Release 2022 R1 Highlights
Ansys Mechanical

Table of Contents

- Mechanical
- External Models
- Meshing
- Post Processing/Graphics
- Composites
- Structural Optimization
- Linear Dynamics
- MAPDL
  - Contacts
  - Elements
  - Linear Dynamics
  - Materials
  - Fracture
  - Nonlinear Adaptivity (NLAD)
  - Solver
- Aqwa
- Workbench Additive
- Explicit Dynamics
- Workbench LS-DYNA
- Rigid Body Dynamics
- Motion Integration
- nCode DesignLife Integration

Mechanical

Rigid Transformations on External Data

- Rigid Transformation inside Mechanical is supported for data imported from an upstream External Data system
- This transformation is added onto the Rigid Transformation defined in External Data system and can be used to further transform the data
- Two options to define the transformation:
  1. Use Origin and Euler Angles
  2. Use Coordinate Systems

Details of "Imported Pressure"

Scope	
Definition	
Graphics Controls	
Settings	
Rigid Transformation	
Mesh Alignment	Use Origin and Euler Angles
Origin X	1. m
Origin Y	1. m
Origin Z	0. m
Theta XY	45 degree
Theta YZ	0. degree
Theta ZX	0. degree
Legend Controls	

Details of "Imported Pressure"

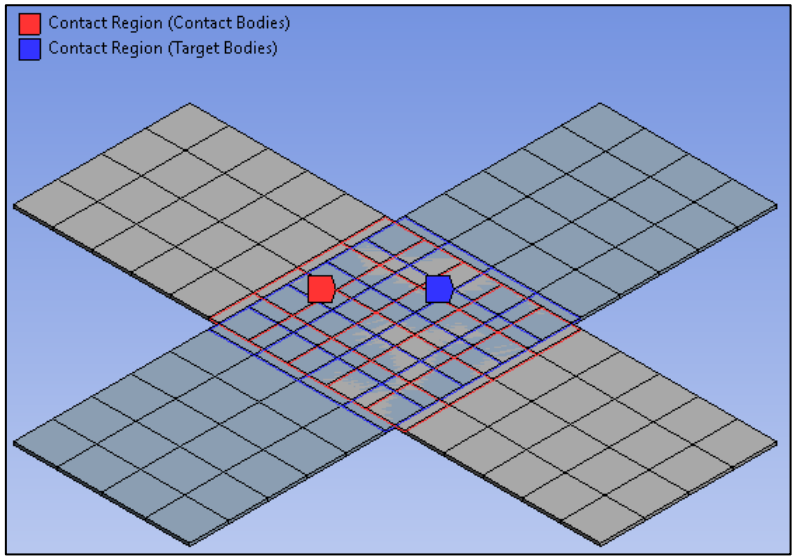
Scope	
Definition	
Graphics Controls	
Settings	
Rigid Transformation	
Mesh Alignment	Use Coordinate Systems
Source Coordinate System	Global Coordinate System
Target Coordinate System	Coordinate System
Legend Controls	

Contact Shell Element Scoping

- 3D shell element scoping is supported for both **Contact** and **Target** sides of a “Contact Region” when using the Mechanical APDL solver
- This scoping capability supports both Geometry and Named Selection scoping methods

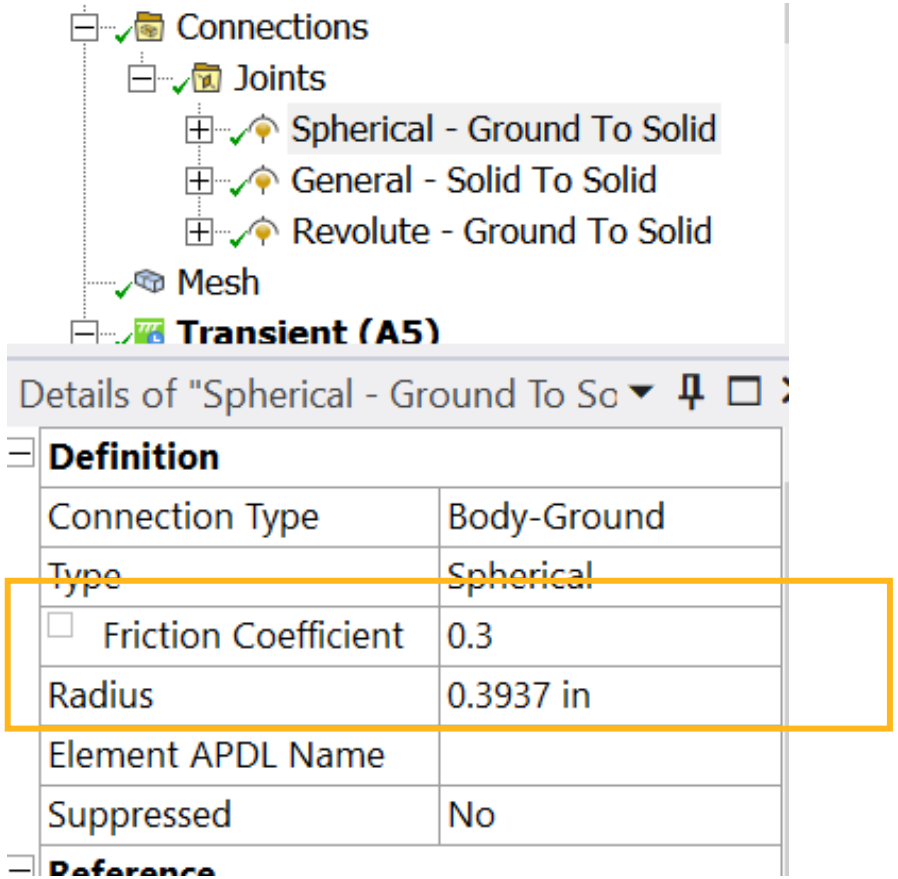
Details of "Contact Region"

Scope	
Scoping Method	Geometry Selection
Contact	20 Elements
Target	20 Elements
Contact Bodies	Surface1
Target Bodies	Surface2
Contact Shell Face	Program Controlled
Target Shell Face	Program Controlled
Shell Thickness Effect	No



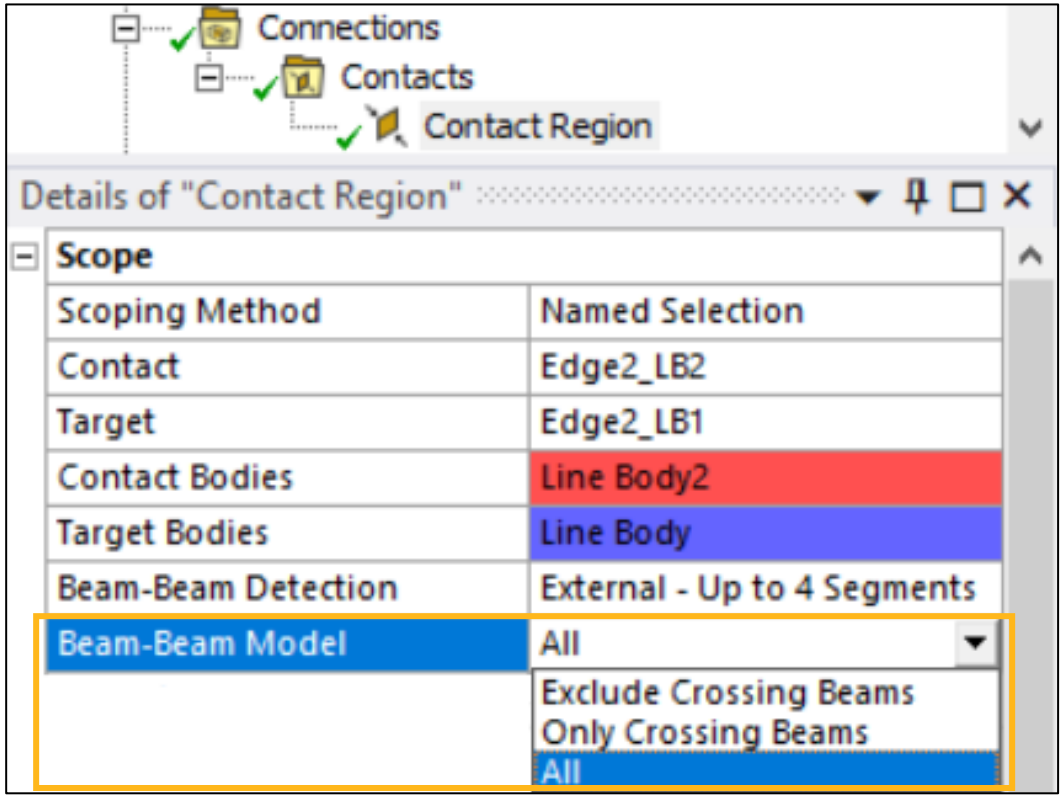
Spherical Joint: Friction properties

- Supported the friction for MAPDL solver using Friction Coefficient and Radius properties



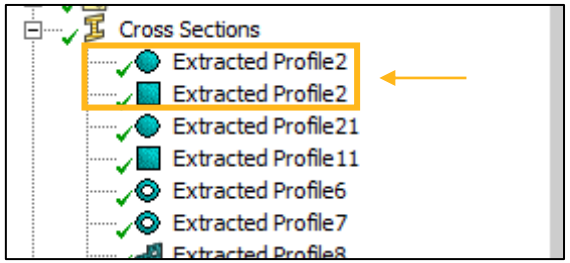
Beam to Beam Model for Contacts

- Displays this property when it detects beam-to-beam contact.
- Enables you to specify the contact traction-based model the application uses for Beam-to-Beam contact
- Beam-Beam Model:
  - ***Exclude Crossing Beams:*** Excludes the contact for any beams that cross one another.
  - ***Only Crossing Beams:*** Includes only the contact for beams that cross one another
  - ***All (Default):*** includes all beam contact scenarios, which are: beam/edge to surface contact, parallel beam-to-beam contact, and crossing beam-to-beam contact

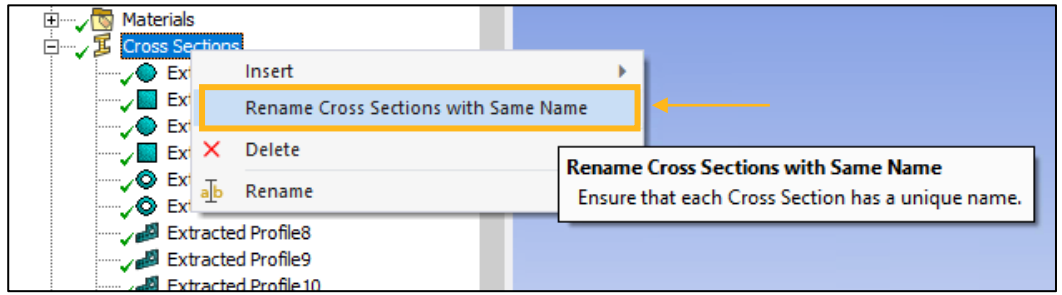


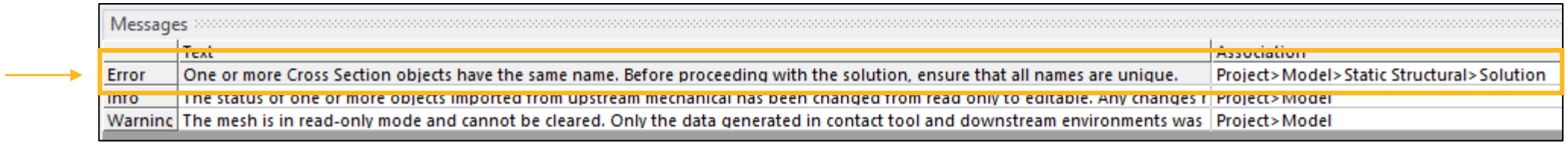
Better Handling of Duplicate Cross Section Names

- Immediate error message if a solve is started with duplicate cross sections



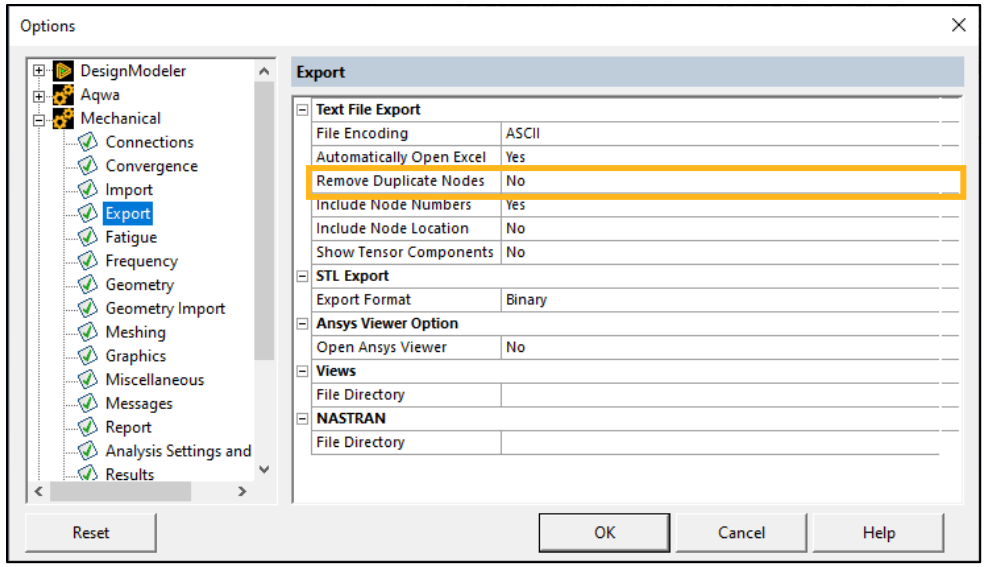
- New option on the Cross Sections context menu to automatically rename any duplicate names

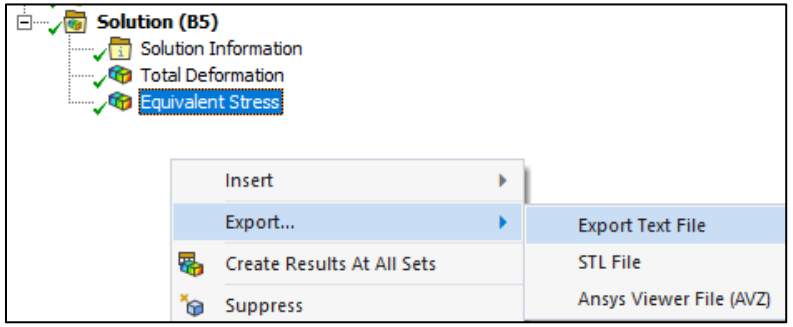




Remove Duplicate Nodes Option

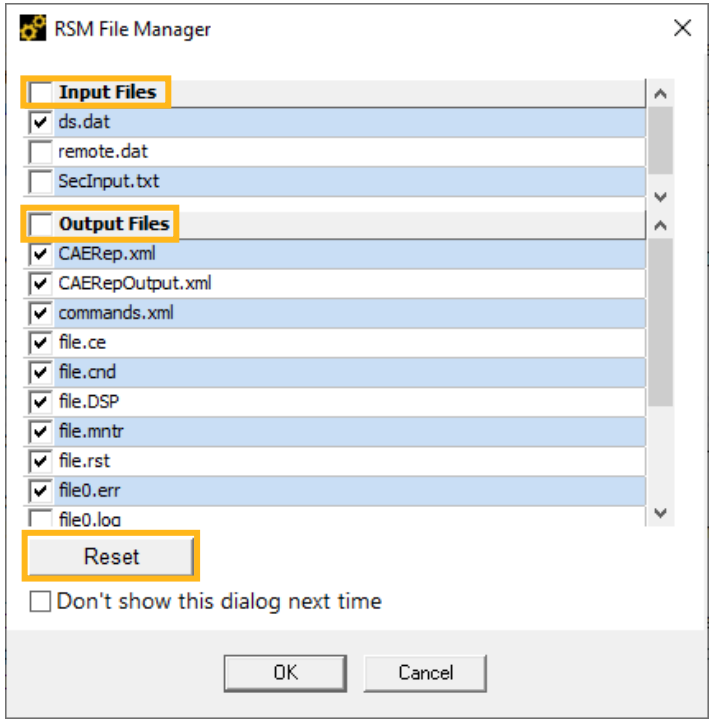
- Added option **Remove Duplicate Nodes** under Export
- When set to ‘Yes’, duplicate nodes will be removed when exporting results to a text file





RSM File Manager Enhancements

- Added **Select-all option** for both Input and Output files
- Added ‘Reset’ button to load the default checkbox settings
- Checkbox settings will now persist between solves, including for files that are no longer present
  - For example, input/output files that are present during one run but not present during another will still have their checkbox setting saved

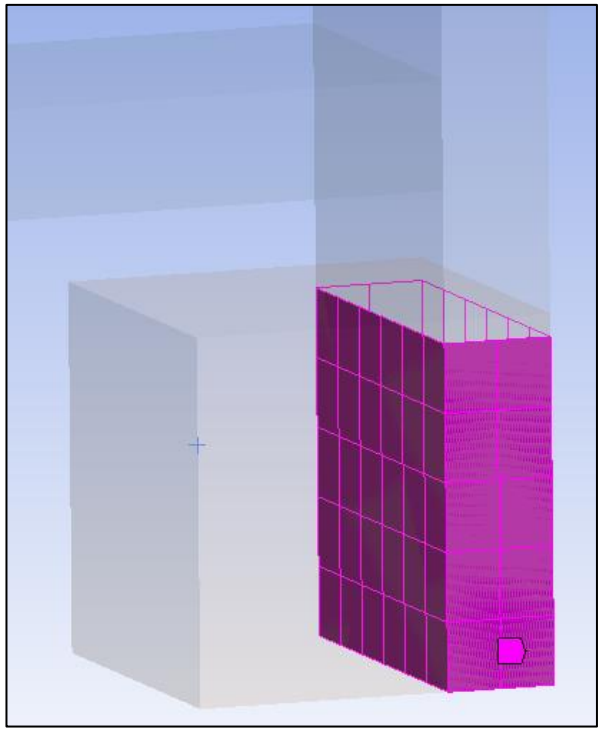


Within Body: Named Selections Criterion

- Added **Within Body** criterion to the Named Selections worksheet. This criterion is only available for nodes, elements, and element faces. Additionally, only geometry bodies with Treatment set to **Construction Body** may be chosen

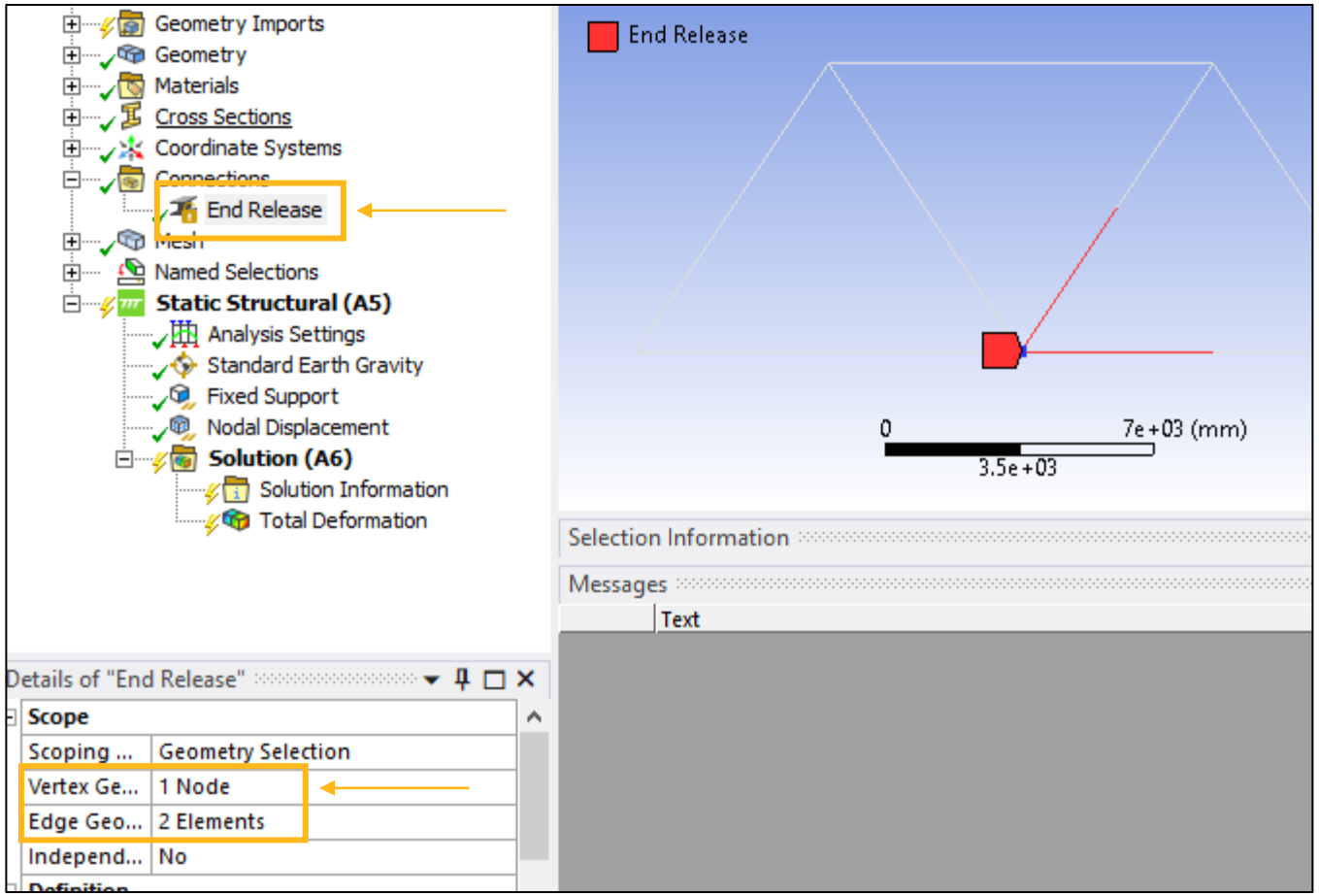
- This criterion will generate a named selection of the chosen Entity Type that are contained within the **Construction Body**

	Action	Entity Type	Criterion	Operator	Units	Value
☑	Add	Element Face	Within Body	Equal	N/A	Cube



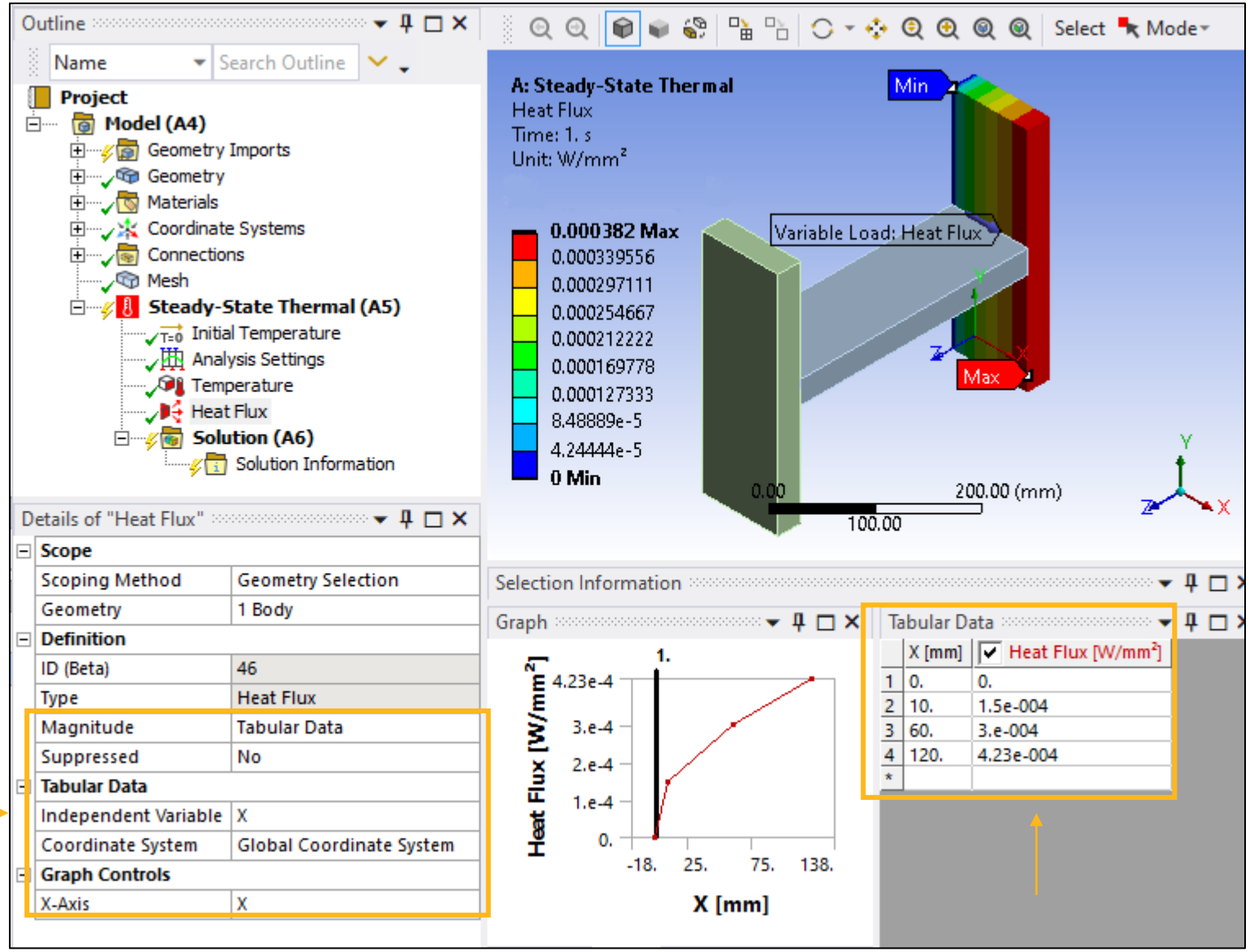
End Release FE Scoping

- Exposed the ability to scope nodes and elements to be released via "End Release"



Heat Flux Tabular Data

- Heat Flux can now be defined via spatially varying tabular data

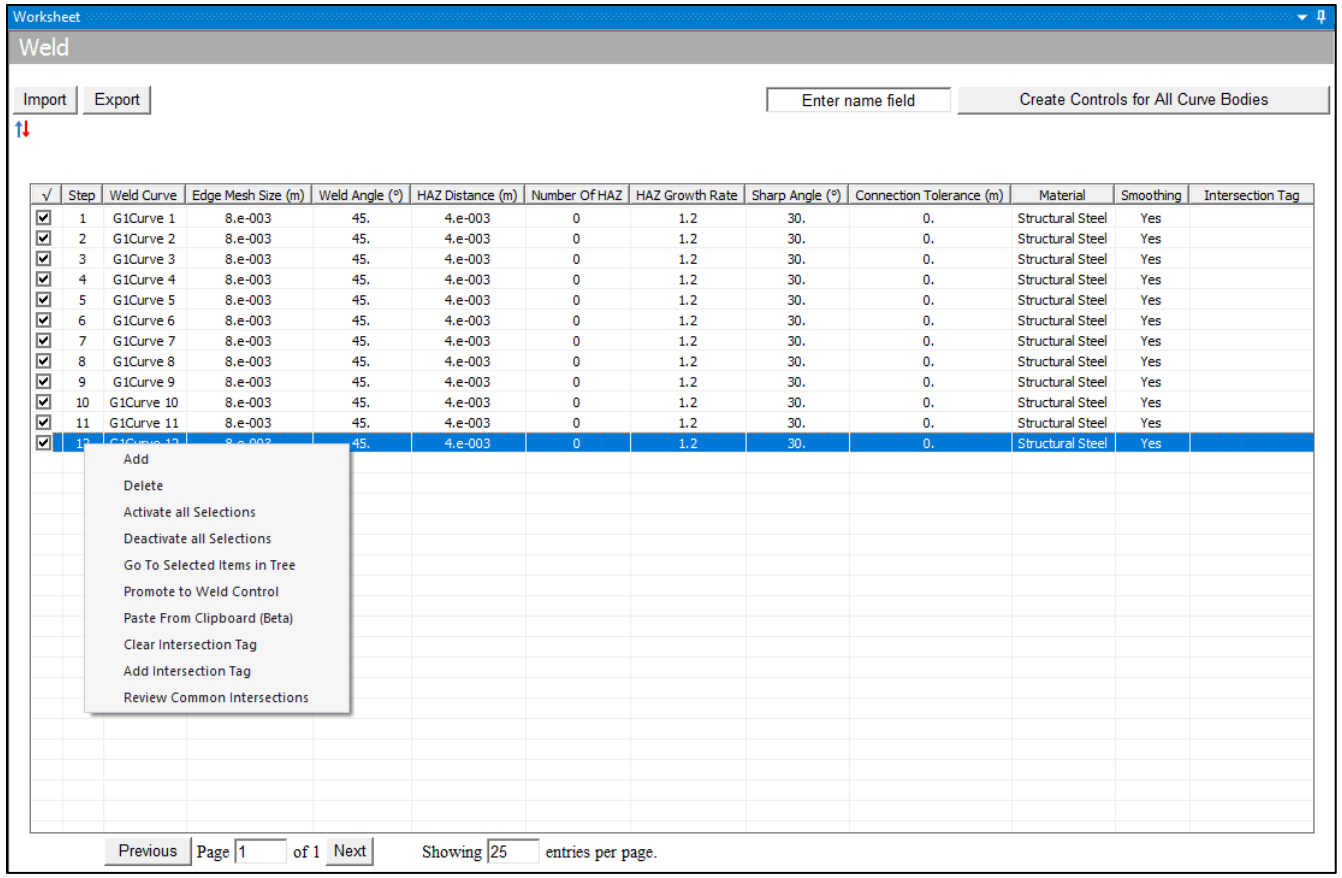


Support for Solid-Shell Elements in Trace Mapping

- Geometries meshed with solid-shell elements are now supported in the trace mapping workflow
- The behavior is like that of shell trace mapping, where trace data is controllable on a per-layer basis
- Dielectric and Trace materials can be defined on a per-layer basis
- This approach should provide users the accuracy they are targeting with less computational expense compared to solid element trace mapping

Seam Weld Worksheet Enhancements

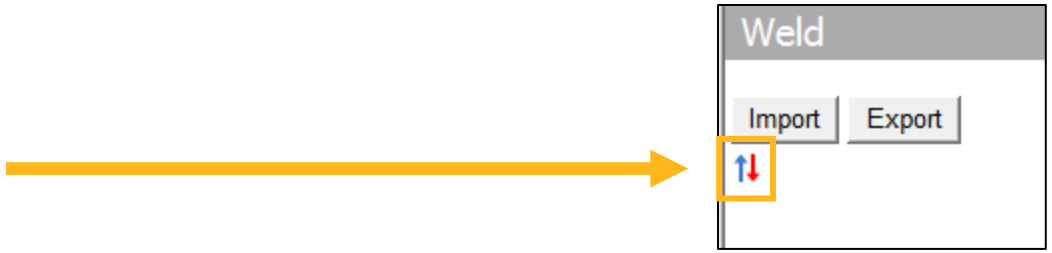
1. Activate all Selections turns on all the curves/edges in the worksheet
2. Deactivate all Selections turns off all the curves/edges in the worksheet
3. Go To Selected Items in Tree activates the weld curves in the geometry folder in the tree
4. Promote to Weld Control cuts and pastes all selected curves and puts them in a new Weld
5. Paste from Clipboard enables copying information from an excel file and pasting the information from a given cell. It is important that the correct cell from where the paste should happen is “right-clicked”. Information flows to the right and then downward from the selected cell
6. Clear Intersection Tag removes the intersection tags for the selected weld curves
7. Add Intersection Tag adds an intersection tag for the selected weld curves. Multiple tags are to be separated by a semi-colon
8. Review Common Intersections selects all the curves that have the same Intersection tag

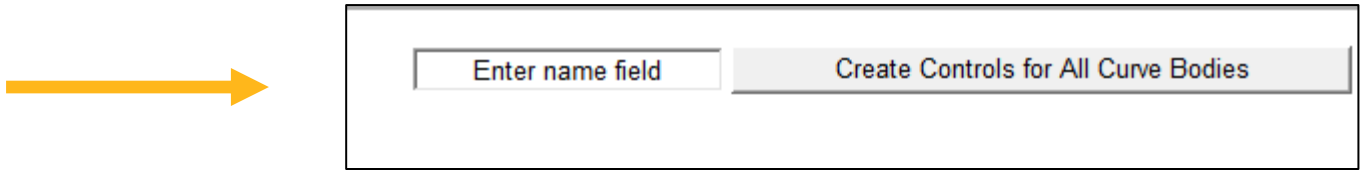


√	Step	Weld Curve	Edge Mesh Size (m)	Weld Angle (°)	HAZ Distance (m)	Number Of HAZ	HAZ Growth Rate	Sharp Angle (°)	Connection Tolerance (m)	Material	Smoothing	Intersection Tag
☑	1	G1Curve 1	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	2	G1Curve 2	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	3	G1Curve 3	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	4	G1Curve 4	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	5	G1Curve 5	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	6	G1Curve 6	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	7	G1Curve 7	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	8	G1Curve 8	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	9	G1Curve 9	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	10	G1Curve 10	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	11	G1Curve 11	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	
☑	12	G1Curve 12	8.e-003	45.	4.e-003	0	1.2	30.	0.	Structural Steel	Yes	

Seam Weld Worksheet Enhancements

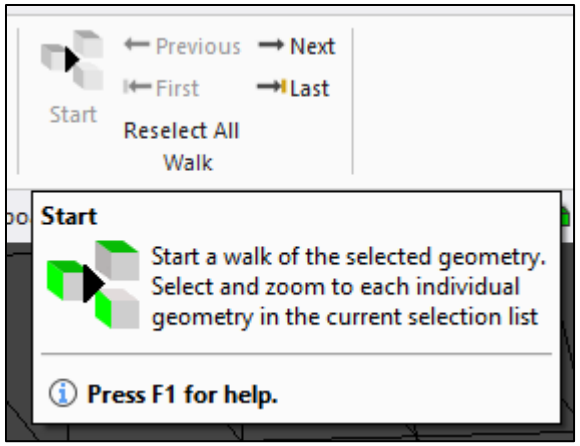
1. By pressing the **Sort by Selection** button, all selected weld curves (for that corresponding weld object) in the worksheet or graphics window are shown in page 1 in ascending order
2. Enter a string in the **Enter name field** box and click on the **Create Controls for All Curve Bodies** button. All weld curves that contain the string entered in this text field are only displayed in the worksheet. Leave this empty to generate all weld curves. This operation gets recorded/journaled and can be used for playback
3. Selecting one/multiple entries in the worksheet will show the selection in the graphics window and vice-versa

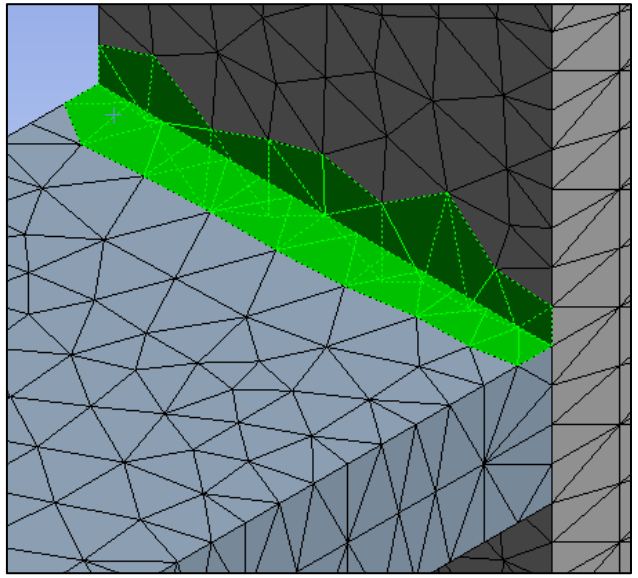


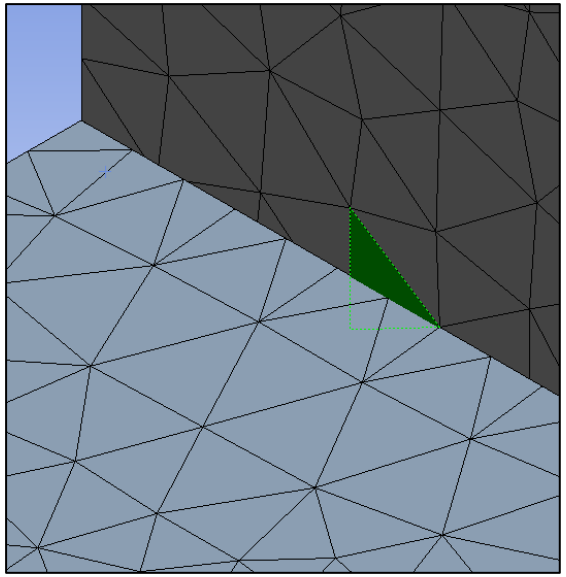


Model Walk for FE Data

- Model Walk visits each individual part of a selection
- Expanded to include Nodes, Elements, and Element Faces
- Automatically zooms in to make identification easy



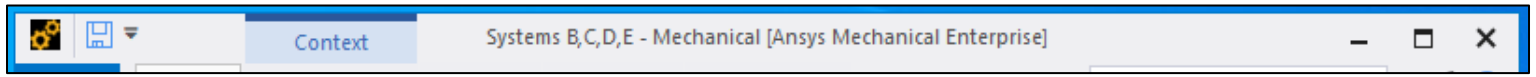




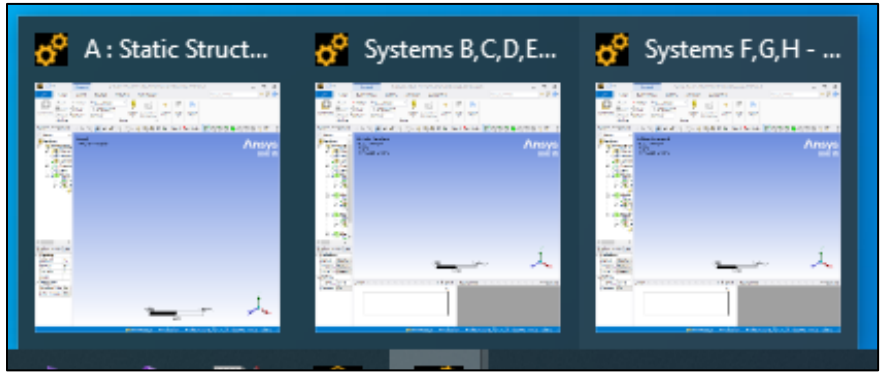
- Especially useful for walking selection sets created by error messages and criteria based named selections

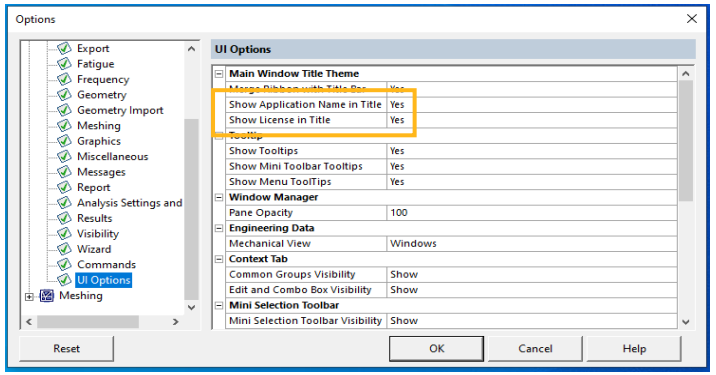
Mechanical Window Titles

- Instead of only showing “Multiple Systems”, the title bar now shows the system identifiers just as it does for single systems



- With several sessions open, it is easier to find a specific window. Hovering the mouse over the Mechanical icon in the task bar will show the entire title

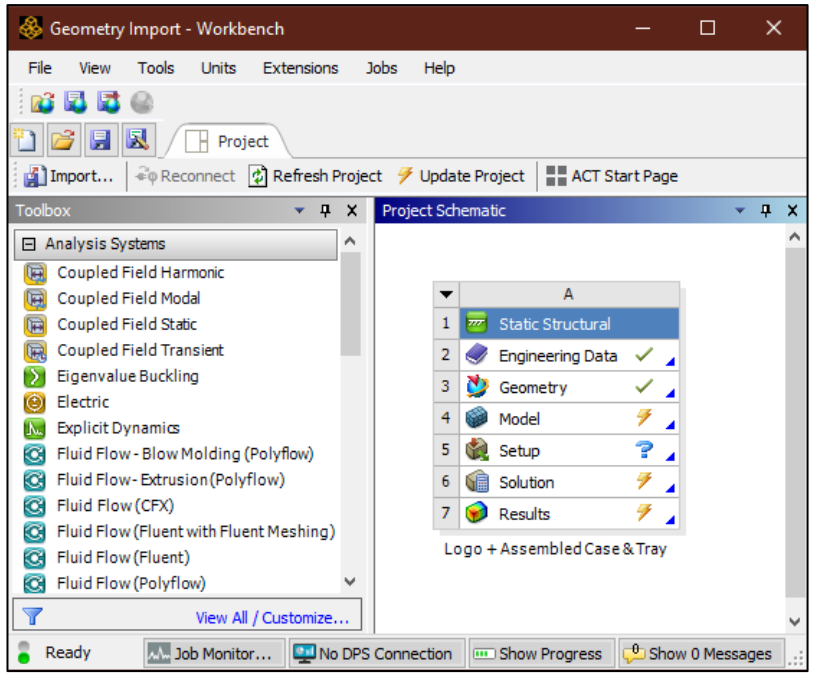




Control setting for window names in “Options” panel

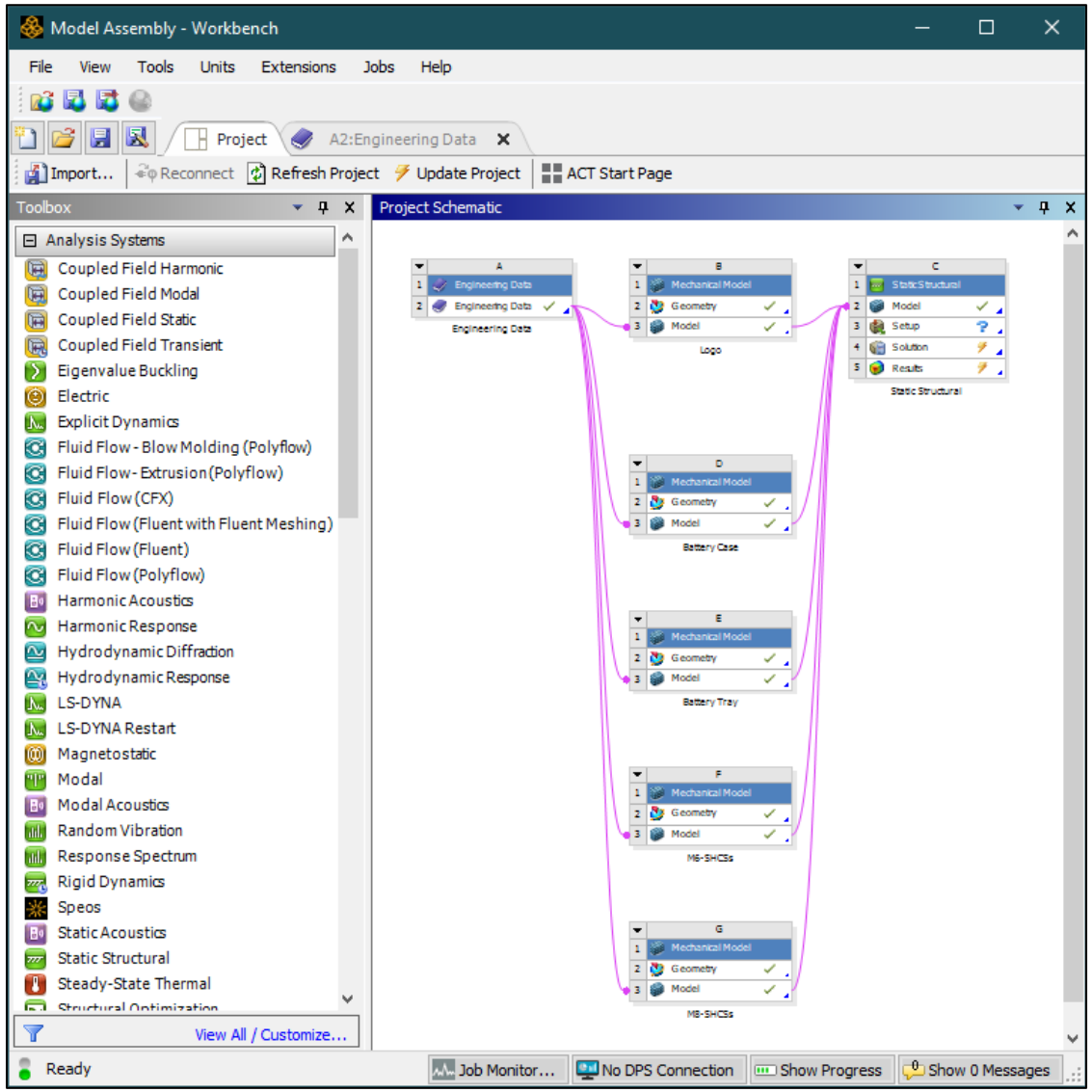
Geometry Import

- An easier, more unified approach to importing multiple geometry sources into Mechanical



