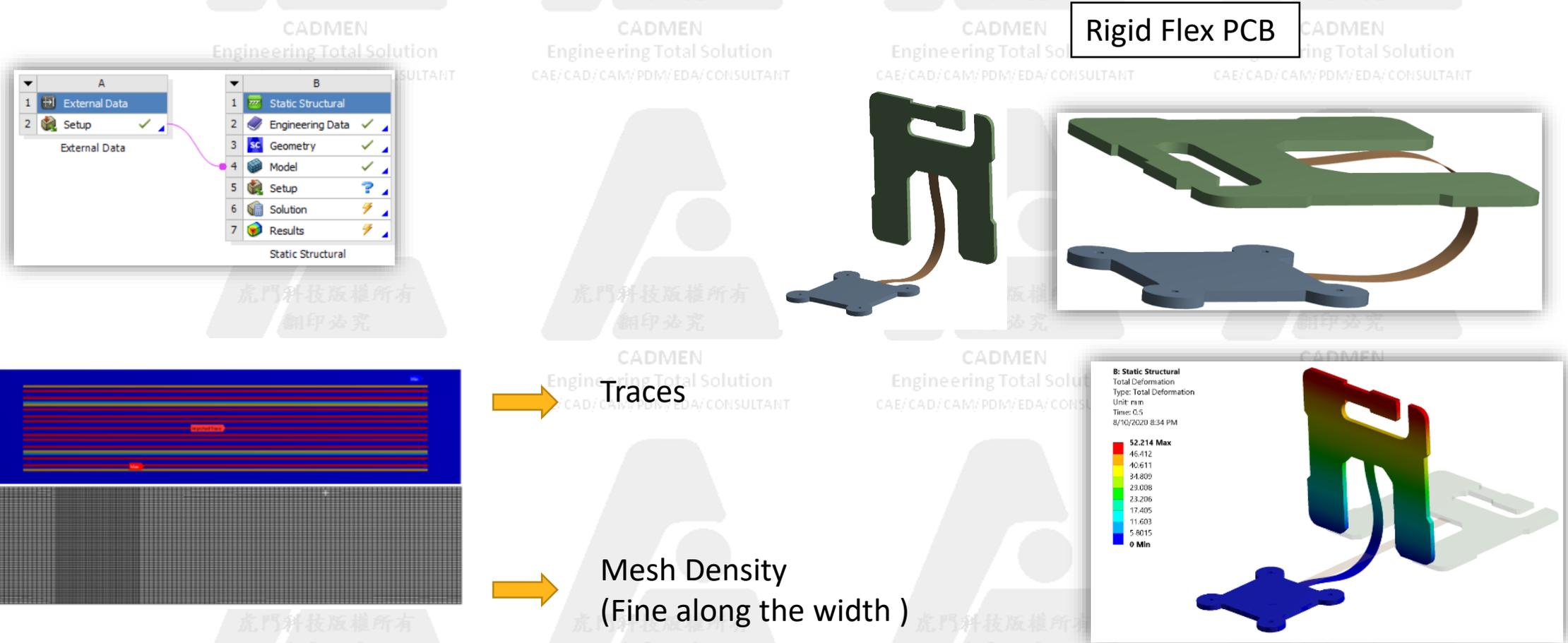
- ANSYS provides leading solutions for the modeling of flexes and electronic components. Multiple levels of fidelity supported
- Start by importing ECAD files directly into ANSYS tools, such as Sherlock, SpaceClaim and Mechanical



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
- Rigid Flex PCB: Modeling of Operating phase (Cyclic Loading)

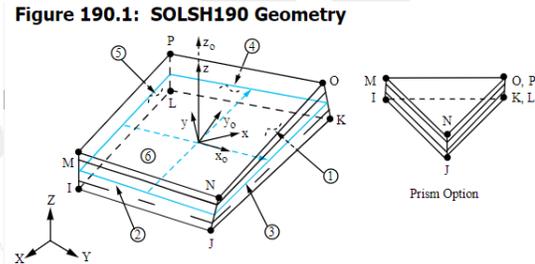


Flex Bending – Trace Reinforcement Approach

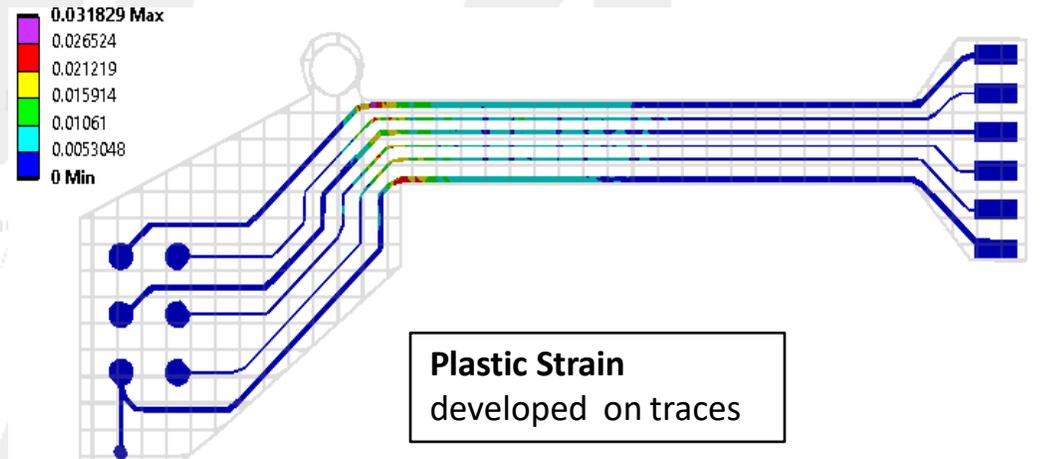
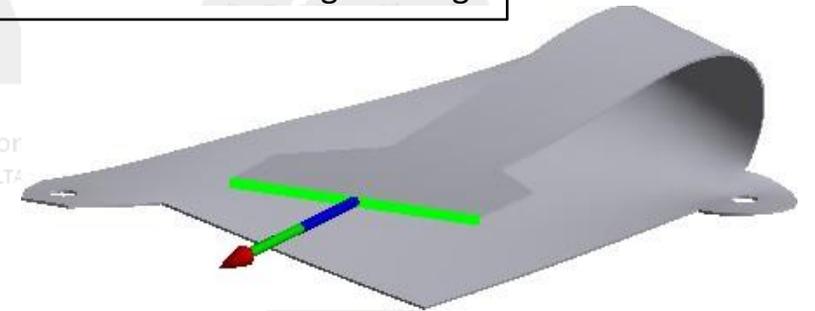
- Goal:
 - To demonstrate the Trace-Reinforcement approach
 - To understand the preprocessing in Sherlock for this method
 - To compare results with 3 different base elements:

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

- Model Description:
 - 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 [page 5](#) for flex model description)