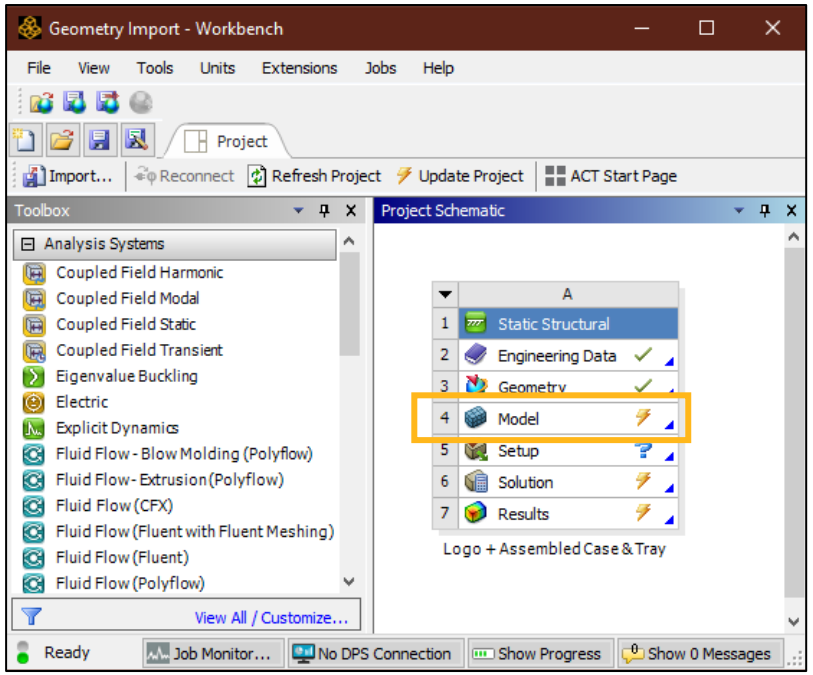
Geometry Import

Model Assembly



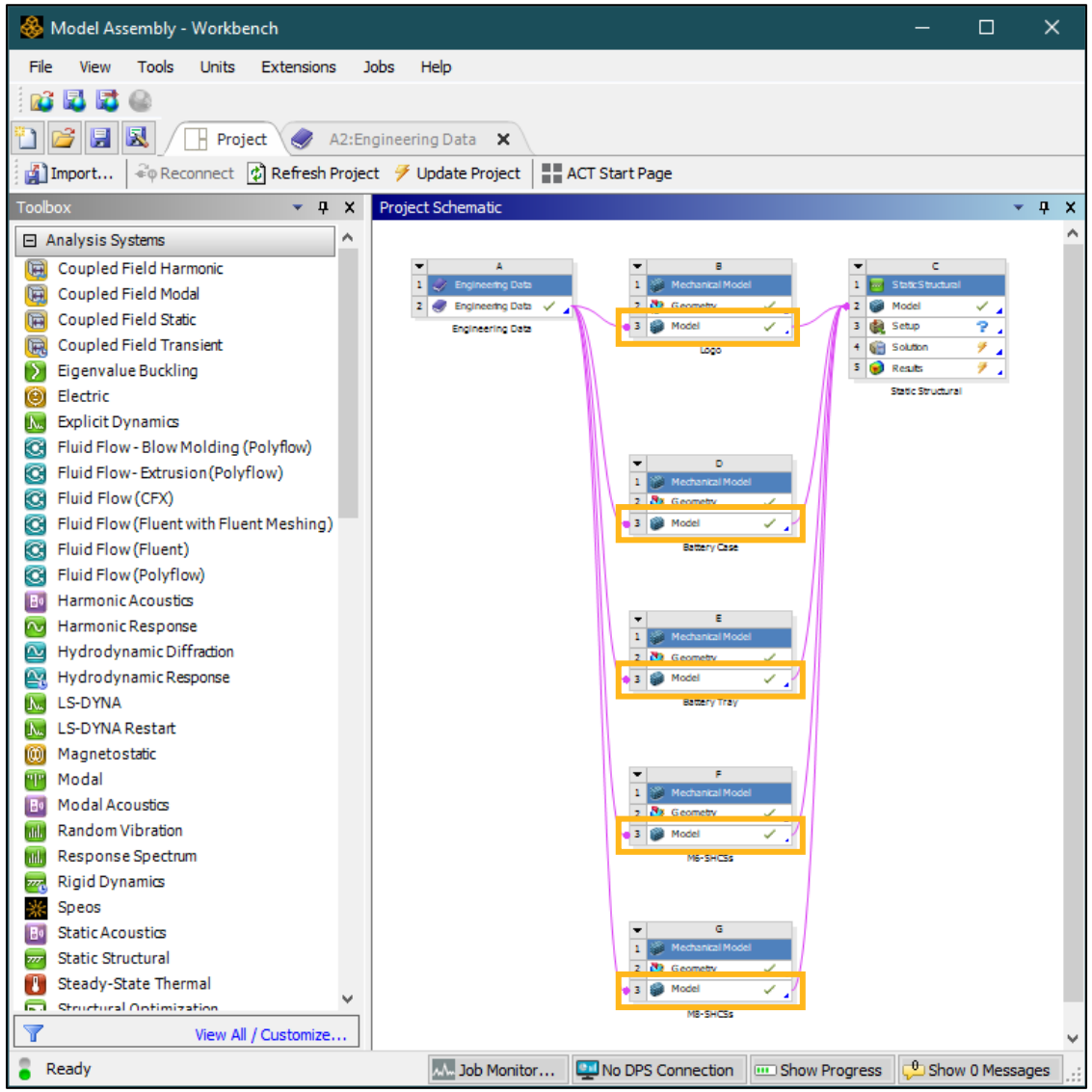
Geometry Import

- Geometry Import uses an "Assemble First, Mesh Later" workflow, which significantly reduces the “time-to-assembly” footprint



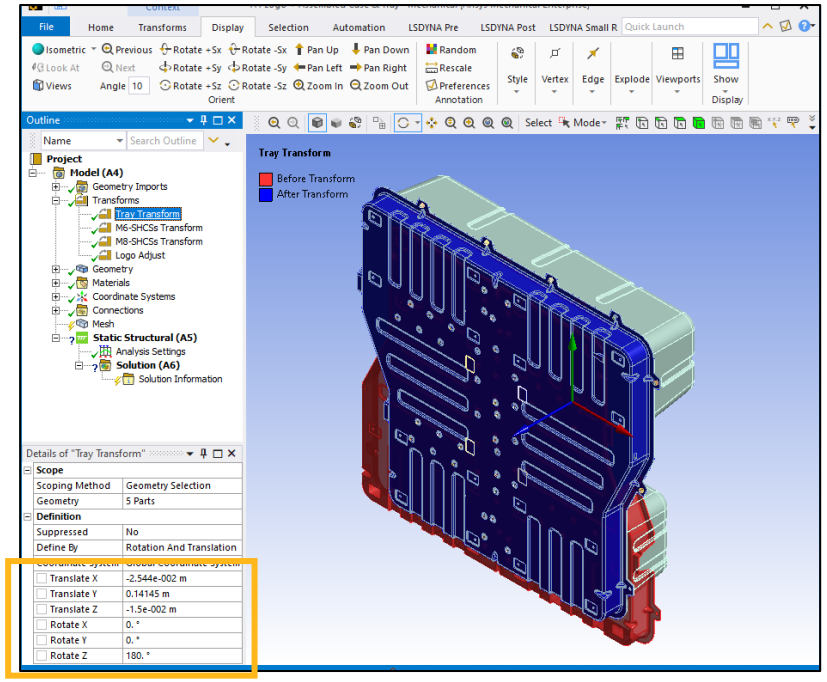
Geometry Import

Model Assembly



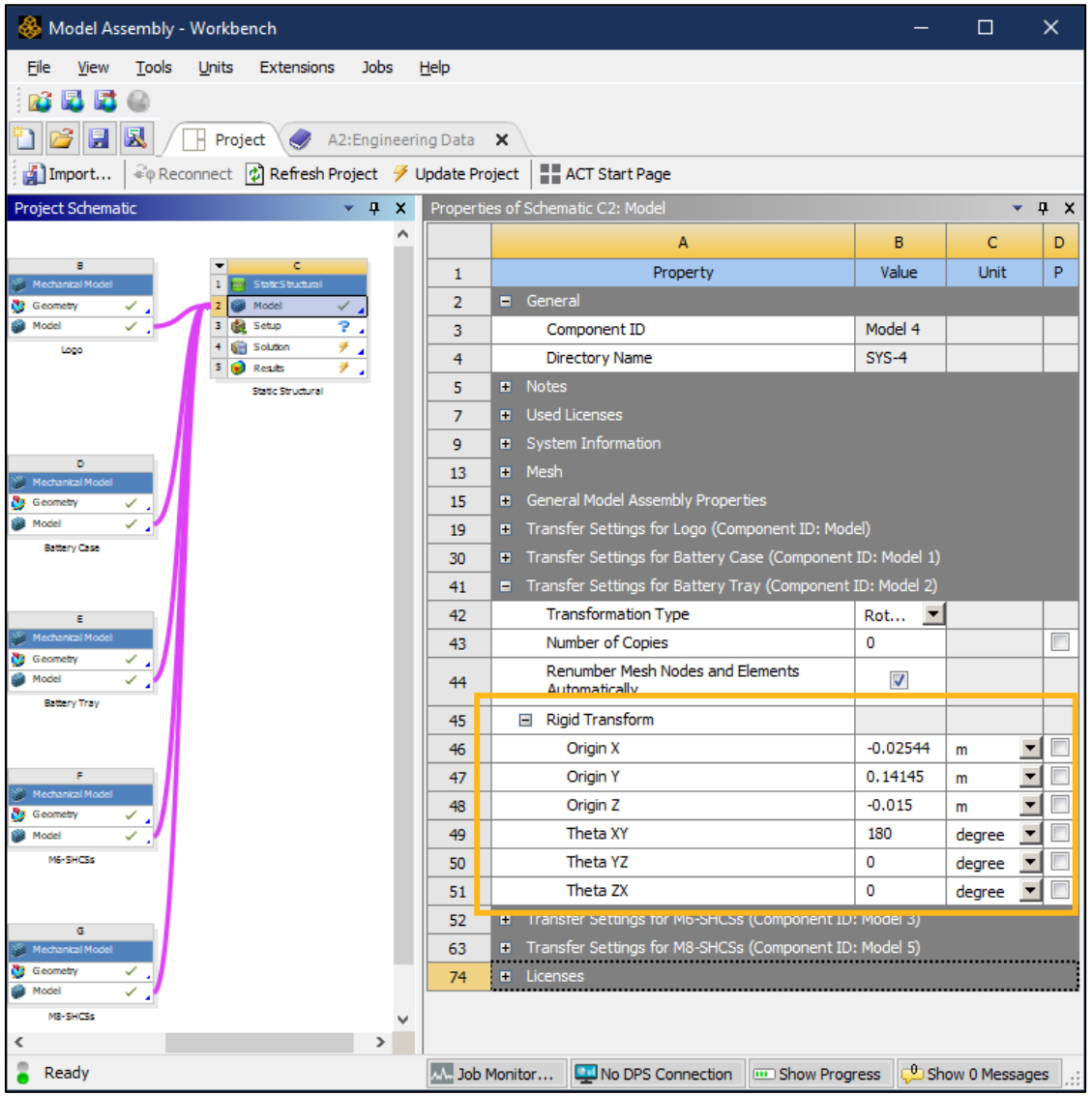
Geometry Import

- In Mechanical, part transformations provide continuous visual feedback during the assembly process



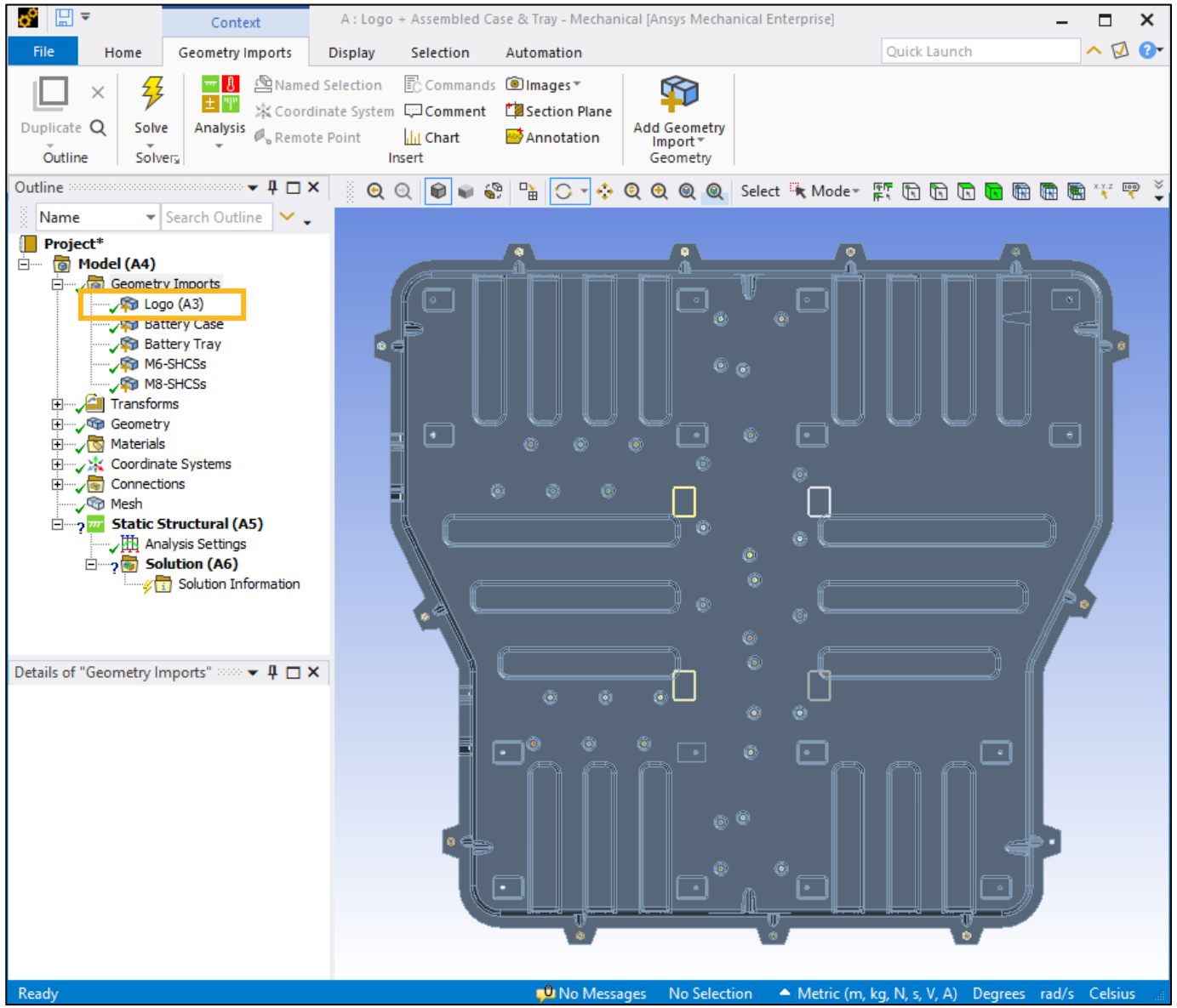
Geometry Import

Model Assembly



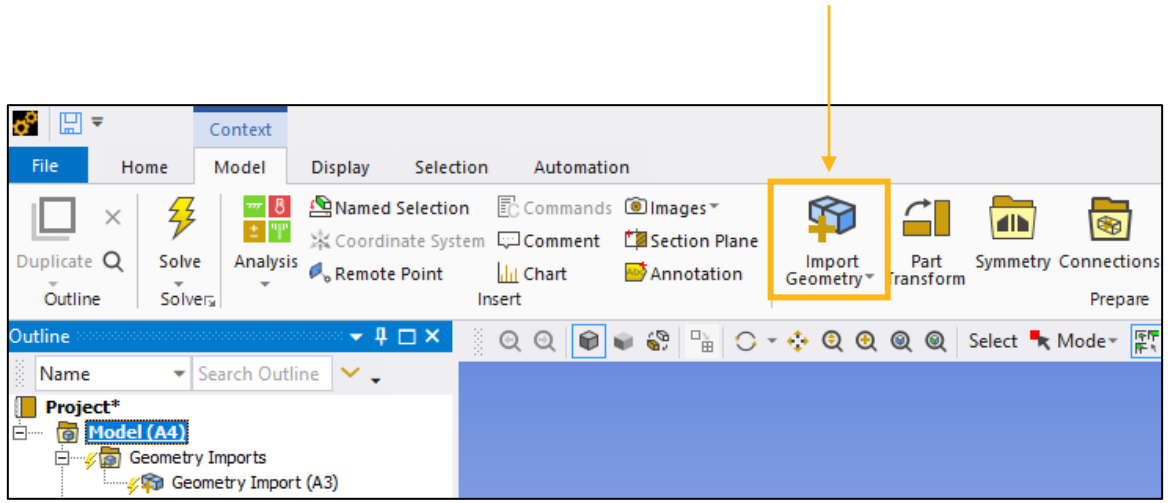
Geometry Import

- This first Geometry Import object represents the geometry specified in Workbench
- This is sometimes referred to as the "Primary Source" and is always shown in the Tree along with the "Geometry Imports" folder
- The primary source name also includes the cell ID of the corresponding "Geometry" cell in Project Schematic, which is "A3" in this example

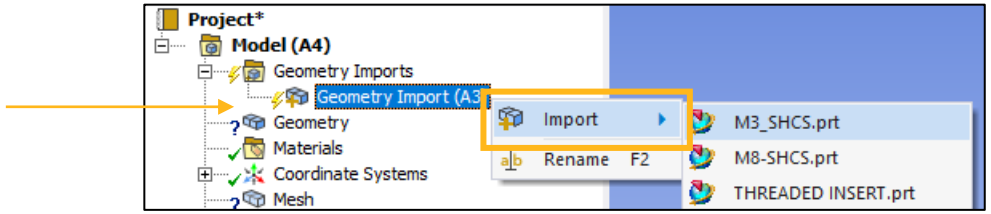


Geometry Import

- In Mechanical, the "Primary Source" import is started via either the Ribbon or Right-Mouse-Button (RMB) context menus

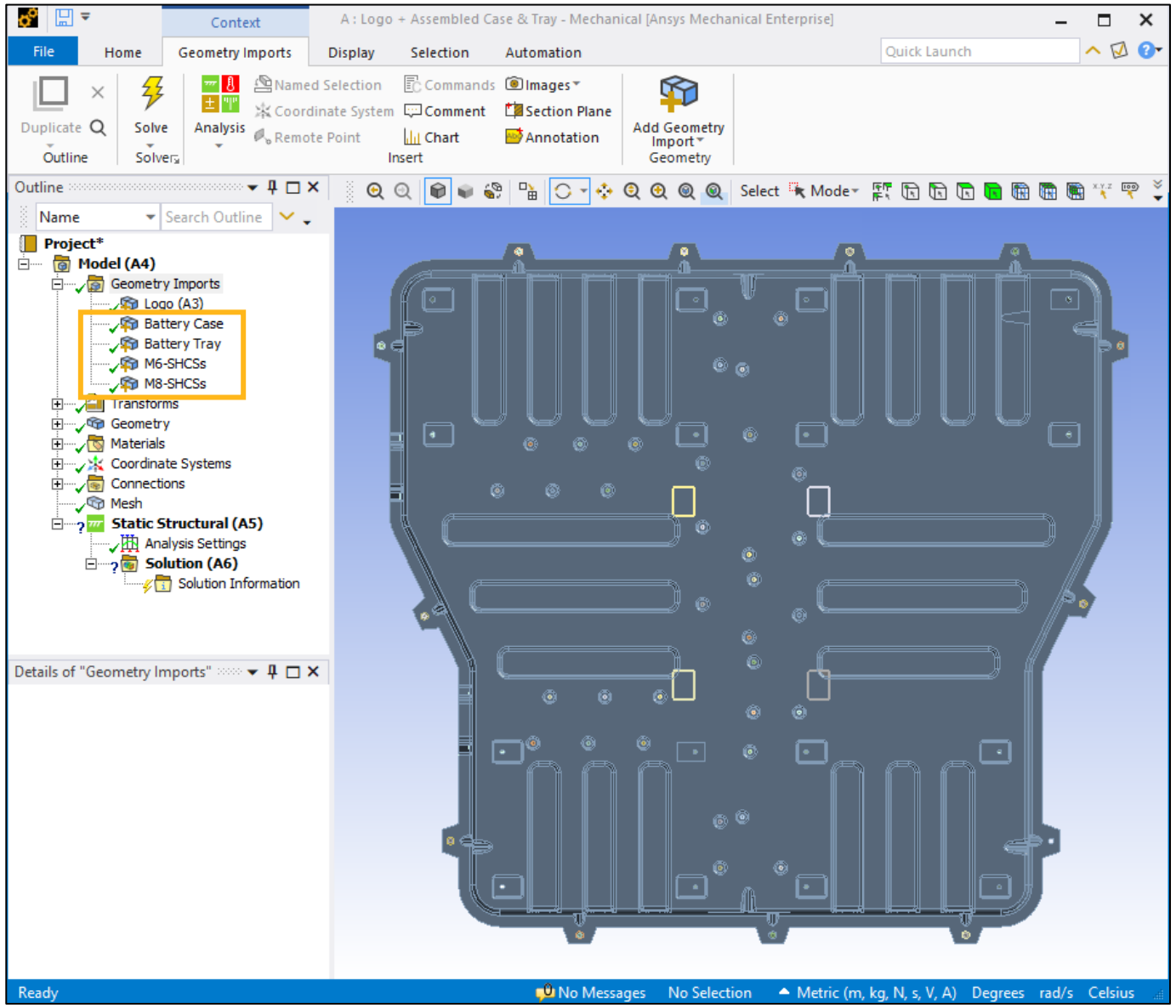


Pro Tip
Use the expanded menu to select from recently used geometry sources



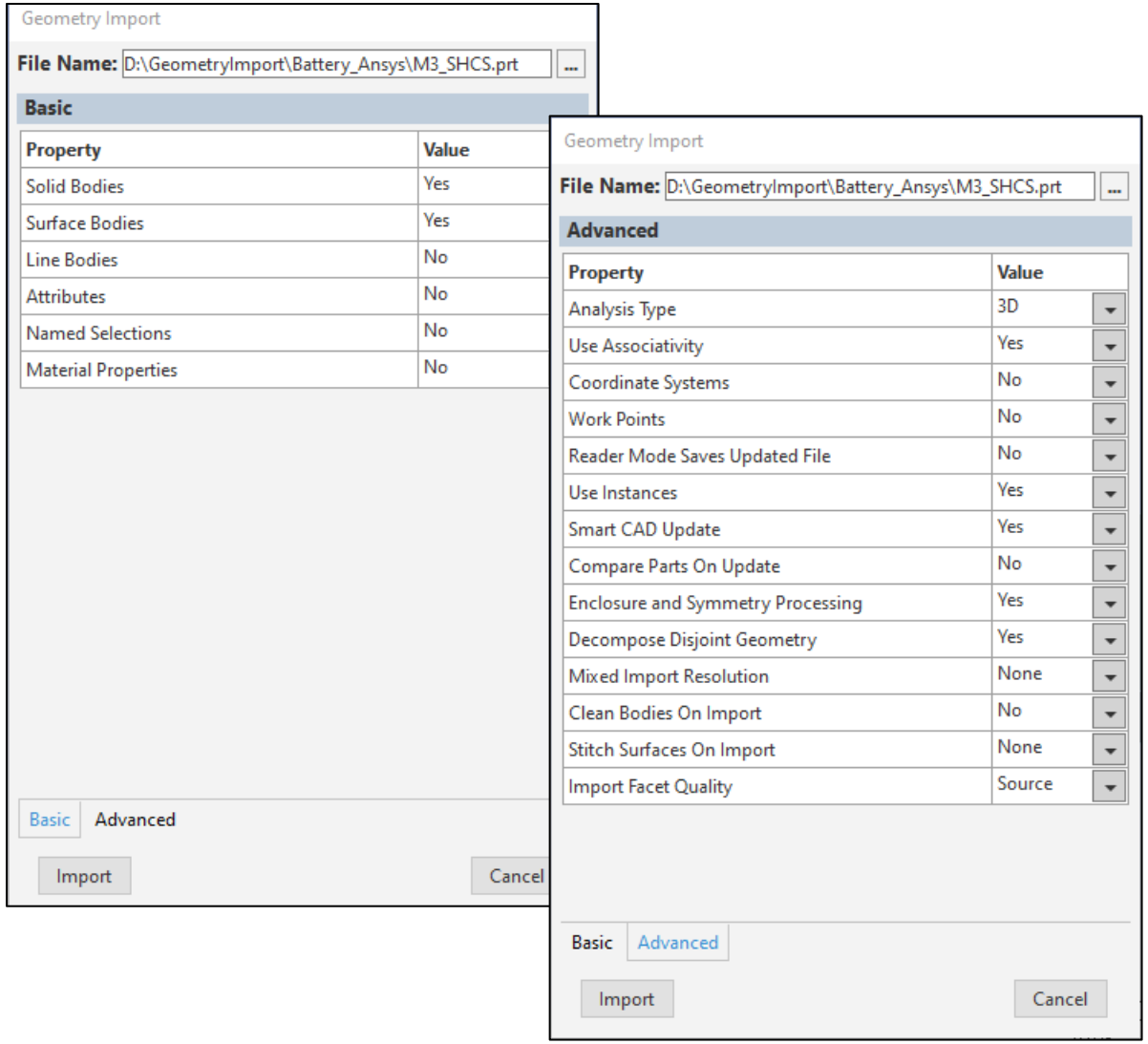
Geometry Import

- Additional Geometry Import objects can be added under the "Geometry Imports" folder
- These are sometimes referred to as "Secondary Imports" and will only appear when explicitly added



Geometry Import

- As before, CAD preferences specify how each import should be processed
- The "Primary Source" import respects the preferences set in the Properties page of the "Geometry" cell in Workbench
- For secondary Geometry Import objects, the preferences are specified *per import* after selecting the geometry source



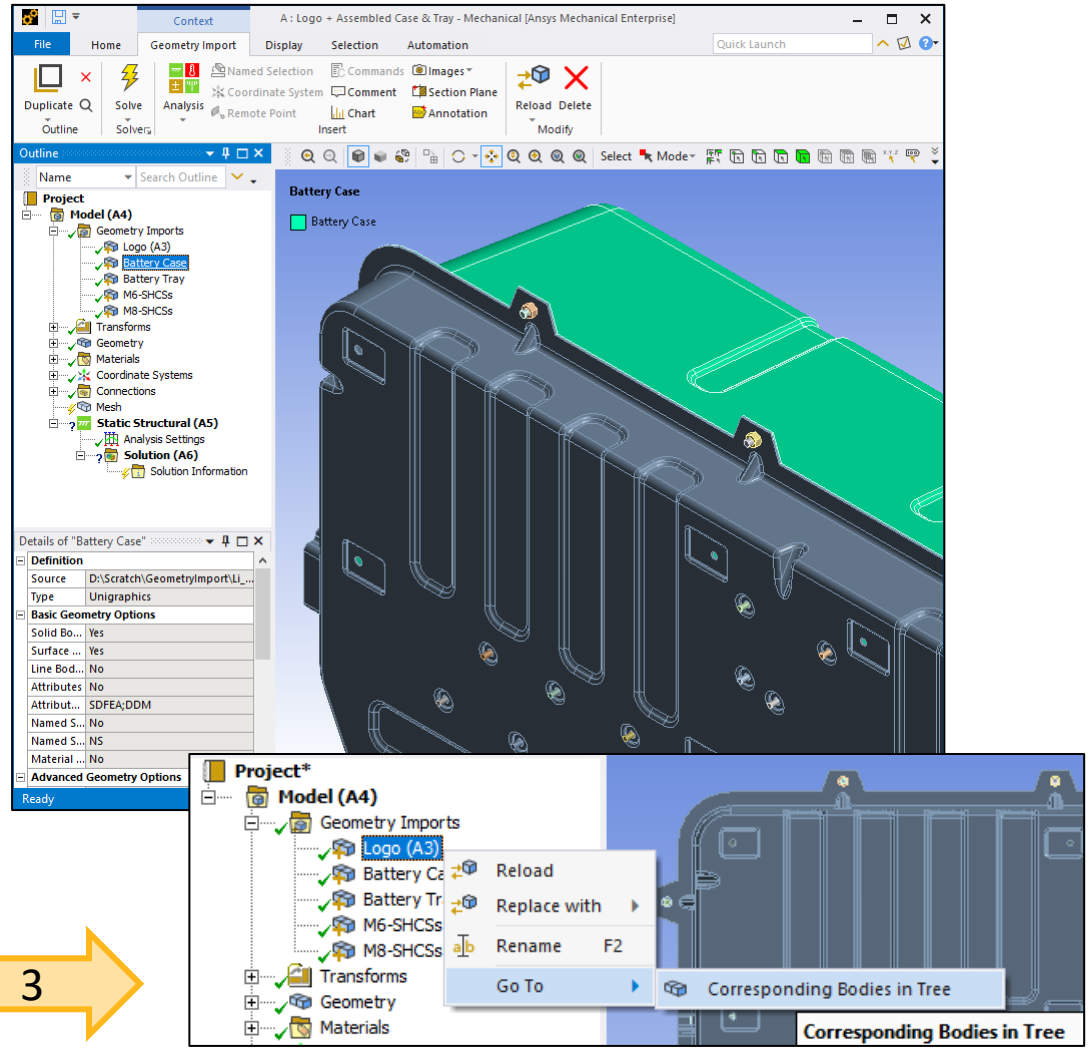
Geometry Import

1. Selecting a **Geometry Import** objects highlights the corresponding bodies
2. Information about each Geometry Import, such as the source file and CAD preferences, are shown in the details view
3. The corresponding bodies in the Tree can be navigated to via the RMB menu “Go To > Corresponding Bodies in Tree”

1

2

3

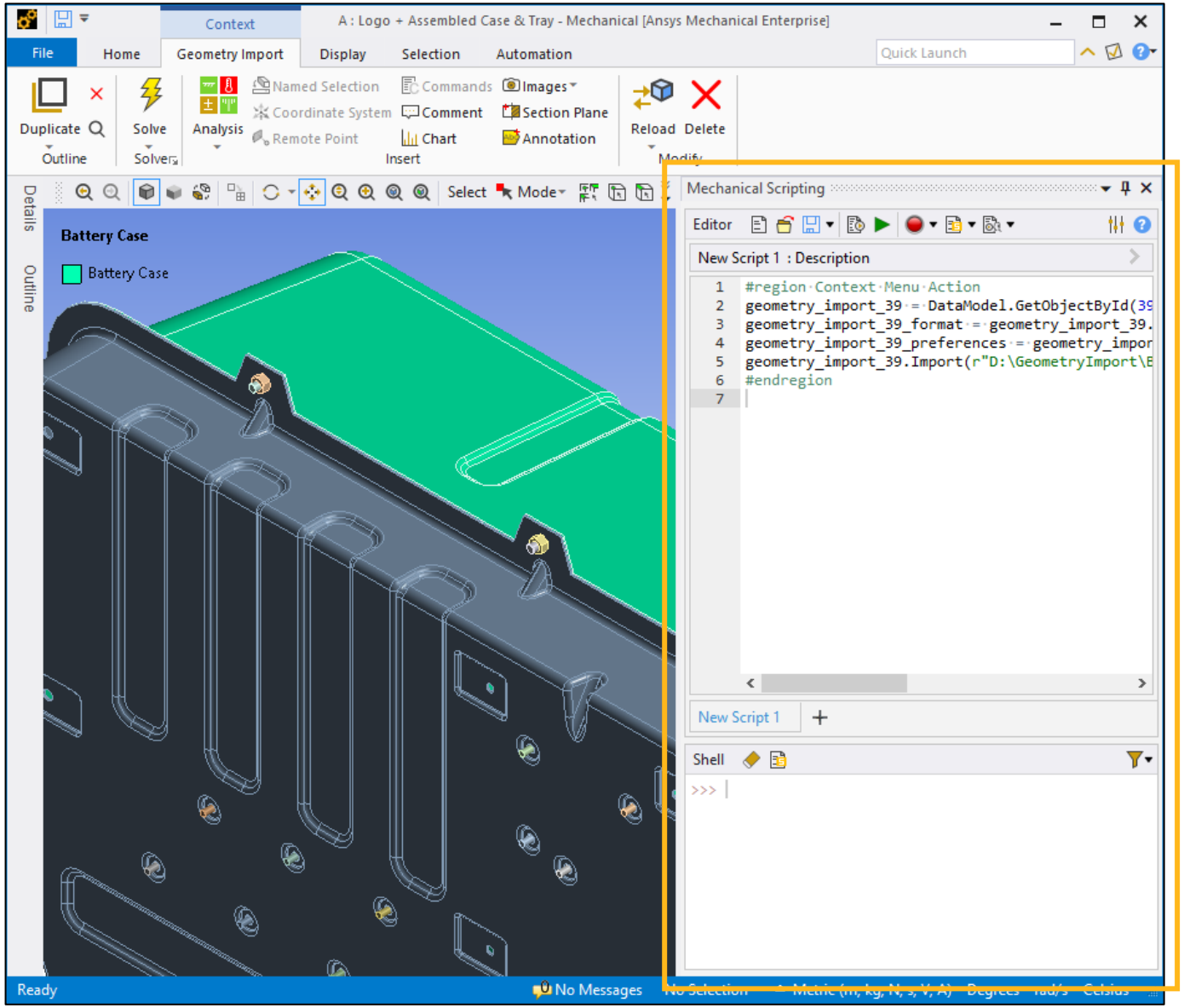


Pro Tip

Use “Go To > Corresponding Bodies in Tree”, then create a group with the selected parts, to easily organize parts under "Geometry" by import

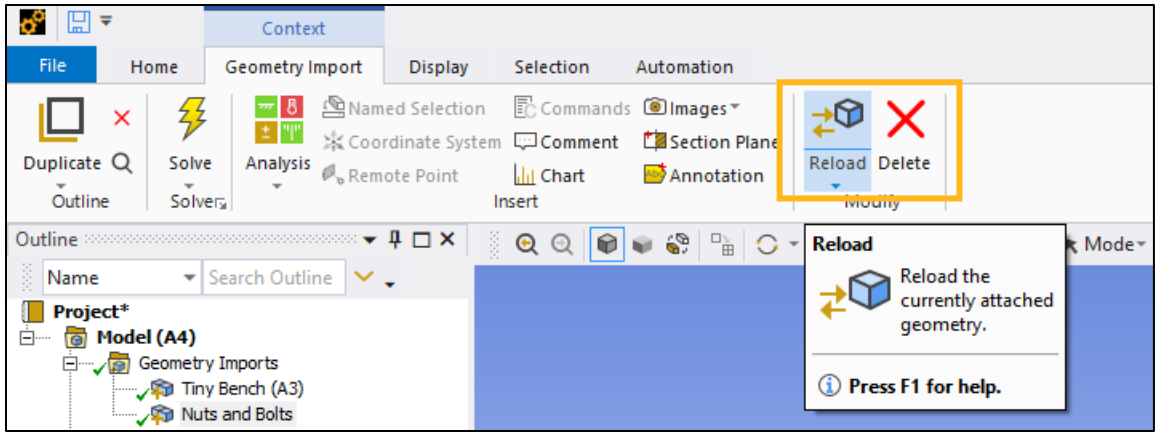
Geometry Import

- Geometry Import is a fully-recorded feature
- The included Geometry Import Python API enables powerful, scriptable operations



Geometry Import

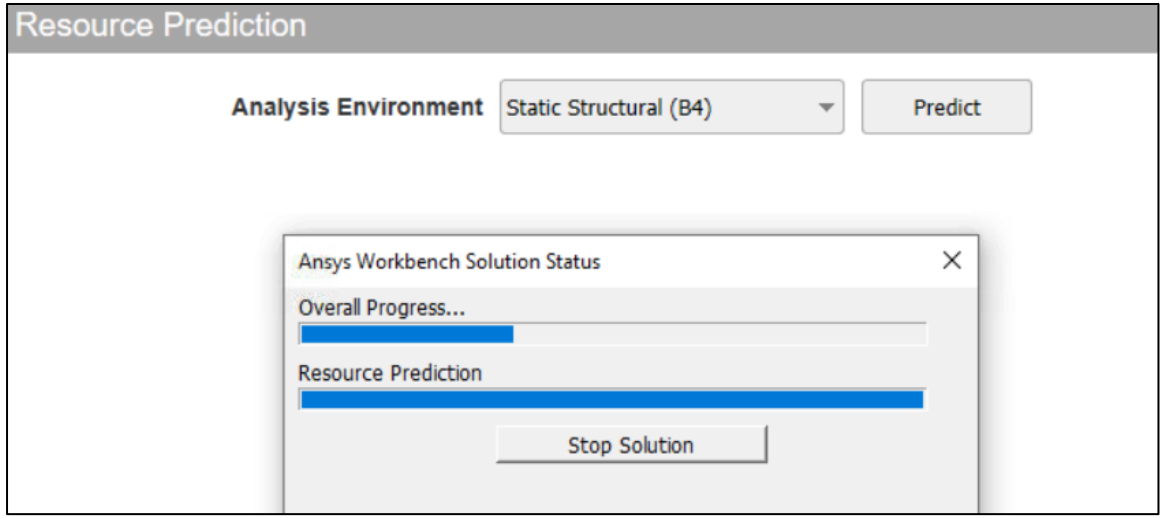
- Geometry sources can easily be reloaded or replaced via the “Reload” button in the Geometry Import Ribbon Context menu
- Secondary sources can also be deleted. Deleting a Geometry Import object deletes the corresponding parts

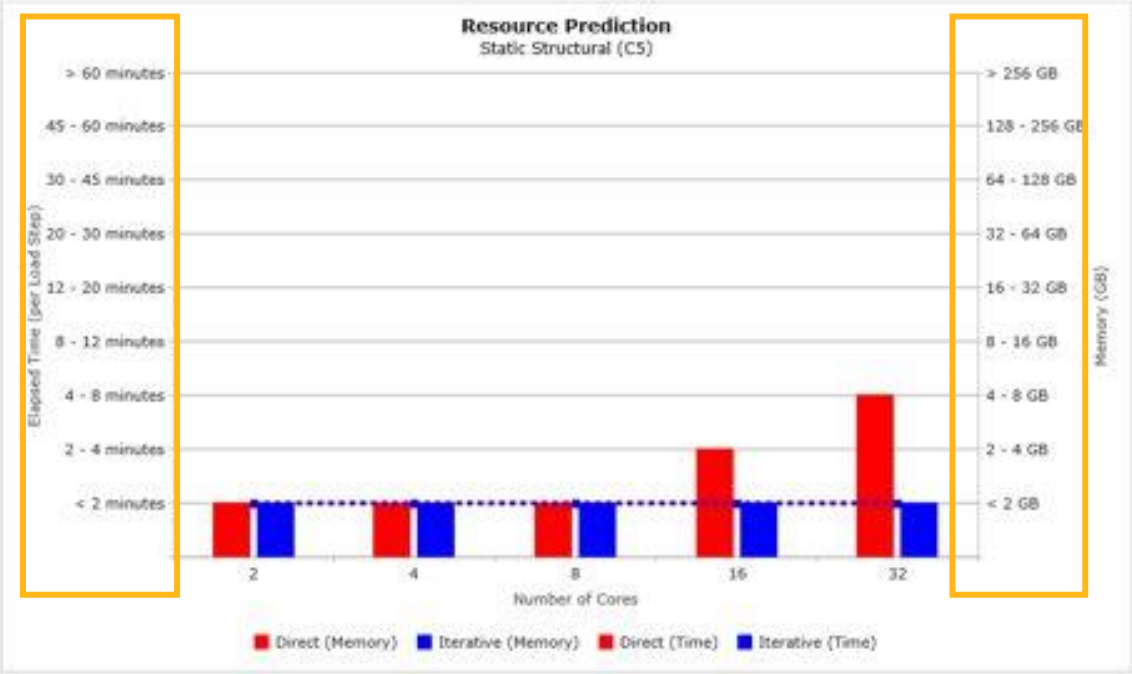


Pro Tip
Click the top-half of the Ribbon Context "Reload" button to automatically reuse the same file and CAD preferences

Resource Prediction Enhancements

- Resource Prediction is now run on a separate thread and progress is updated. The default for memory prediction is changed from Random Forest to Neural network algorithm
- With Beta options enabled:
  The number of bins for memory and time is increased to 9 bins. This increase in number of bins helps to better estimate the memory and time requirements for the simulation



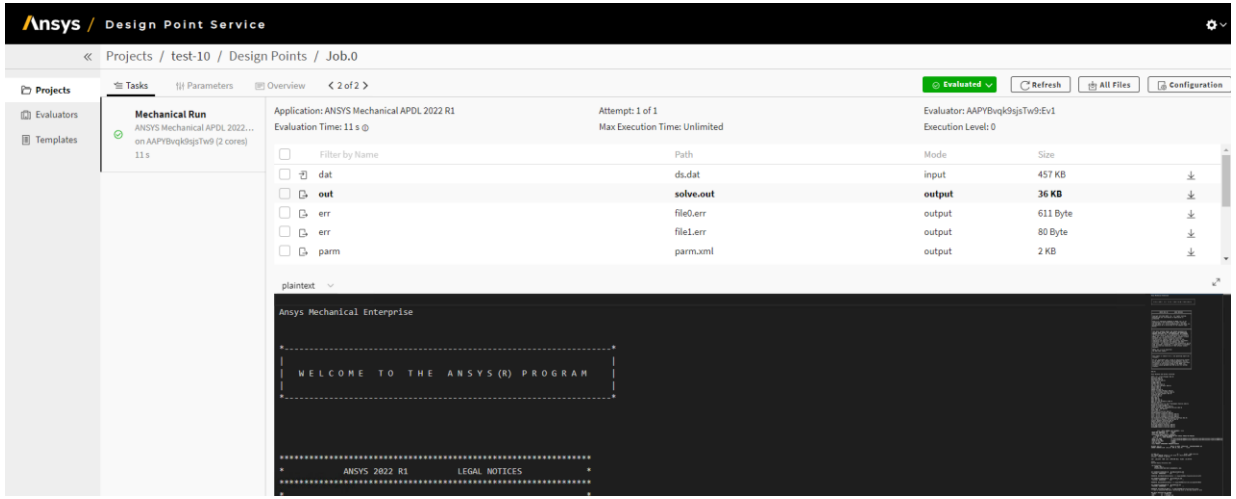


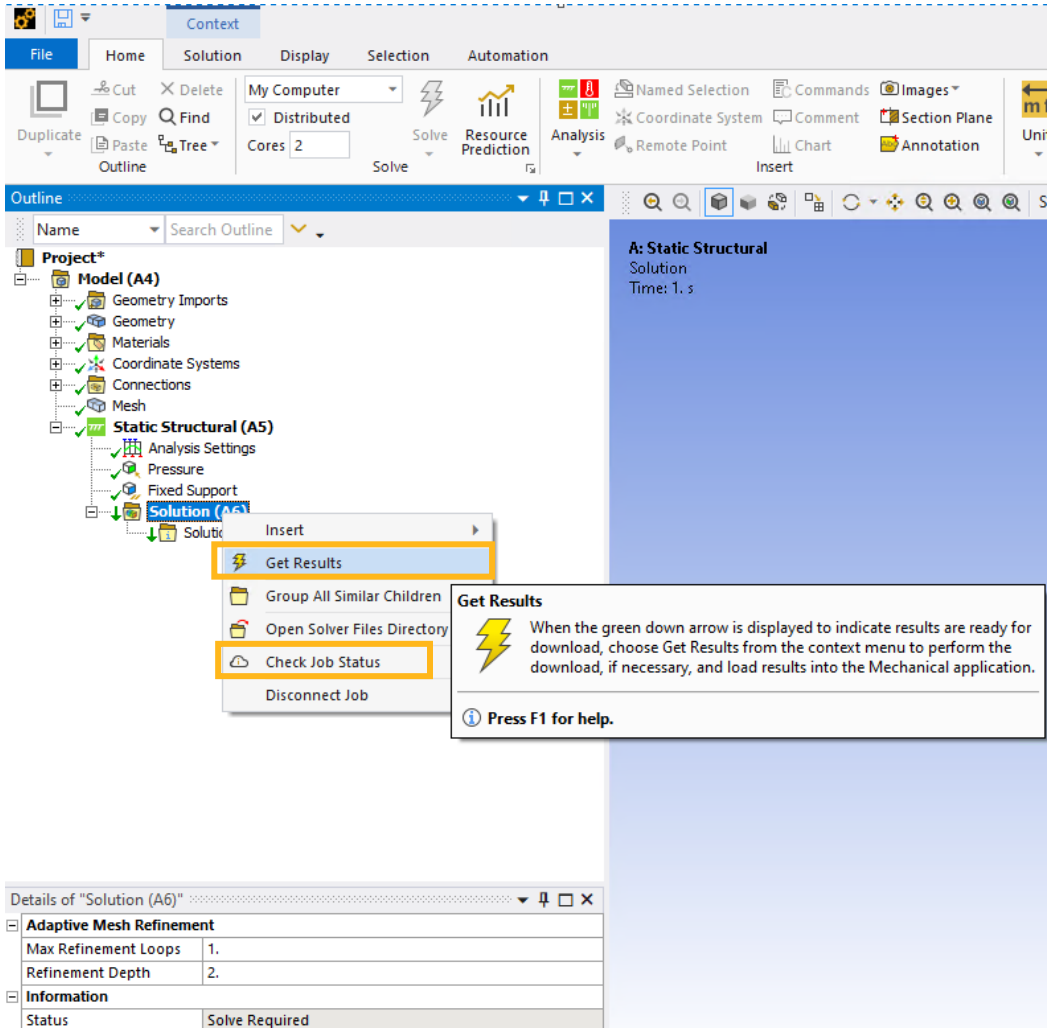
Core Capabilities of DCS (Distributed Compute Services):

- One or many design points of a Workbench project can be submitted to DPS and solved on an evaluator
- A solution update in Mechanical can be submitted to DPS and solved on an evaluator
- A Mechanical APDL input file can be uploaded directly in DPS and solved on an evaluator
- An LS-DYNA input file can be uploaded directly in DPS and solved on an evaluator
- optiSLang can be used to run optimizations and parameter and sensitivity studies on DPS projects
- Arbitrary clients can interact with DPS
- With the DCS Python API, a custom workflow of an arbitrary executable and a set of inputs can be uploaded to DPS and run on an evaluator

DCS (Distributed Compute Services)

- Under “Solver Process Settings”, add a queue and select the target as “DCS”
- To check the status of the job, right click on the solution and select “Check Job Status”
- Once evaluation is done, Mechanical is ready to get the results
- Also review job status on DCS Web UI





Workflow to Send APIP Data: Collaboration with RSM

- Previously when jobs were remotely launched through RSM for Mechanical, we were unable to send the APIP data generated in remote nodes to Ansys Analytics Server, because of no internet connection.
- From 2022 R1 the workflow is changed when launching jobs remotely using RSM:
  - APIP executable is launched by RSM after solver has finished running job.
  - APIP executable tries to send already sitting APIP data in HPC cluster. If it fails to send from HPC Cluster (eg: no internet connection), moves the files from app data to HPC staging directory.
  - RSM then moves the files from HPC staging directory to client's project scratch.
  - Files moved from client's solver directory to client's %appdata% by Mechanical.

Note: This is not a Mechanical feature. This enables better APIP data management for RSM workflows

Performance and Disk Space Improvements using /fclean

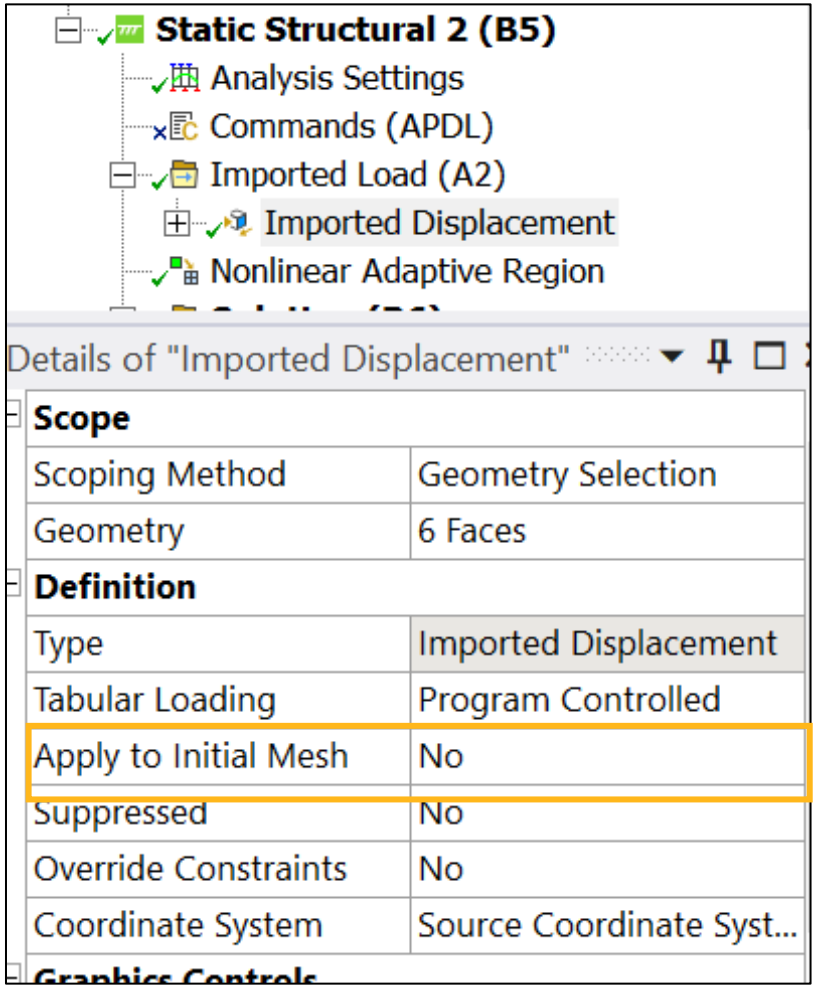
- Performance and disk space improvements by issuing /FCLEAN command which will delete the solution generated redundant distributed files in a distributed parallel processing run. This reduces the disk space to up to half. This gives additional benefit to the users by avoiding the file download when run on a remote machine using RSM
- The files are not cleaned using /FCLEAN command, when files are needed for distributed solution of downstream linked analysis, or when the solution does not run to completion for the existing distributed run

NLAD: Apply Loads to Initial Mesh

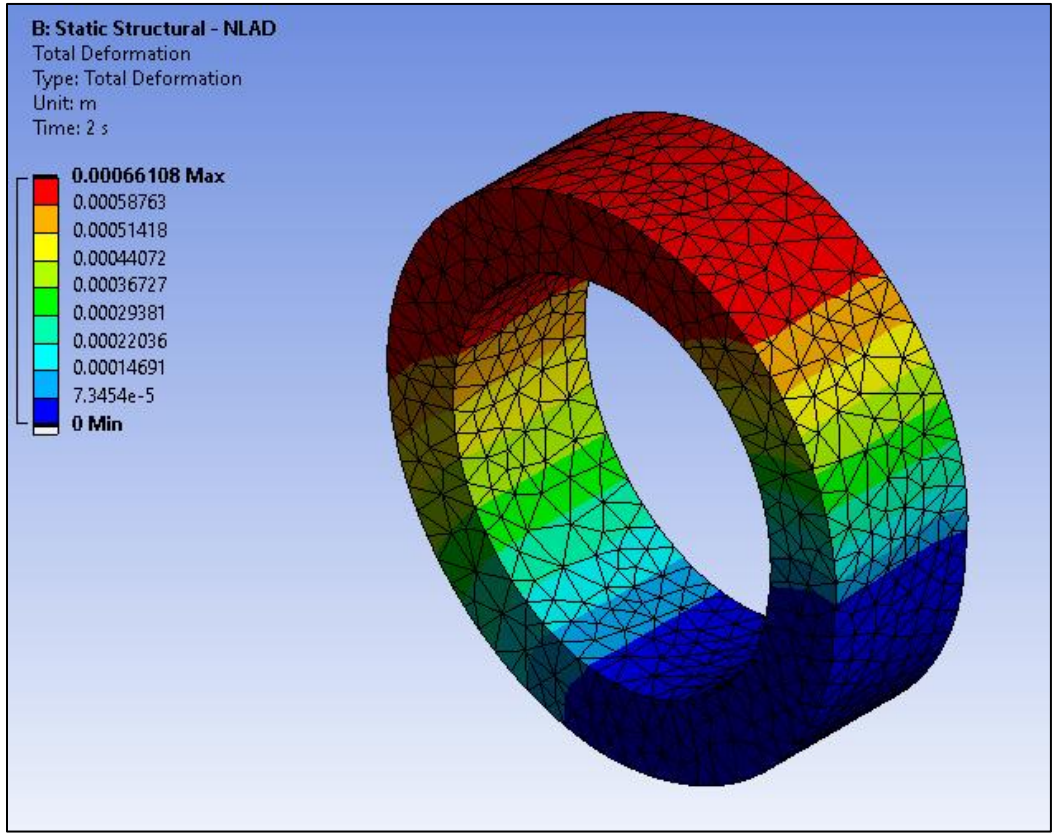
- In the presence of non-linear adaptive region, this property enables you to apply the imported loading and constraint directly on the initial mesh for each load step even when re-mesh occurs. This property enables you to map the loads to initial mesh

Supported Loads

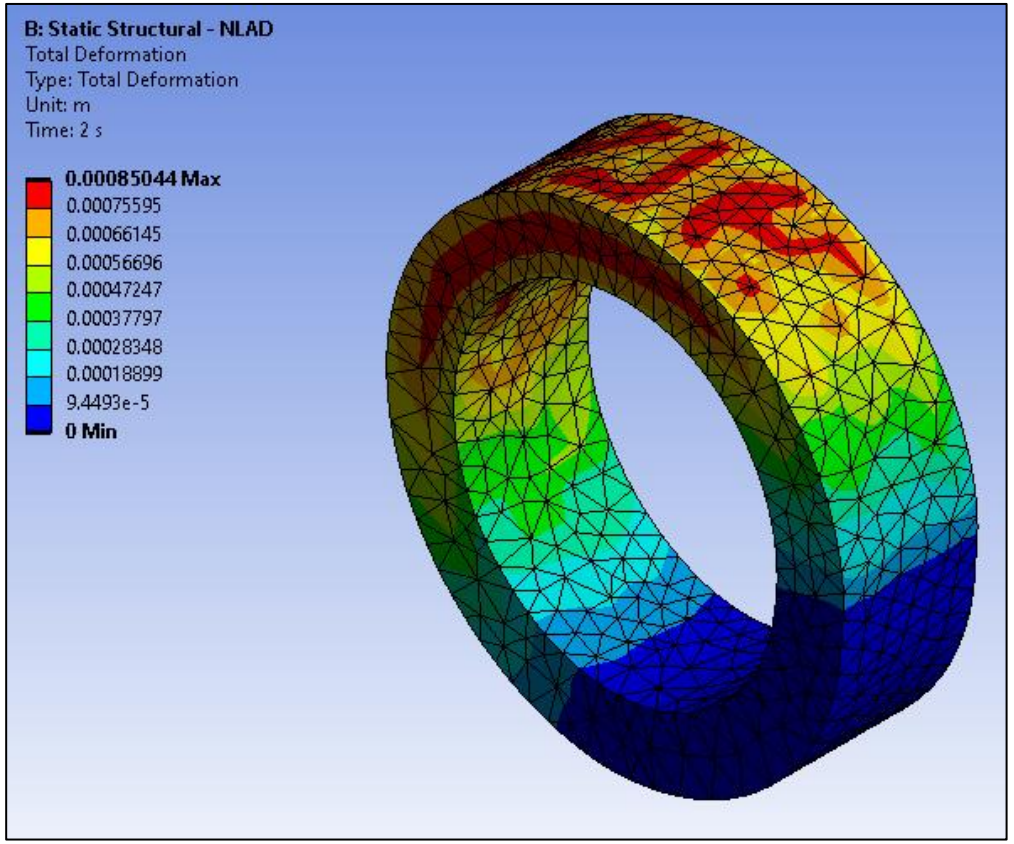
- Imported Displacement
- Imported Body temperature



NLAD: Apply Loads to Initial Mesh - Results



Apply To Initial Mesh set to “Yes”

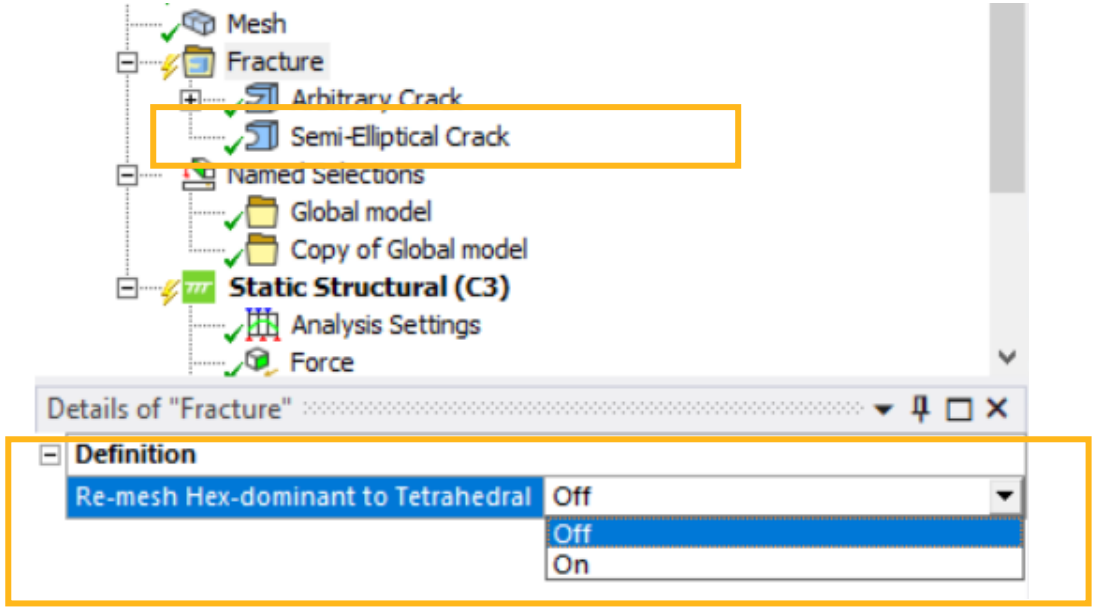


Apply To Initial Mesh set to “No”

Loads are mapped to initial mesh

Fracture Enhancements

- **Semi-Elliptical Crack** mesh generation on an imported base mesh
- We can automatically re-mesh an imported hex or hex-dominant base mesh as a tetrahedral mesh (as required by the **Semi-Elliptical Crack**) using the **Re-mesh Hex-dominant to Tetrahedral** property of **Fracture** folder

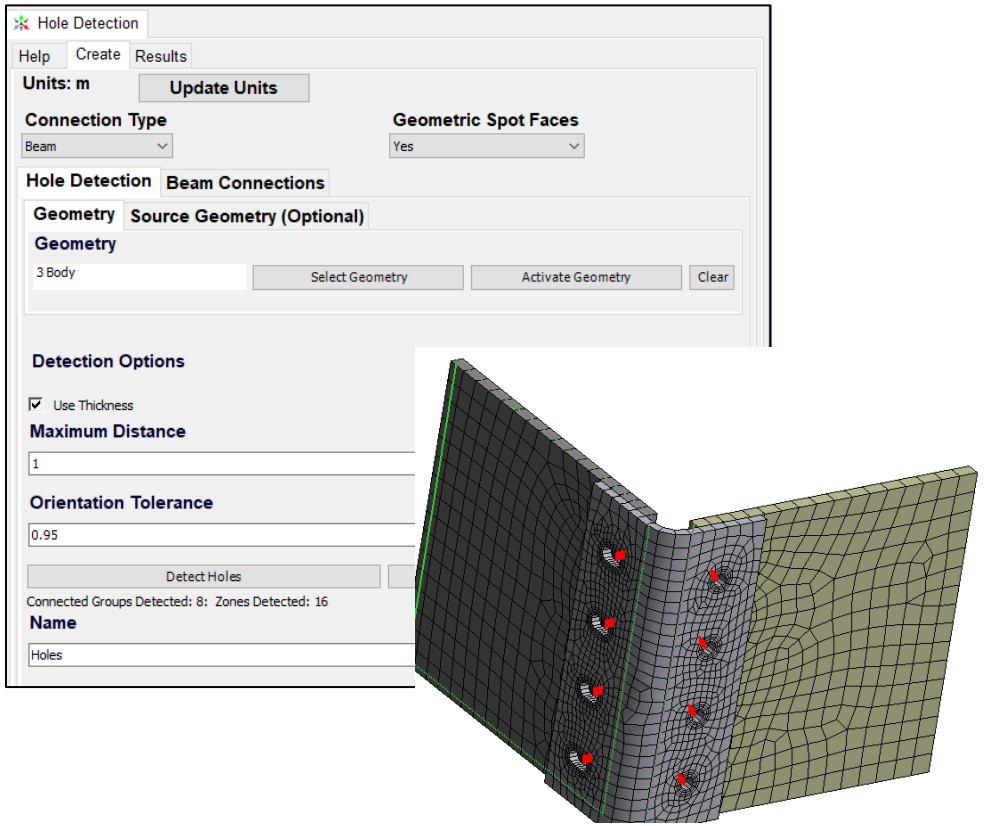


Bolt Tool

Bolted Joint Simulation

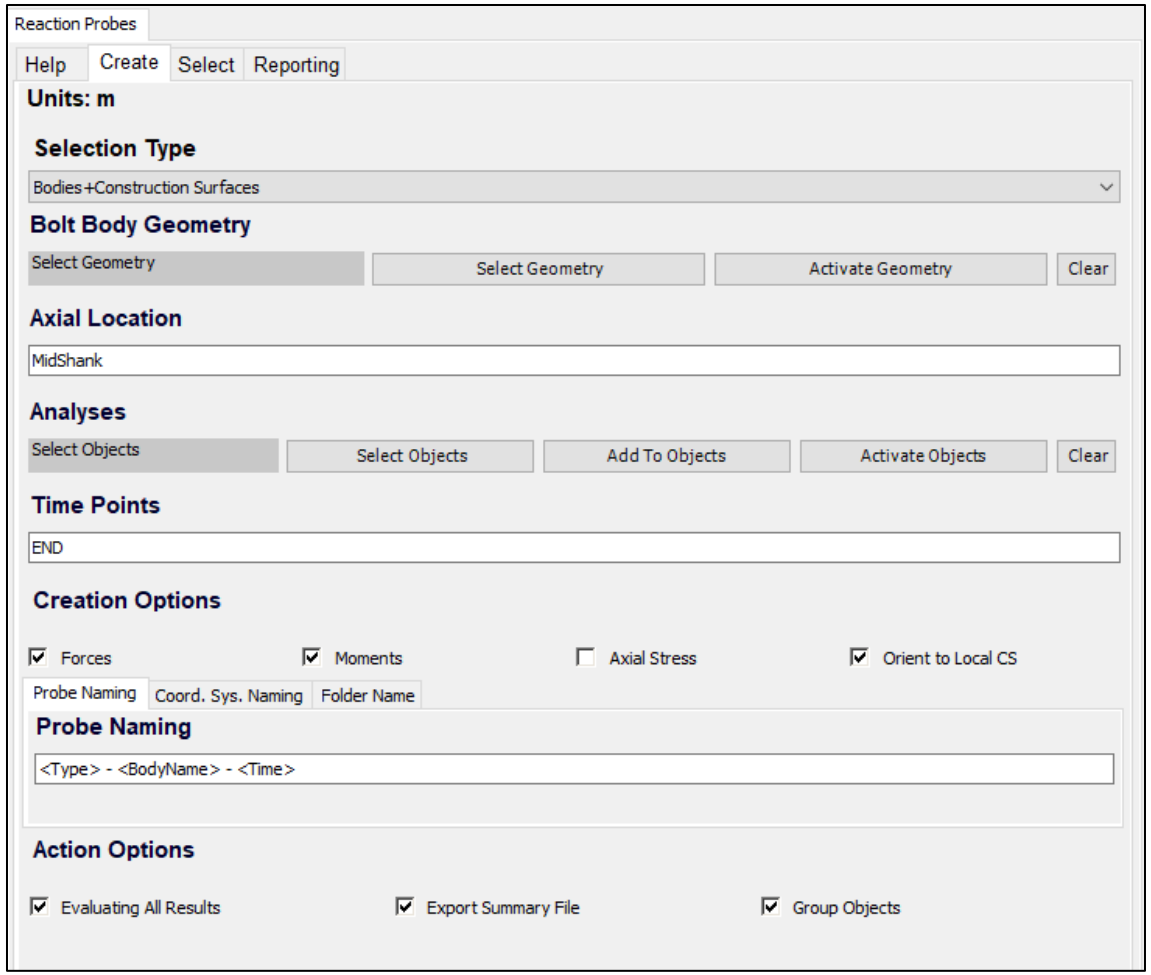
Hole Detection/Connection Creation (Surface Bodies)

- Auto hole detection
  - Surface bodies only
- Creation of connections automatically
  - Creates a “spot face” based on mesh inflation
  - Connections via beams or contacts
  - Create mesh controls around hole entities
- Reduces/Eliminates the need for geometric imprints, and tedious creation of multiple connections manually.
- More robust than Object Generator



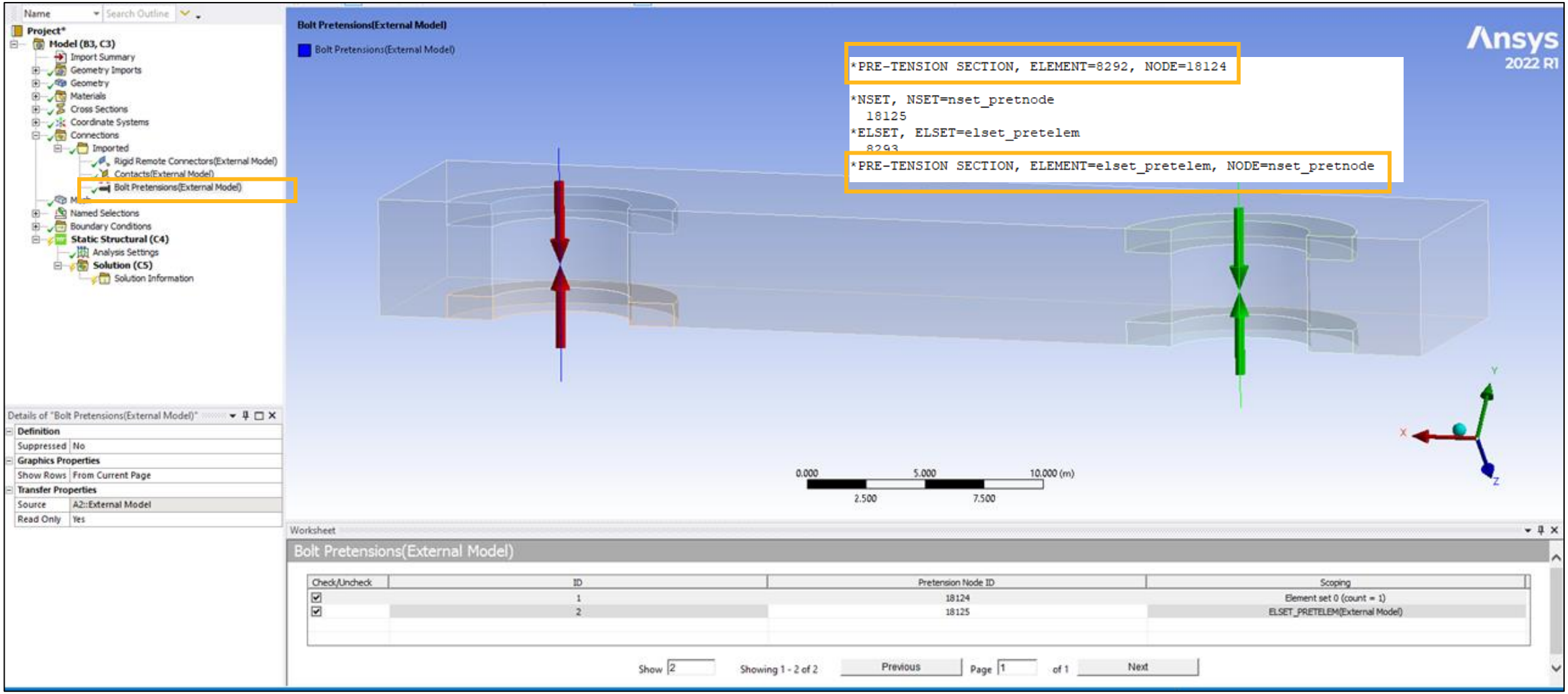
Reaction Probe Wizard

- Auto creation and export of reaction probes for multiple locations
- Allows users to easily export lots of force/moment data for connections to spreadsheet for further processing
- Auto creation of local CS for contextual results
- Multiple scoping/creation methods
  - Contacts
  - Construction surfaces
  - Geometry Faces



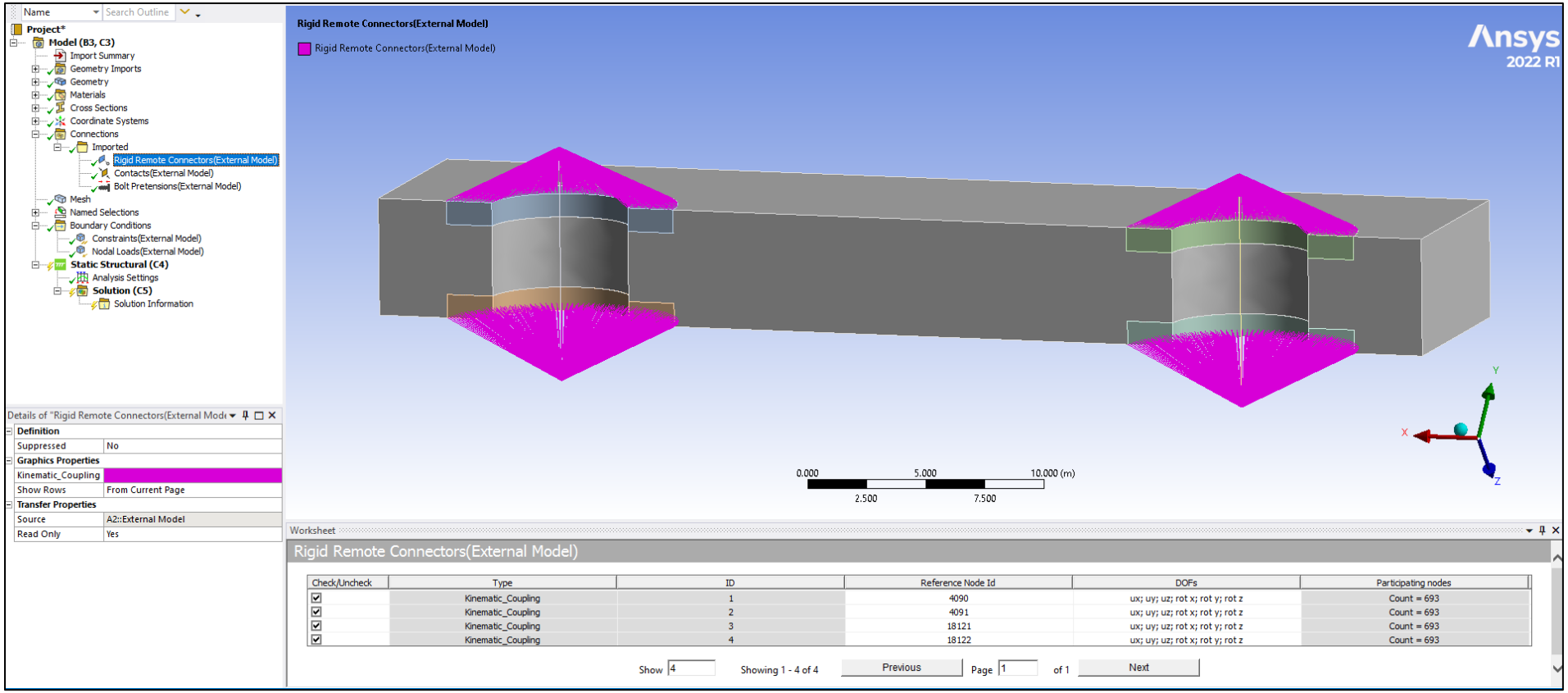
External Model

Abaqus: Support Bolt Pretensions on Beam Elements



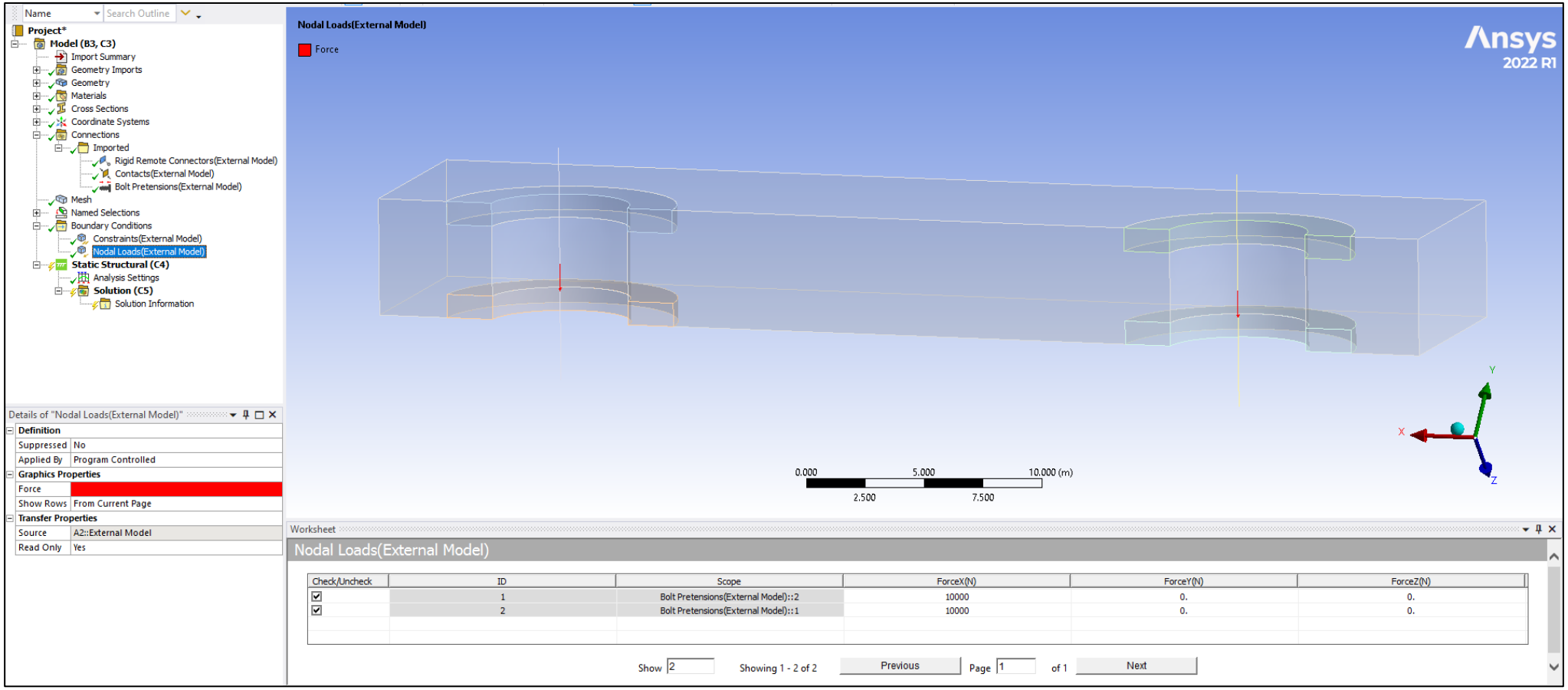
Abaqus: Support Bolt Pretensions on Beam Elements

- Connections can be applied to Beam nodes used for Bolt Pretension



Abaqus: Support Bolt Pretensions on Beam Elements

- Loads can be applied to the Beam Pretension nodes



Abaqus: Support *HYPERELASTIC Keyword for Materials

The following mutually exclusive parameters (material models) are supported:

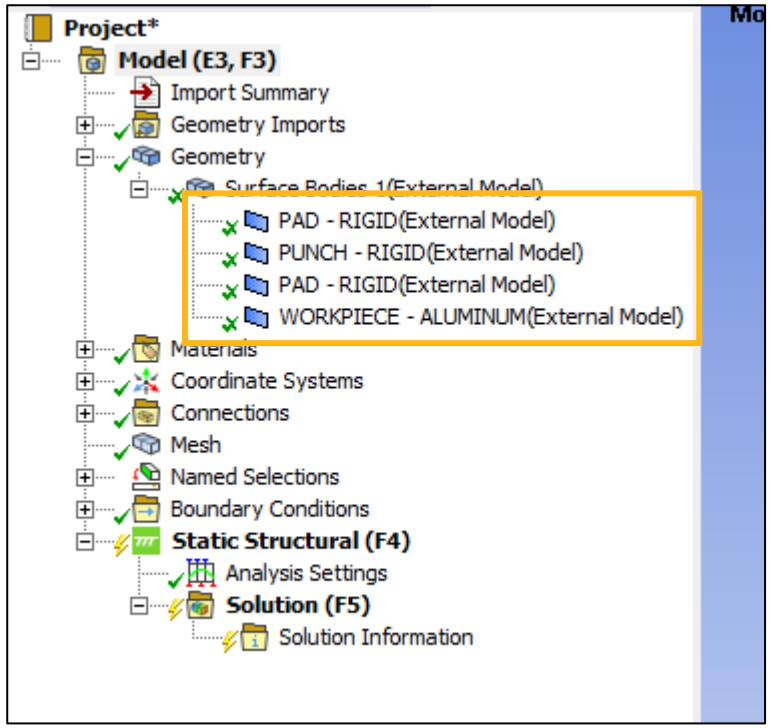
- MOONEY-RIVLIN
- NEO HOOKE: This parameter is equivalent to *HYPERELASTIC, REDUCED POLYNOMIAL and as a result, is imported as Yeoh 1st Order.
- OGDEN: Supports parameter N = 1 (default), 2, or 3.
- POLYNOMIAL (default): Supports parameter N = 1 (default), 2, or 3.
- REDUCED POLYNOMIAL: Supports parameter N = 1 (default), 2, or 3.
- YEOH

Limitation: Temperature dependence is not supported

LS-DYNA: Set Body Names Using *PART Names

- Imported Bodies are named using *PART names from the .k input files

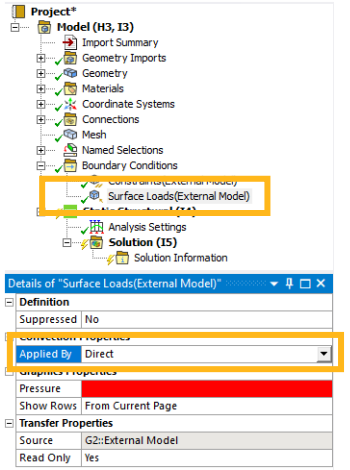
```
*PART
$ c
$ head
 PUNCH - RIGID
$       i       i       i       i       i       i
$     pid     sid     mid   eosid    hgid   adpopt
        1       2       2       0       0       0
*PART
$ c
$ head
 PAD - RIGID
$       i       i       i       i       i       i
$     pid     sid     mid   eosid    hgid   adpopt
        2       2       2       0       0       0
*PART
$ c
$ head
 WORKPIECE - ALUMINUM
$       i       i       i       i       i       i
$     pid     sid     mid   eosid    hgid   adpopt
        3       1       1       0       0       0
*PART
$ c
$ head
 PAD - RIGID
$       i       i       i       i       i       i
$     pid     sid     mid   eosid    hgid   adpopt
        4       2       2       0       0       0
```

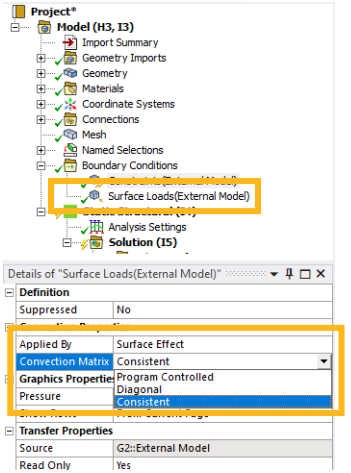


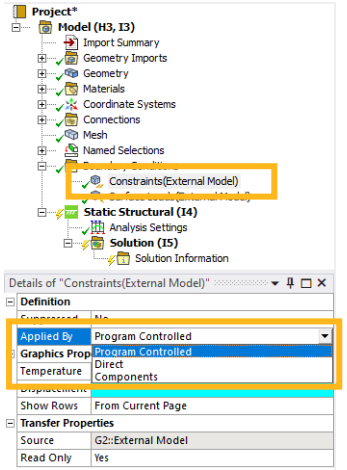
New Imported Boundary Conditions Options

Thermal convection

- Surface Loads object now has 2 new properties for "Applied By": Surface Effect or Direct
  - This specifies how to send the thermal convections to the solver file
  - "Surface Effect" has the options:
    - Convection Matrix: Program Controlled, Diagonal or Consistent
    - This specifies which film coefficient to use for the convection loads







Imported Nodal Loads and Constraints

- New property available:
- Definition → Applied By: Program Controlled, Direct or Components
  - This specifies how to send the nodal loads to the solver file

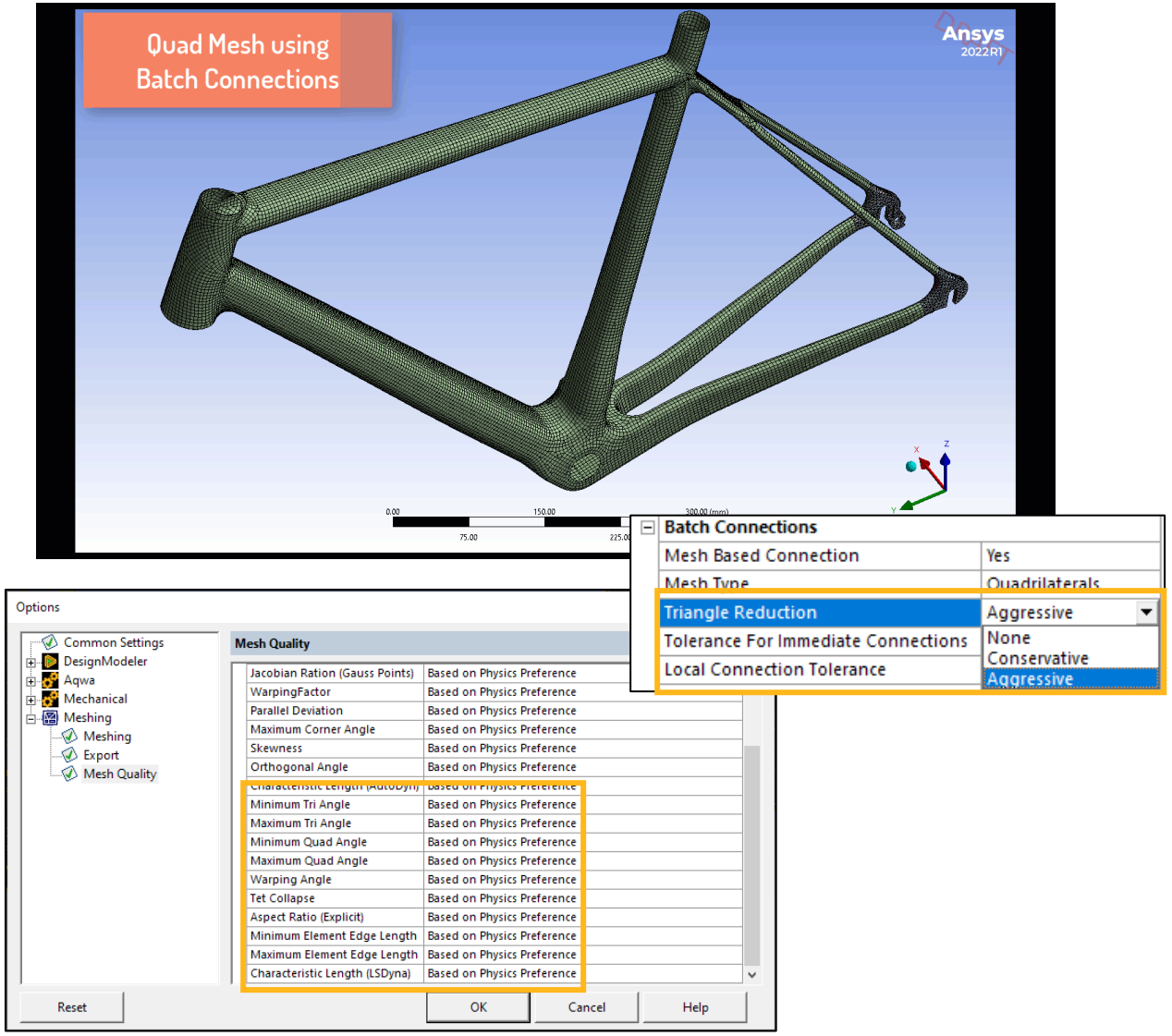
Meshing

New Features

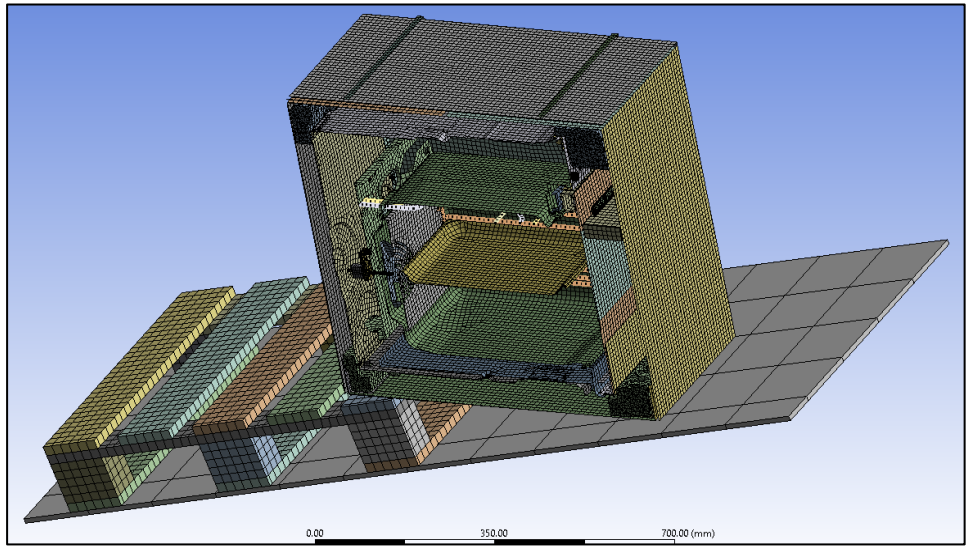
- **Shell Meshing**
  - Tri Reduction
  - Support for new metric visualization:
    - Warping angle
    - Min quad/tri angle
    - Max quad/tri angle
    - Min/Max Element Edge Length
- **Welds Meshing**
  - Enhancements to Weld Worksheet
  - Usability and error handling improvements
  - Visualisation 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

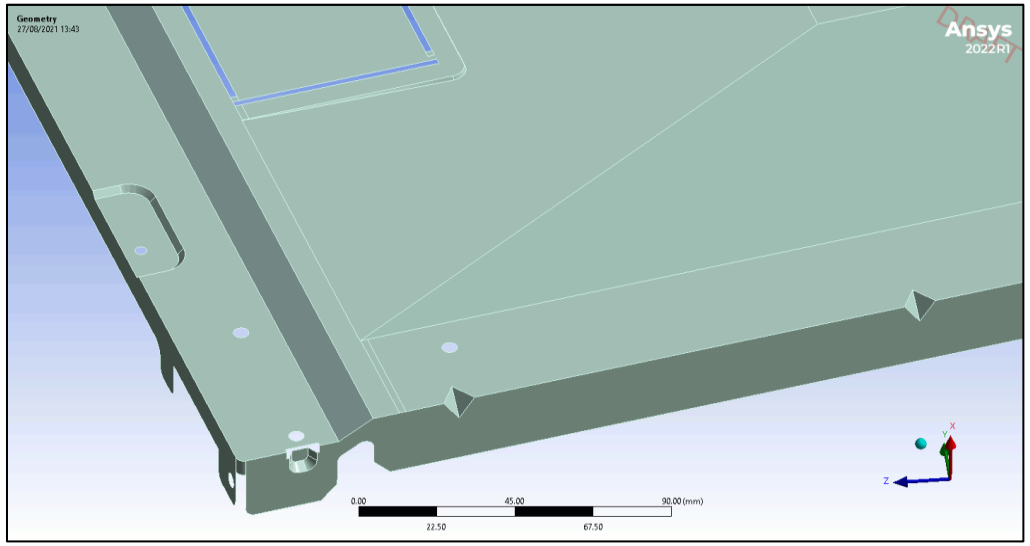
Shell Meshing: Triangle Reduction & Quality

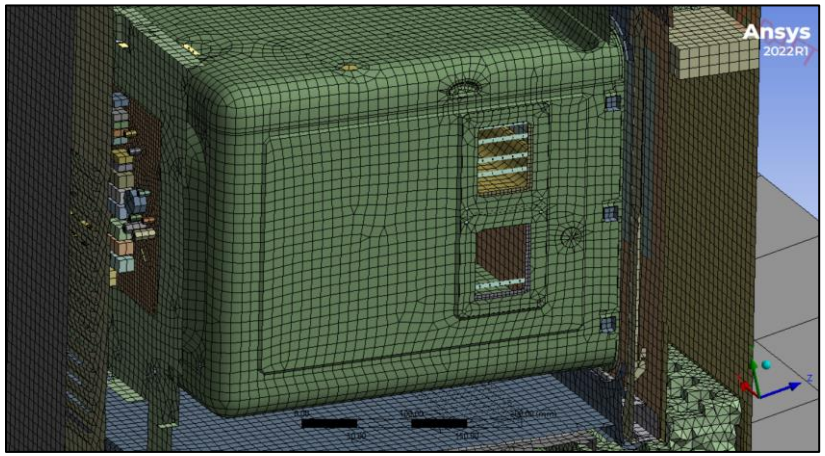
- Available with Batch Connections
- Option to control level of triangles in quad mesh
  - None
  - Conservative (default)
    - Remove triangles near shell edges
  - Aggressive
    - Remove as many triangles as possible sometimes at cost of quality
    - Seen to reduce tri count by up to 80% in some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

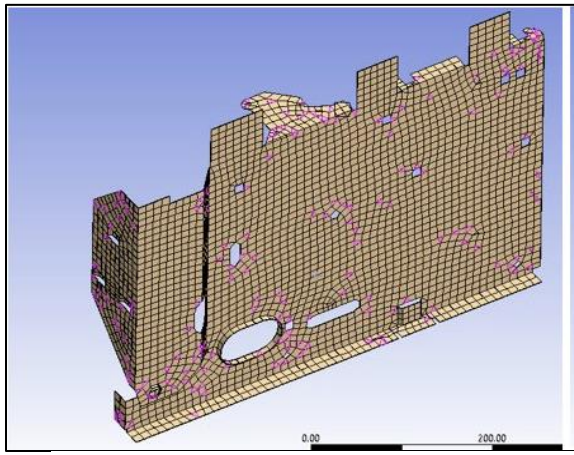


Steamer Shell Mesh: Tri Reduction

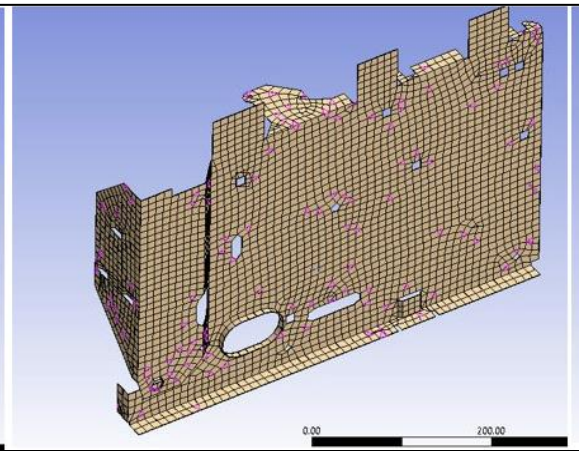




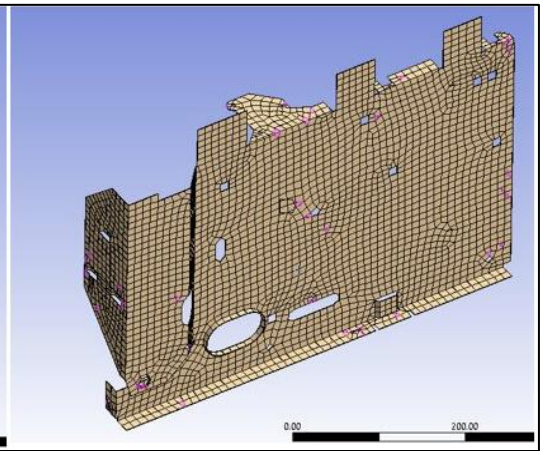




Tri Reduction = None
Number of tri = 184



Tri Reduction = Conservative
Number of tri = 126



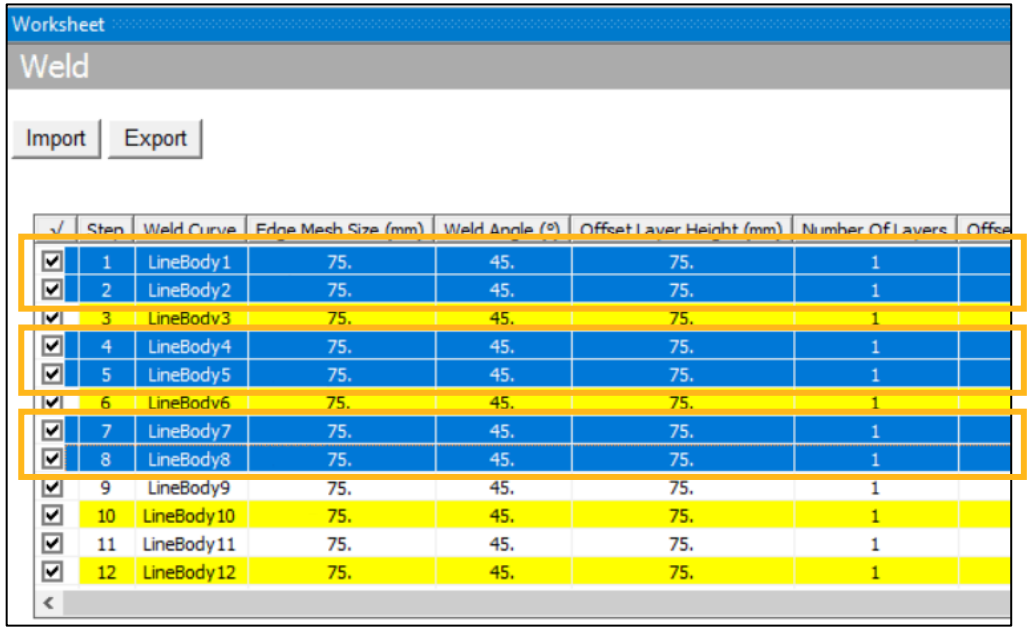
Tri Reduction = Aggressive
Number of tri = 48

Terminology Changes to Align with Industry

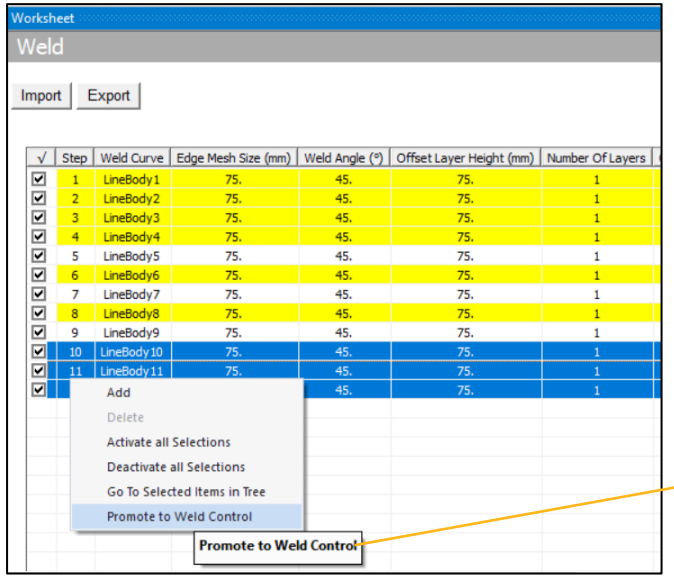
Old Terminology	New Terminology
Tent	Angled
Extension	Normal
Tent and Extension	Normal and Angled
Seam	Continuous Seam
Skip	Intermittent Seam
Create Offset Layer	Create HAZ Layer
Number of Layer	Number of HAZ [1,2,3, No]
Offset Layer Growth Rate	HAZ Growth Rate
Offset Layer Height	HAZ Distance
Weld Width	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

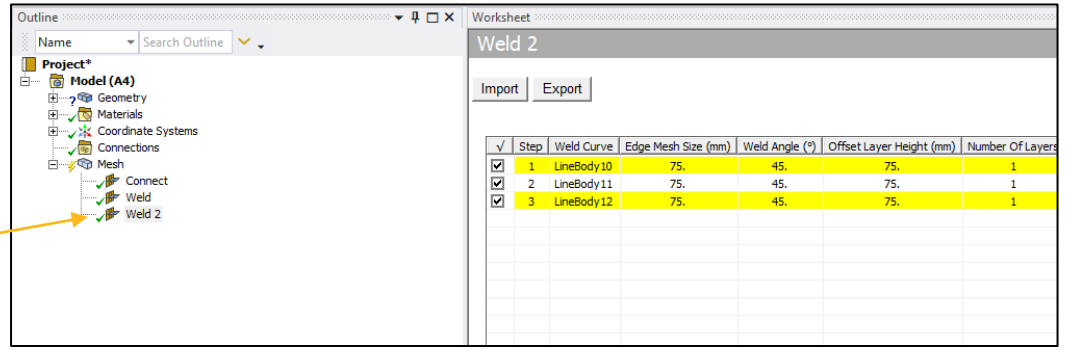
- Work with multiple rows from weld worksheet
  - Selection, Activation, Deactivation etc…
  - Go to selected items in tree
- Promote to weld control
  - creates new weld control with worksheet by removing selected rows from original.