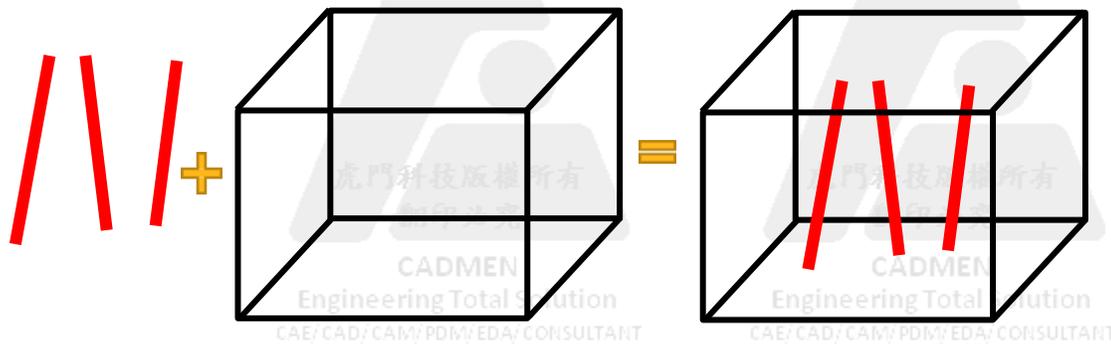


Reaction Force Vector during Bending



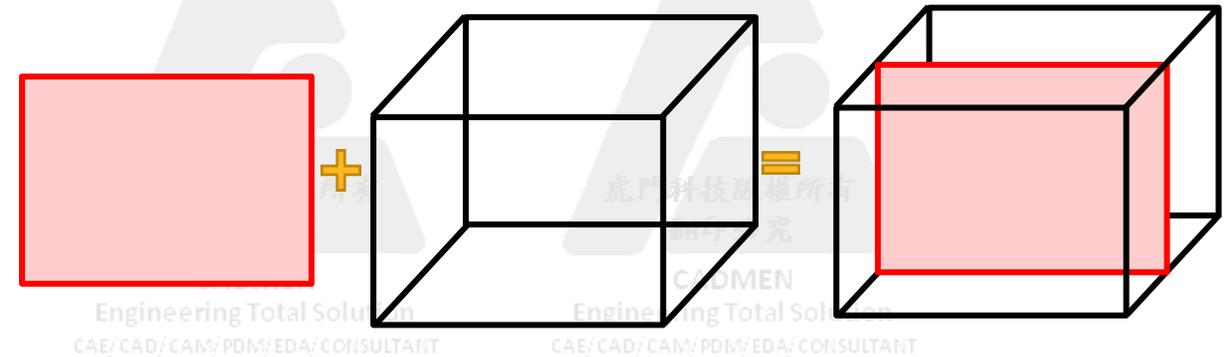
Reinforcement Element Technology

- 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



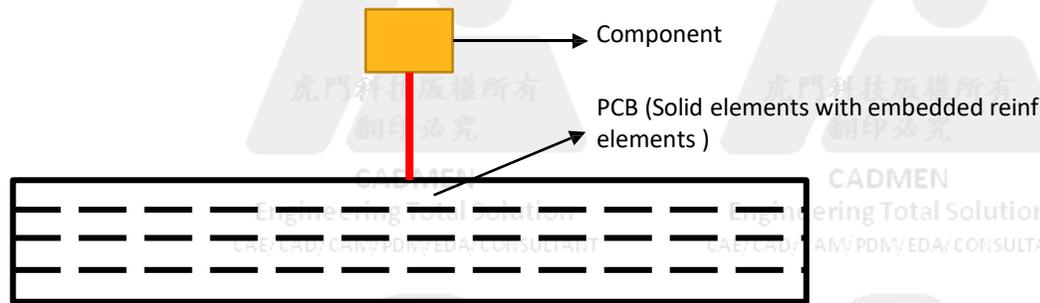
Smear Reinforcement:

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

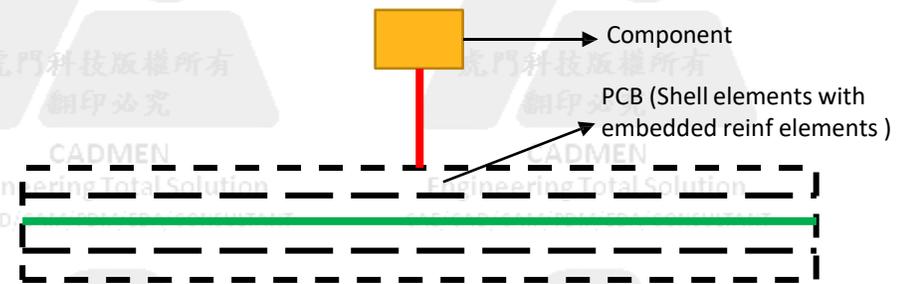
Reinforcement Element Technology

Reinforcement element with Solid and Shell base elements

- 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)

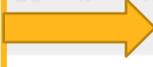
- Refer to Ansys Help to read more on Reinforcement Elements: [REINF263](#) , [REINF264](#) and [REINF265](#)

4. Workflow Overview

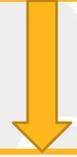
- Sherlock:**
- Import ECAD file and packages
 - Create Solid PCB CAD (step file)
 - Create traces as surface bodies (step file)
 - Create wbjn and py file



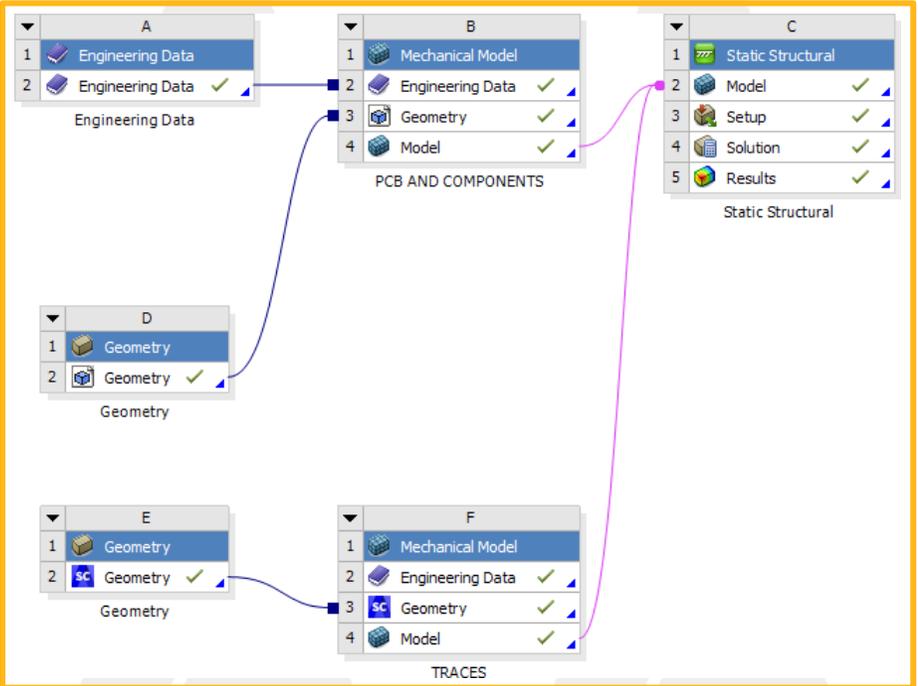
- Workbench:**
- Read the wbjn file.
 - The WBJN file calls the PY file and the STEP file to create the basic workflow



- 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



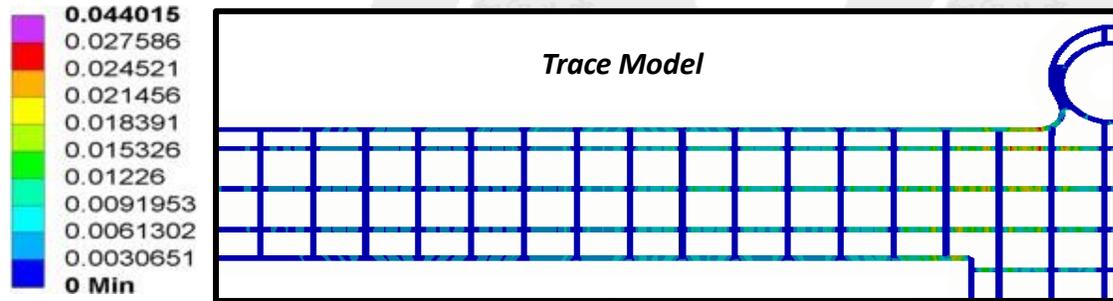
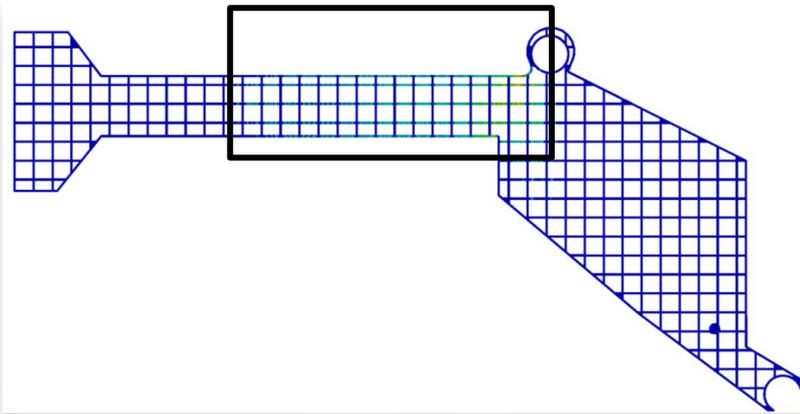
- Mechanical:**
- Assign materials
 - Contact between the components and PCB
 - Boundary Conditions
 - Loading condition
 - Solve and post process.