√	Step	Weld Curve	Edge Mesh Size (mm)	Weld Angle (°)	Offset Layer Height (mm)	Number Of Layers	Offse
☑	1	LineBody1	75.	45.	75.	1	
☑	2	LineBody2	75.	45.	75.	1	
☑	3	LineBody3	75.	45.	75.	1	
☑	4	LineBody4	75.	45.	75.	1	
☑	5	LineBody5	75.	45.	75.	1	
☑	6	LineBody6	75.	45.	75.	1	
☑	7	LineBody7	75.	45.	75.	1	
☑	8	LineBody8	75.	45.	75.	1	
☑	9	LineBody9	75.	45.	75.	1	
☑	10	LineBody10	75.	45.	75.	1	
☑	11	LineBody11	75.	45.	75.	1	
☑	12	LineBody12	75.	45.	75.	1	



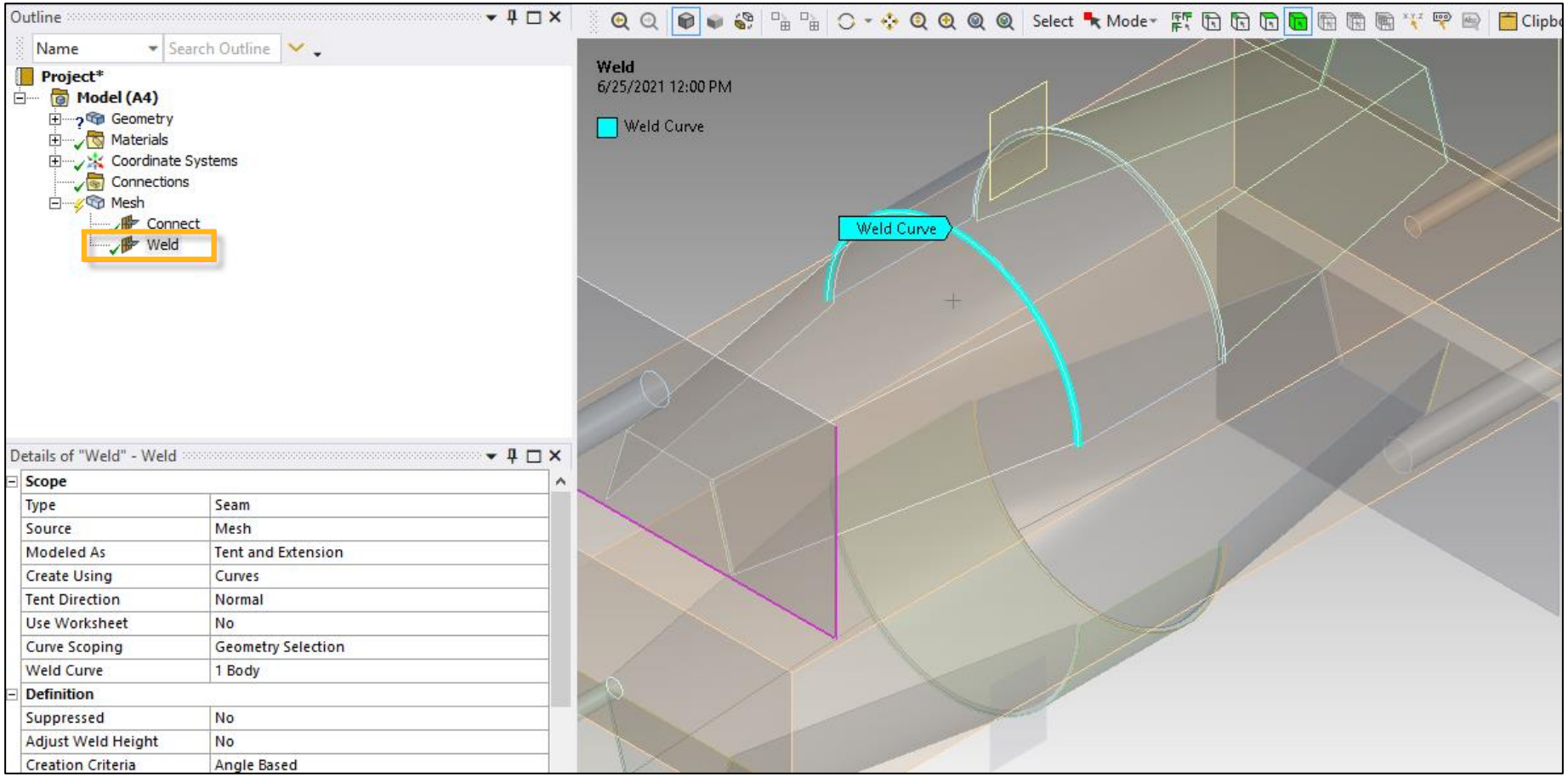
√	Step	Weld Curve	Edge Mesh Size (mm)	Weld Angle (°)	Offset Layer Height (mm)	Number Of Layers
☑	1	LineBody1	75.	45.	75.	1
☑	2	LineBody2	75.	45.	75.	1
☑	3	LineBody3	75.	45.	75.	1
☑	4	LineBody4	75.	45.	75.	1
☑	5	LineBody5	75.	45.	75.	1
☑	6	LineBody6	75.	45.	75.	1
☑	7	LineBody7	75.	45.	75.	1
☑	8	LineBody8	75.	45.	75.	1
☑	9	LineBody9	75.	45.	75.	1
☑	10	LineBody10	75.	45.	75.	1
☑	11	LineBody11	75.	45.	75.	1
☑	12	LineBody12	75.	45.	75.	1



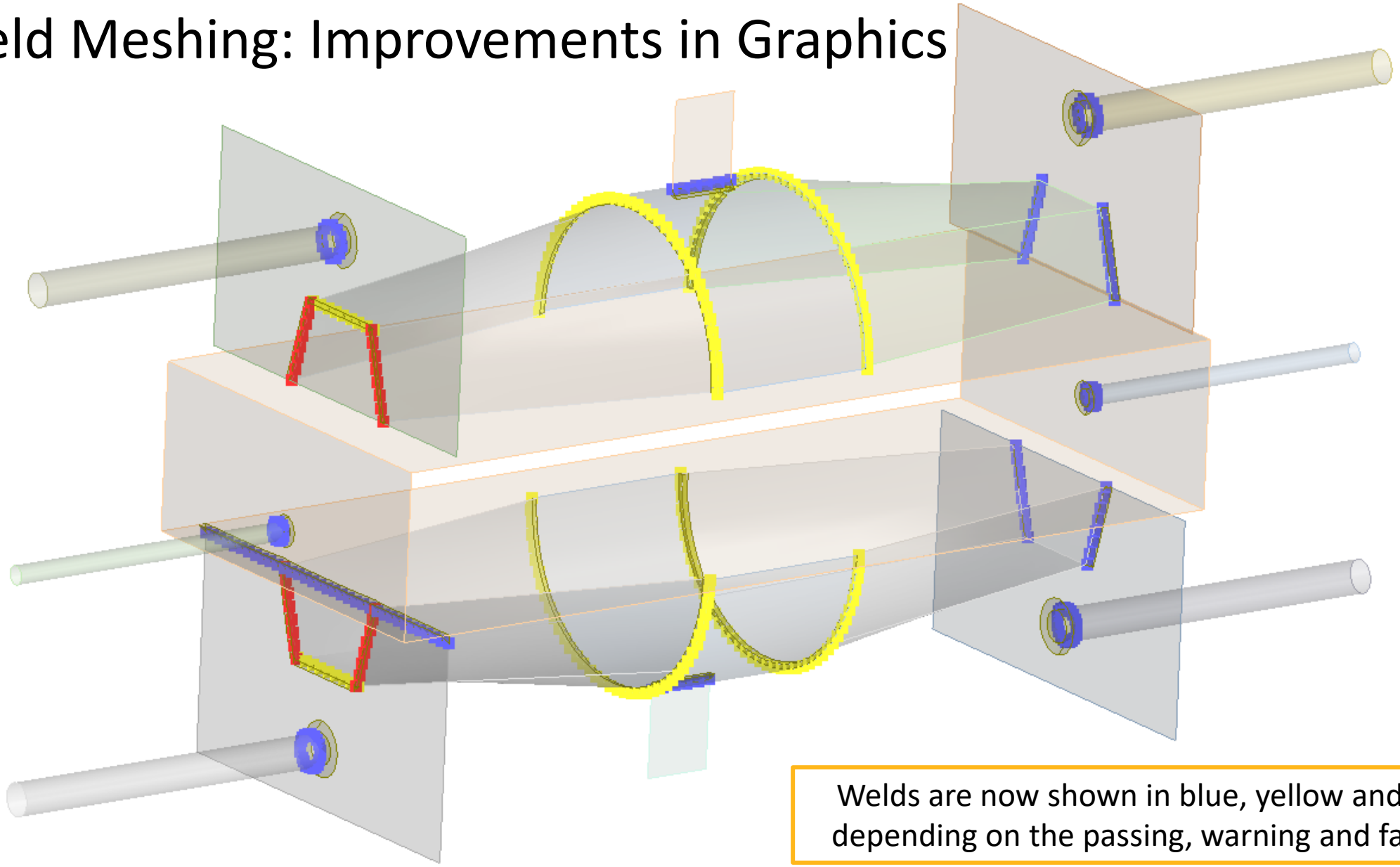
√	Step	Weld Curve	Edge Mesh Size (mm)	Weld Angle (°)	Offset Layer Height (mm)	Number Of Layers
☑	1	LineBody10	75.	45.	75.	1
☑	2	LineBody11	75.	45.	75.	1
☑	3	LineBody12	75.	45.	75.	1

Highlighting Weld Curve with Annotations

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



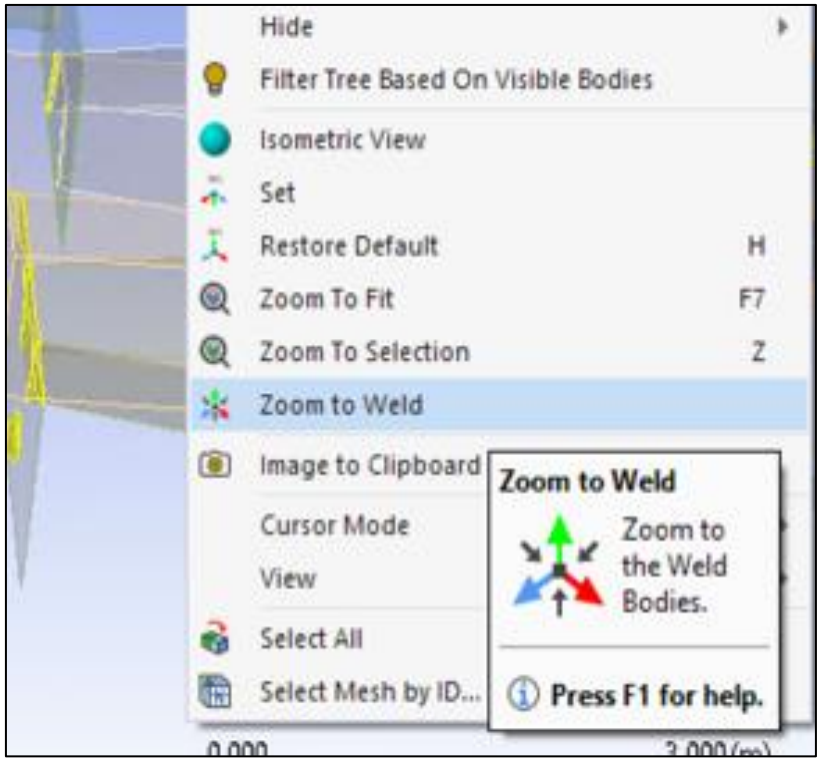
Weld Meshing: Improvements in Graphics

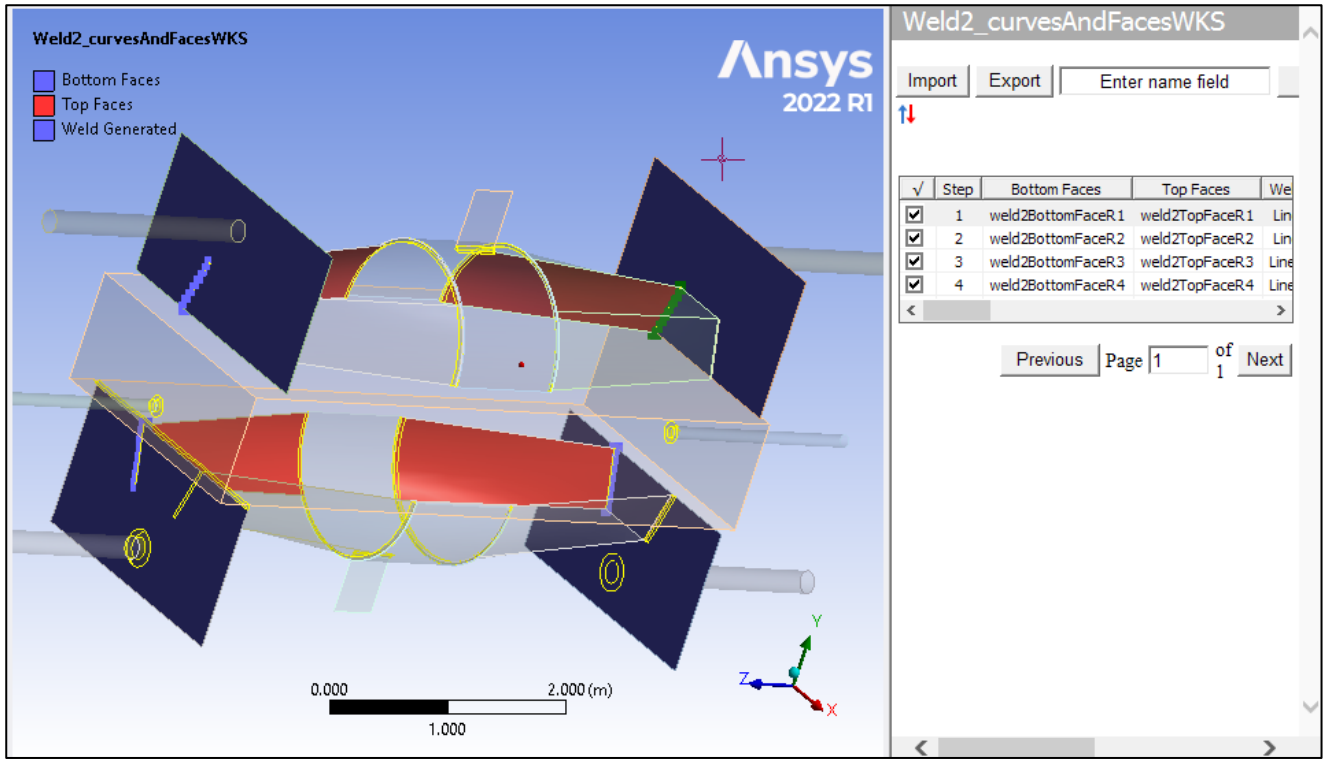


Welds are now shown in blue, yellow and red colors depending on the passing, warning and failing status

Weld Meshing: Improvements in Graphics

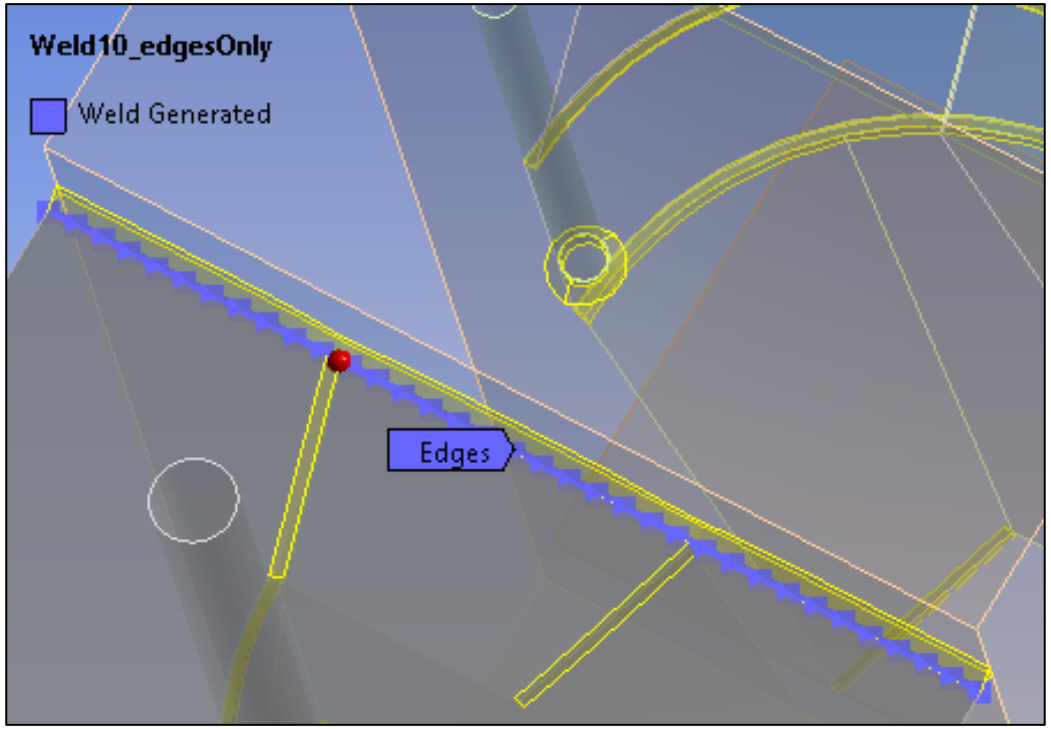
- “Zoom to Weld” option is added in the graphics context menu which zooms to fit the scoped references of the weld component. For the object using worksheet “Zoom to Weld” will be applied to the selected items in the worksheet
- For interaction with the worksheet, Picking and Highlight of the weld components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

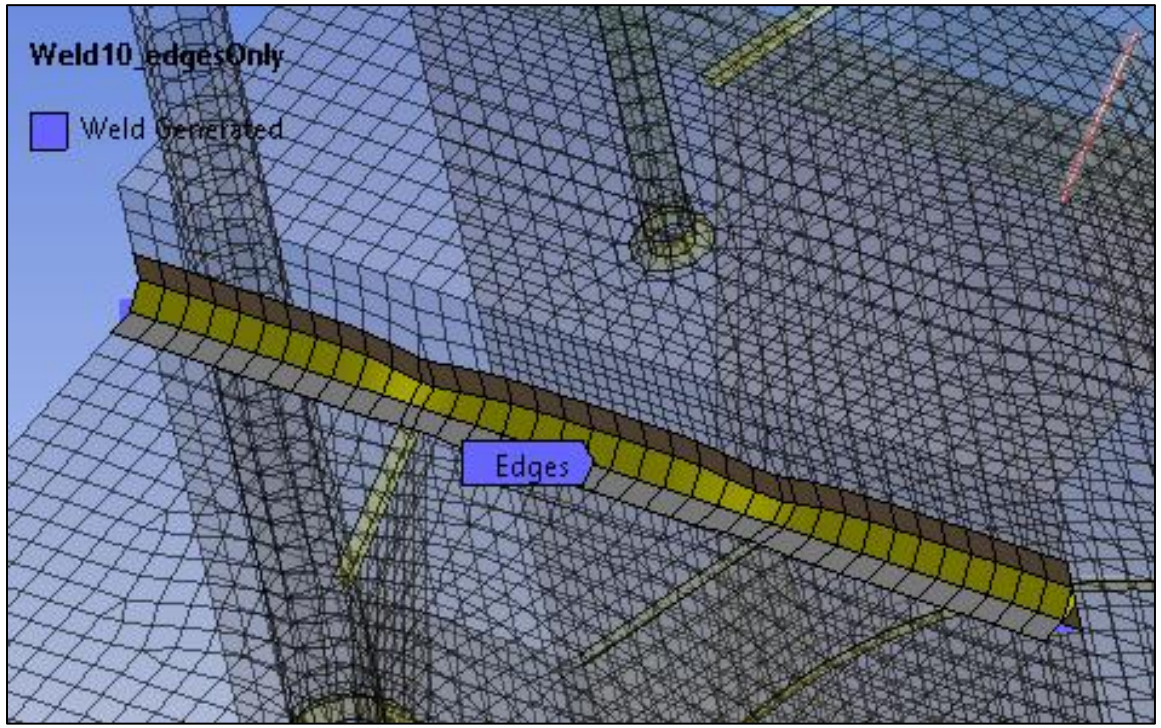




Weld Meshing: Improvements in Graphics

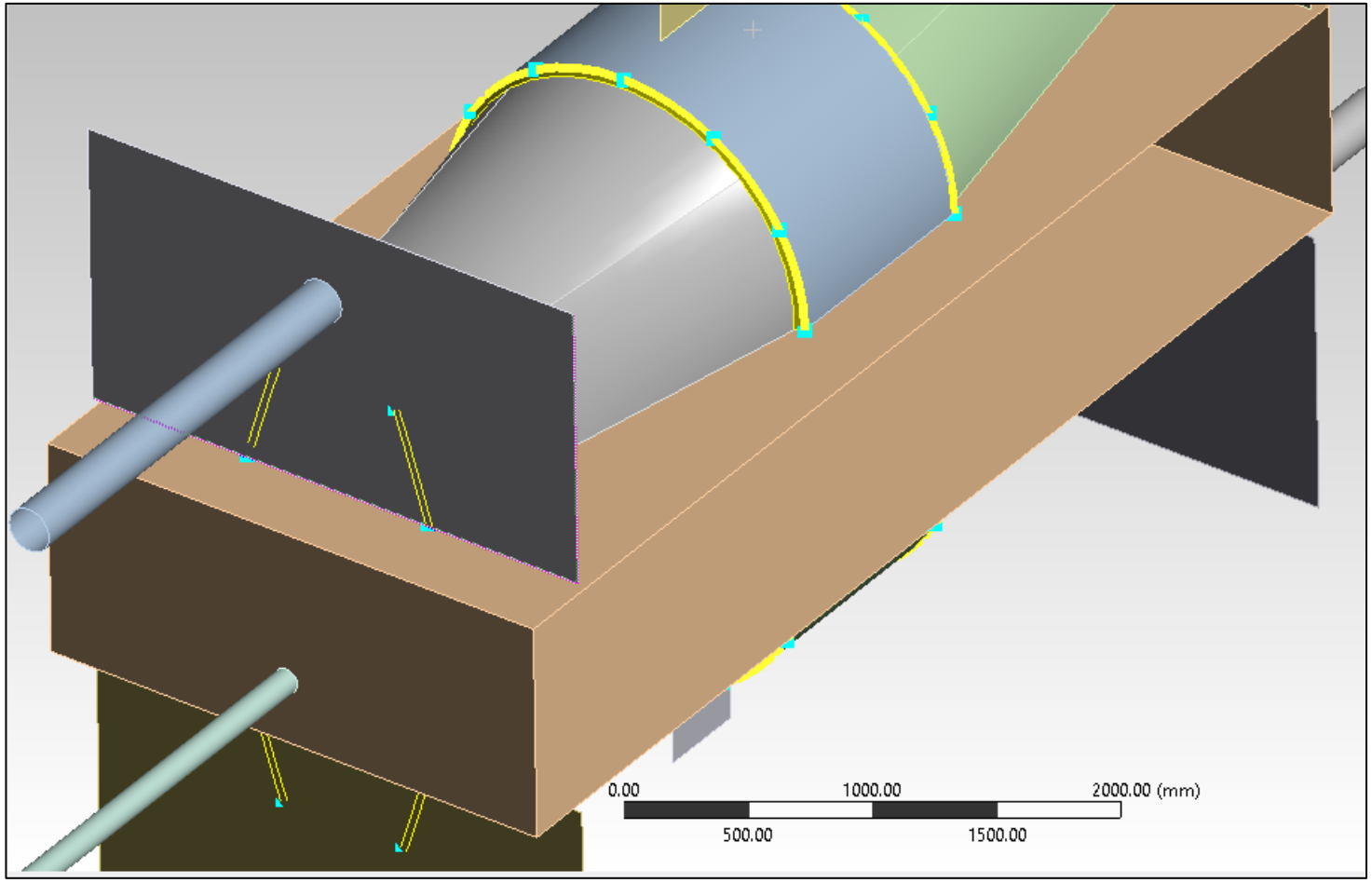
- The geometry references for different weld types scoping are highlighted based on the Mesh States. Dots along the weld edges/curves is displayed which represent the edge mesh size
- Turn on “Show Mesh” displays the Weld and HAZ layer elements as solid and other elements as translucent





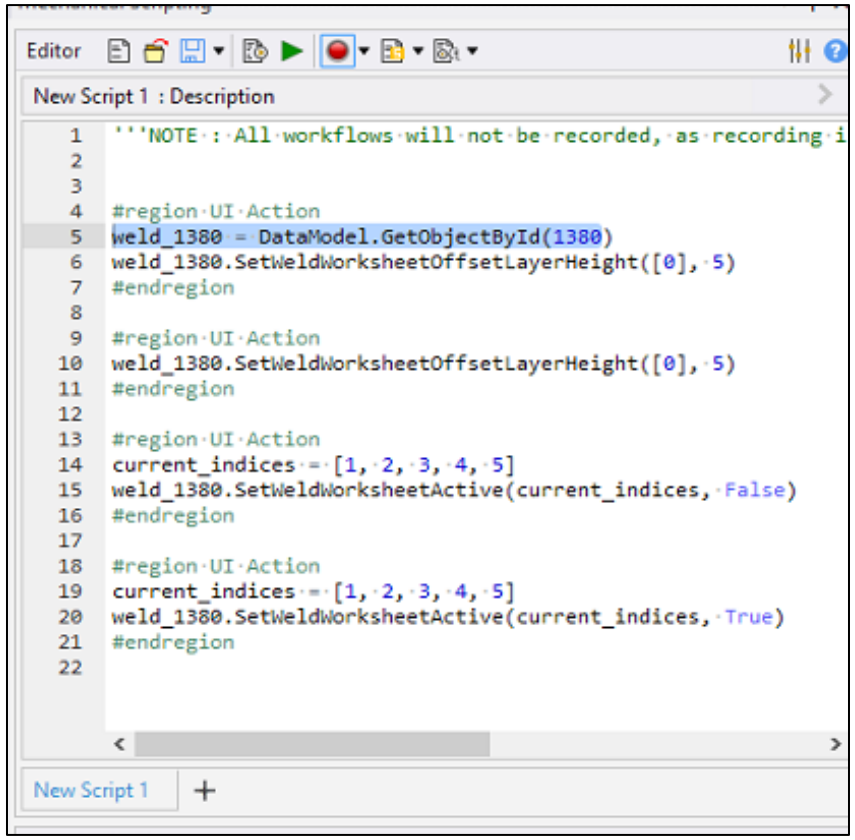
Preview Weld

- 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



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)
```

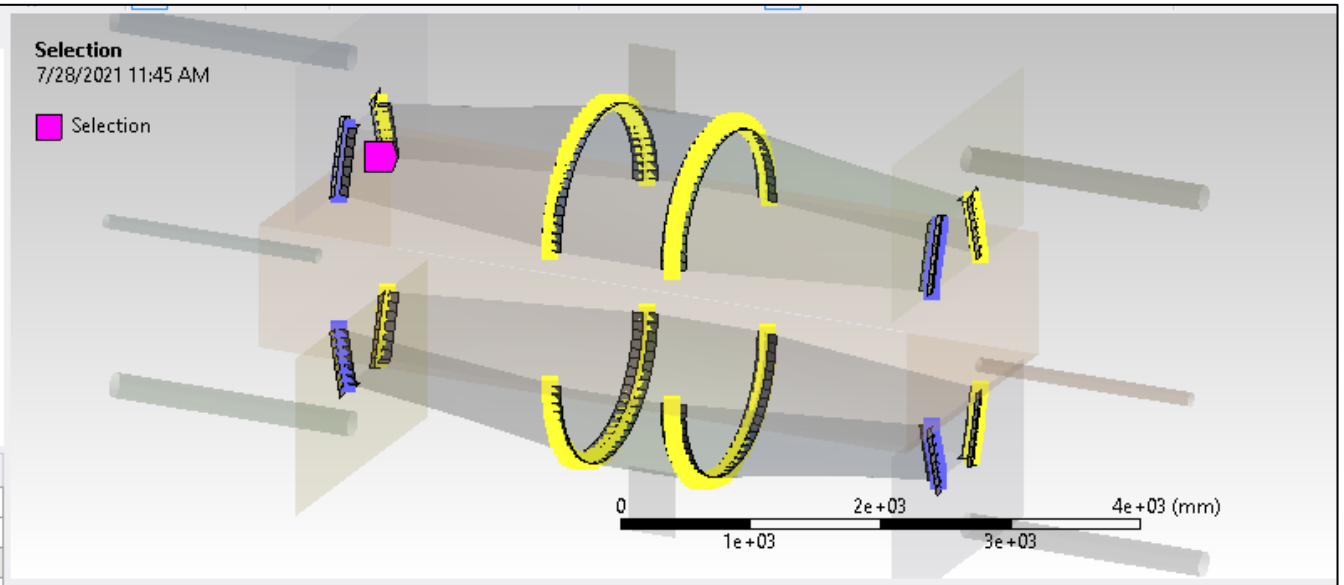
Weld Meshing: Criteria Based HAZ Named Selection

Name | Search Outline

- 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
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

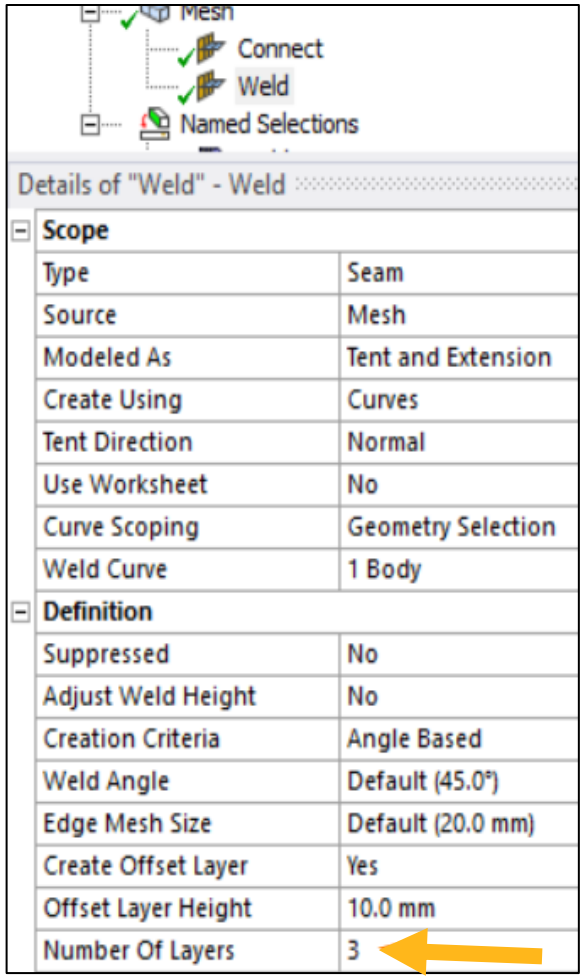
Selection

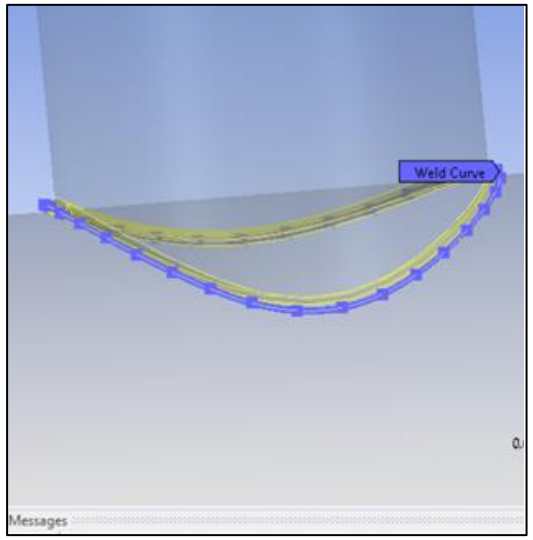
Generate

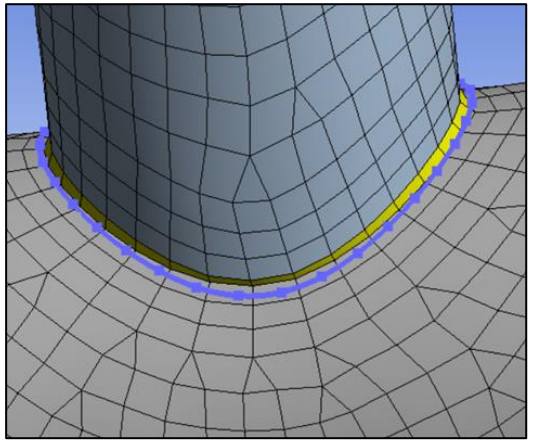
Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

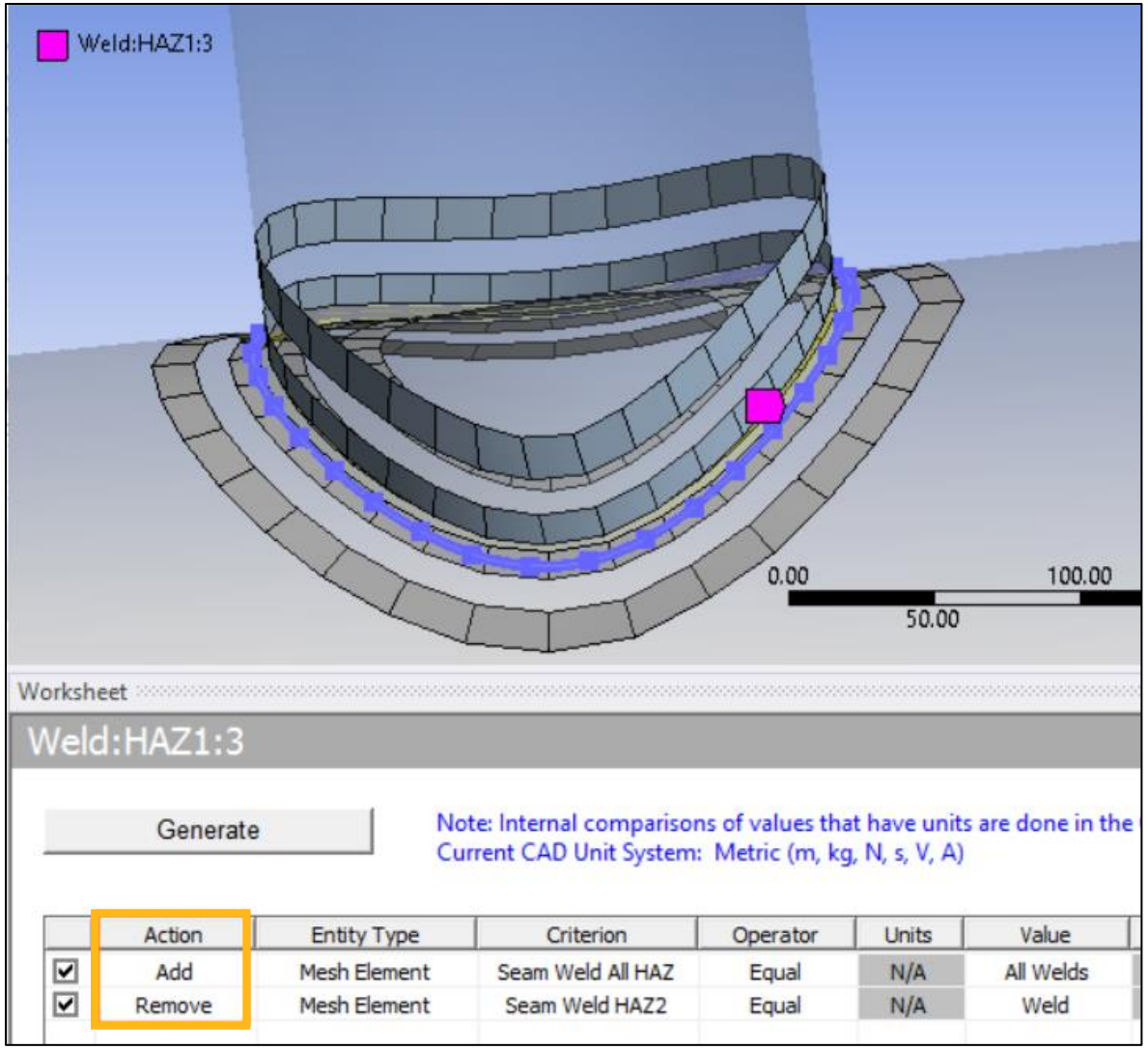
	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound
☑	Add	Mesh Element	Seam Weld HAZ1	Equal	N/A	All Welds	N/A

Weld Meshing: Criteria Based HAZ Named Selection

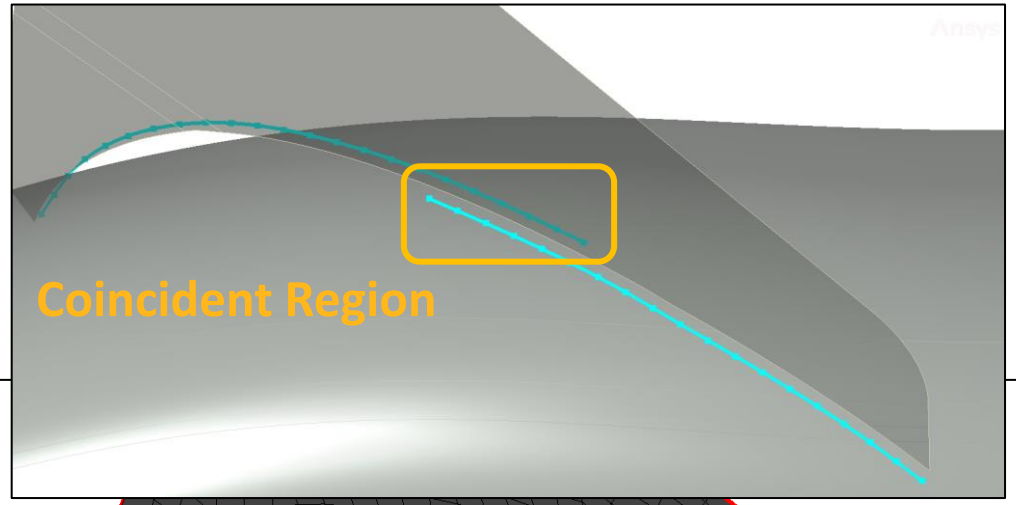


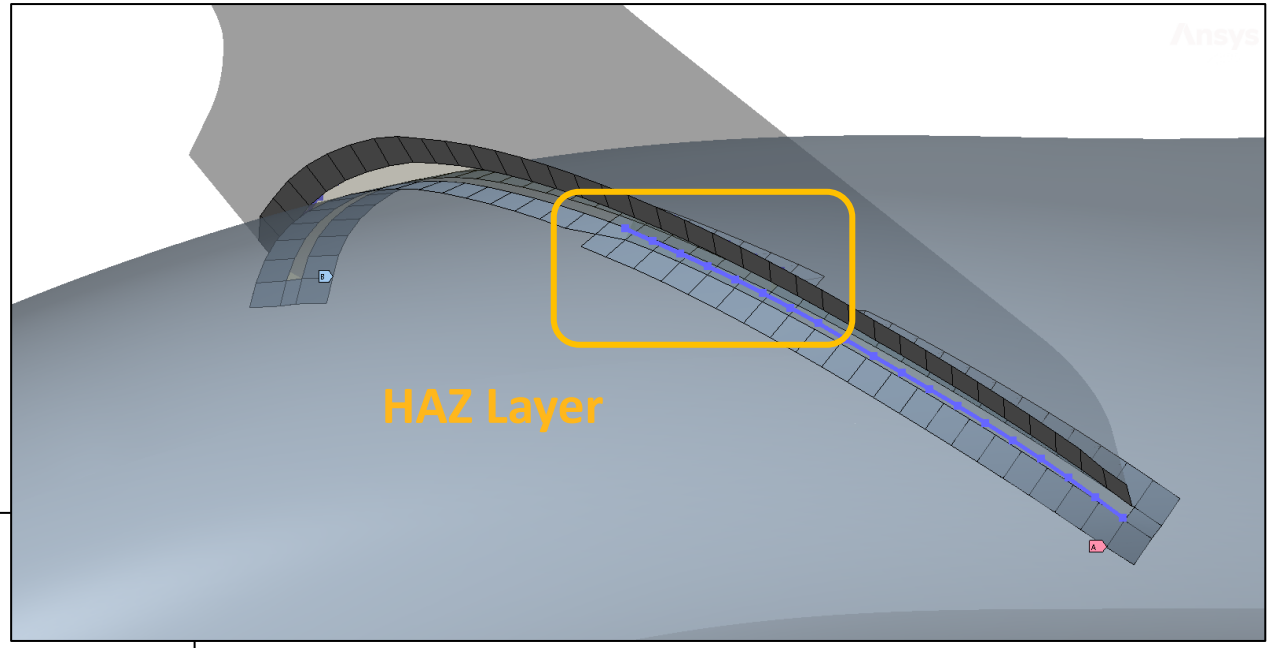


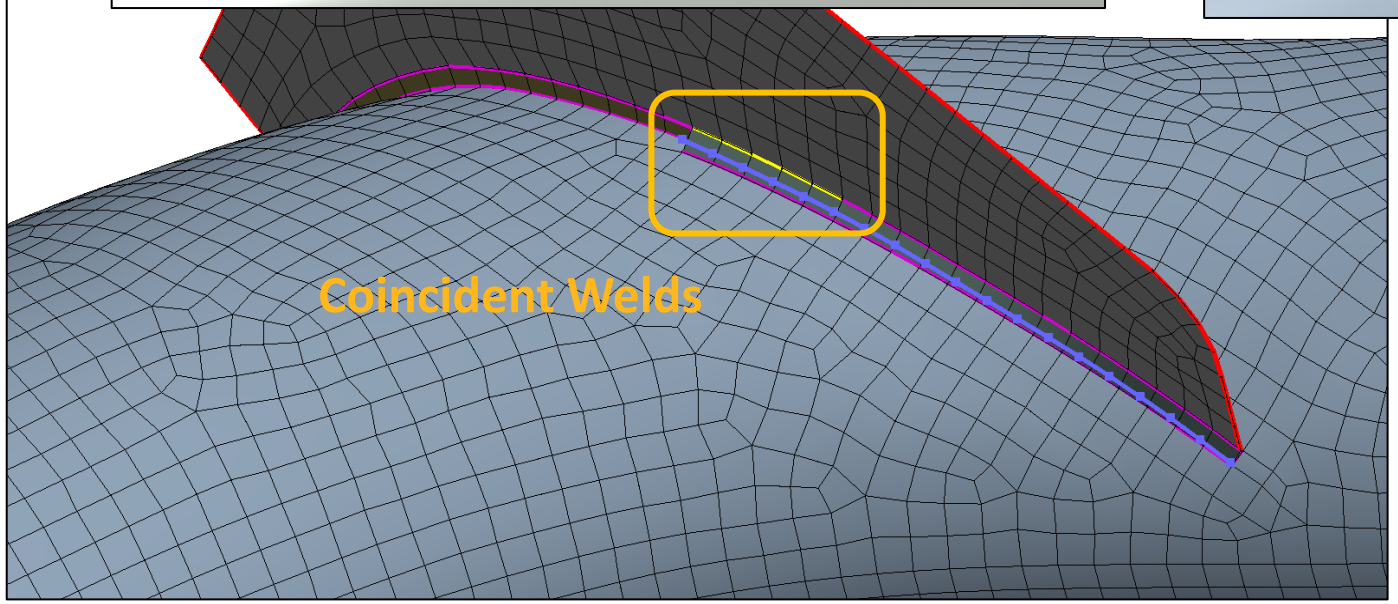


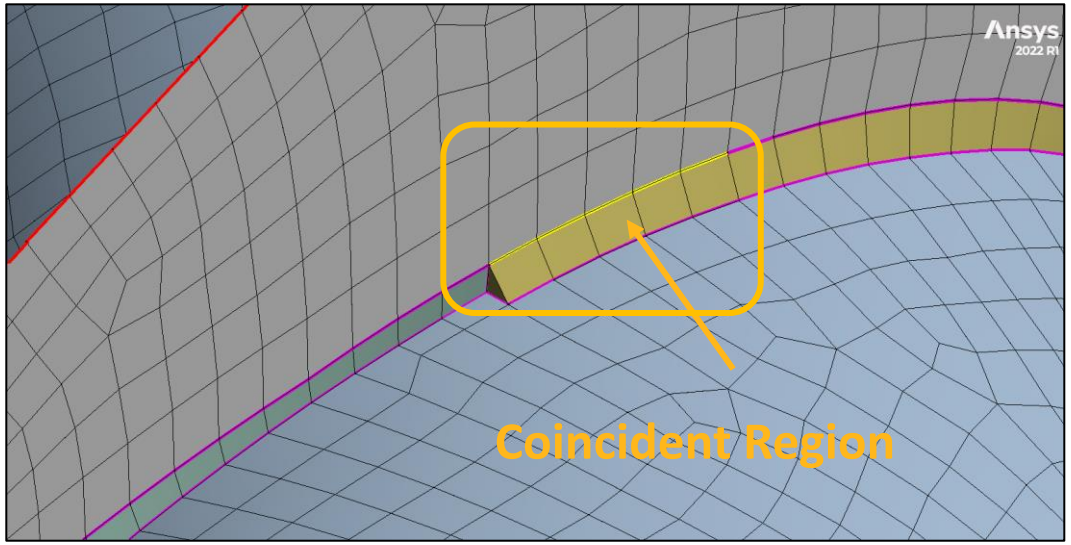


Coincident Welds



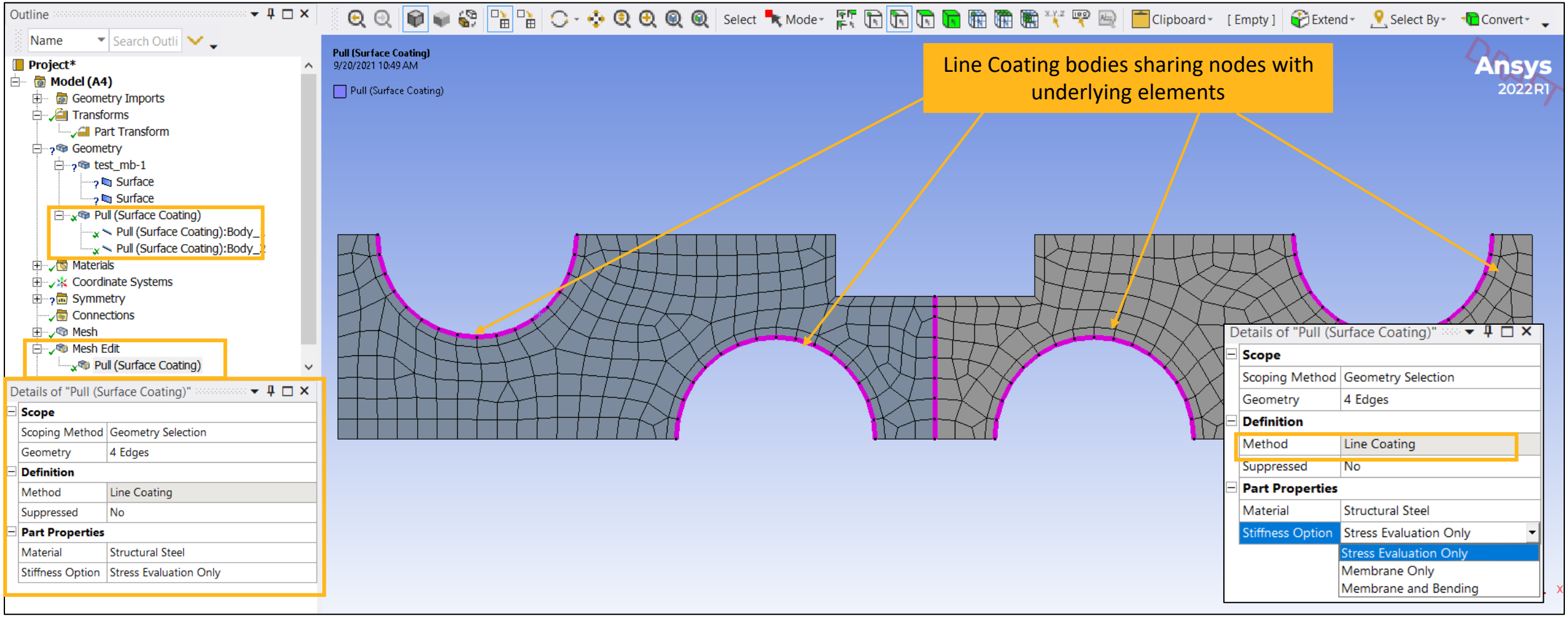






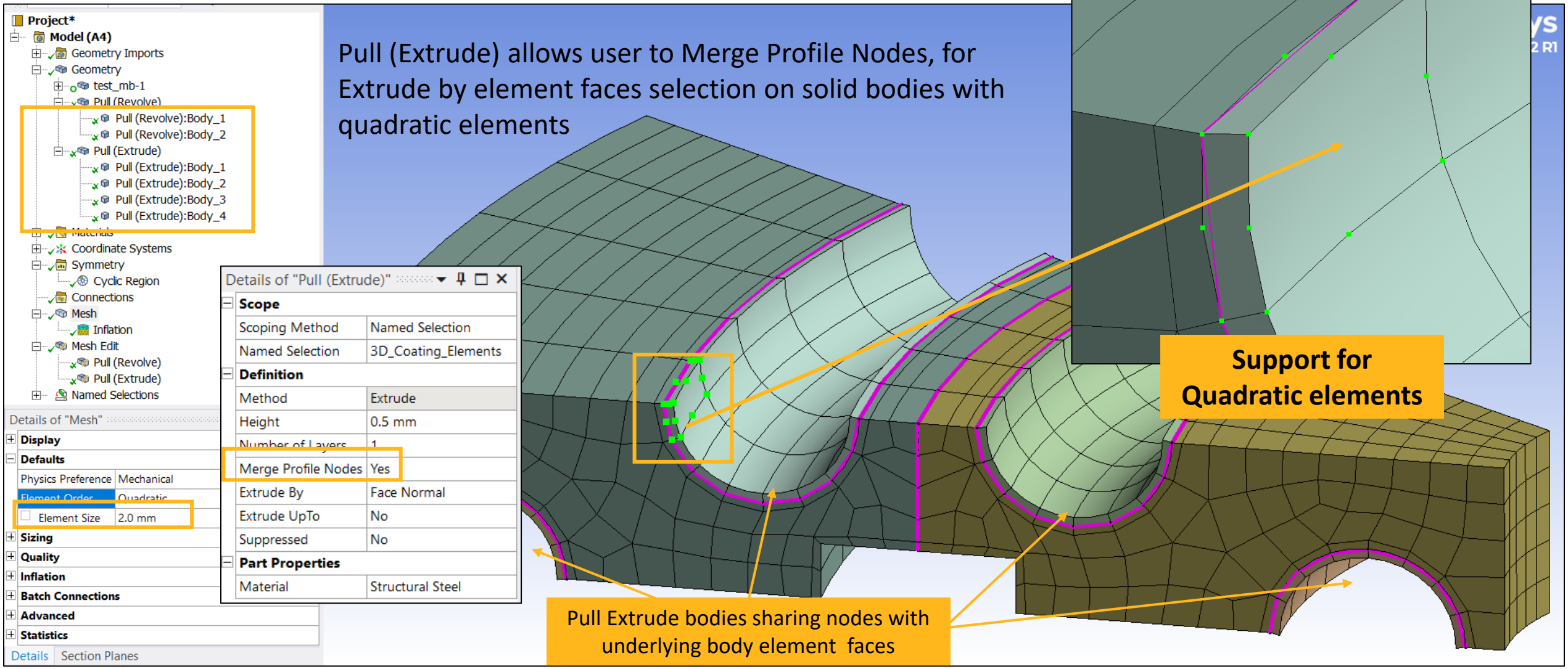
Pull (Line Coating)

- 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



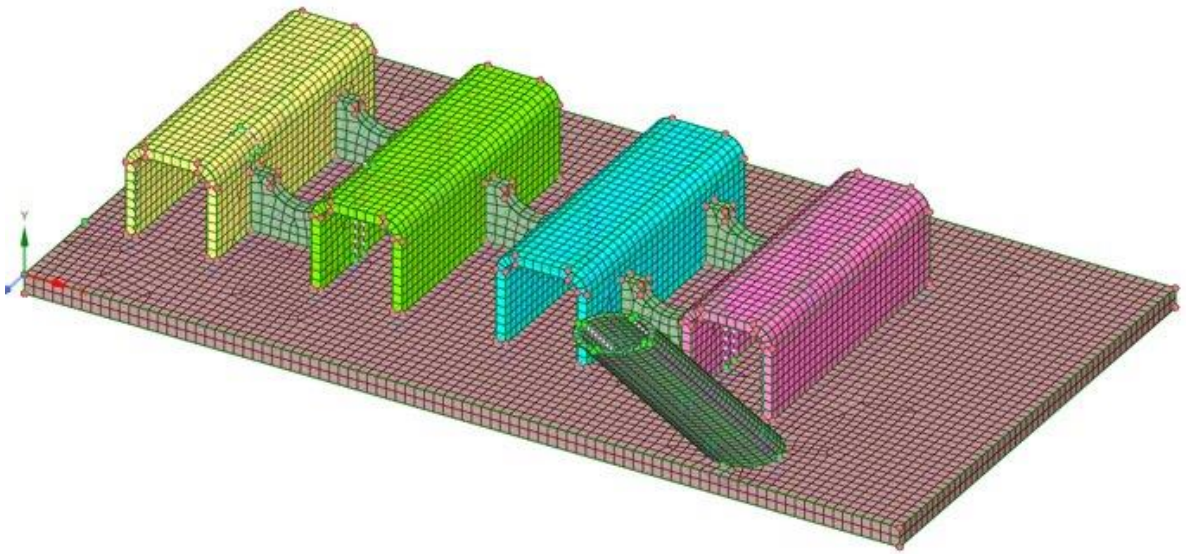
Pull: Support for Quadratic Elements (Curvilinear Mid-Nodes)
Pull (Extrude): Merge Profile Nodes

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



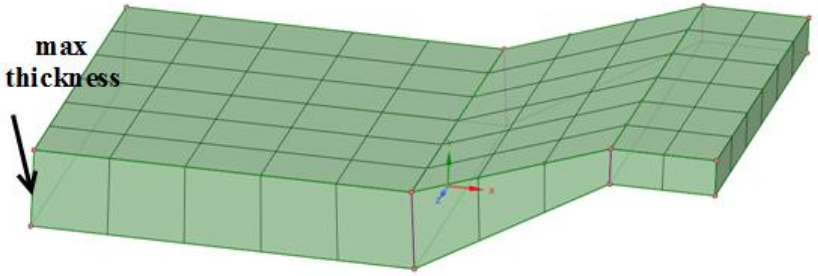
Spaceclaim Meshing: Overview

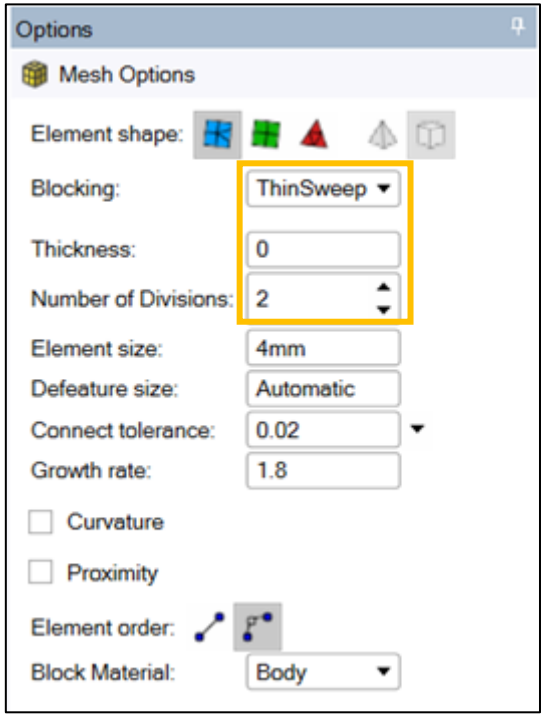
- Thin Body Meshing
  - Full Release of ThinSweep Block Decomposition
- Robustness, Performance, Usability
  - Pull improvements
  - New Mapping Options at block controls level
  - Improved quality for All Quadrilateral method
  - Improved performance
    - Faster “activate” for meshing in SpaceClaim
    - 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

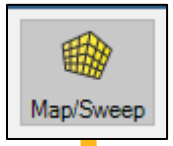
- Full release of ThinSweep option which was previously Beta
- Set number of divisions across thickness and (optionally) thickness of material
- For models with varying thickness, specify a Thickness value slightly more than the maximum thickness of the model.

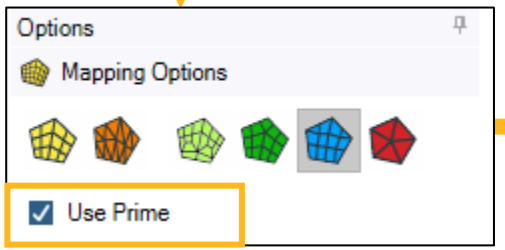




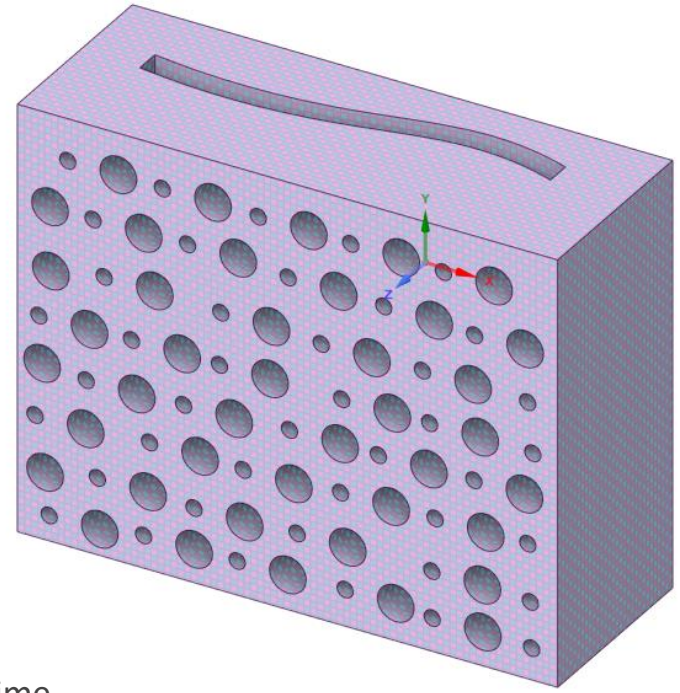
Spaceclaim Meshing: Surface Meshing Performance

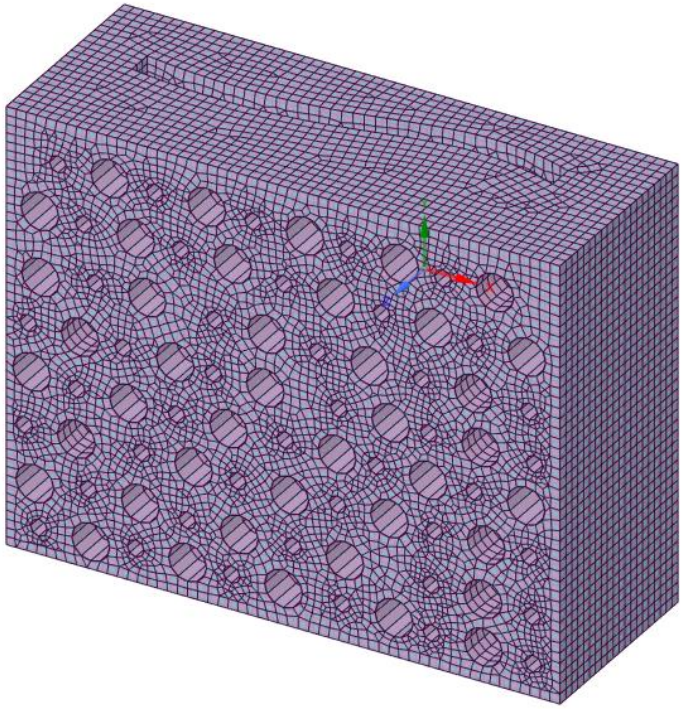
New option: Prime surface meshing at local control level





Apply faster Prime Surface meshing by Toggling ON the "Use Prime" option



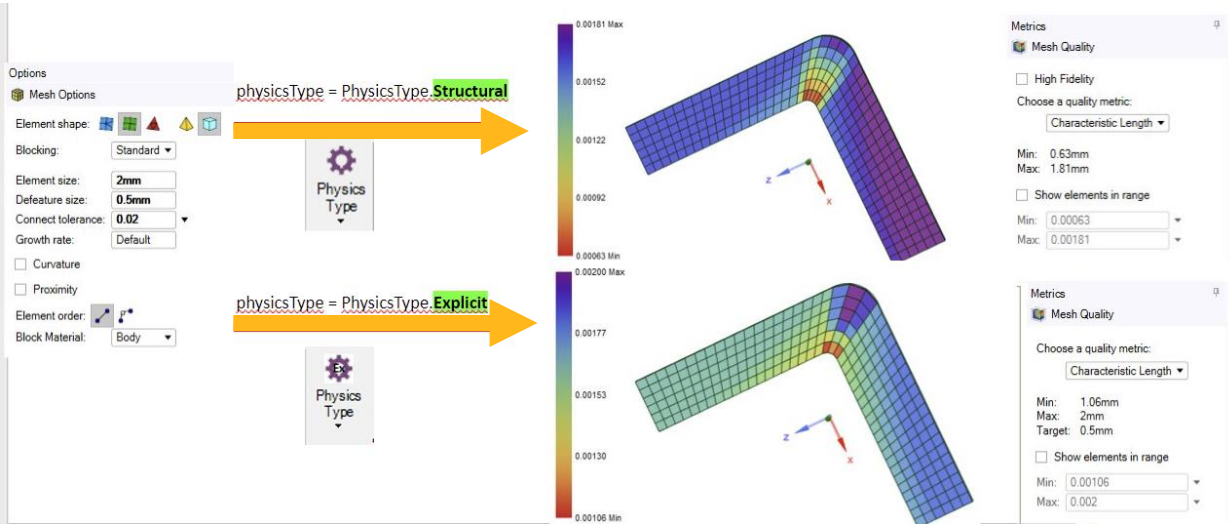


Geometry Surfaces
options.Method = MapMeshMethod.QuadsPrime

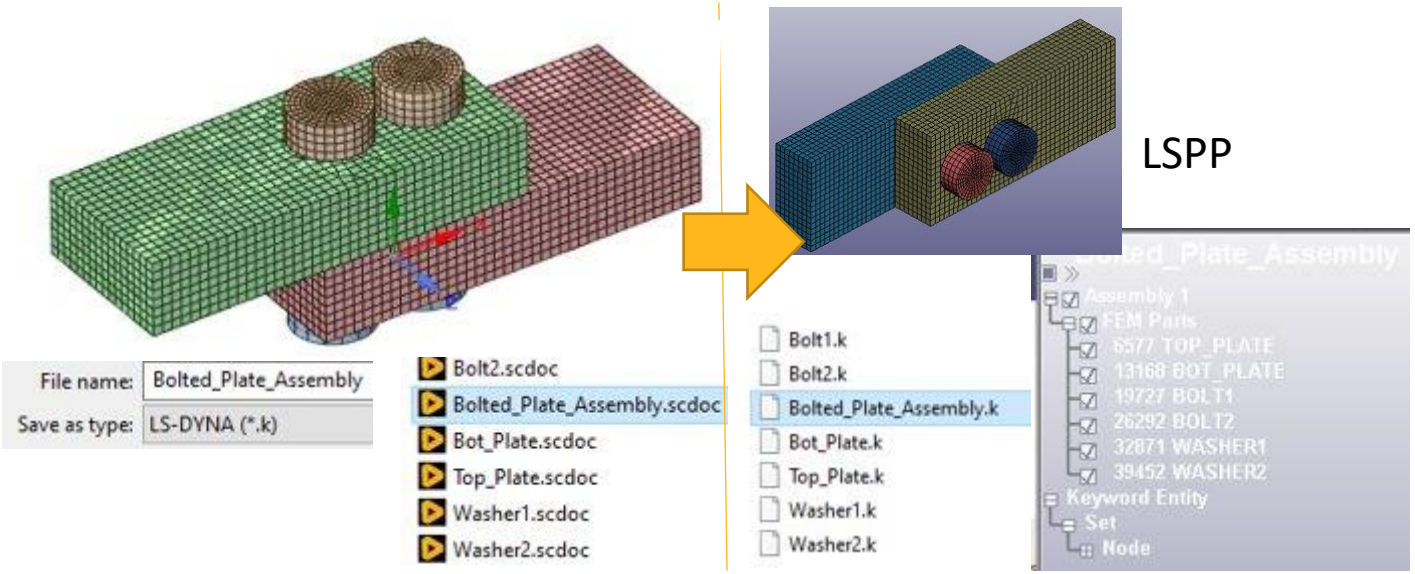
Block Faces:
options.ConvertType = ConvertBlockType.ToFreeQuadDom
options.ElementShape = ElementShapeType.PrimeQuadDominant

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



LSPP

Explicit Meshing

Tetrahedral Meshing

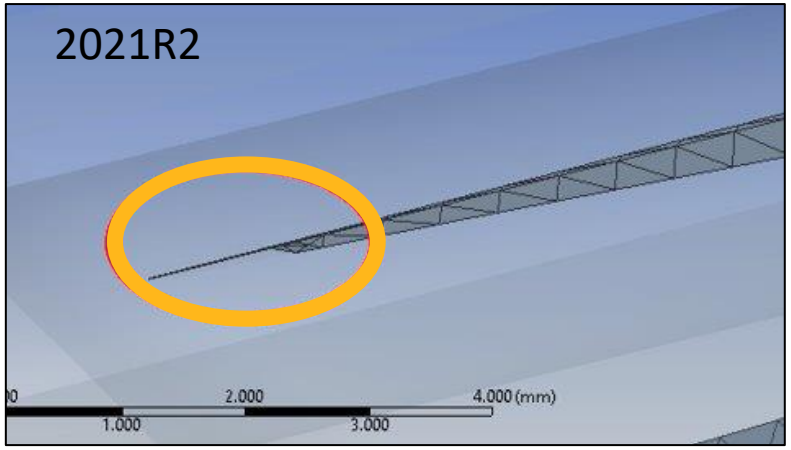
Mesh Diagnostics

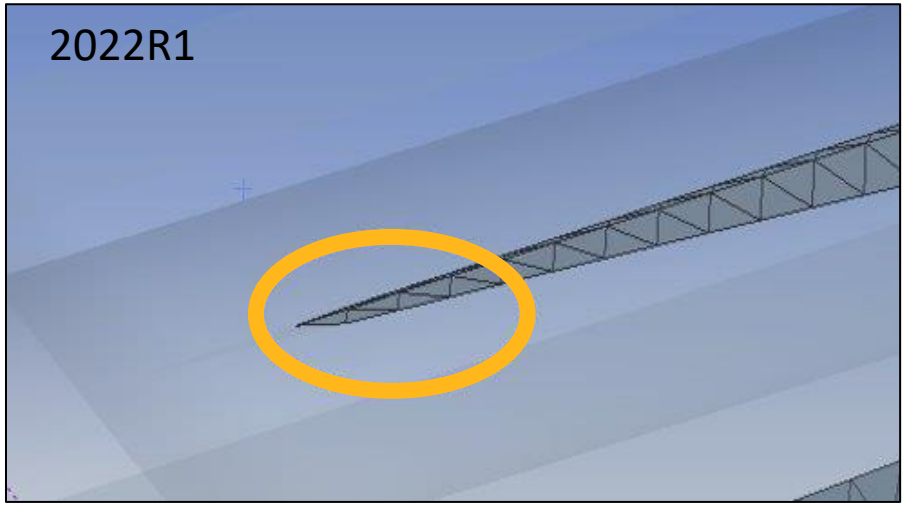
Feature Based Meshing

Improved Robustness for Patch Conforming Tetrahedra

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

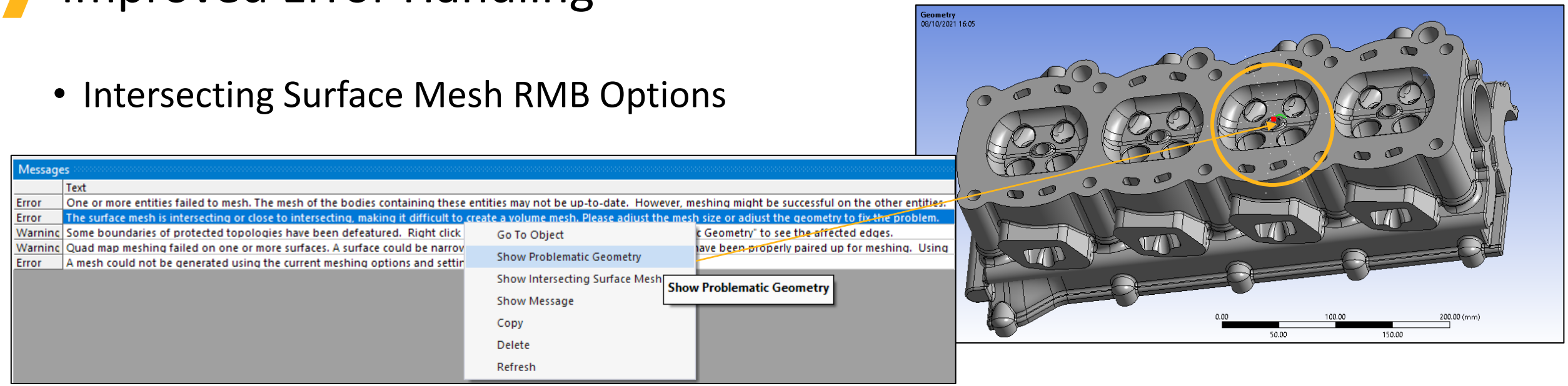
- In 2022 R1, many improvements to robustness have been made for heavy defeaturing required by Explicit users

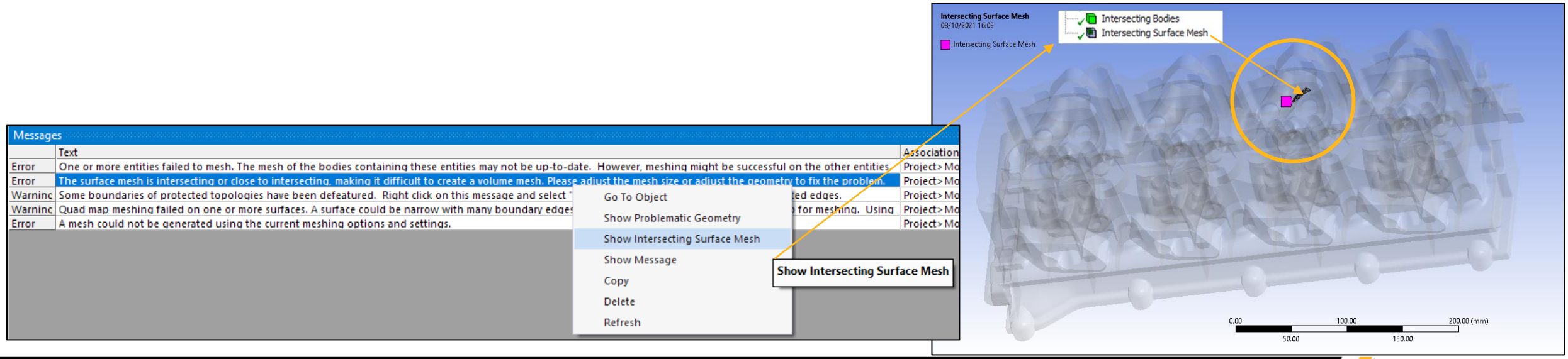




Improved Error Handling

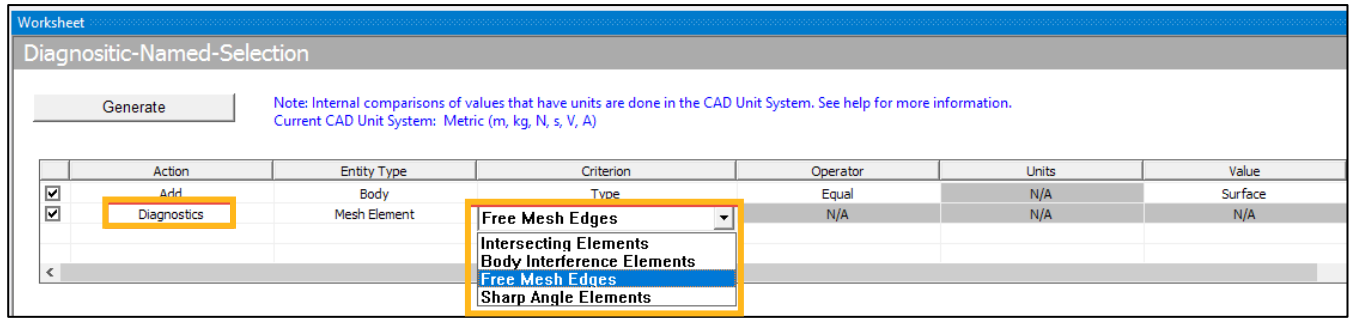
- Intersecting Surface Mesh RMB Options

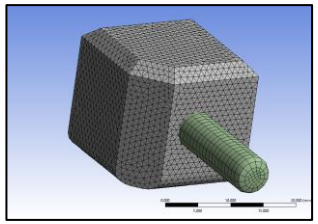




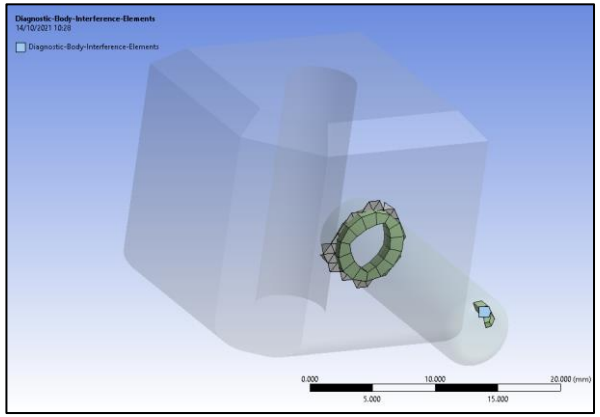
Diagnostics Tools

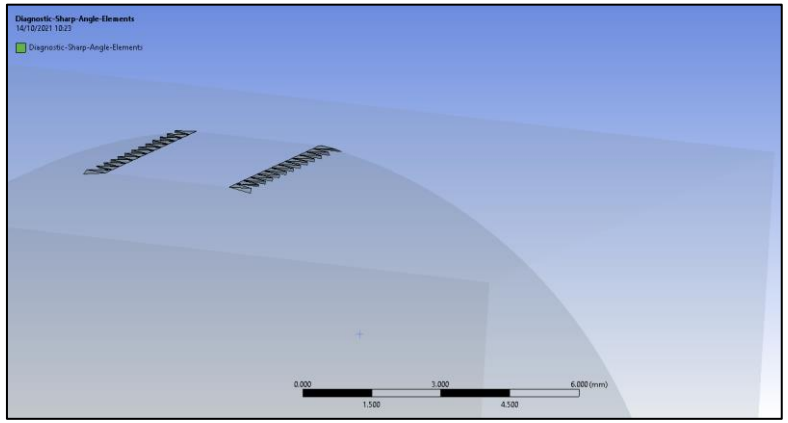
- Use worksheet Named Selection to select bodies and then run Diagnostics for visualisation of the problem
- New tools to find issues and fix via additional settings or return to geometry tool for modifications