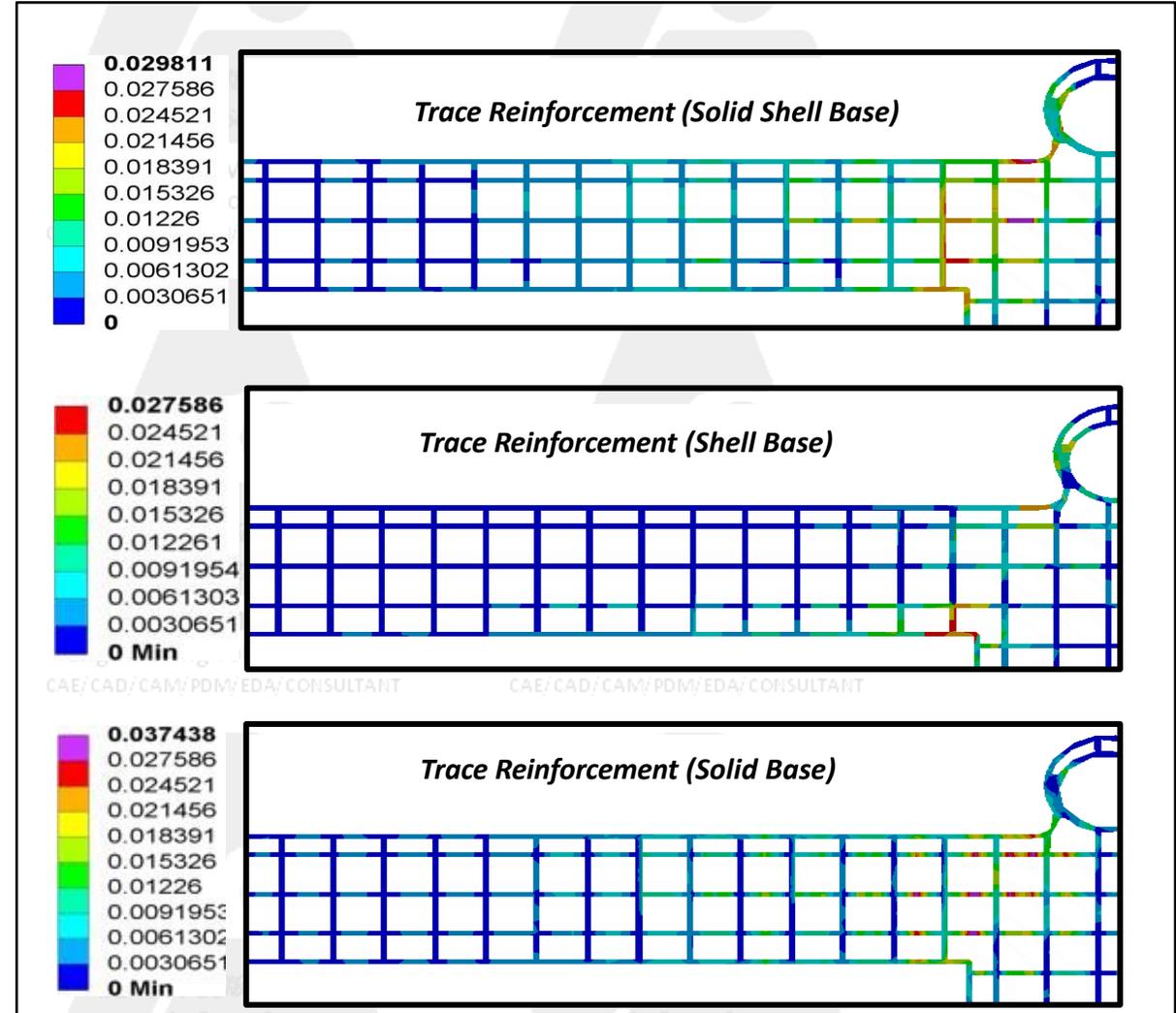
Workbench project Schematic

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

Plastic Strain (Layer 1)

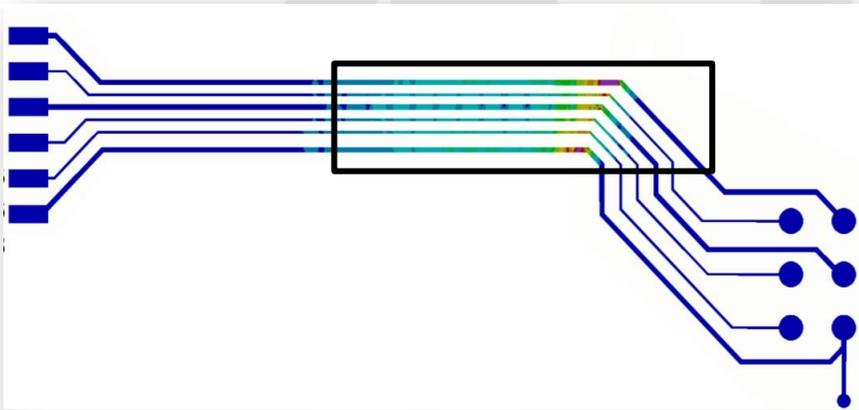


Trace Model approach is considered as benchmark and Trace-reinforcement approach with different base elements is compared against it

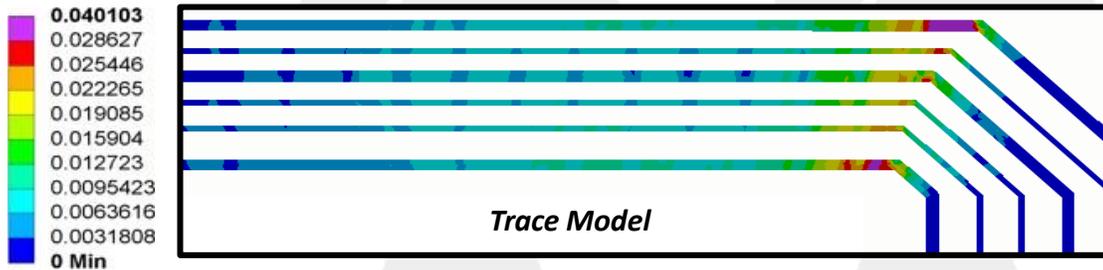


Comparative Analysis: Solid Vs Shell Vs Solid-Shell Base Element

Plastic Strain (Layer 2)



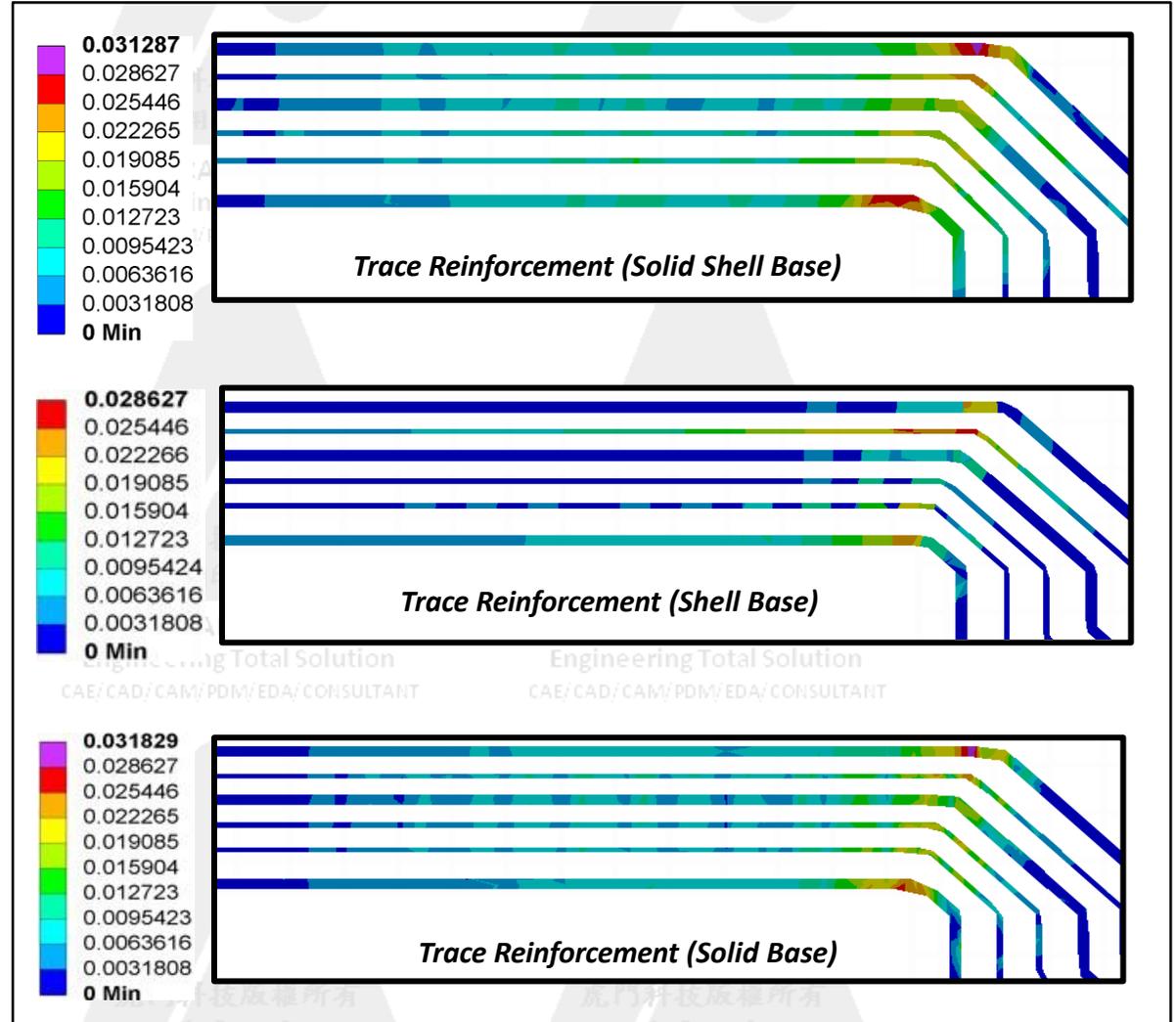
虎門科技版權所有
研究
CADMEN
Engineering Total Solution
CAE/CAD/CAM/PDM/EDA/CONSULTANT



虎門科技版權所有

虎門科技版權所有

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



CADMEN

Engineering Total Solution

CAE/CAD/CAM/PDM/EDA/CONSULTANT



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

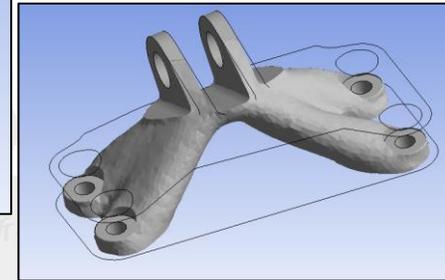
(2) Run the optimization

(3) Post:
Define your display

The screenshot shows the 'Structural Optimization (F5)' tree with 'Analysis Settings' selected. Below it is a 3D model of a mechanical part with a fixed support and a force of 25000 N. The 'Details of "Analysis Settings"' dialog is open, showing the 'Output Controls' section where 'Export Design Properties' is set to 'All Accepted Iterations' and 'Export Design Properties File Format' is set to 'HDF5 File'.