- Options available at 2022 R1:
  - Mesh Element:
    - Intersecting surface mesh failures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





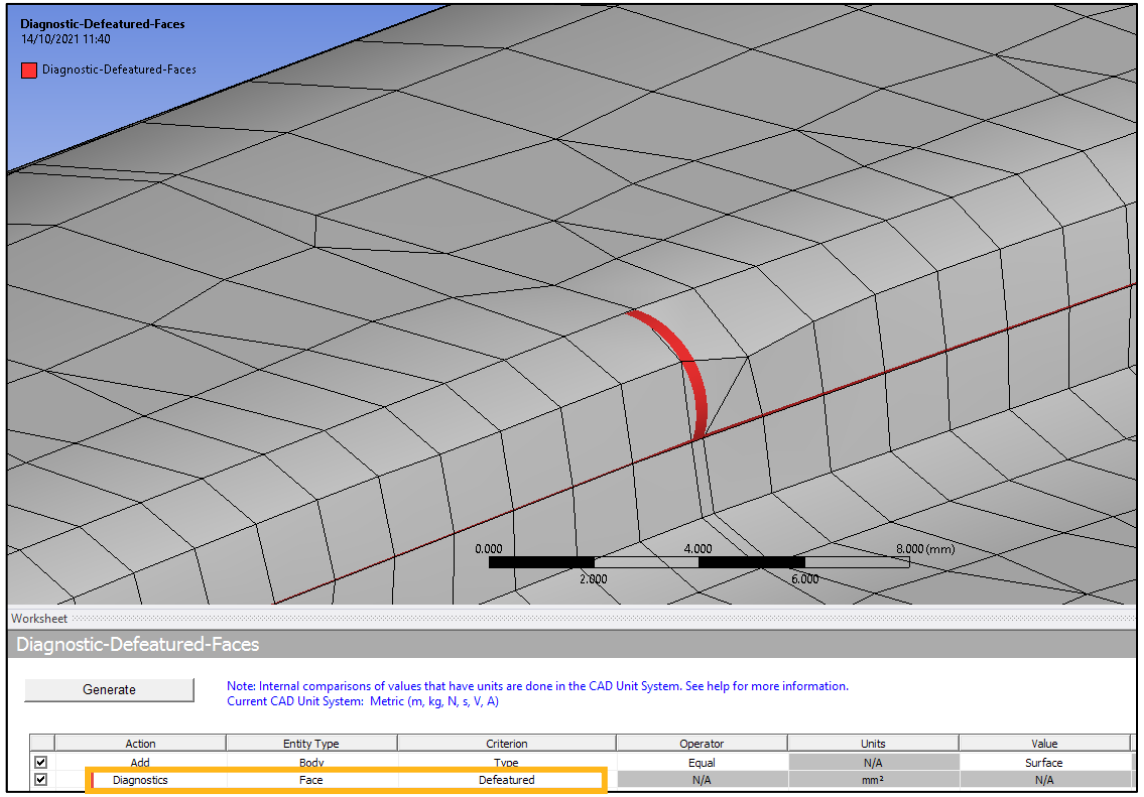
Body Interference

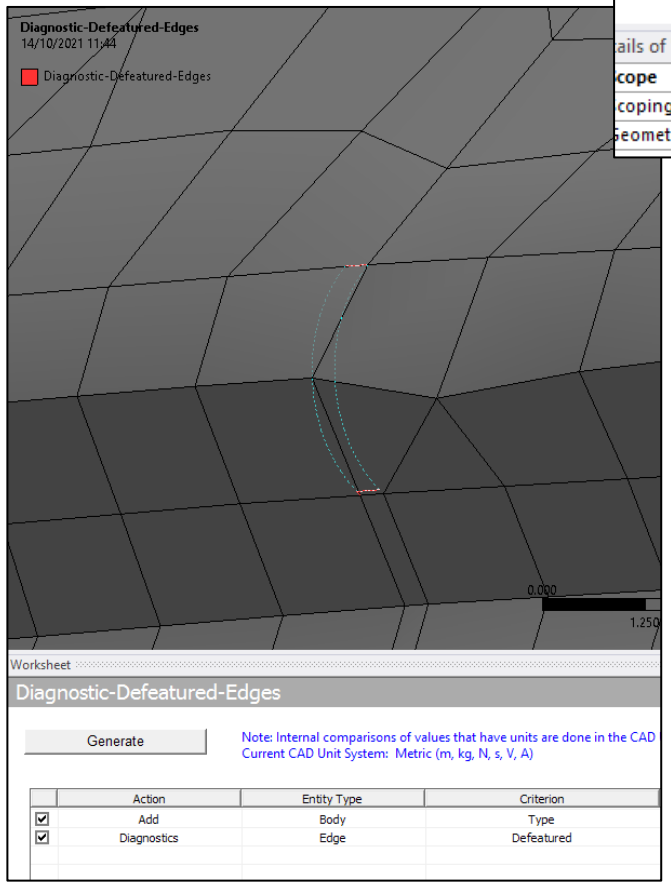


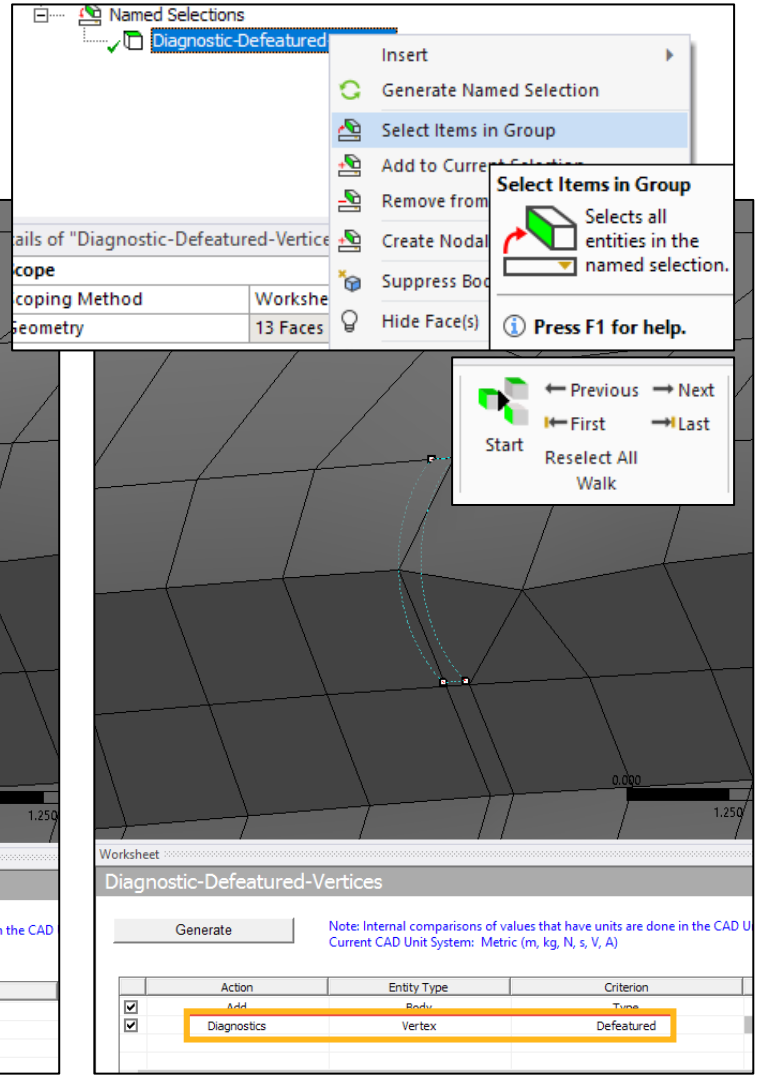


Sharp Angle Elements

Mesh Diagnostics Tools: Defeaturing Checks



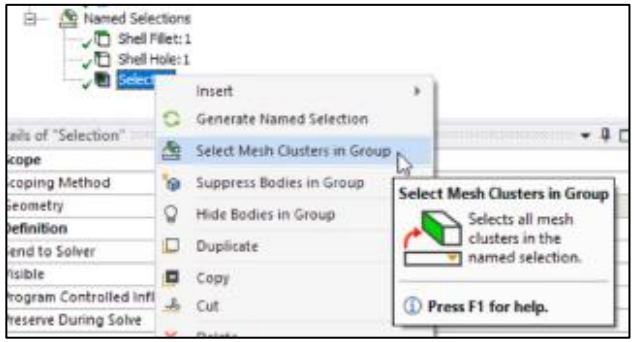


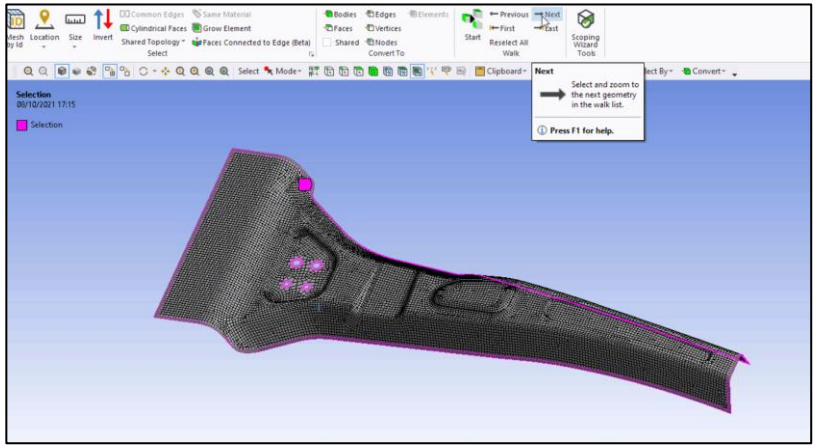


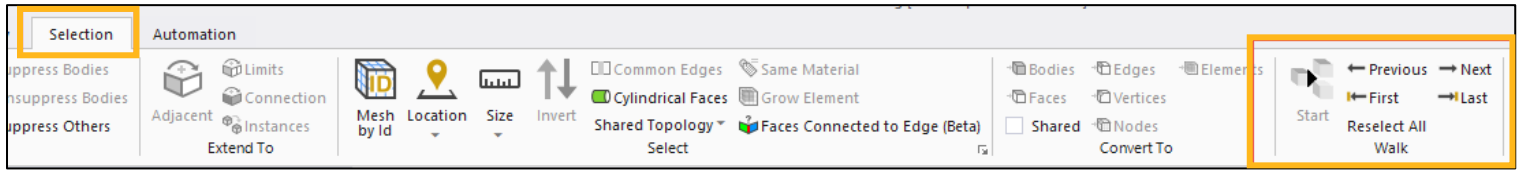
- Detect and examine where defeaturing has occurred after meshing
- Use Selection Walk to traverse defeatured faces

Mesh Cluster Walk

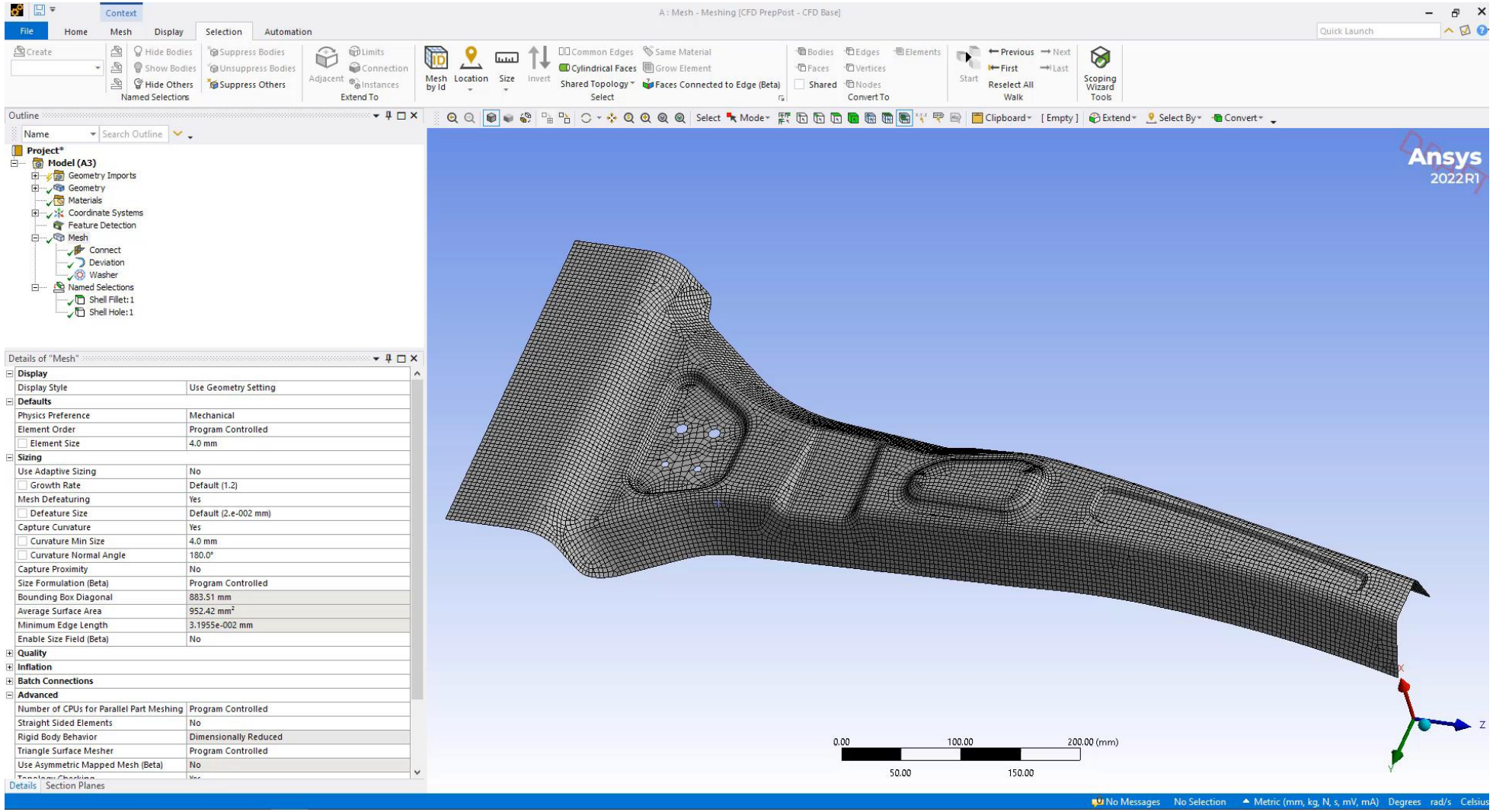
- Model Walk has been enhanced to walk through
  - Mesh Elements
  - Mesh Element Clusters
- Useful for new diagnostics tools to traverse multiple issues
- New RMB option on Element based NS from Diagnostics
  - Select Mesh Clusters in Group







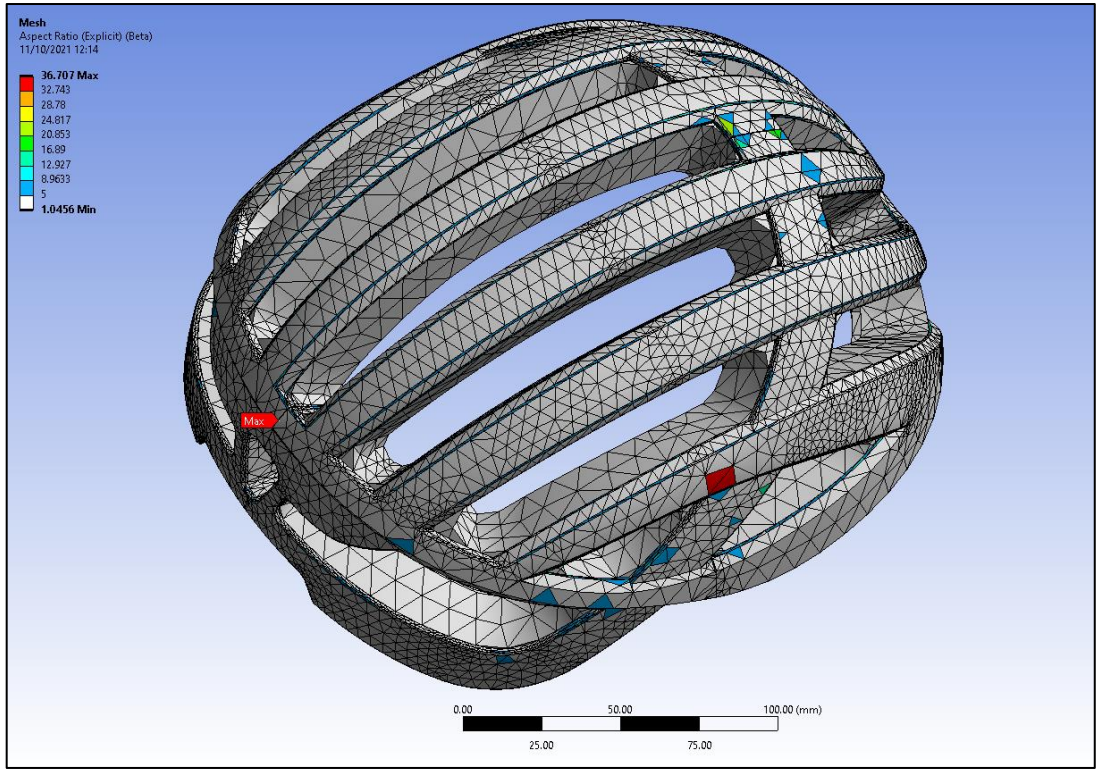
Mesh Cluster Walk: Short Demo with Free Face Diagnostics



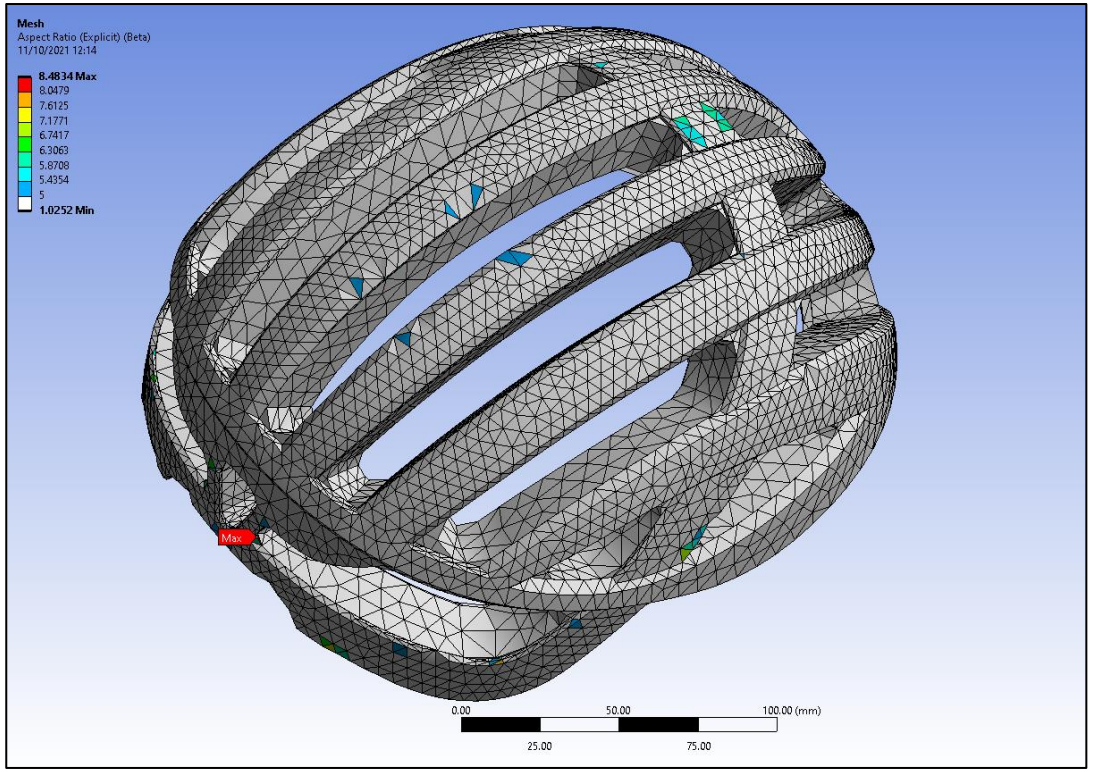
Default Surface Mesher Explicit Physics Pref. = Adv. Front

- 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

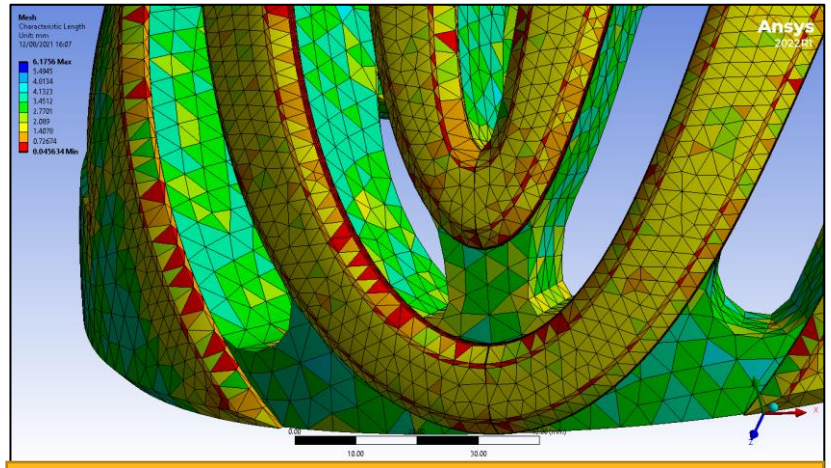
Tet Meshing Aspect Ratio Targeting (Explicit Physics Pref.)

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

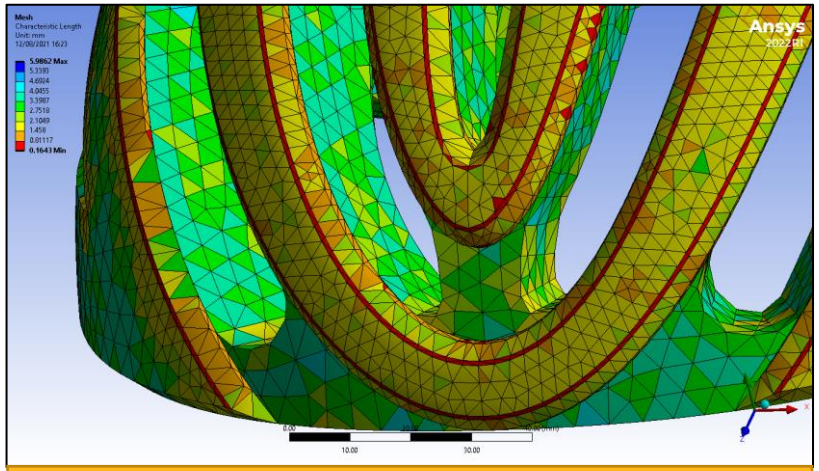
Quality	
Check Mesh Quality	Yes, Errors and Warnings
Target Element Quality	Default (0.200000)
Target Characteristic Length (LSDyna)	Default (0.5 mm)
Target Aspect Ratio (Explicit)	5.

Challenges:

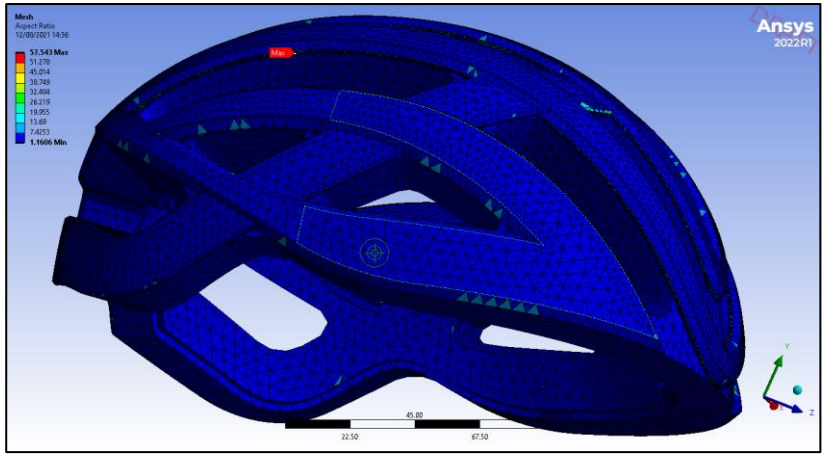
- Bike Helmet Example: A complex geometry with several intricate features



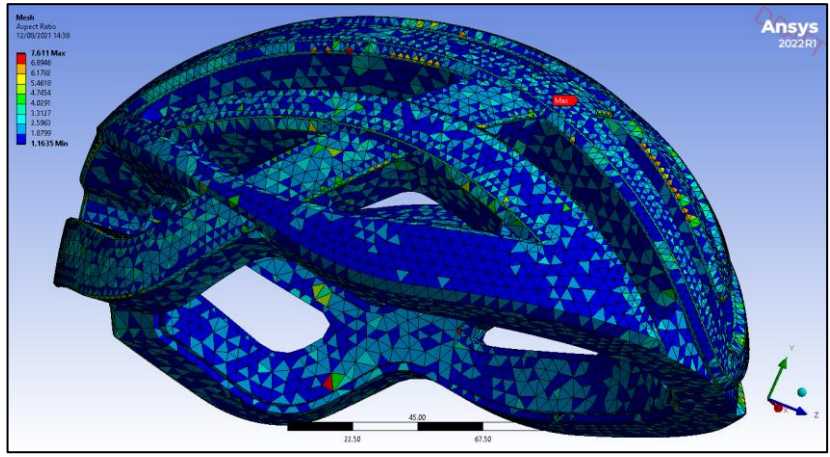
2021 R2 – Max AR=53, Min CL=0.05mm



2022 R1 – Max AR=7.6, Min CL=0.16mm



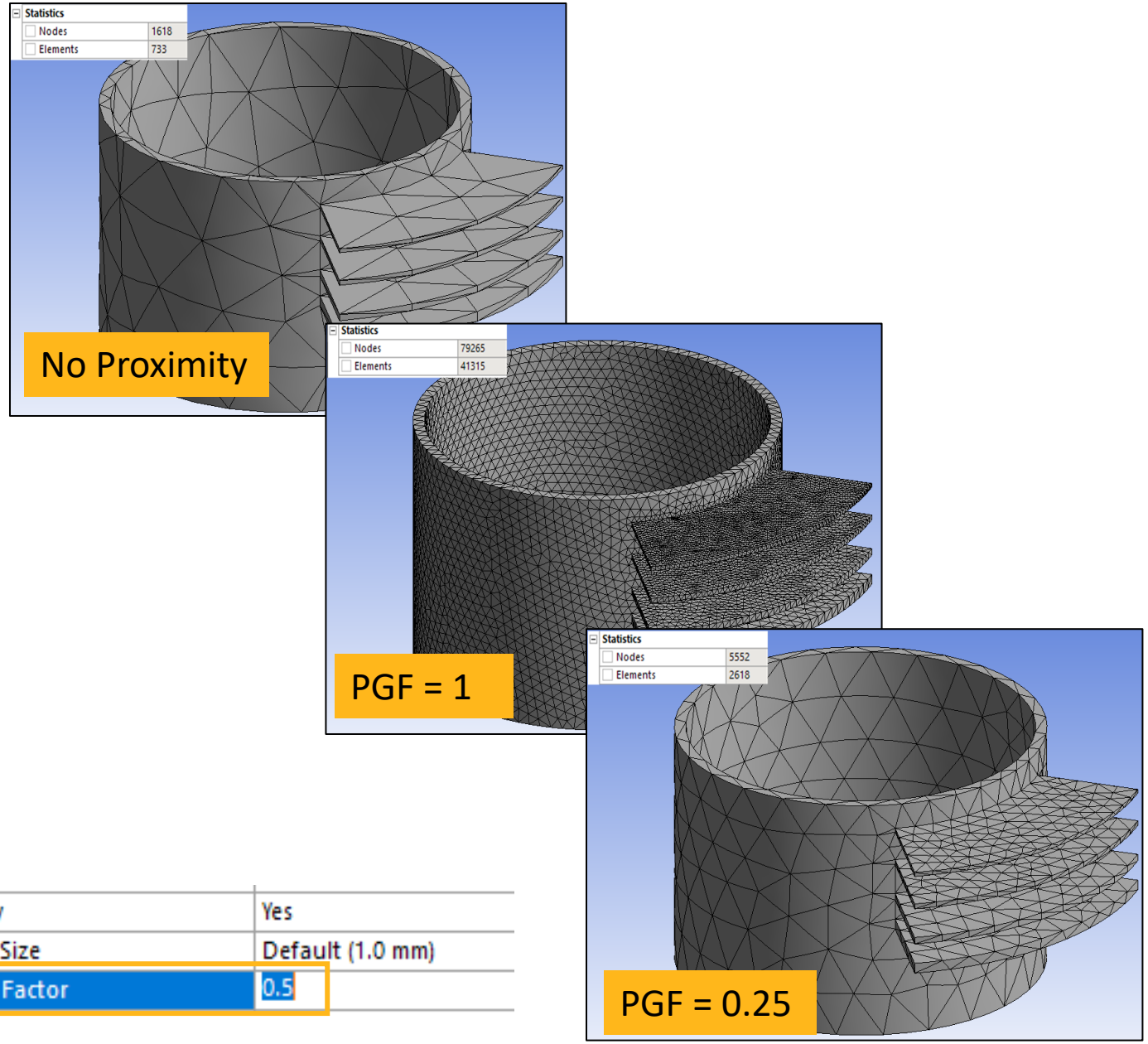
BEFORE



AFTER

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

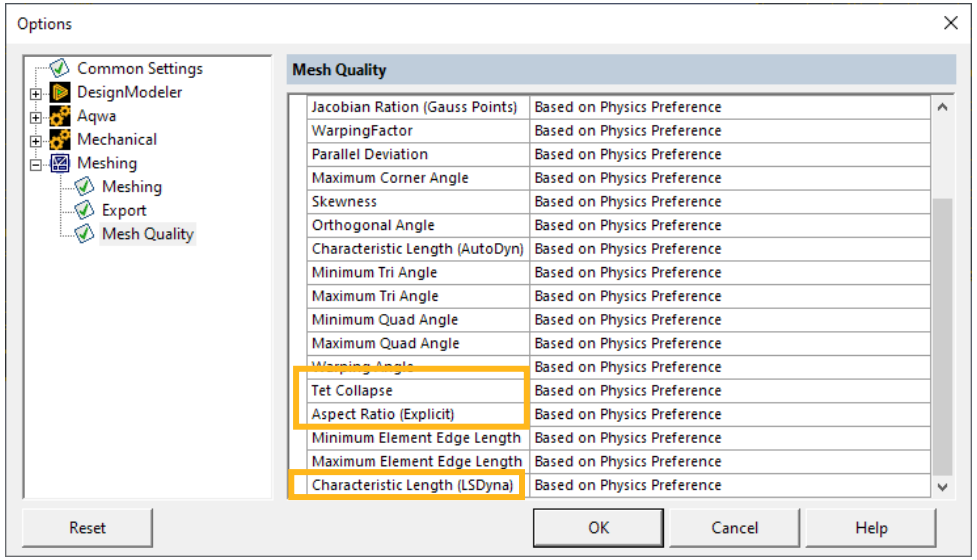


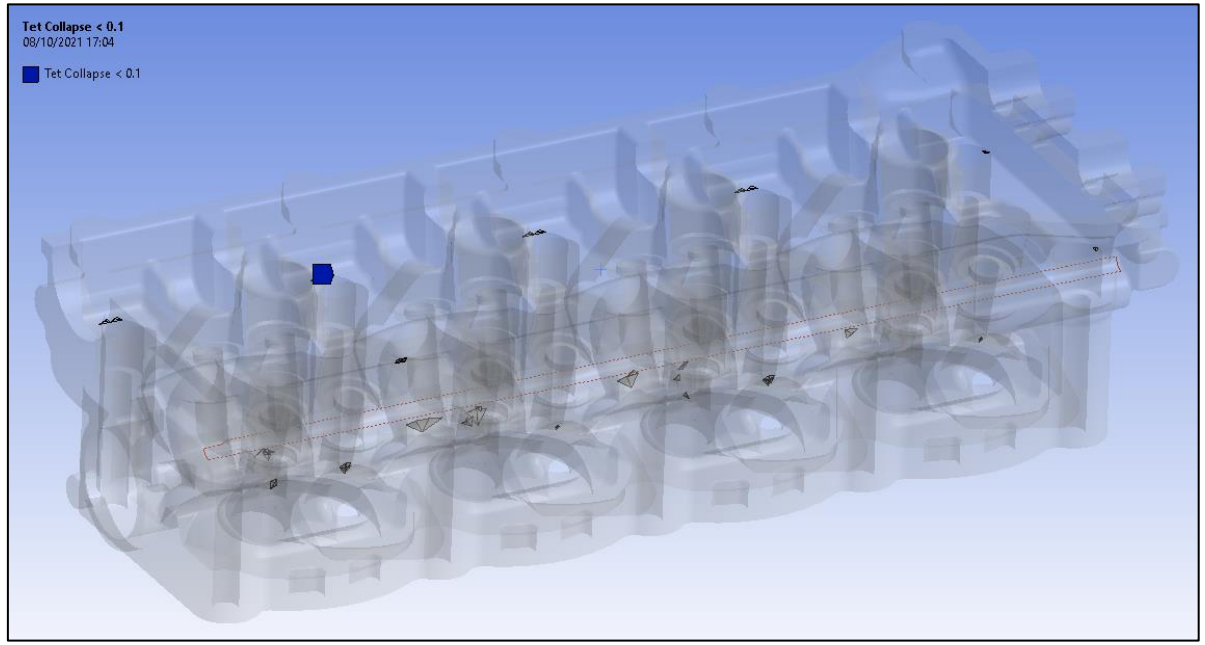
Capture Proximity	Yes
Proximity Min Size	Default (1.0 mm)
Proximity Gap Factor	0.5

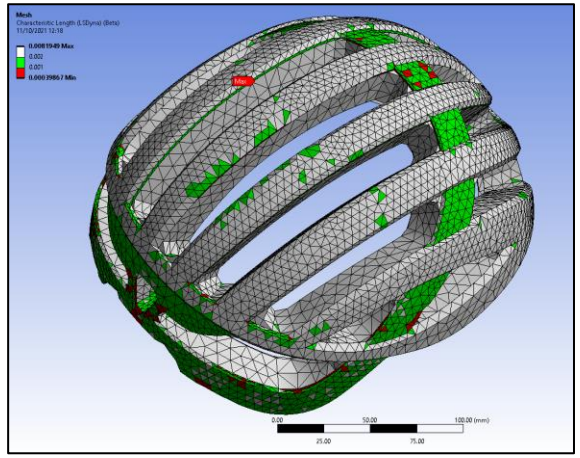
Quality

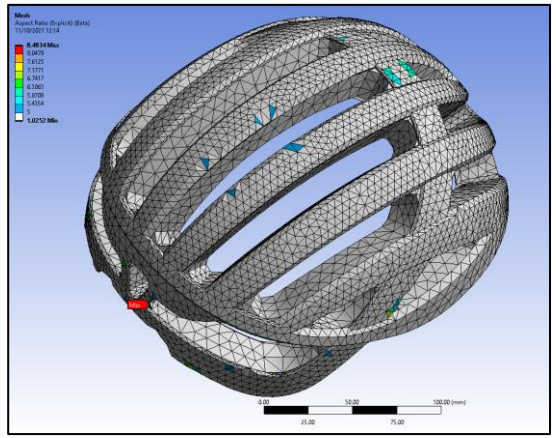
- Exposure of many more metrics as mentioned
  - Visibility of metrics is based on physics preference
  - New include LS Dyna Characteristic Length and Explicit Aspect Ratio

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.









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

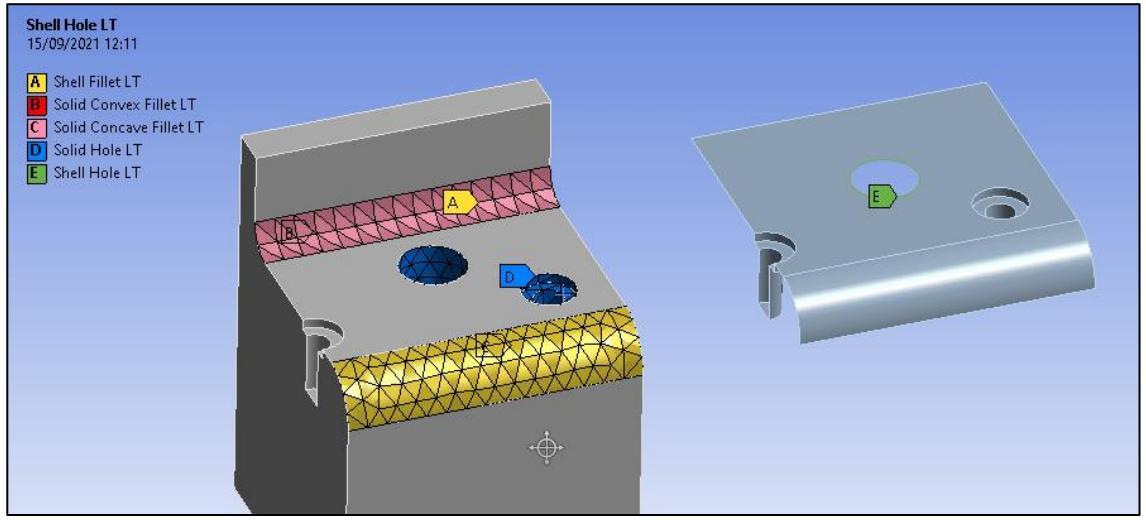
Worksheet

Feature Detection

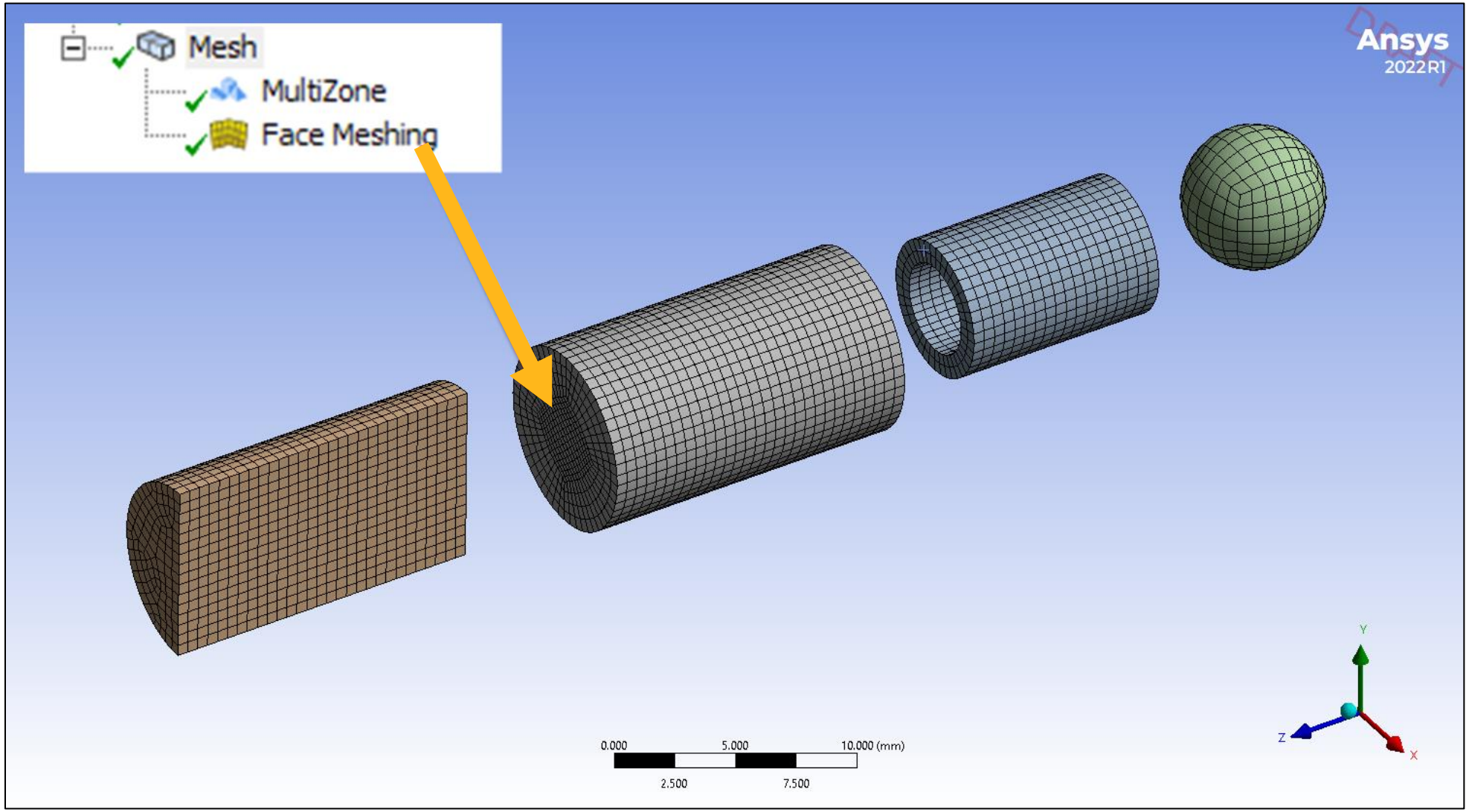
*Right click on the grid to add/delete a row.

	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

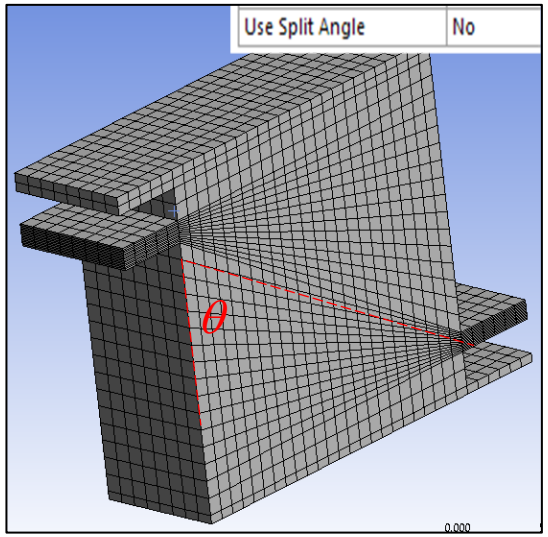


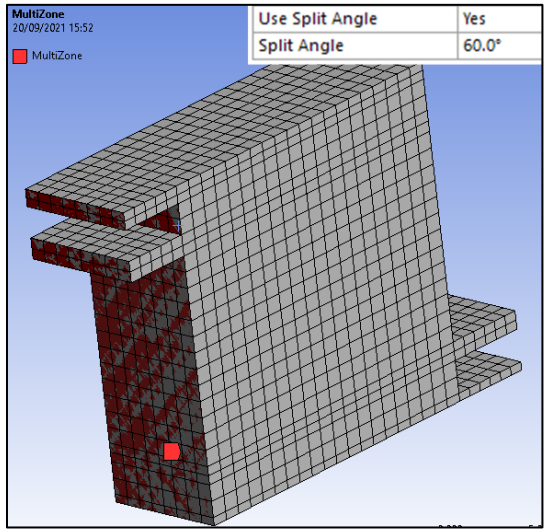
Hex Meshing: Less Decomposition in Mechanical MultiZone

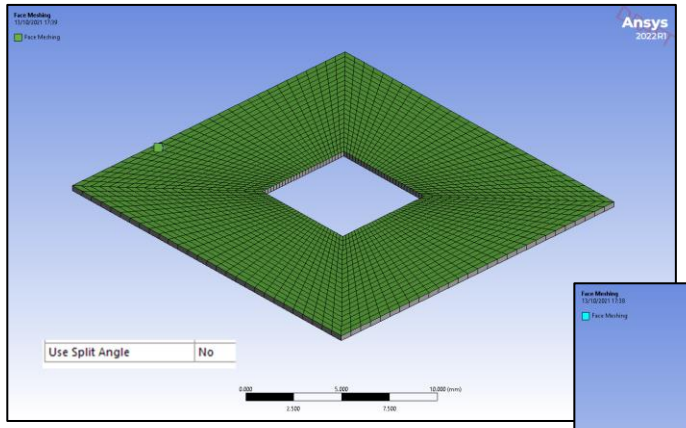


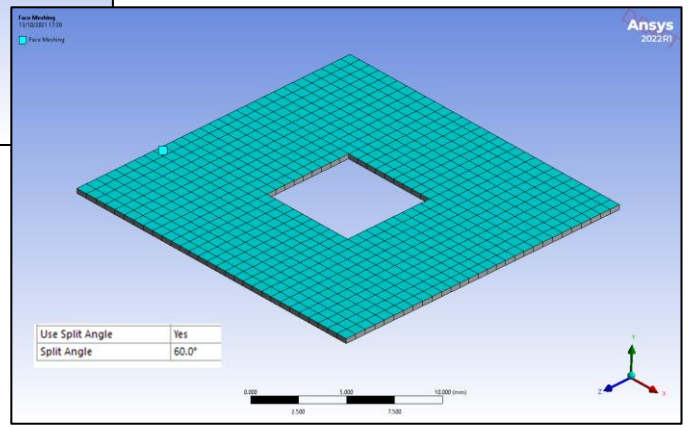
Hex Meshing: Split Angle

- 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





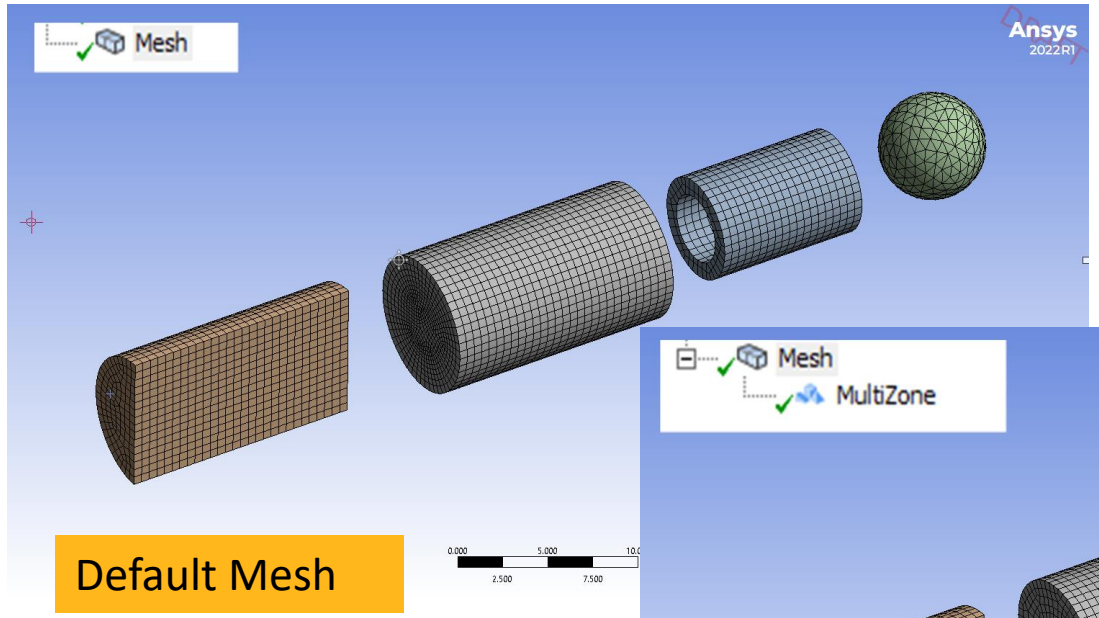


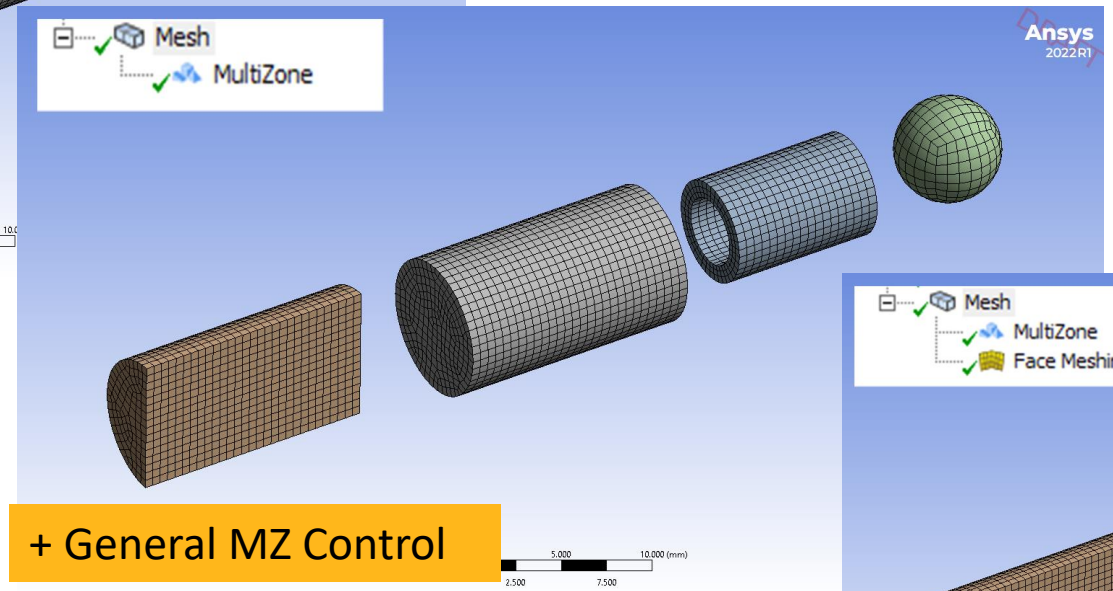


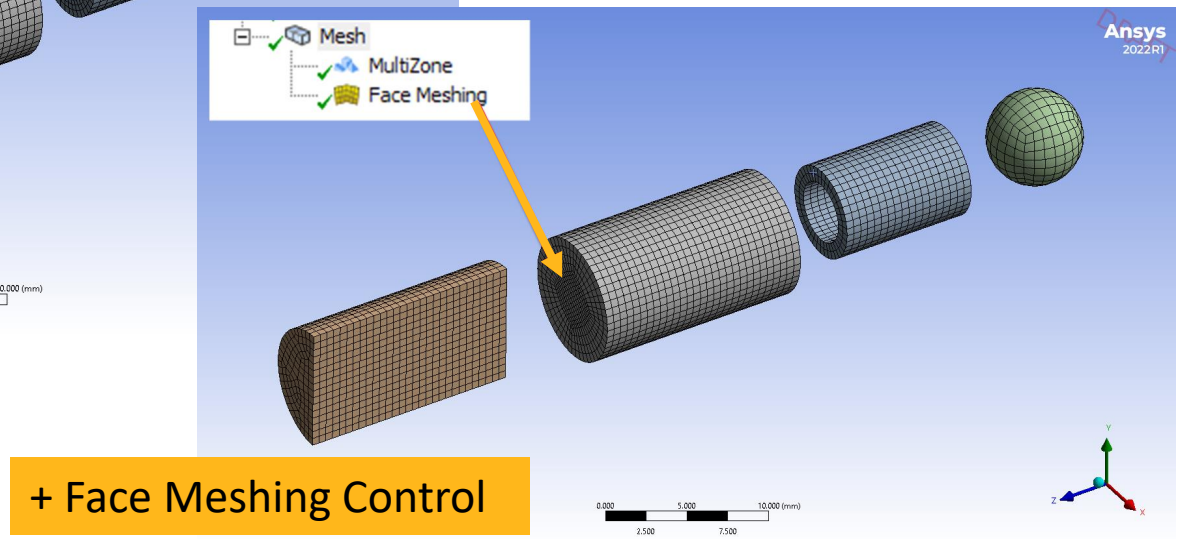
Details of "MultiZone" - Method

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
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

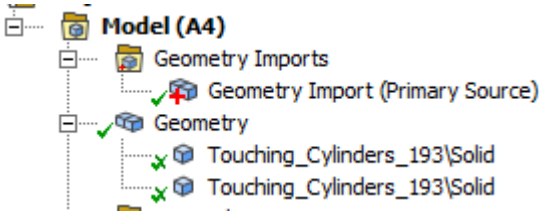






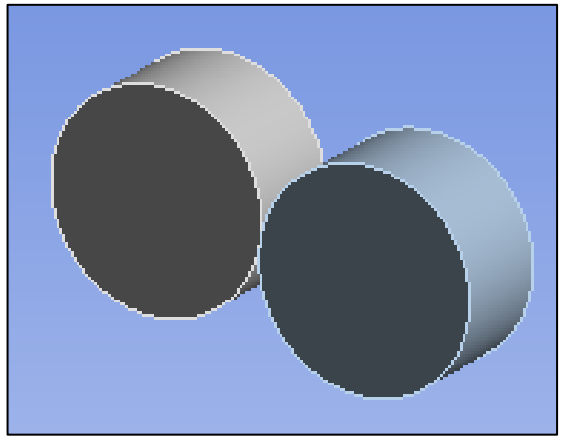
Hex Meshing: Better Default Mesh for Cylinders, Circles, and Spheres

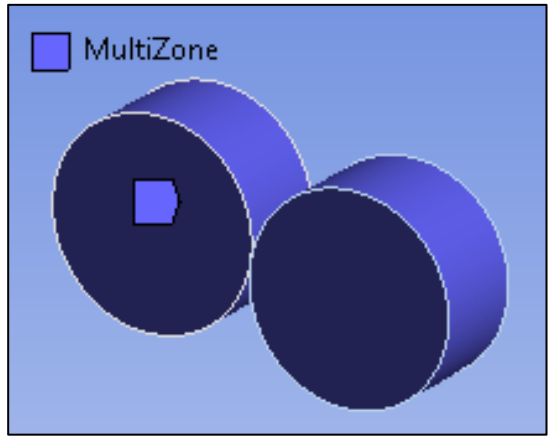
Touching Cylinders

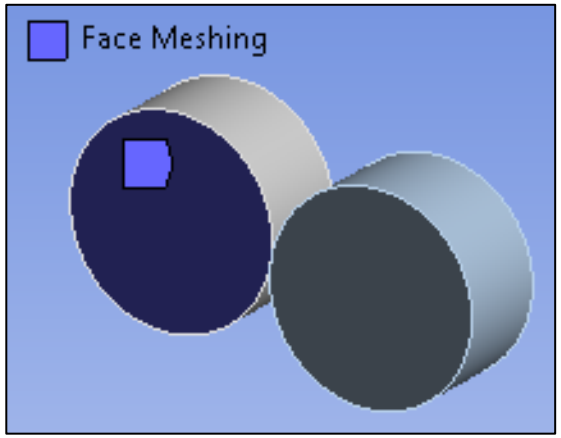


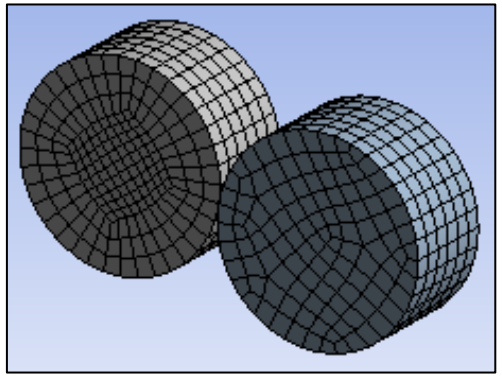
MultiZone Applied with no special inputs/selections – Automatic Hex

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