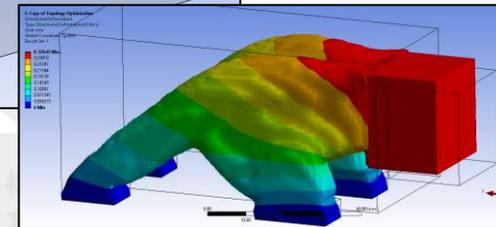
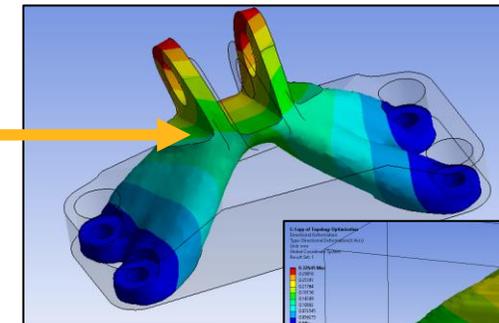
Details of "Analysis Settings"	
Definition	
Maximum Number Of Iterations	500.
Convergence Accuracy	0.1 %
Output Controls	
Export Design Properties	All Accepted Iterations
Export Design Properties File Format	HDF5 File
Store Results At	All Iterations
Solver Controls	
Solver Type	Program Controlled
Analysis Data Management	

The screenshot shows the 'Structural Optimization (F5)' tree with the 'Deformation' menu open. The 'Total' option is selected, and the 'Details of "Solution (F6)"' dialog is open, showing 'Adaptive Mesh Refinement'.



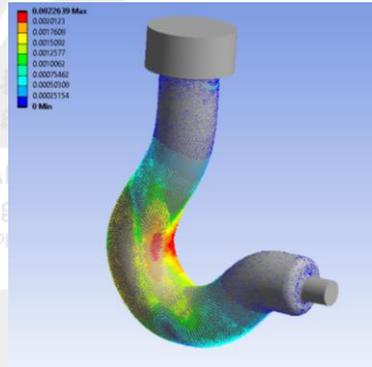
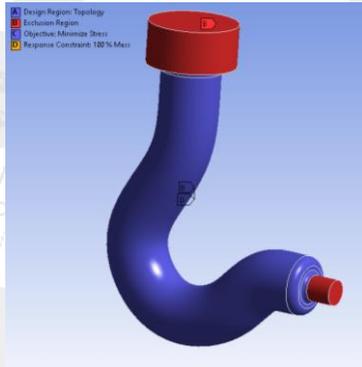
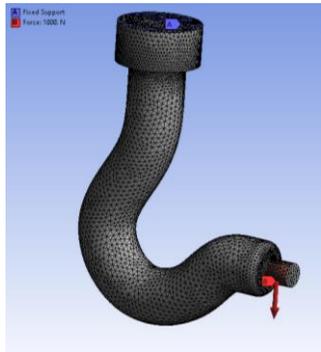
The screenshot shows the 'Structural Optimization (B5)' tree with 'Solution (B6)' selected. The 'Total Deformation' option is selected in the 'Solution Information' section. The 'Details of "Total Deformation"' dialog is open, showing the 'Scope' and 'Definition' sections.

Details of "Total Deformation"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
Type	Total Deformation
Environment Selection	Static Structural 2
By	Iteration
Iteration	Last
Suppressed	No



Maximum Principal Stress

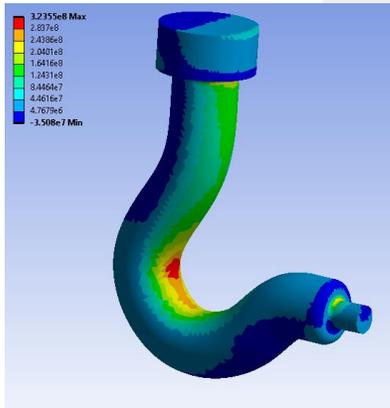
- ✓ MPS is available alongside Von Mises eq. norm
- ✓ Available for Topology (SIMP or Level-Set), Shape and Lattice Optimization
- ✓ Can be consumed as an objective or as a constraint



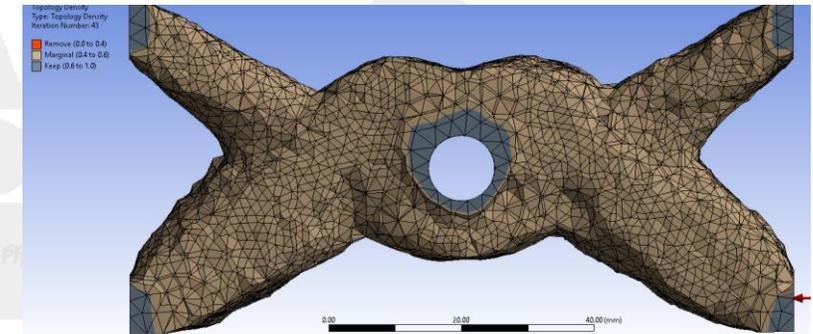
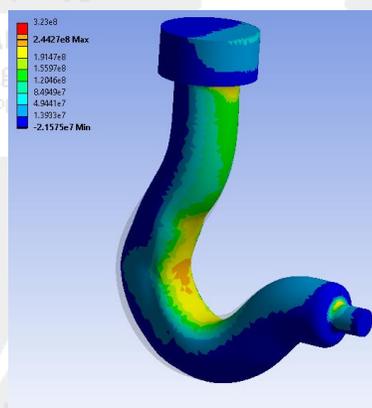
Mesh & Structural Set-up

Shape Optimization Set-up

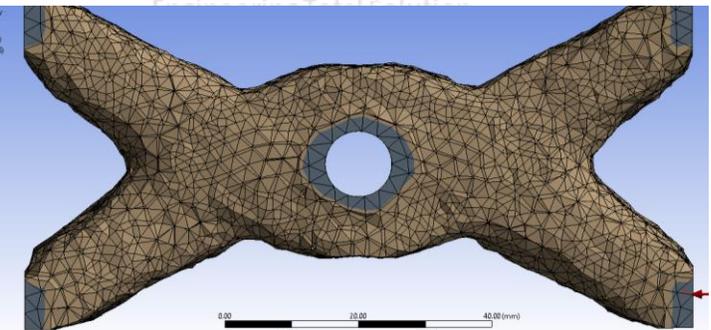
Results & direct validation



MPS
- 25 %



MPS-driven optimization



VonMises-driven optimization

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

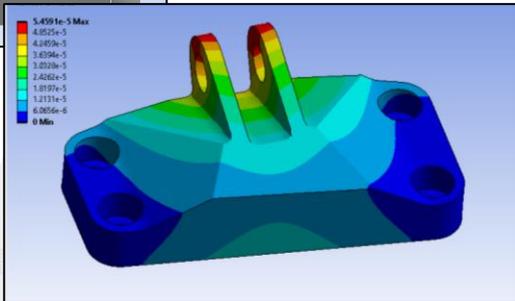
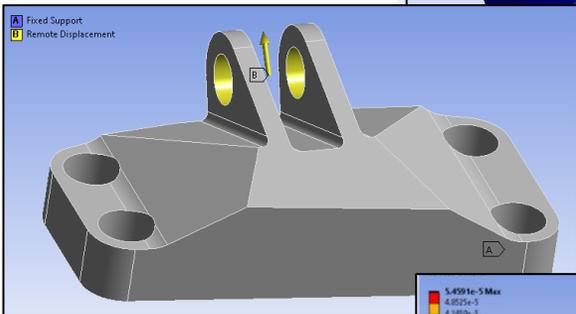
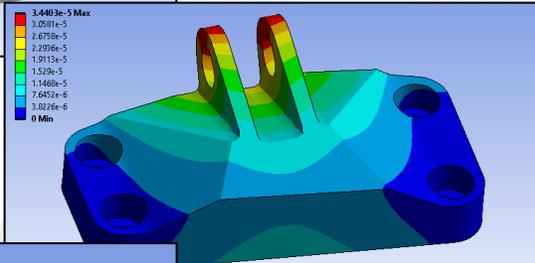
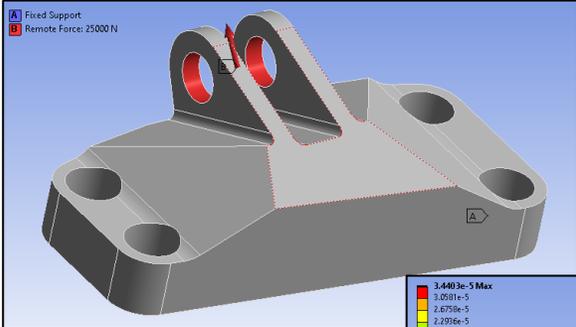
The screenshot shows the ANSYS Engineering Total Solution interface. The tree view on the left includes:

- Static Structural (D5)_remoteDisp (D5)
 - Analysis Settings
 - Fixed Support
 - Remote Displacement
 - Solution (D6)
 - Solution Information
 - Total Deformation
 - Primary Criterion
- Structural Optimization 3 (E5)
 - Analysis Settings
 - Optimization Region
 - Objective
 - Response Constraint
 - Solution (E6)
 - Solution Information
 - Topology Density

The 'Details of "Primary Criterion"' panel is expanded, showing the following configuration:

Details of "Primary Criterion"	
Definition	
Base Result	Reaction Force
Suppressed	No
Results	
<input type="checkbox"/> Value	39670 N
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Displacement
Load Step Selection	
Step	1
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	Z
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete

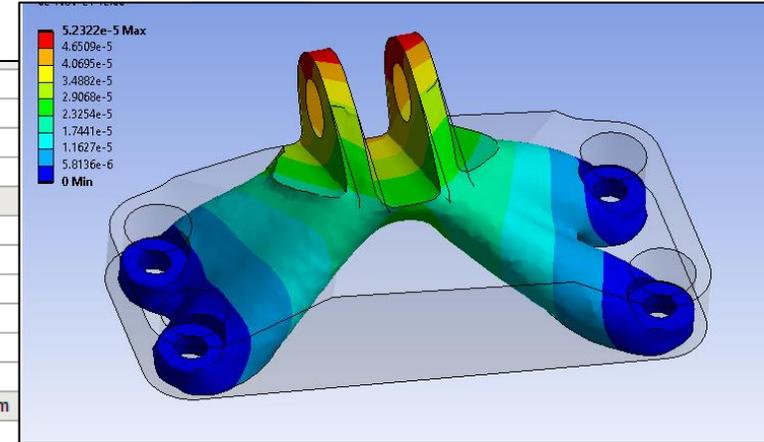
Example #1-2



$\left\{ \begin{array}{l} \min(\text{primary criterion}) \\ \Omega \\ \text{vol} < 40\% \end{array} \right.$

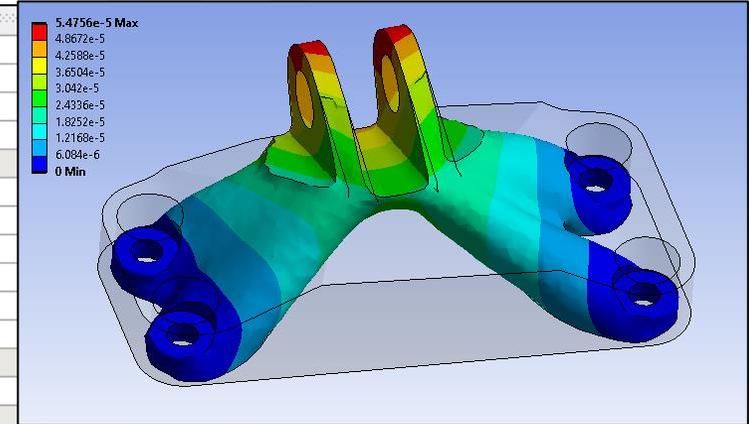
$\left\{ \begin{array}{l} \max(\text{primary criterion}) \\ \Omega \\ \text{vol} < 40\% \end{array} \right.$

Definition	
Base Result	Displacement
Suppressed	No
Results	
<input type="checkbox"/> Value	2.1427e-005 m
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Force
Load Step Selection	
Step	1
Element Reduction	
Coordinate System	Nodal Coordinate System
Reference Type	Constant
Reference Value	0.
Vector Reduction	Z
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete



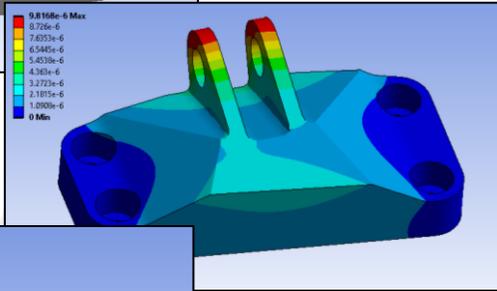
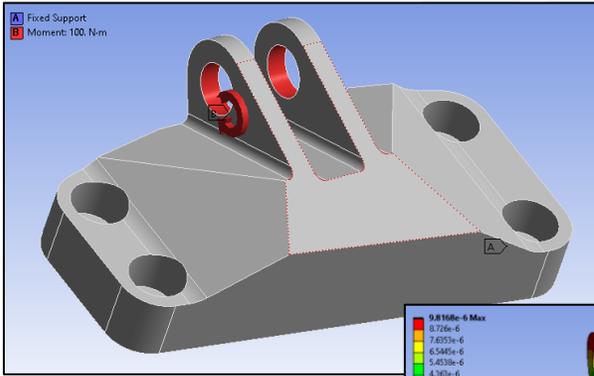
z-disp of the remote-force support

Details of "Primary Criterion"	
Definition	
Base Result	Reaction Force
Suppressed	No
Results	
<input type="checkbox"/> Value	39670 N
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Displacement
Load Step Selection	
Step	1
Element Reduction	
Coordinate System	Nodal Coordinate System
Reference Type	Constant
Reference Value	0.
Vector Reduction	Z
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete



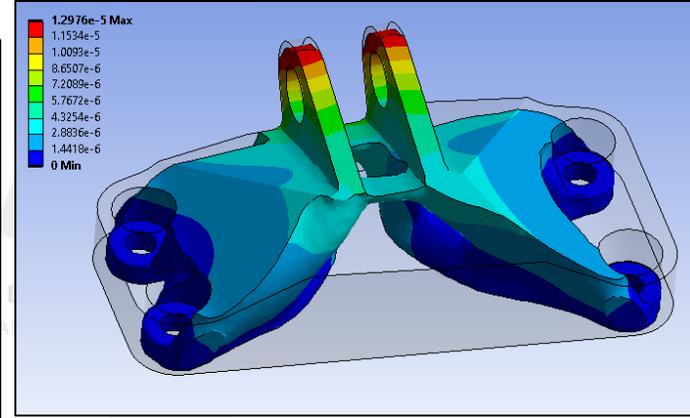
z-reaction-force at the remote-displacement support

Example #3-4

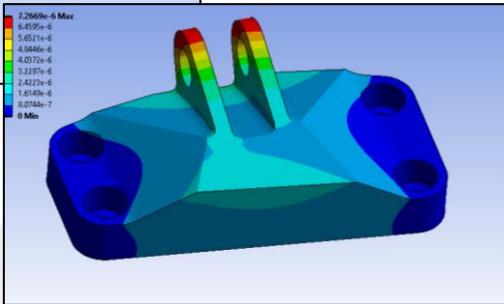
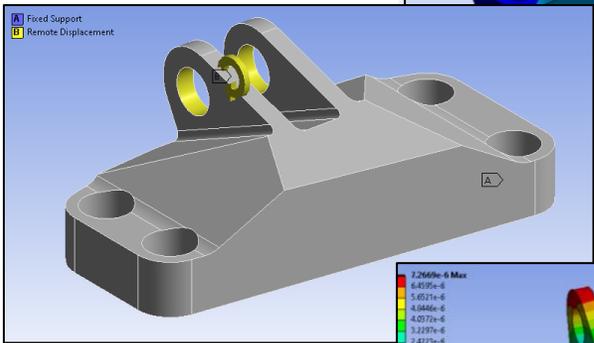


$\max_{\Omega}(\text{primary criterion})$
 $\text{vol} < 40\%$

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
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete

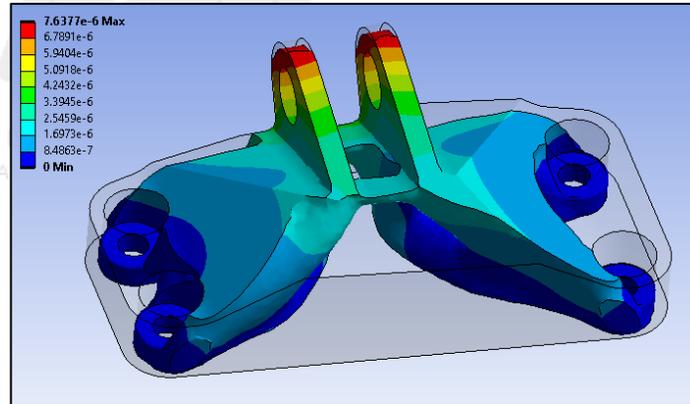


x-rotation of the remote-moment support



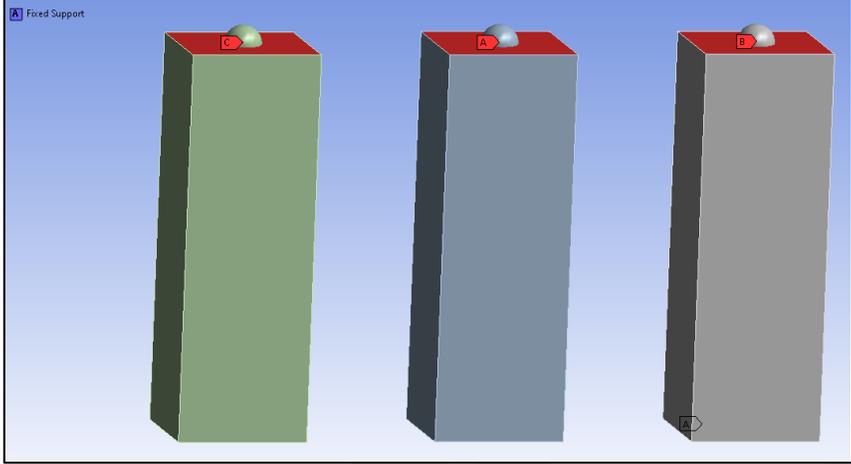
$\min_{\Omega}(\text{primary criterion})$
 $\text{vol} < 40\%$

Details of "Primary Criterion"	
Definition	
Base Result	Reaction Moment
Suppressed	No
Results	
Value	-74.025 N-m
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Displacement
Load Step Selection	1
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete



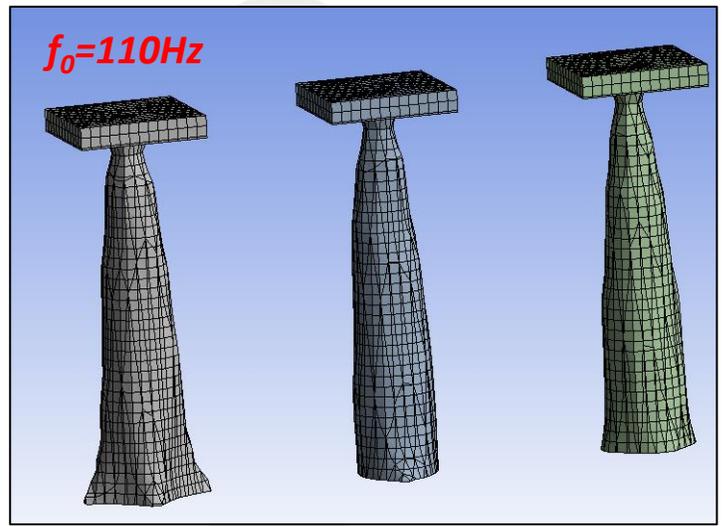
x-reaction-moment at the remote-displacement support

UDC Modal: Robust Frequency Example



optim#1

$$\begin{cases} \max_{\Omega} f_0 & [\text{single freq}] \\ \text{mass} \leq 25\% \end{cases}$$



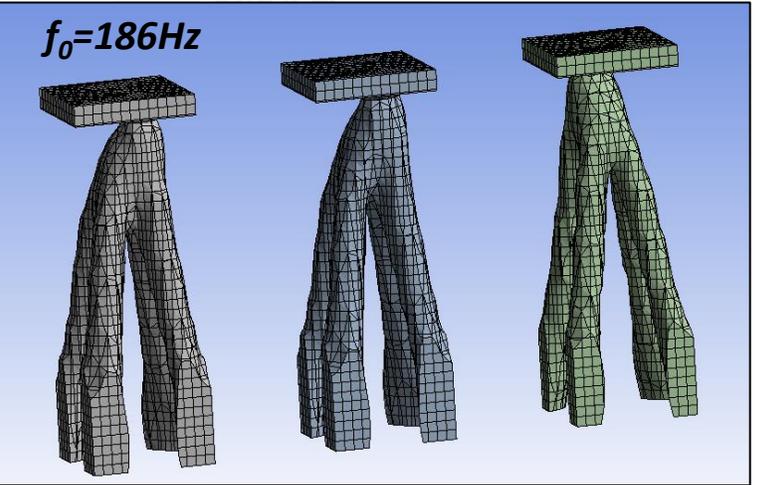
+70%
better

Better performance thanks to
the robust-frequency criterion

optim#2

$$\begin{cases} \max_{\Omega} \tilde{f}_0 & [\text{robust freq}] \\ \text{mass} \leq 25\% \end{cases}$$

Details of "Primary Criterion 2"	
Definition	
Base Result	Robust Frequencies
Suppressed	No
Mode Number	1
Frequency Reduction	Smooth Minimum
Results	
Value	473.64 Hz



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

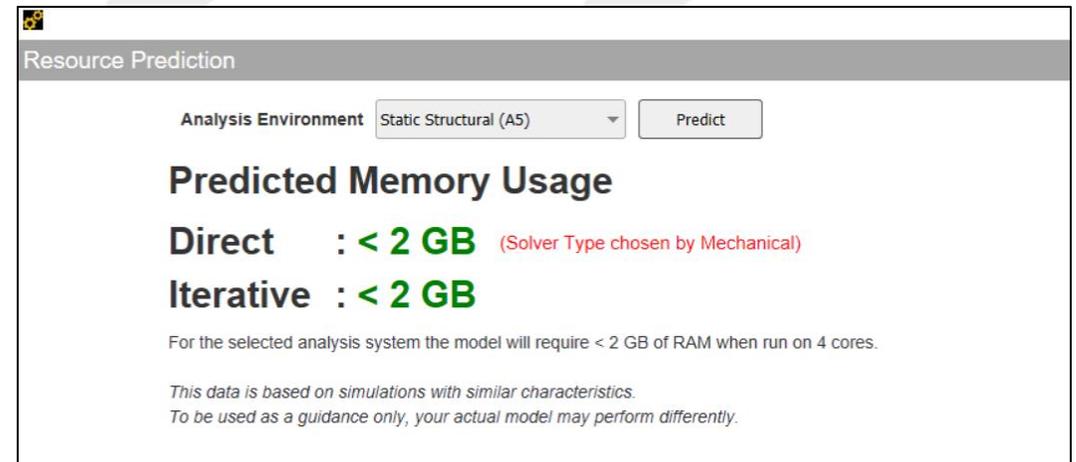
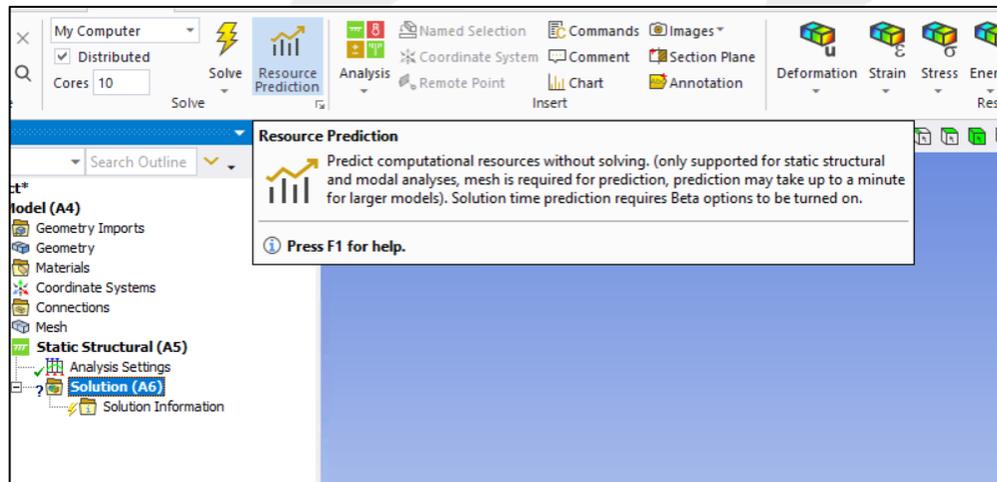
- The optimizer may converge prematurely due to the “mode-crossing” phenomenon
- Roughly speaking, the modes order changes during the optimization and confuses the optimizer
- The “smooth minimum” option manages this context