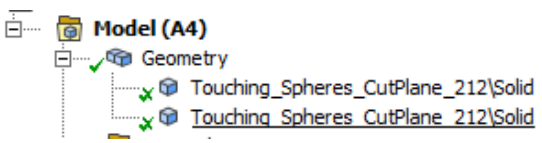


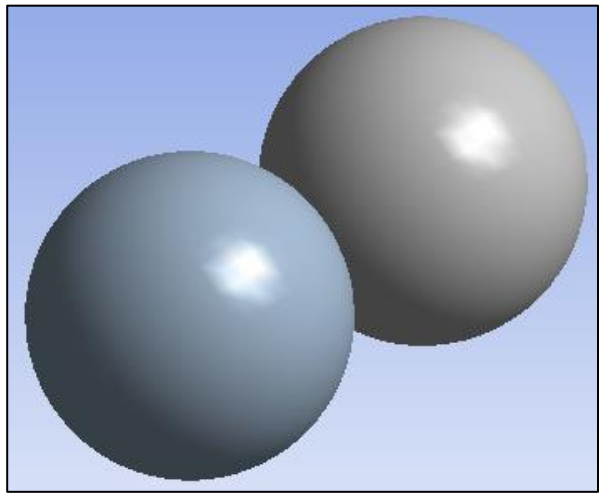


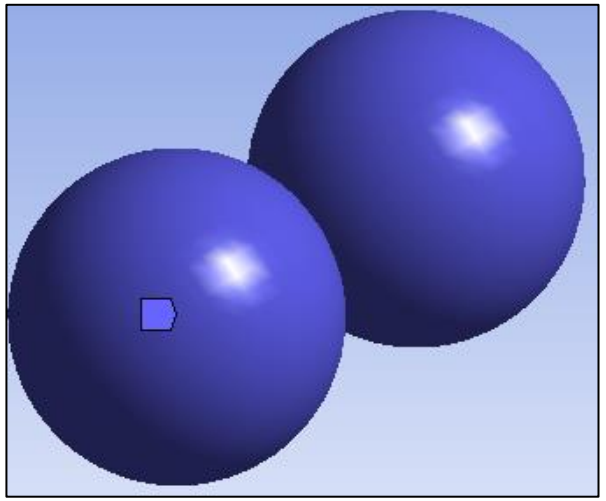
Hex Meshing: Better Default Mesh for Cylinders, Circles, and Spheres

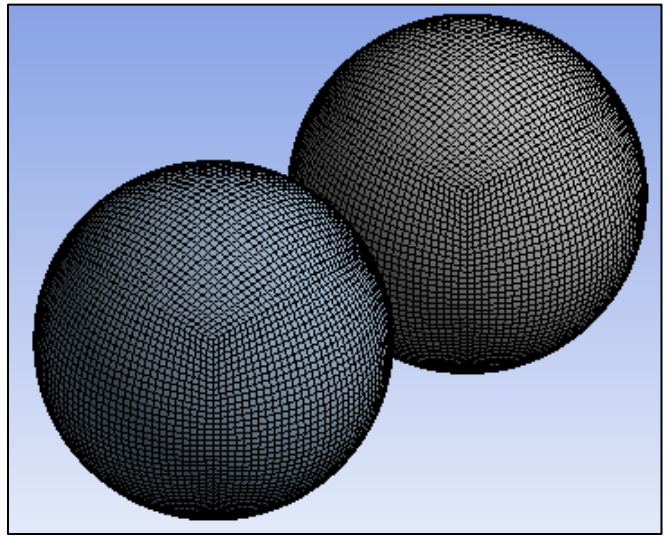
Touching Spheres

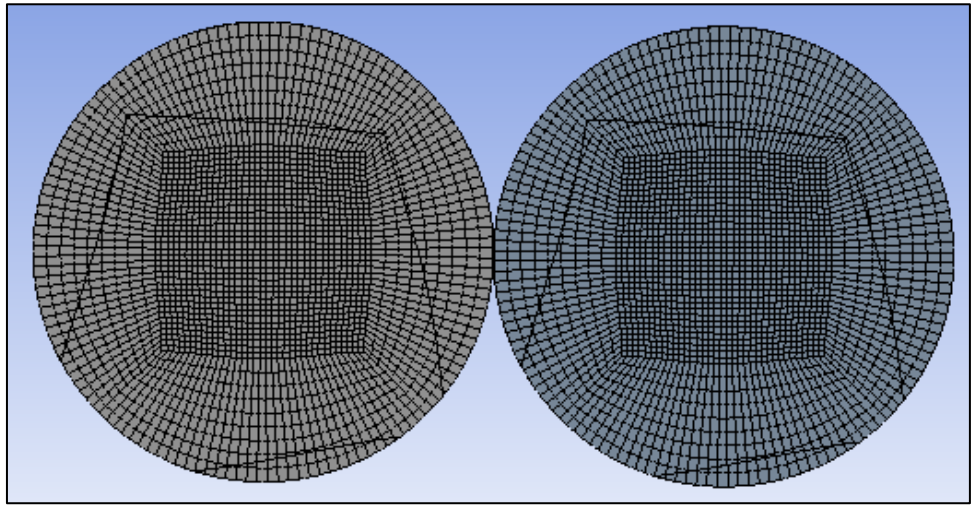
MultiZone Applied with no special inputs/selections – Automatic Hex



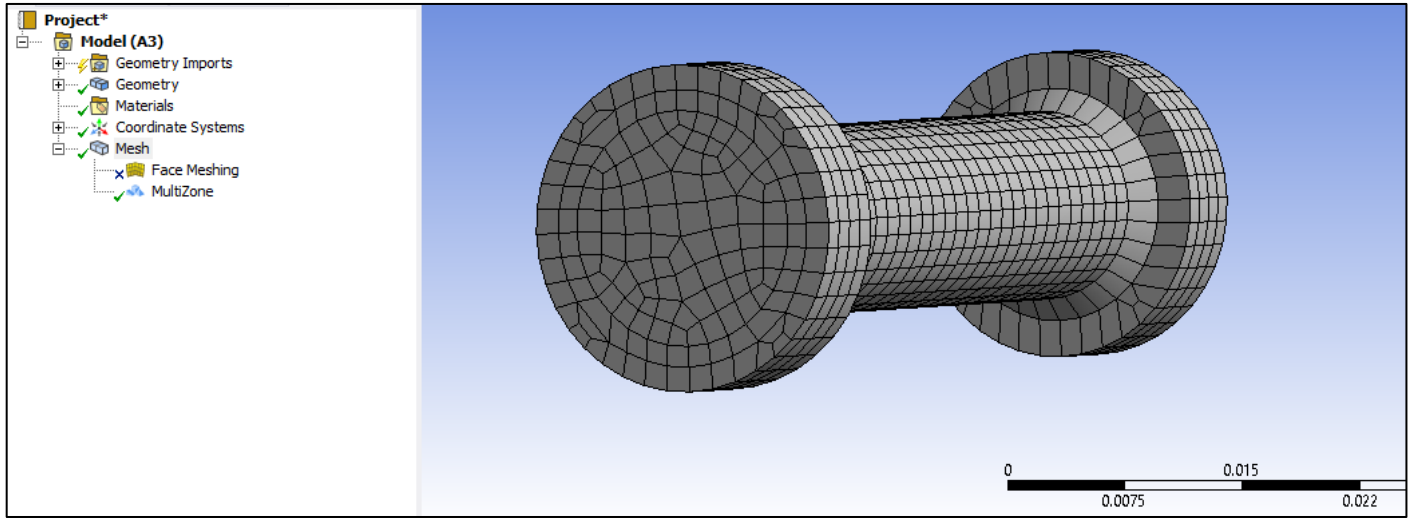




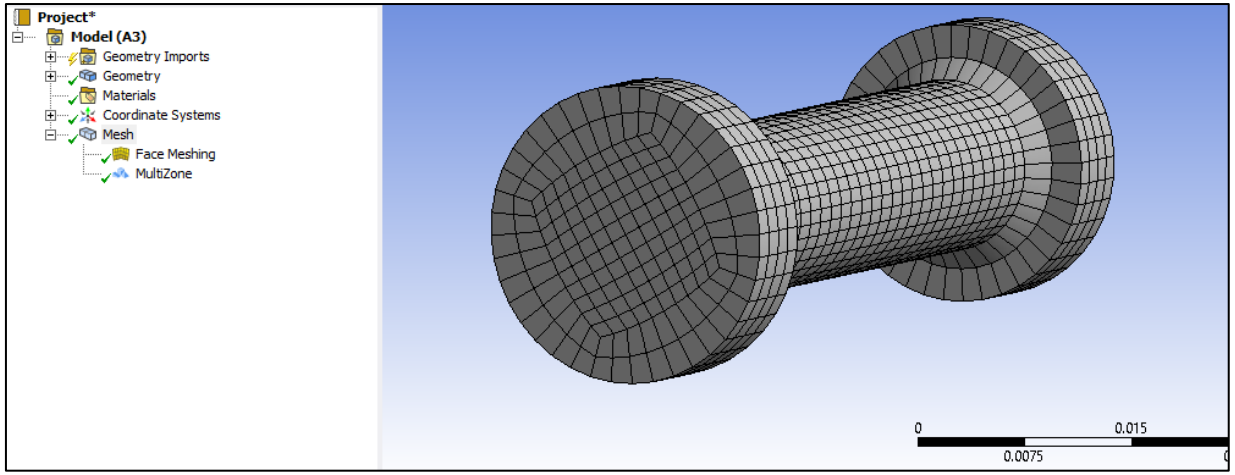




Hex Meshing: Improved Hex Mesh for Simple Shapes



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

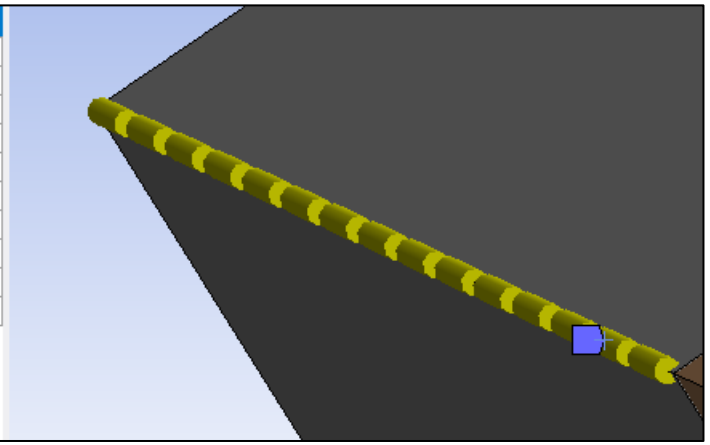


Hex Meshing: Body Fitted Cartesian - Edge Sizing Support

- BF Cart now supports “Edge Sizing” control

Details of "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



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

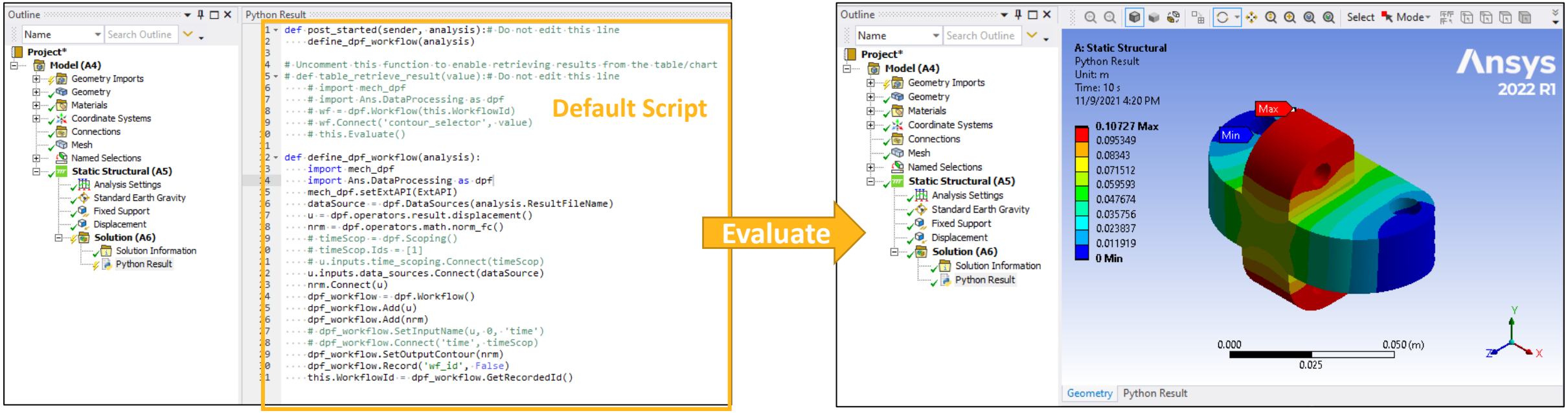
Post Processing/Graphics

Performance Improvements

- Speedup time to click on Named Selections and loads for some models by 90%
- Speedup time to hide and show body groups for some models by 75%
- Improve time to click on Imported Contact on some models from infinity (impossible to click earlier – Mechanical would hang) down to a few seconds
- Improve UI behavior of “Thick Shells and Beams” button for some models – from minutes to instantaneous if there are no shells, beams or SPH particles present (no-op behavior)
- Speedup time to click on mesh for some models with lots of bodies (meshed + unmeshed parts) by 70% when there was a section plane enabled and by 45% when the section plane was disabled
- Speedup time to click on mesh for some models by 37%
- Utilize bulk-data APIs to further speedup the time (on top of the above-mentioned improvement) to click on Imported Contact for some models by 64%

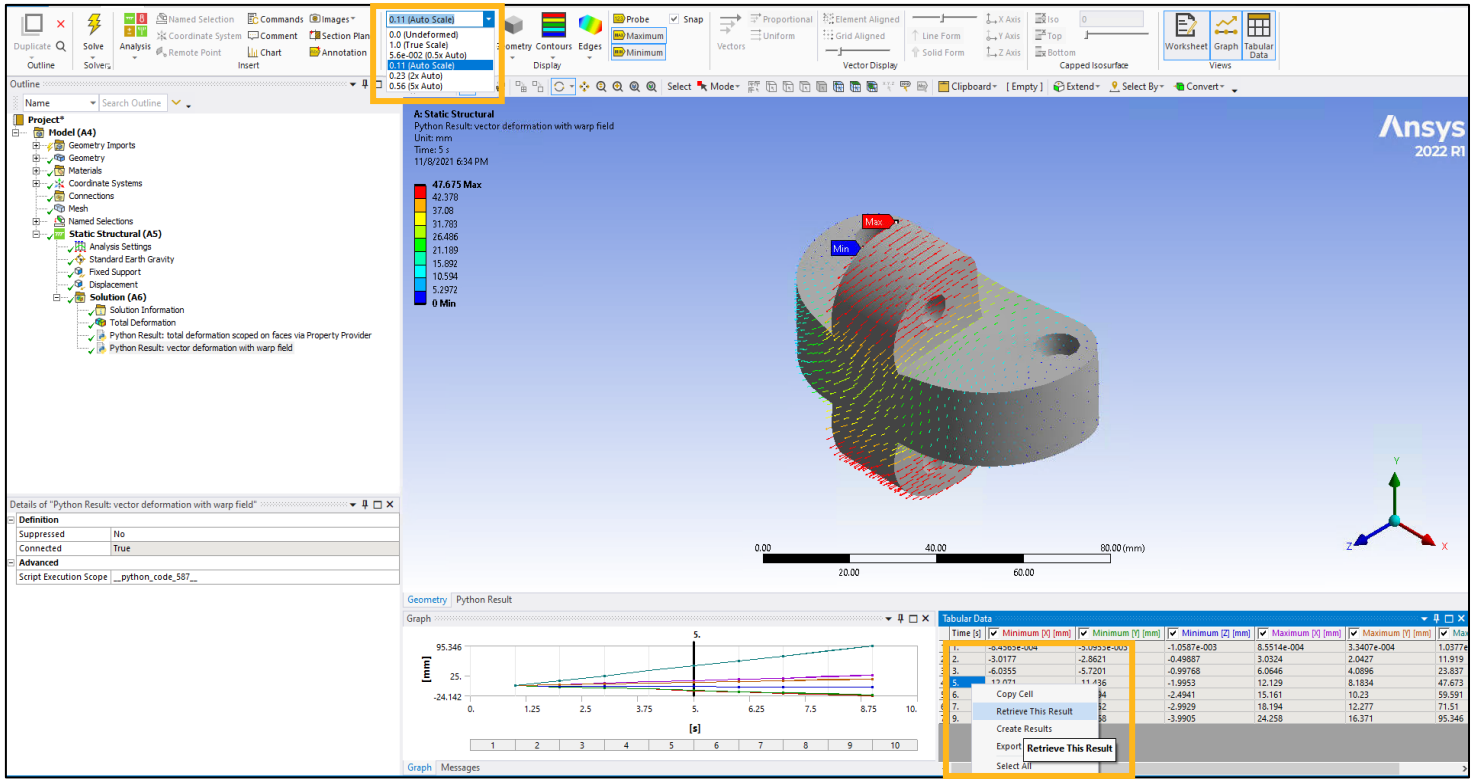
Python Result

- The **Python Result** object enables users to evaluate output quantities by executing an Iron-python script based on the Data Processing Framework (DPF) post-processing toolbox
- A **Python Result** object with a default script can be inserted under the **Solution** object. Users can create and evaluate a variety of scripts and display the results in Mechanical



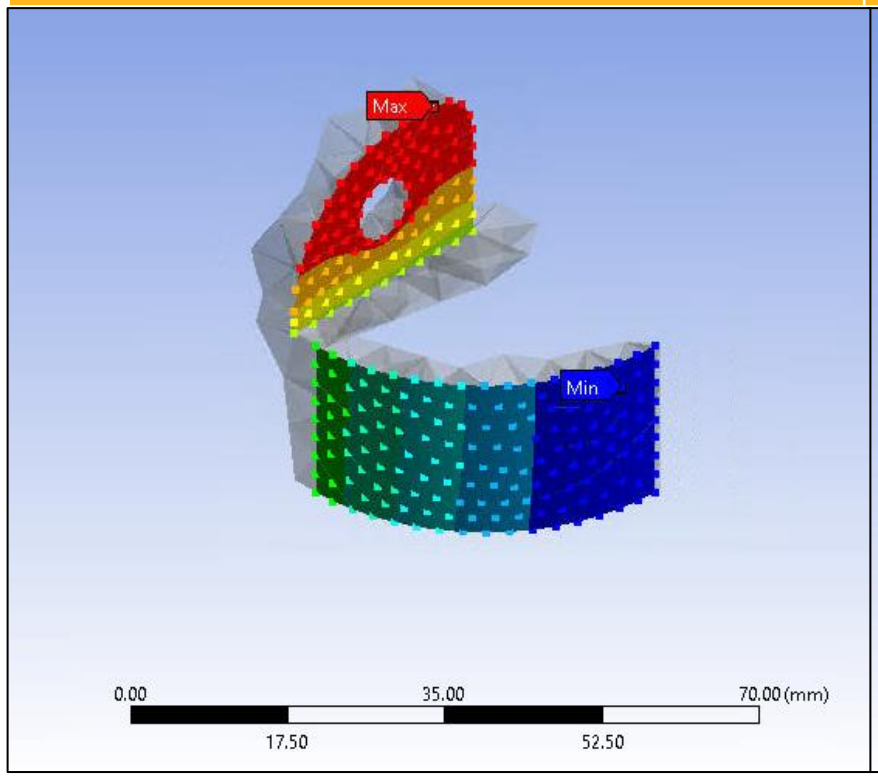
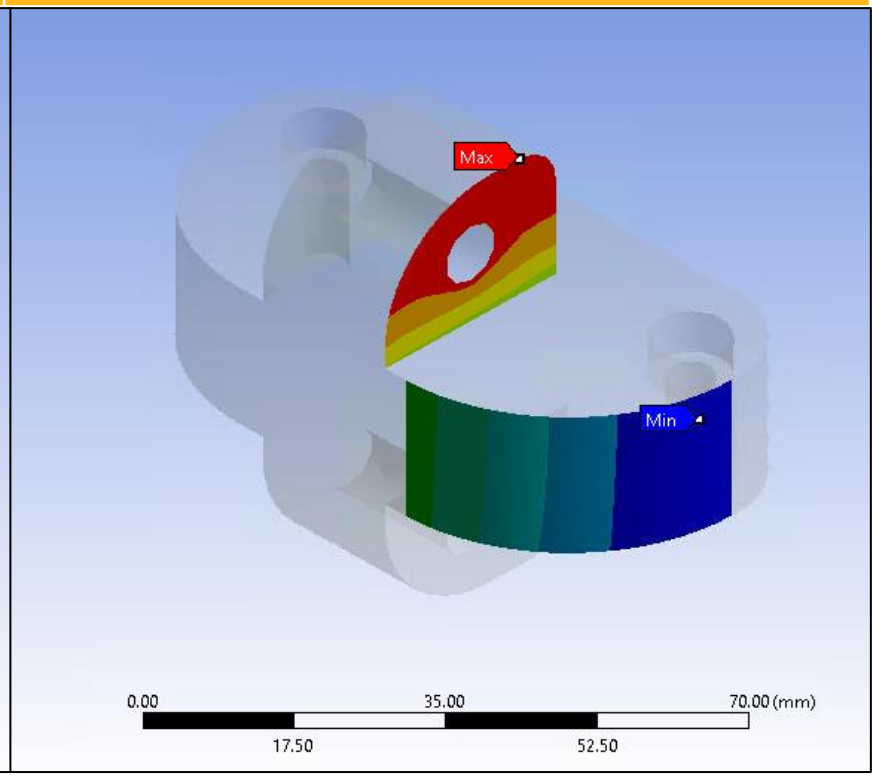
Python Result

- Vector result is supported
- Retrieve result from the **Graph** window or **Tabular Data** window to view the results at a desired time point
- Deformation Scale Factor is available if the warping field is defined



Python Result

- Different types are provided for a better contour result display

dpf_workflow.SetOutputContour(nrm)	dpf_workflow.SetOutputContour(nrm, *dpf.enums.GFXContourType.GeomFaceScoping*)
	

dpf.enums.GFXContourType.FENodalScoping

dpf.enums.GFXContourType.FEElementalScoping

dpf.enums.GFXContourType.FEElementalFaceScoping

dpf.enums.GFXContourType.GeomBodyScoping

dpf.enums.GFXContourType.GeomFaceScoping

dpf.enums.GFXContourType.GeomEdgeScoping

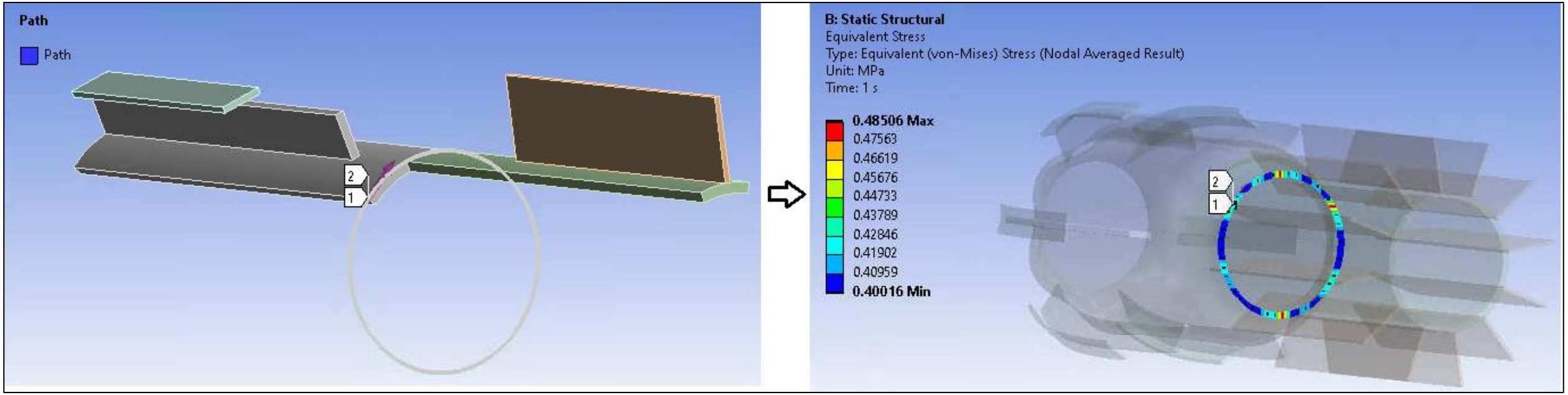
dpf.enums.GFXContourType.GeomVertexScoping

Path Results Extensions: Results scoped to Paths Available for Models with Symmetry Conditions

- Create Paths and scope results to them, even if there are symmetry conditions present
- The model will be expanded, and results will be mapped wherever the Path intersects with the expanded model

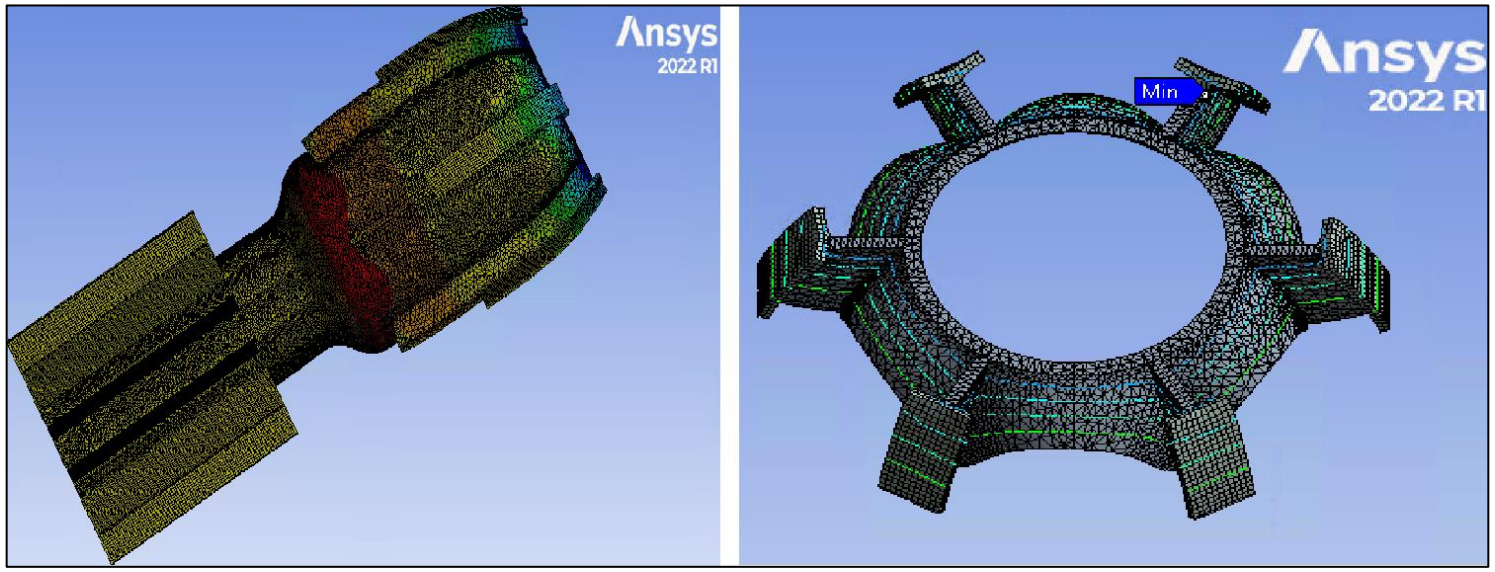
Symmetry conditions include:

- Cyclic
- Pre-meshed cyclic
- General Axisymmetric
- Multi-stage cyclic



Result Preferences API

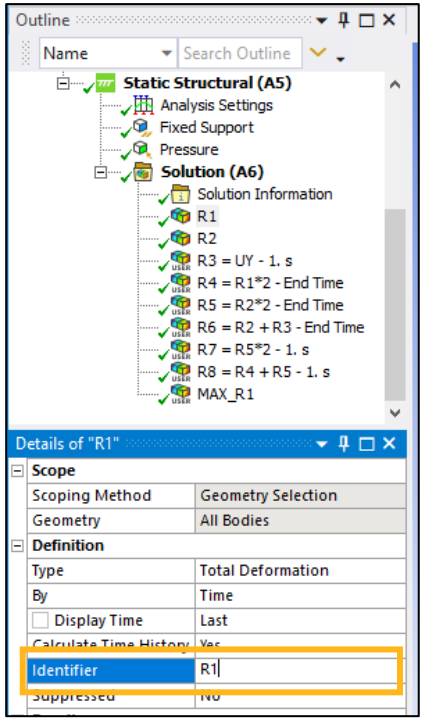
- Using the ResultPreference API, we can control some display properties of result objects



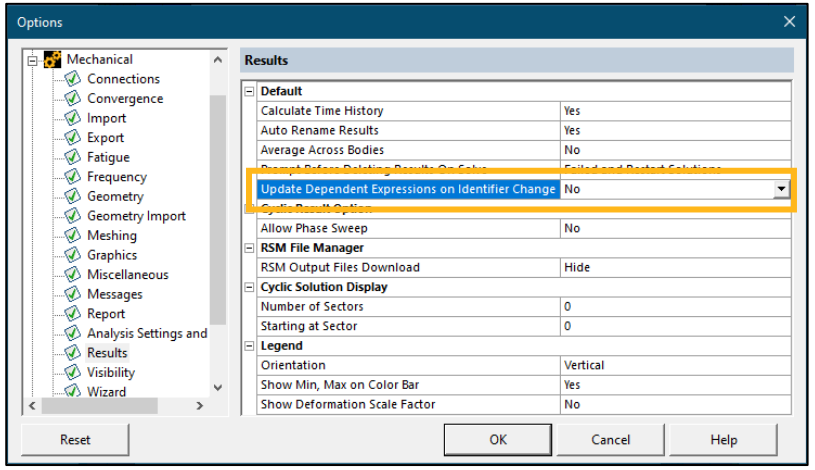