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



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

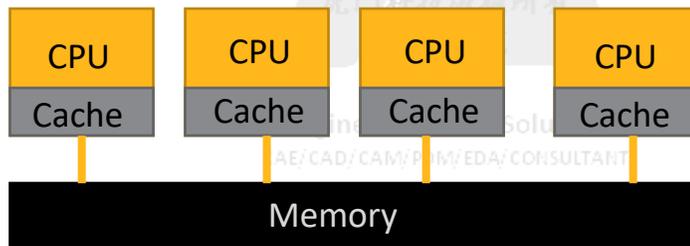
Total core count = $N \times M = P$ cores

`ansys221 -b -np 8 -nt 2 <test.dat> out`

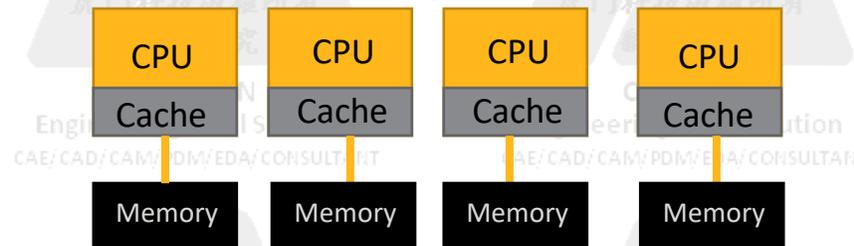
will use 8 processes with 2 threads per process for a total of 16 CPU cores

Distributed Ansys Enhancements

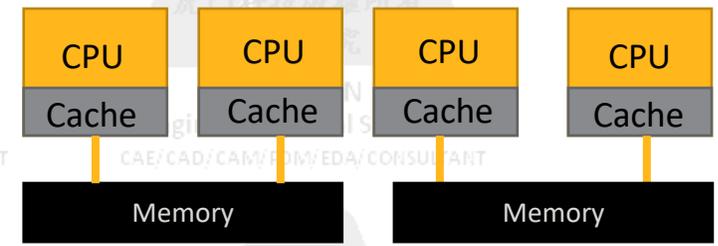
- Hybrid parallel
 - Wide applicability → Works for all features supported in DMP mode
 - Supported with all platforms and MPI libraries
 - Reduces memory requirements → Run larger jobs on your cluster
 - Improves scalability → Utilize more cores in your compute nodes



SMP



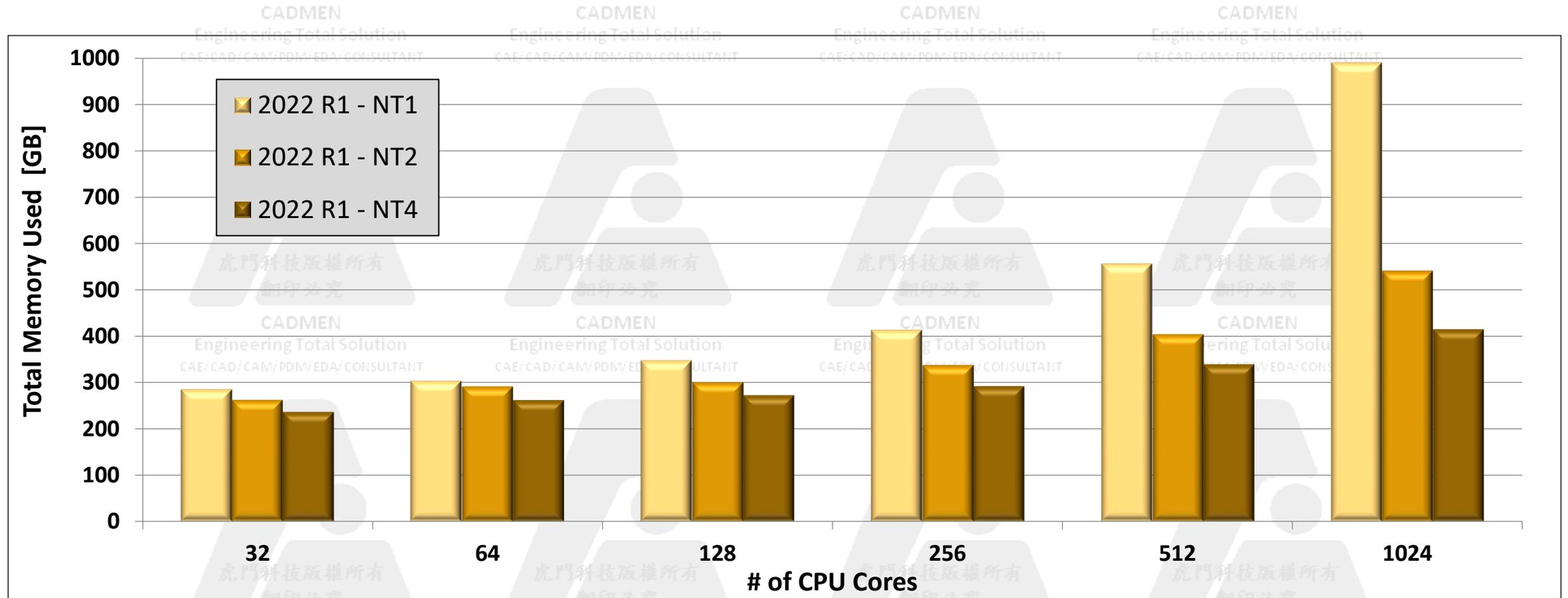
DMP



Hybrid

Distributed Ansys Enhancements

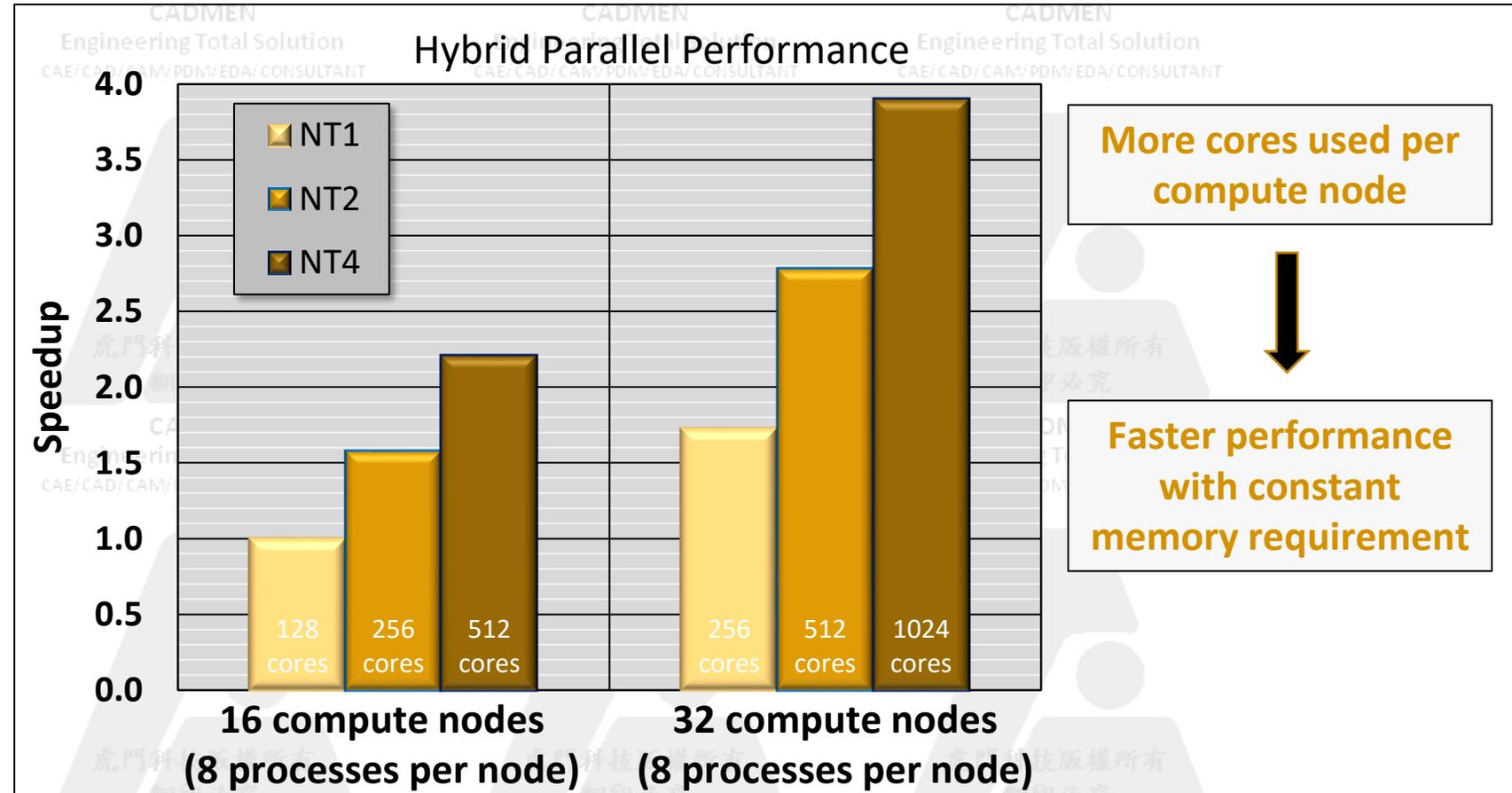
- Hybrid parallel reduces total memory requirements (Engine benchmark)



Distributed Ansys Enhancements

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

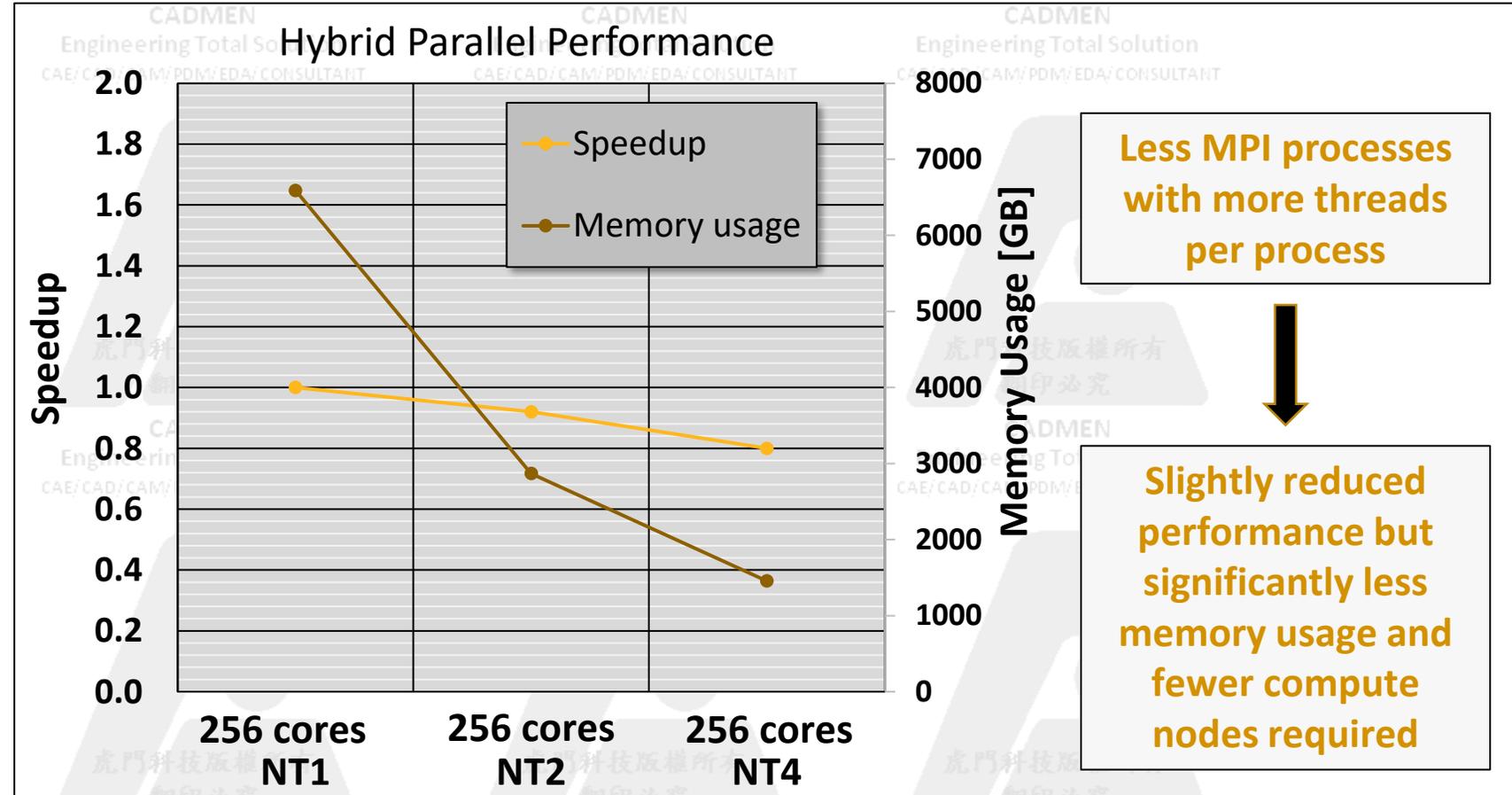
- 88 million DOF; sparse solver
- Nonlinear static analysis involving contact, large deflections, 126 Newton-Raphson iterations to converge
- Linux cluster; each compute node contains 2 Intel Xeon Platinum 8168 processors (44 cores), 346GB RAM, SSD, CentOS 7.9



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

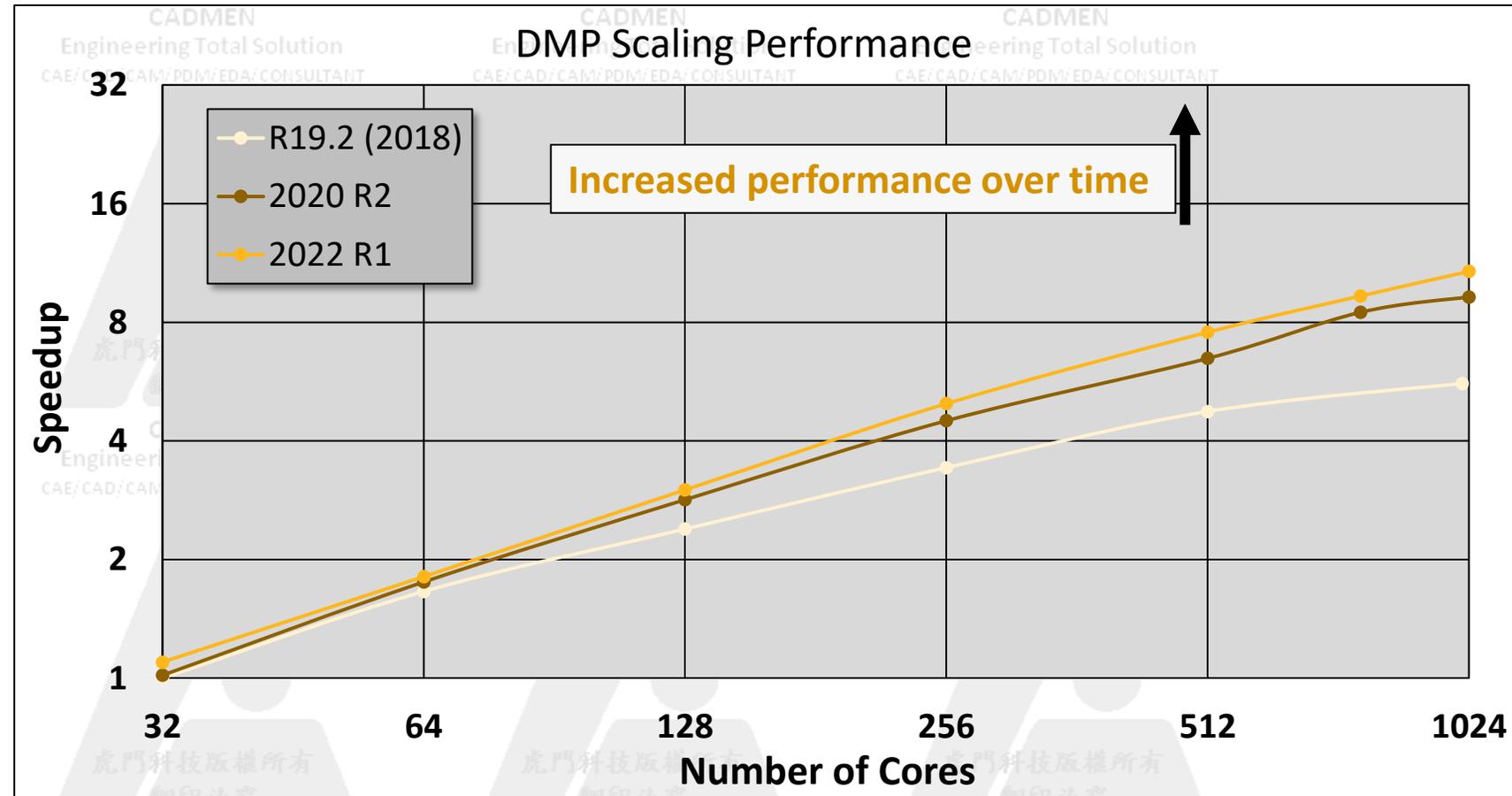


Distributed Ansys Enhancements

- 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



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