```
# Displays the result minimum annotation label
Graphics.ViewOptions.ResultPreference.ShowMinimum = True

# Sets the geometry view to CappedIsoSurface
Graphics.ViewOptions.ResultPreference.GeometryView = MechanicalEnums.Graphics.GeometryView.CappedIsosurface

# Sets the capping value for CappedIsoSurface view
Graphics.ViewOptions.ResultPreference.IsoSurfaceValue = Quantity('0.55767')

# Sets the contour view to Isolines
Graphics.ViewOptions.ResultPreference.ContourView = MechanicalEnums.Graphics.ContourView.Isolines
```

Result - Identifiers Editing

- Users can now edit Identifiers after a result has been evaluated.
- Changes will automatically update and reevaluate all the expressions where that Identifier was used.



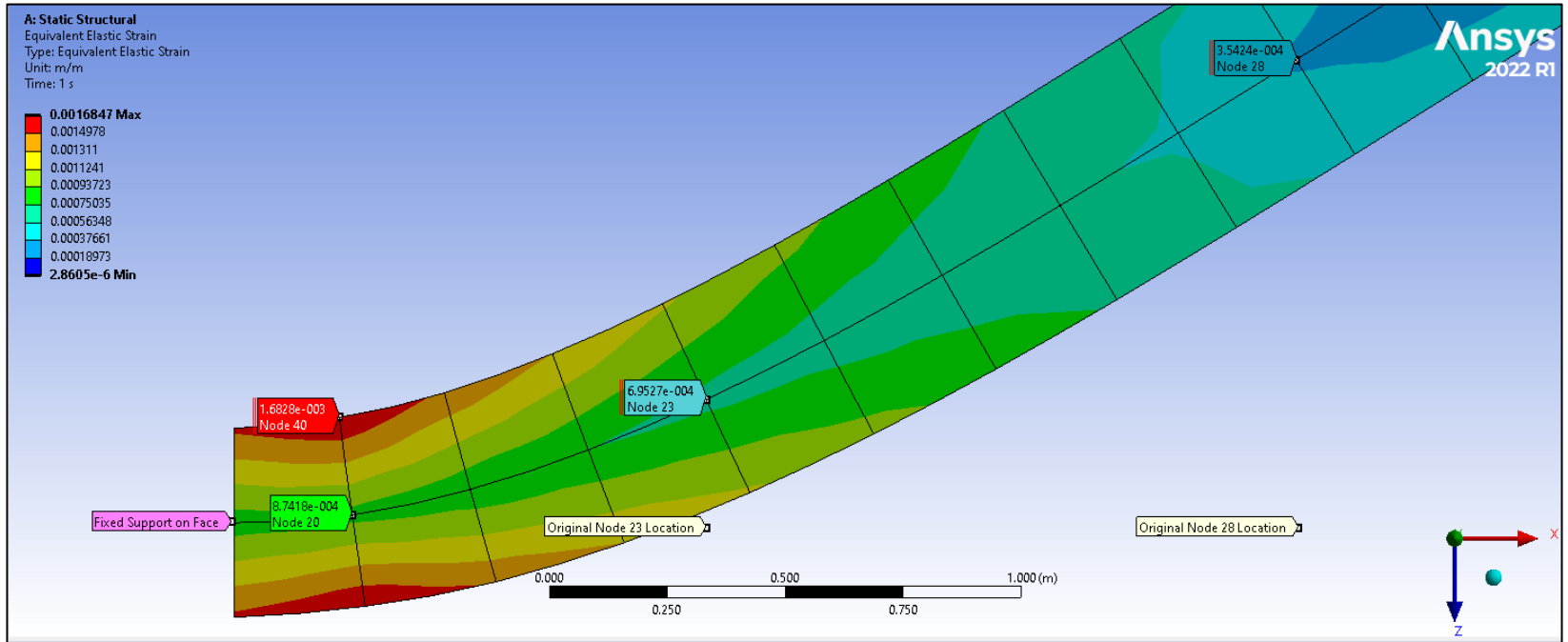
- Changing the Identifier in a result will never invalidate that result. The effect on the dependent results is controlled by the setting:



- *Mechanical → Results → "Update Dependent Expressions on Identifier Change"*

Enhancements to User Annotations and Result Probe Labels

- The new color column allows user annotations and result probe labels to have customized colors
- The new “Show Always” column allows a user annotation to be displayed while on any object in the tree
- Snapping result probe labels now show the snapped-to node’s ID (if any mesh node was hit)

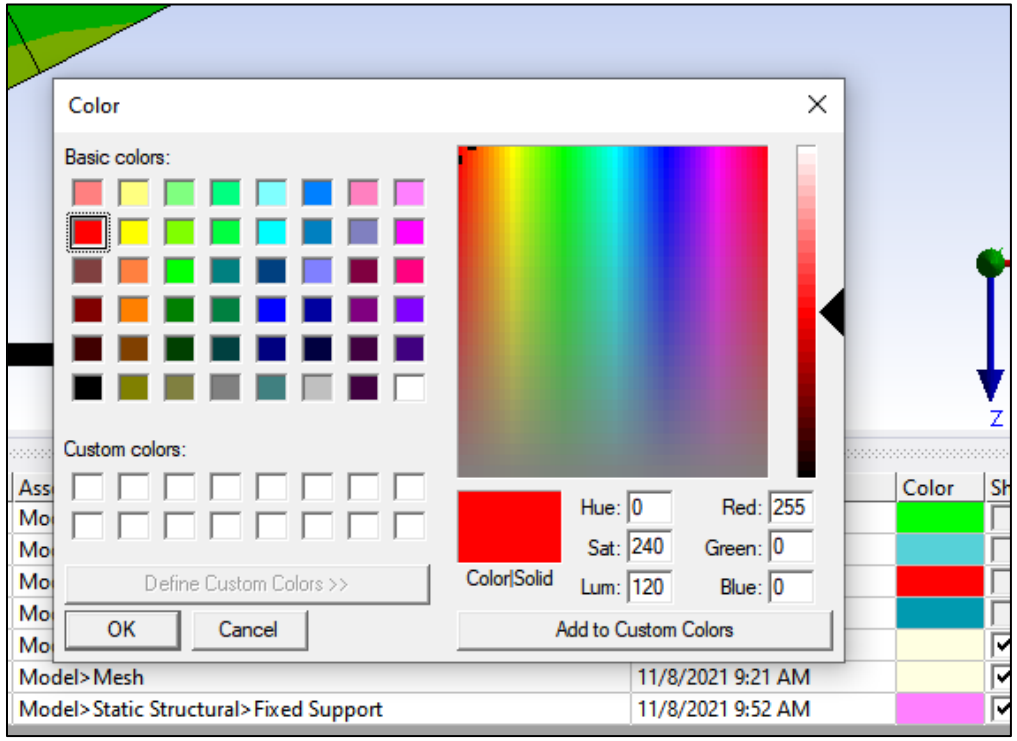


Graphics Annotations

Type	Value	Note	Unit	Location X	Location Y	Location Z	Node ID	Association	Timestamp	Color	Show Always
Result	8.7418e-004		m/m	0.250021	0.010000	0.183721	20	Model>Static Structural>Solution>Equivalent Elastic Strain	11/8/2021 9:15 AM		☐
Result	6.9527e-004		m/m	1.000100	0.010000	-0.060924	23	Model>Static Structural>Solution>Equivalent Elastic Strain	11/8/2021 9:15 AM		☐
Result	1.6828e-003		m/m	0.222725	0.009906	-0.023147	40	Model>Static Structural>Solution>Equivalent Elastic Strain	11/8/2021 9:15 AM		☐
Result	3.5424e-004		m/m	2.250203	0.010000	-0.780051	28	Model>Static Structural>Solution>Equivalent Elastic Strain	11/8/2021 9:15 AM		☐
User Defined	N/A	Original Node 23 Location		1.000581	0.009997	0.209995		Model>Mesh	11/8/2021 9:20 AM		☑
User Defined	N/A	Original Node 28 Location		2.254619	0.009982	0.210182		Model>Mesh	11/8/2021 9:21 AM		☑
User Defined	N/A	Fixed Support on Face		-0.005994	0.010004	0.197427		Model>Static Structural>Fixed Support	11/8/2021 9:52 AM		☑

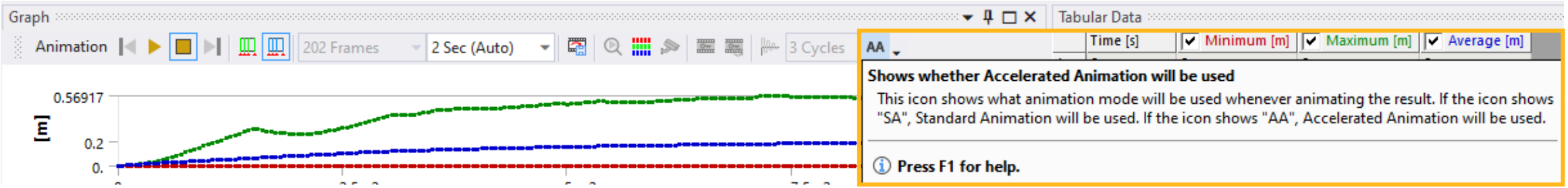
Enhancements to User Annotations and Result Probe Labels

- To change the color of a label, double-click its color cell and select a new color in the pop-up window

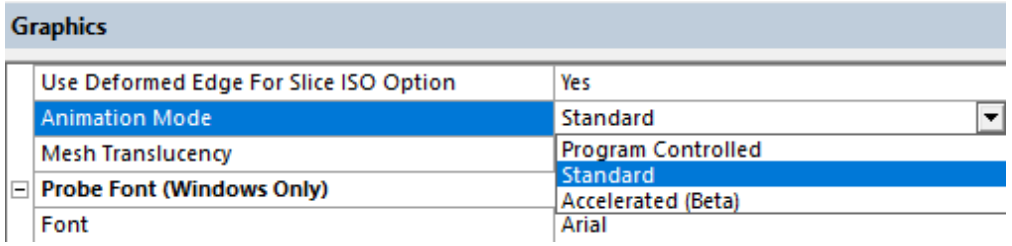


Accelerated Animation

- While not all results, models, or display modes are supported, those that do (most standard structural contour results scoped to bodies displayed with no wireframe, for example) can animate much faster with less memory overhead. Furthermore, it supports full user interaction during animation creation (such as model rotation) rather than locking up the UI as in Standard Animation.

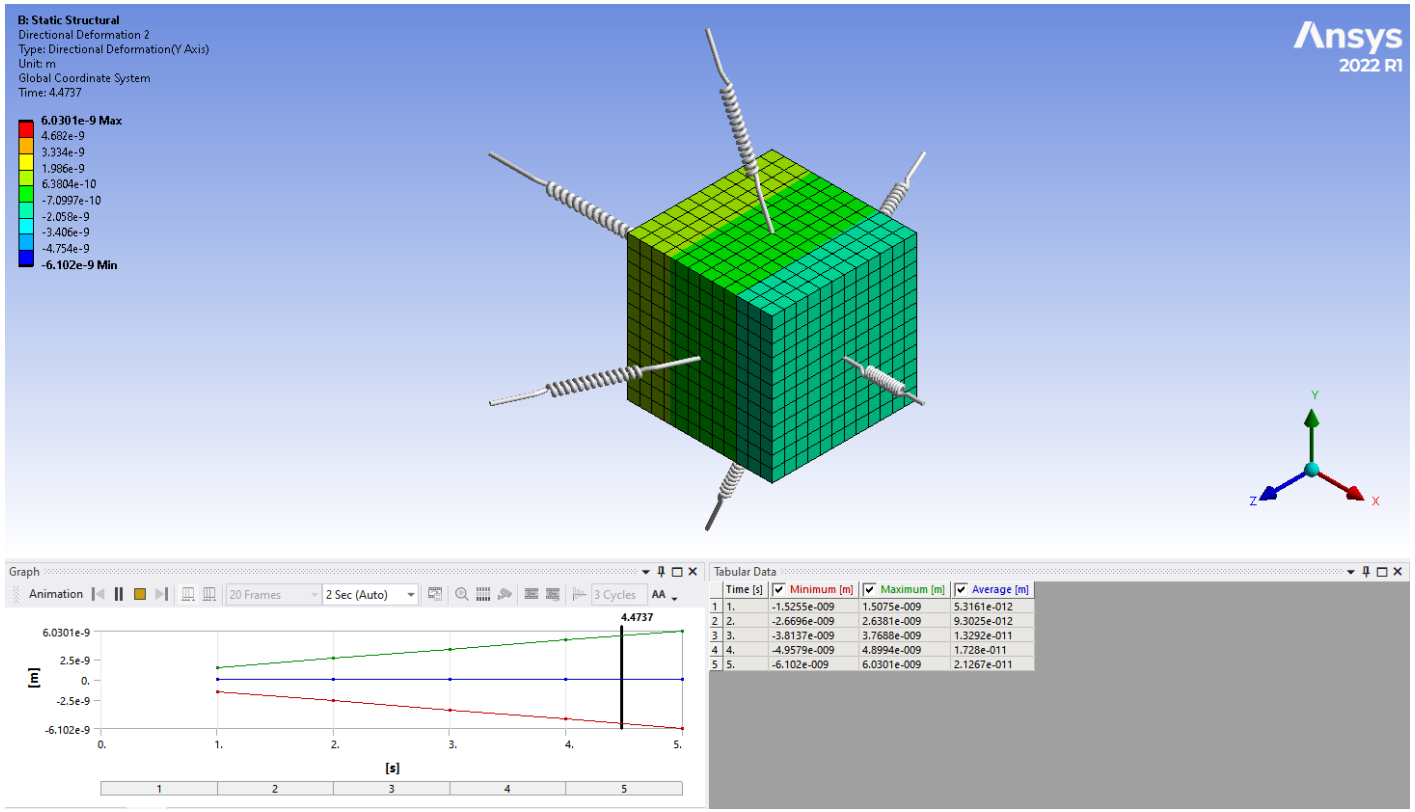


- Can be controlled via the Animation Mode Mechanical Graphics preference



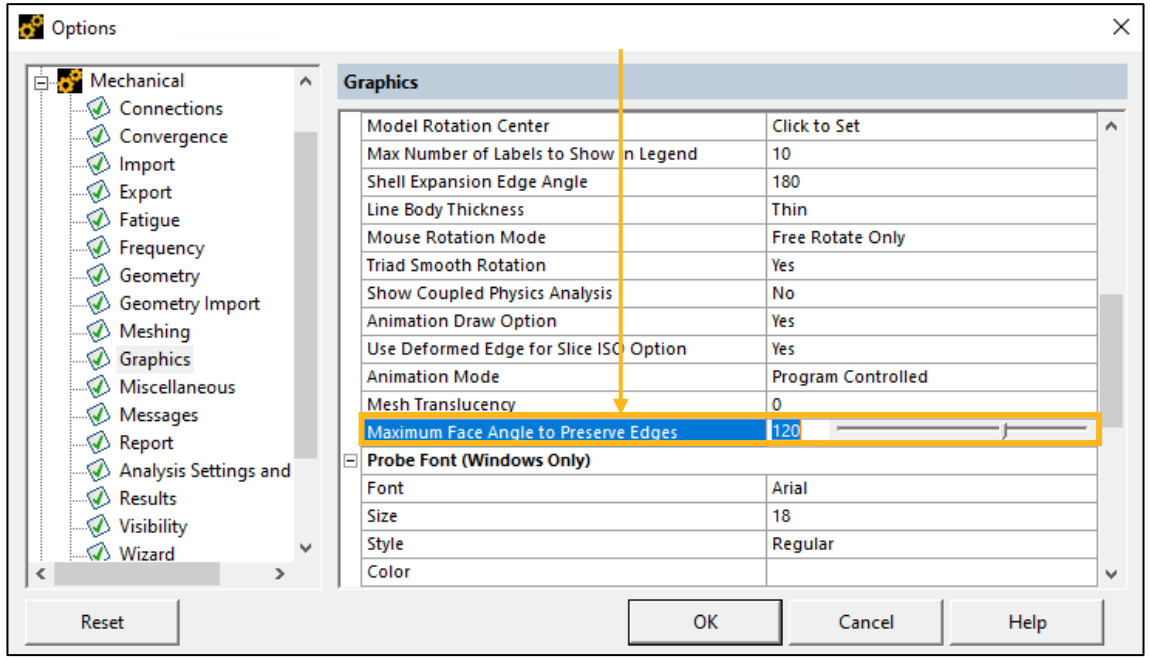
Accelerated Animation

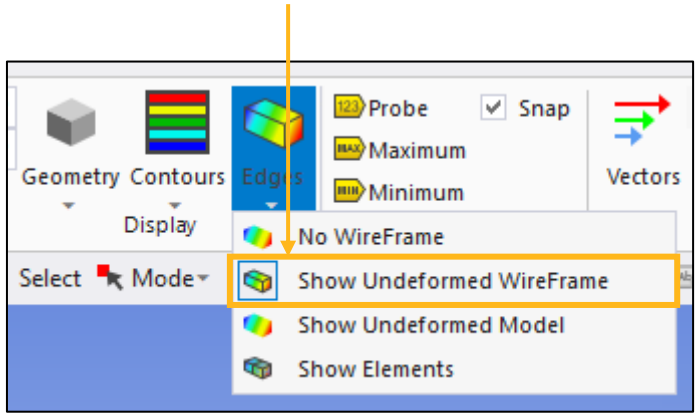
- Accelerated Animation is now the default animator for supported scenarios
- Accelerated Animation now supports rendering mesh element outlines and springs



Face Angle Display Preference

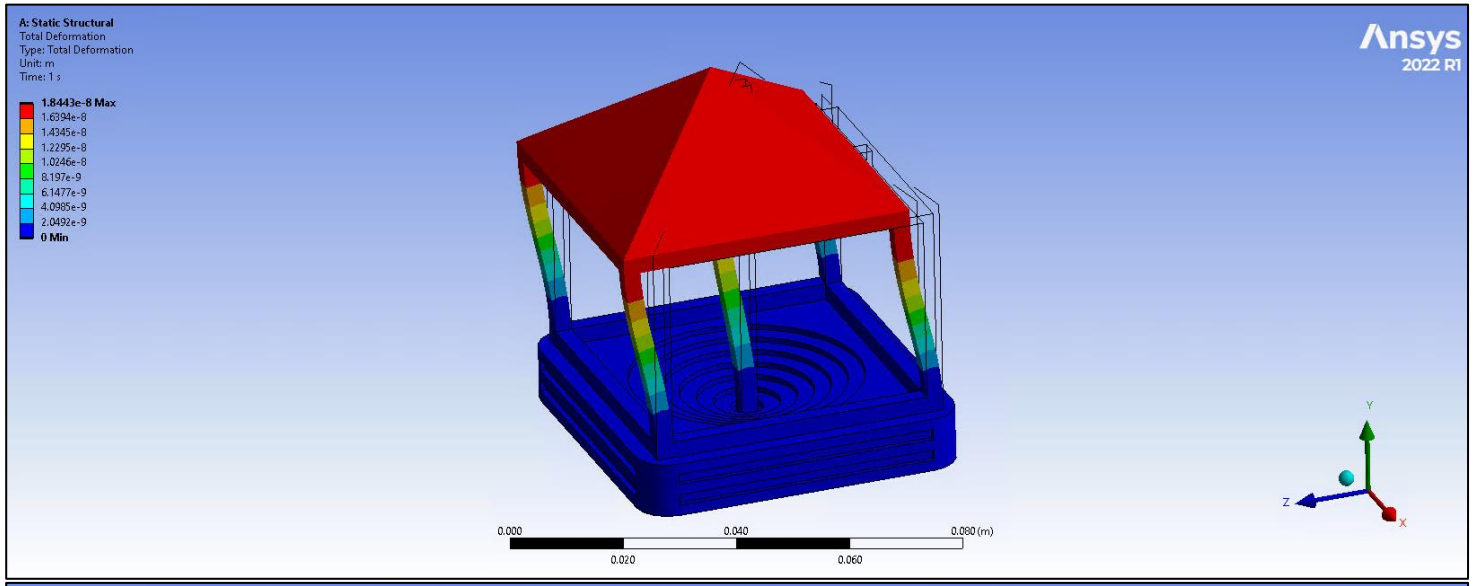
- Provide display preference to change the maximum face angle in order to better preserve the edges between faces when showing undeformed wireframe on results
- Default is 120 degrees and range is from 0 to 180 degrees



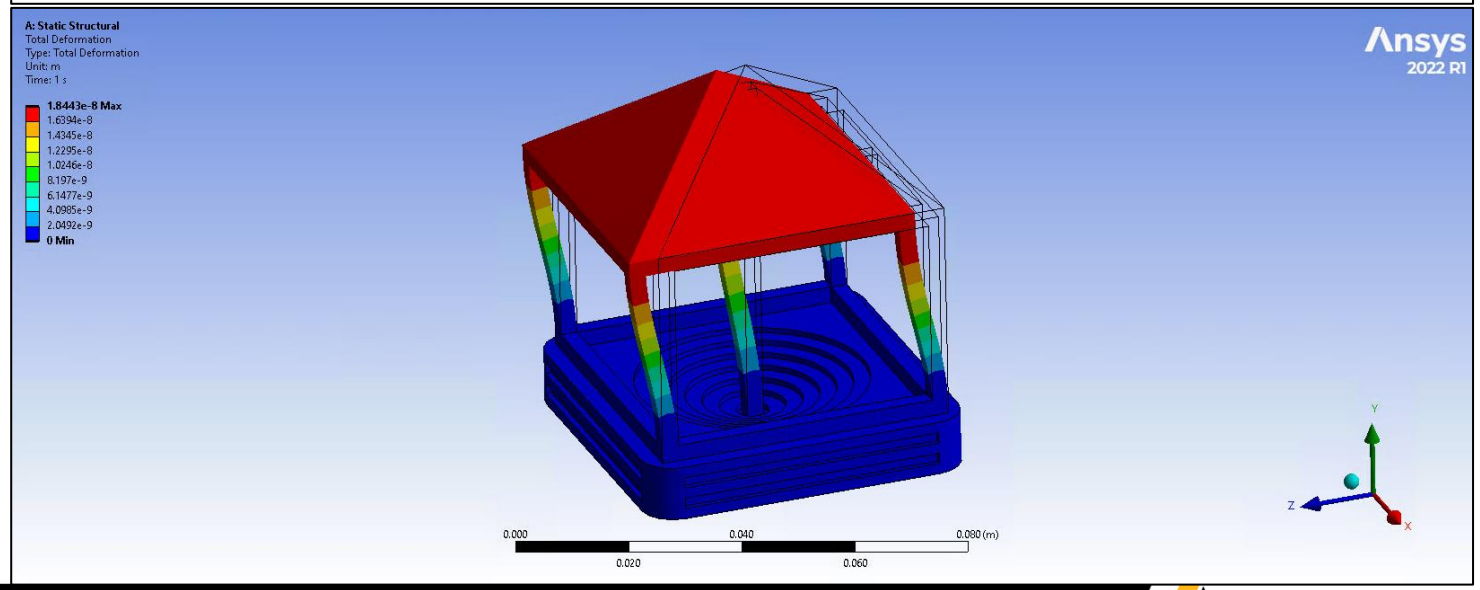


Face Angle Display Preference

Default undeformed wireframe on result shows missing edges due to insufficient maximum face angle of 120 degrees



Undeformed wireframe display is fixed with a maximum face angle of 130 degrees



Composites

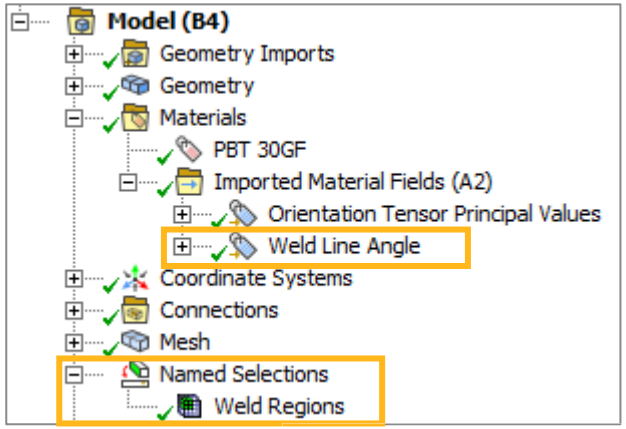
Short Fiber Workflow

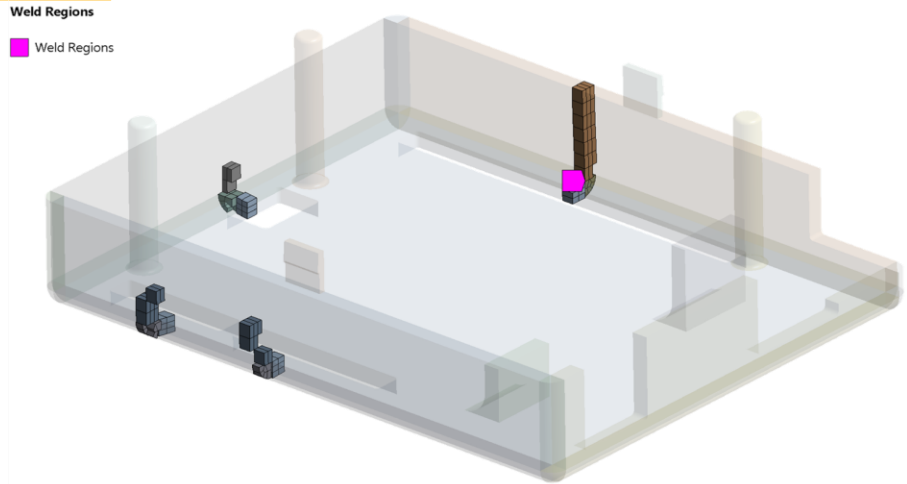
Ansys

Import of Weld Lines

- With the Injection Molding Data system, you can now import Weld Lines results from Moldex3D® and Moldflow®
- You can map Weld Lines as element-based Named Selections in Mechanical. You can use them for visualization and post-processing purposes, as well as to assign degraded material properties
- The Weld Line Angle is imported and mapped as a Material Field

17	Injection Molding Simulation Data	
18	Mesh File	D:\casing.cdb
19	Fiber Orientation Tensor File	D:\casing.o2d
20	Initial Stress File	
21	Fiber Volume Fraction File	
22	Weld Lines File	D:\casing.nwd

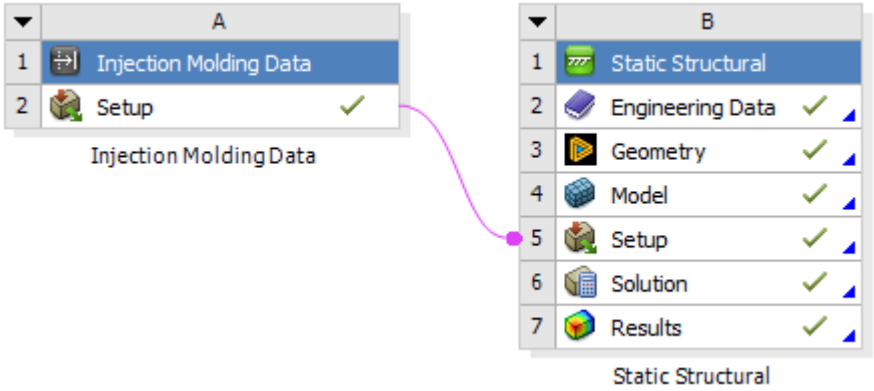




Injection Molding Data: Support for Unreinforced Materials

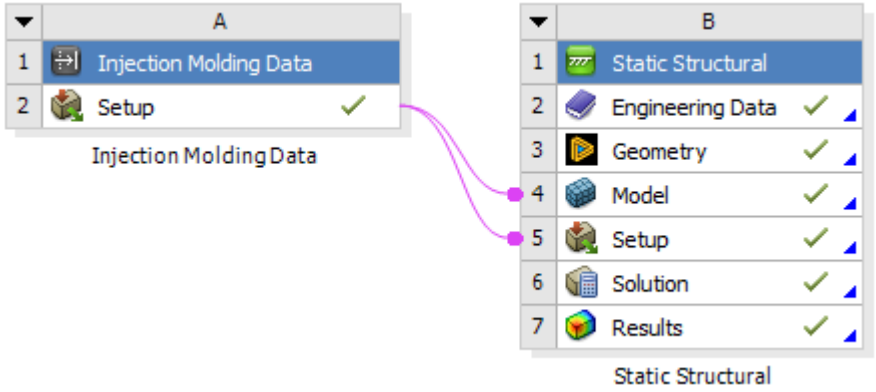
- Import Initial Stress and Weld Line results independently of the fiber orientation tensor. This is especially useful if you are dealing with unreinforced materials

Import Initial Stress



Injection Molding Simulation Data	
Mesh File	D:\casing.cdb
Fiber Orientation Tensor File	
Initial Stress File	D:\casing.ist
Fiber Volume Fraction File	
Weld Lines File	

Import Initial Stress and Weld Lines



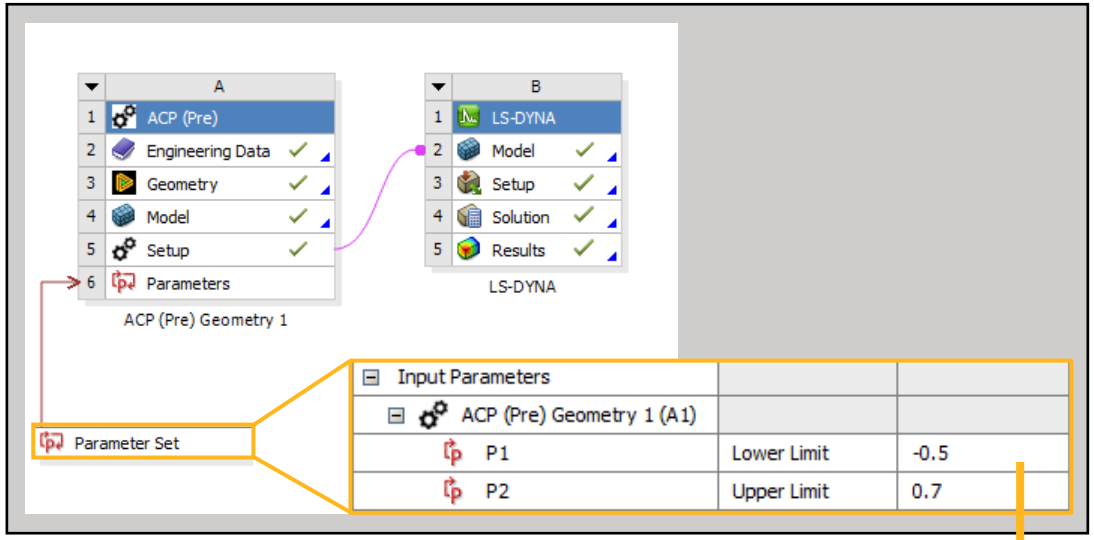
Injection Molding Simulation Data	
Mesh File	D:\casing.cdb
Fiber Orientation Tensor File	
Initial Stress File	D:\casing.ist
Fiber Volume Fraction File	
Weld Lines File	D:\casing.nwd

Ansys Composite PrepPost

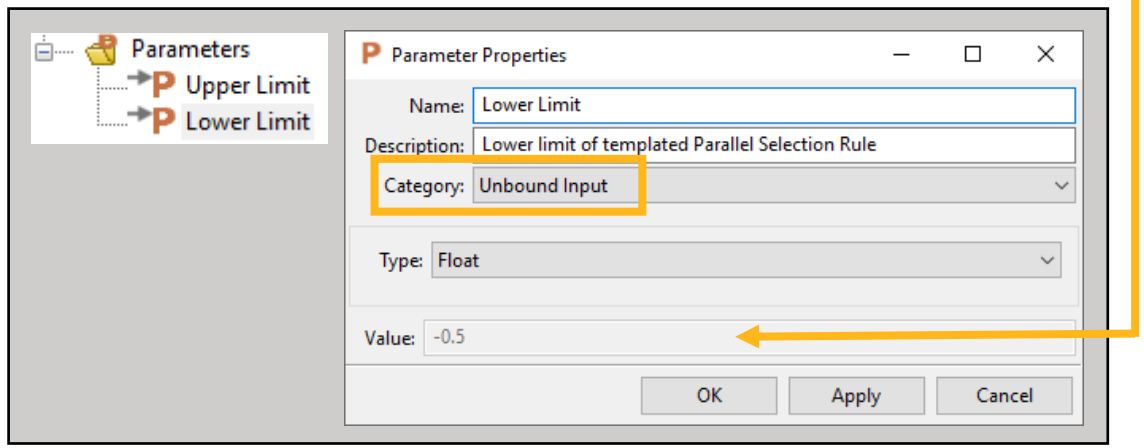
Unbound Input Parameter

- New **Parameter** type **Unbound Input** which is not linked to an ACP object
- It can be used for any parameterization or customization
- The main use of this parameter type is in combination with the new Python Script object (see next slide)
- Supported by Ansys Workbench

Ansys Workbench

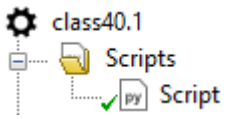


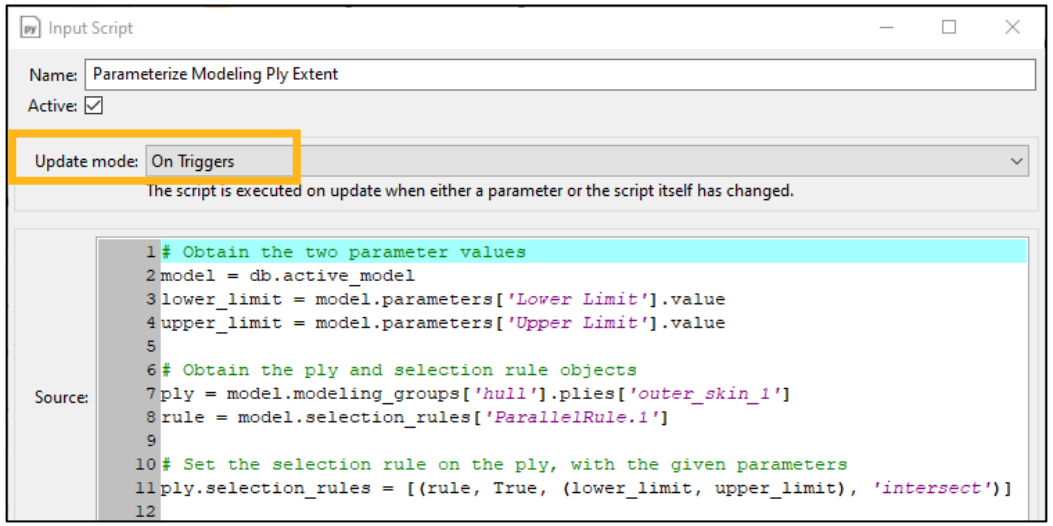
ACP



New Python Script Object

- The **Script Object** serves as a command snippet. It allows the execution of arbitrary Python code.





- The **Update Mode** defines when the code is run:
  a) Either as part of the update logic of ACP (**on triggers**)
  b) or before each update (**always**)
  c) or if manually triggered (**manual**)

- Use cases:
  - Complex parametrization
  - Customization and automatization of the lay-up modelling in ACP Pre
  - Implementation of custom post-processing features in ACP Post

How to Parametrize Ply Coverage

(independent of the CAD geometry)

Parameter Set

Input Parameters		
ACP (Pre) Geometry 1 (A1)		
P1	Lower Limit	-0.5
P2	Upper Limit	0.7

2.

Parameters
- Upper Limit
- Lower Limit

Create Parameter ... 1.

Sort

Parameter Properties

Name: Lower Limit

Description: Lower limit of templated Parallel Selection Rule

Category: Unbound Input

Type: Float

Value: -0.5

OK | Apply | Cancel

ACP Model
- Scripts
  - Ply Extent

Update mode: On Triggers

The script is executed on update when either a parameter or the script itself has changed.

Source:

```
# Obtain the two parameter values
model = db.active_model
lower_limit = model.parameters['Lower Limit'].value
upper_limit = model.parameters['Upper Limit'].value

# Obtain the ply and selection rule objects
ply = model.modeling_groups['hull'].plies['outer_skin_1']
rule = model.selection_rules['ParallelRule.1']

# Set the selection rule on the ply, with the given parameters
ply.selection_rules = [(rule, True, (lower_limit, upper_limit), 'intersect')]
```

3.

4.

Outcome

Modeling Groups
- hull
  - outer_skin_1

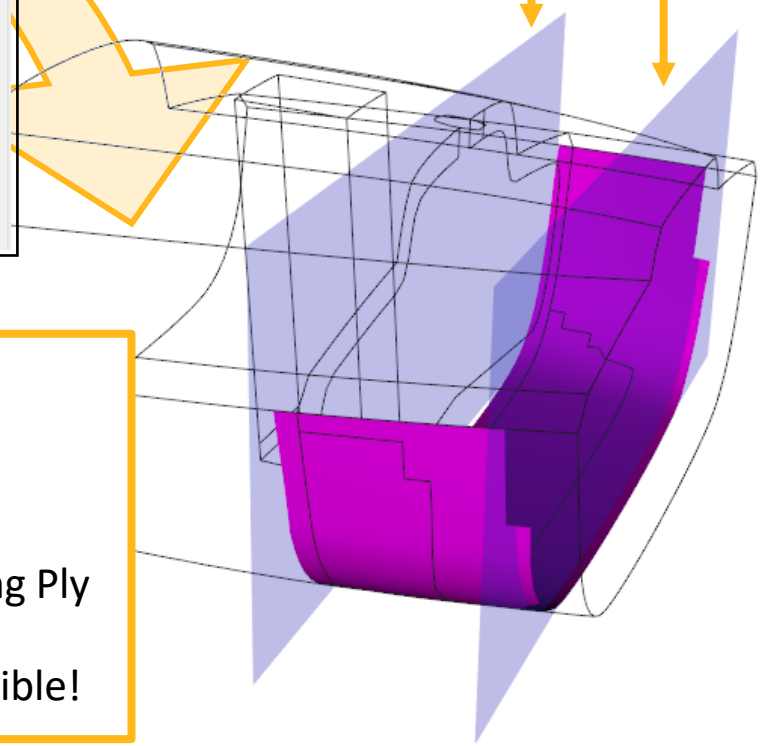
Modeling Ply Properties

Name: outer_skin_1

ID: outer_skin_1

General | Draping | Rules | Thickness

Selection rule	Type	Template	Parameter 1	Parameter 2
ParallelRule.1	Intersect	True	-0.5	0.7



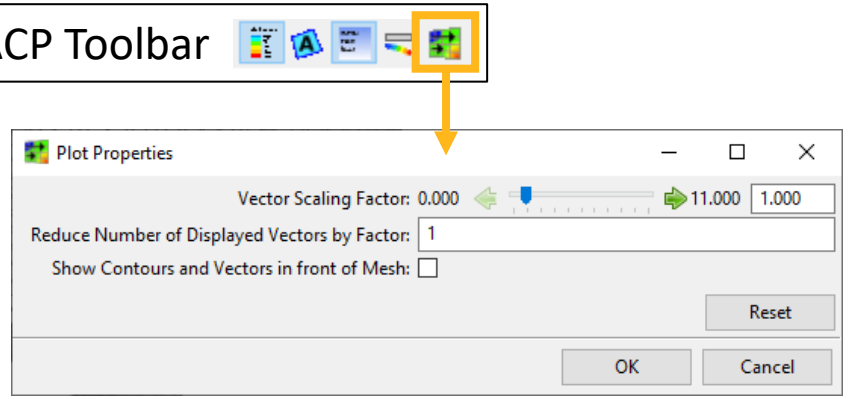
1. Create **Unbound Parameters** in ACP
2. Set values in WB
3. Retrieve them in a **Python Script**
4. Change the template Rule of the Modeling Ply

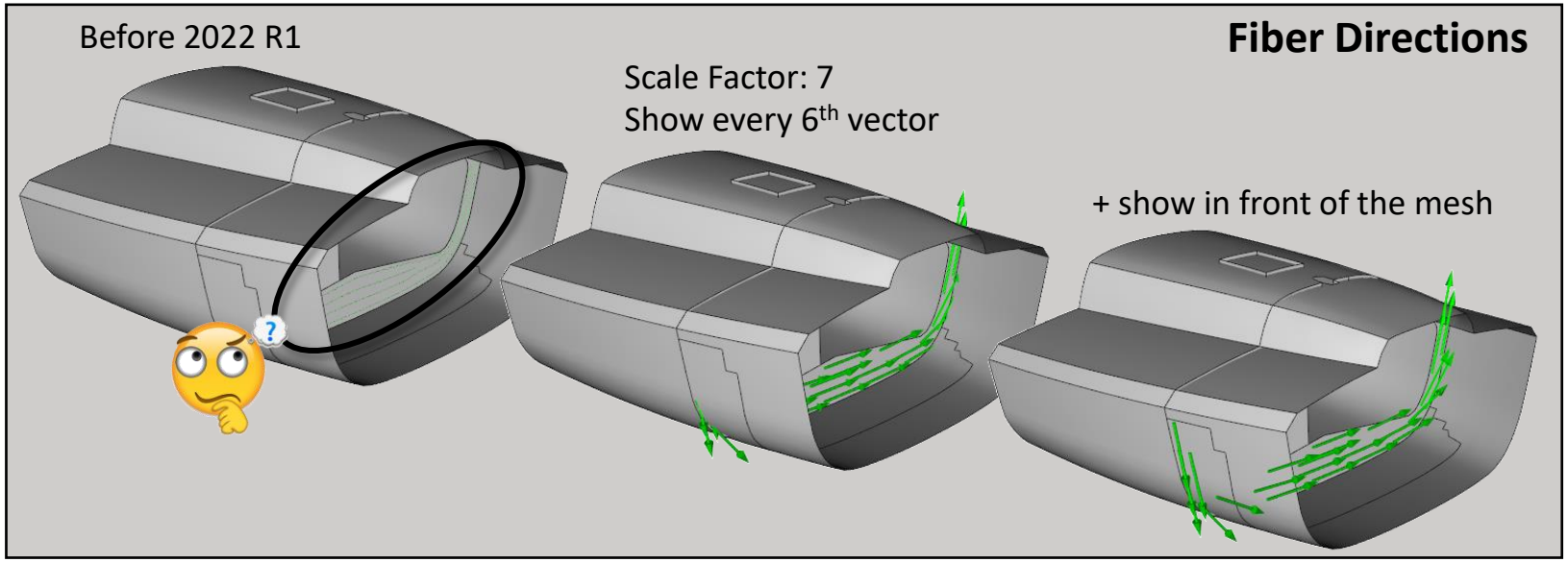
→ Parametrization of template rules now possible!

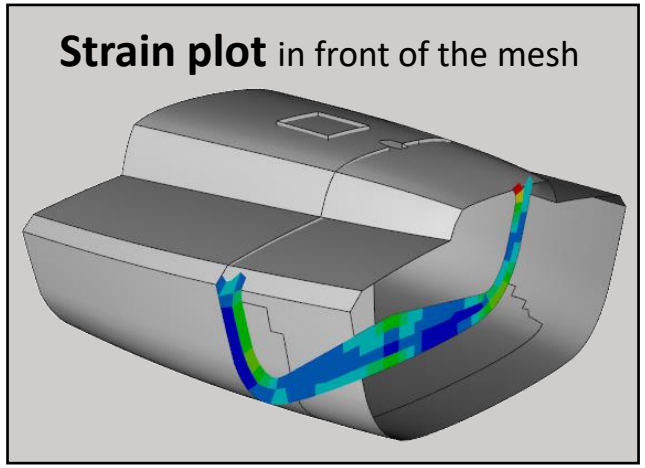
Plot Visualization

- New options to customize all direction plots
  - Scale the vectors
  - Set number of displayed vectors
- And show a plot in front or behind of the mesh!

ACP Toolbar

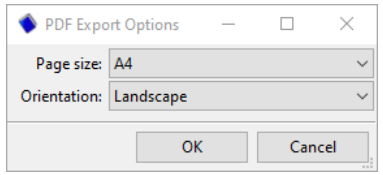




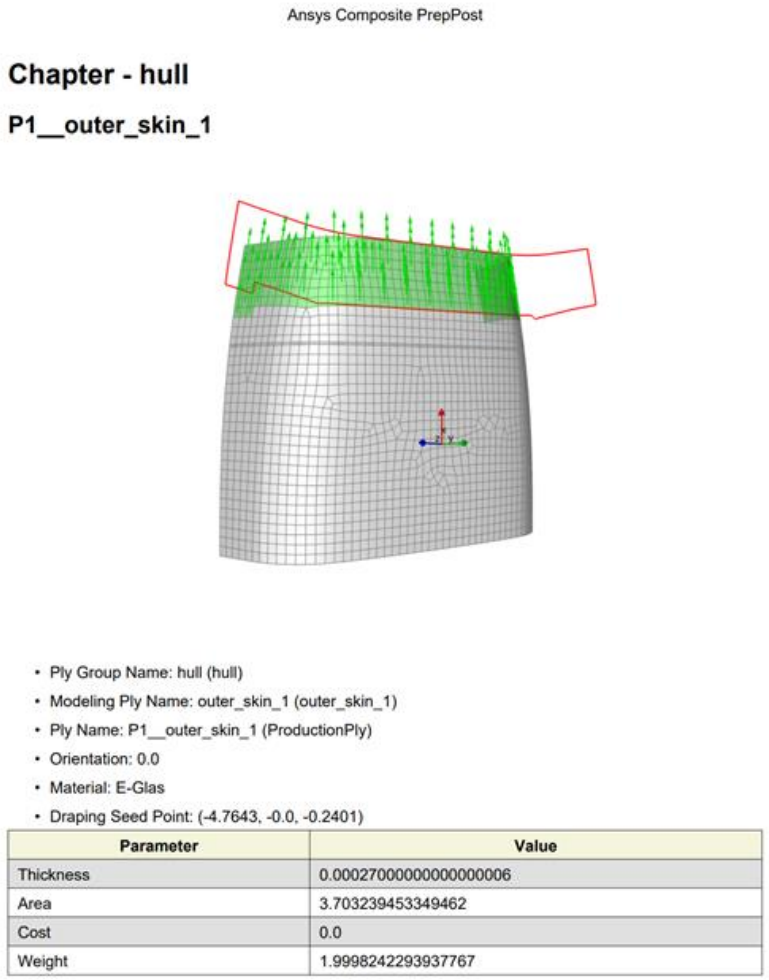


Ply Book

- The Ply Book allows you to create a report for production
- It is improved in various areas
  - Customization
  - New automatic visualization of Rosettes (reference systems)
  - New live preview
  - New PDF export options



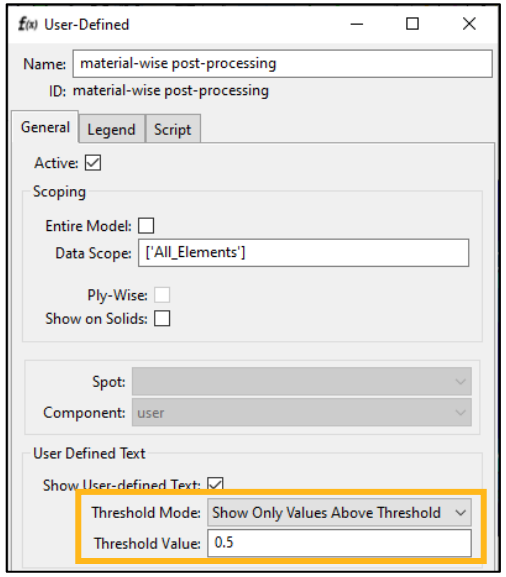
- The new plot visualization also improves the readability of the ply book

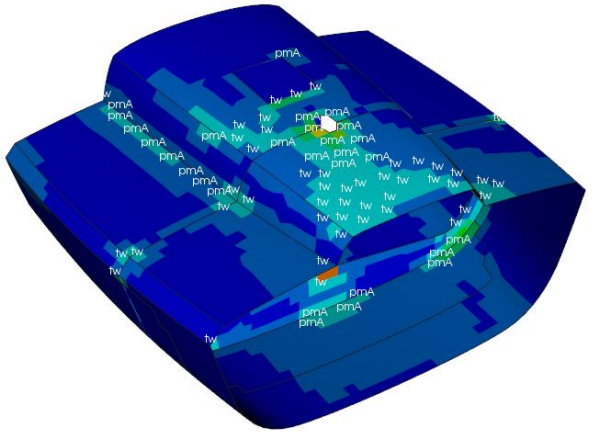


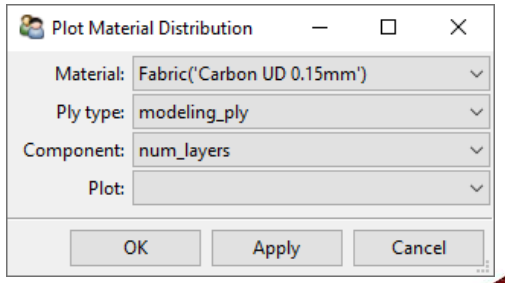
Other Improvements

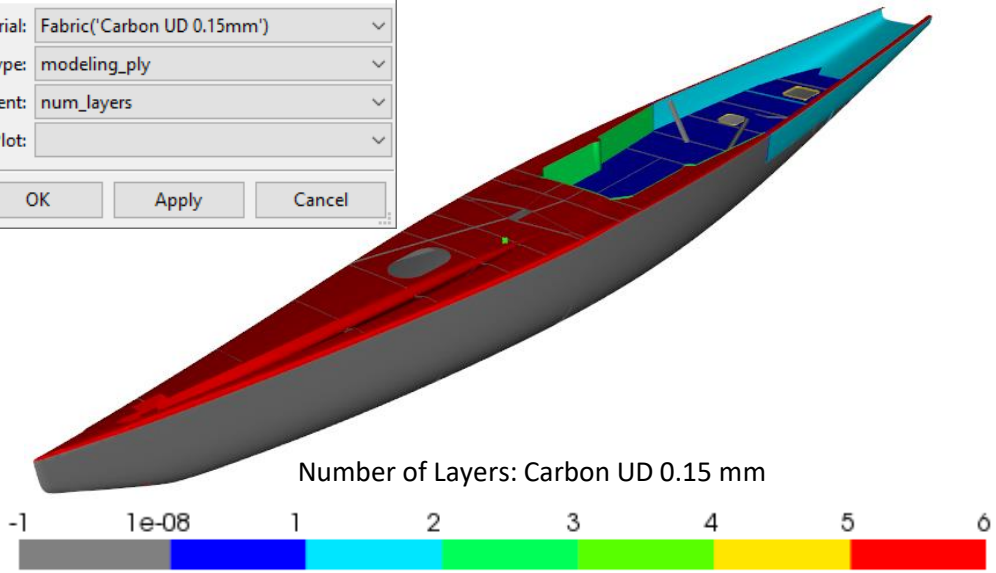
- User-Defined Plots
  - Threshold for elemental text labels to hide labels above or below the value

- Python Scripts and Wizards for customers
  - New examples to get started with scripting
  - Wizard to plot material distributions (see example)
  - Wizard to compute material-wise failure









Structural Optimization

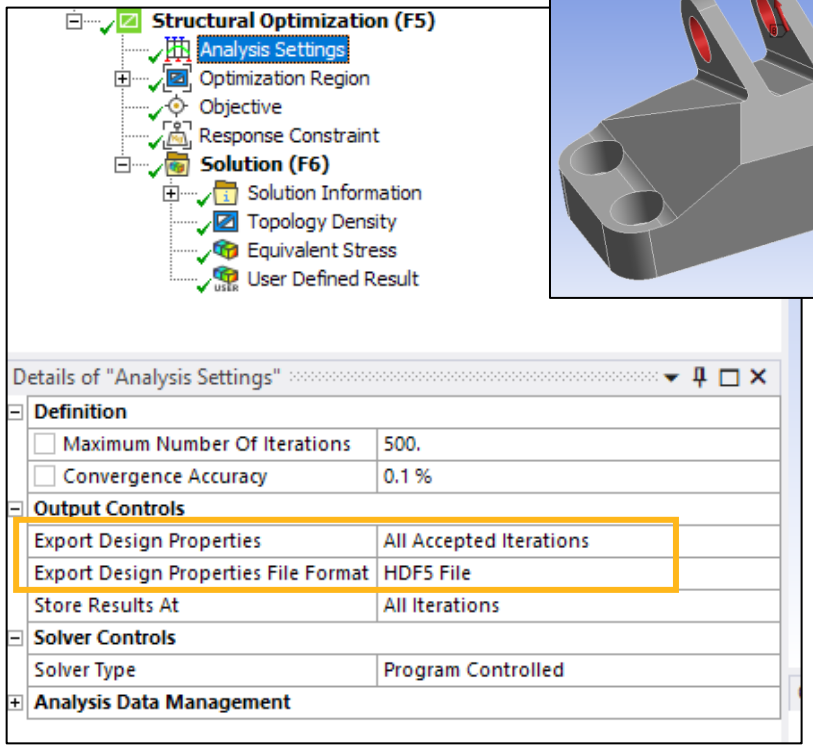
Export Design Properties

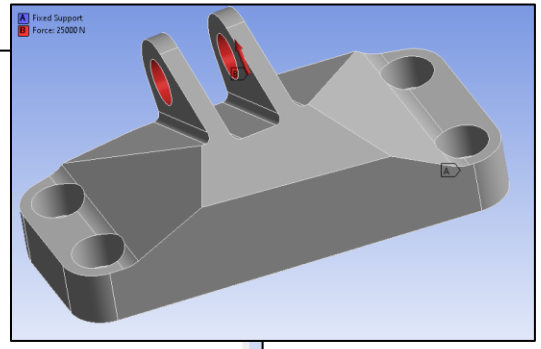
This new feature permits users to visualize the results (deformation, stress, eigen-mode, etc) on the final design, giving them the opportunity to quickly examine and validate the mechanical behavior.

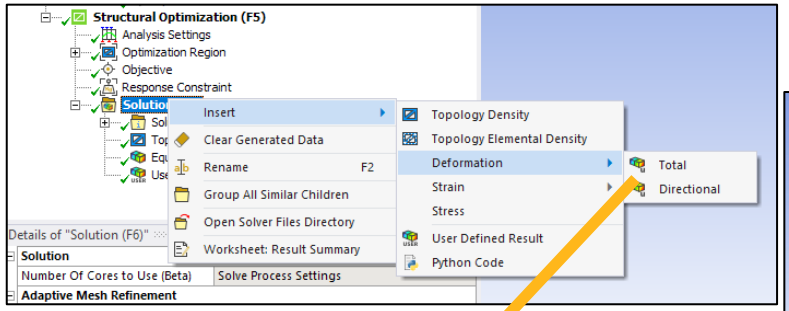
(1) Optimization set-up: Activate the option

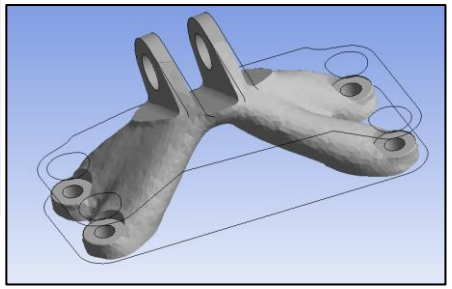
(2) Run the optimization

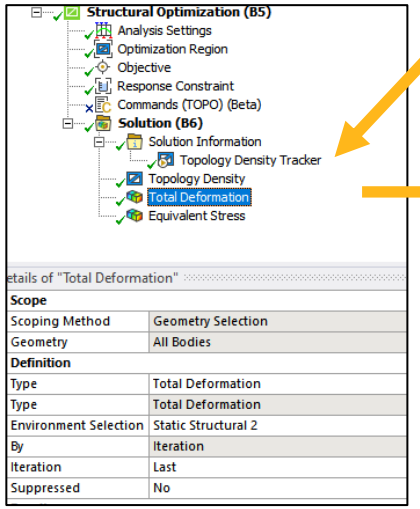
(3) Post: Define your display

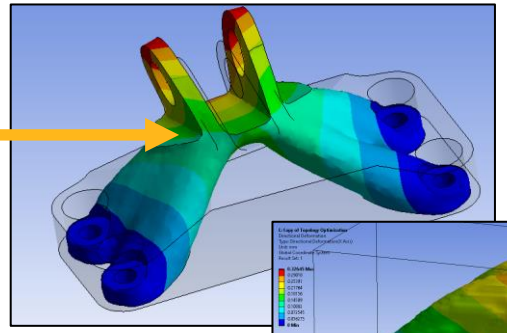


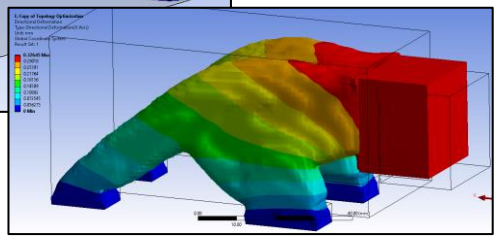




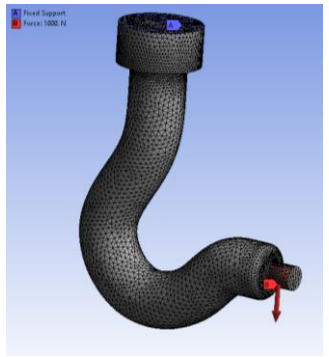


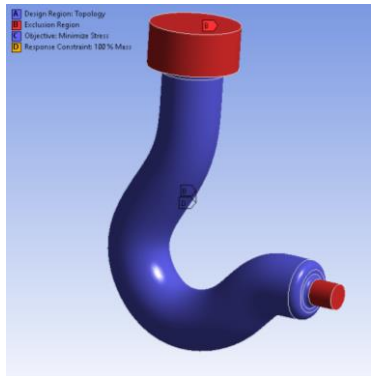


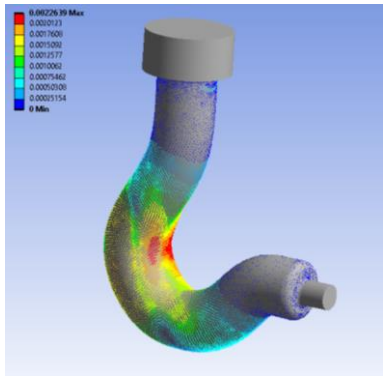




Maximum Principal Stress



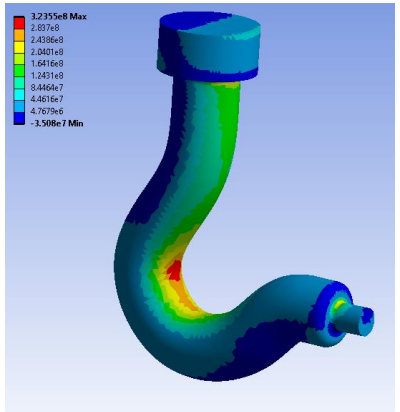




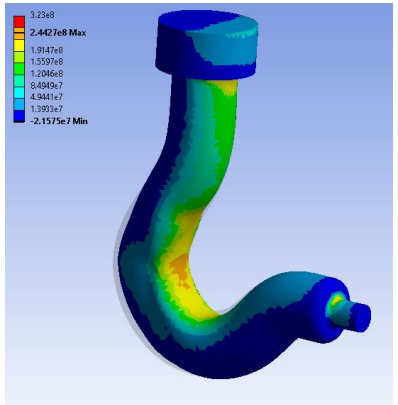
Mesh & Structural Set-up

Shape Optimization Set-up

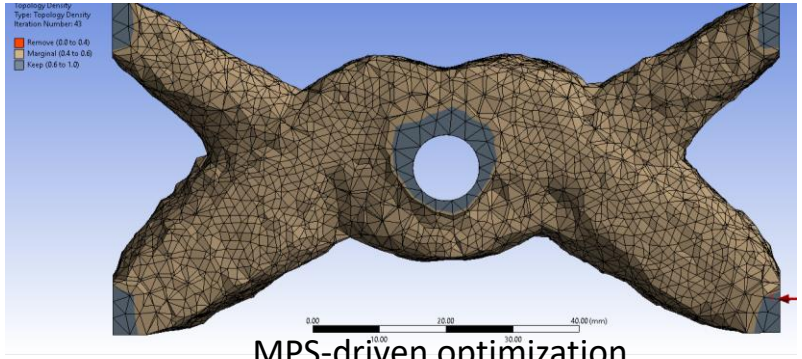
Results & direct validation



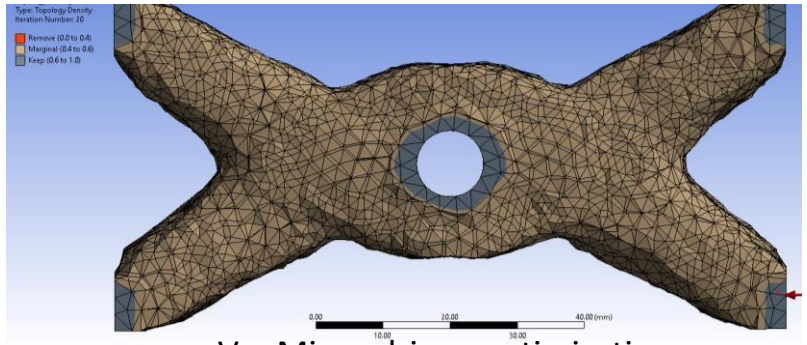
MPS
– 25 %



- ✓ 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



MPS-driven optimization



VonMises-driven optimization

User Defined Criterion

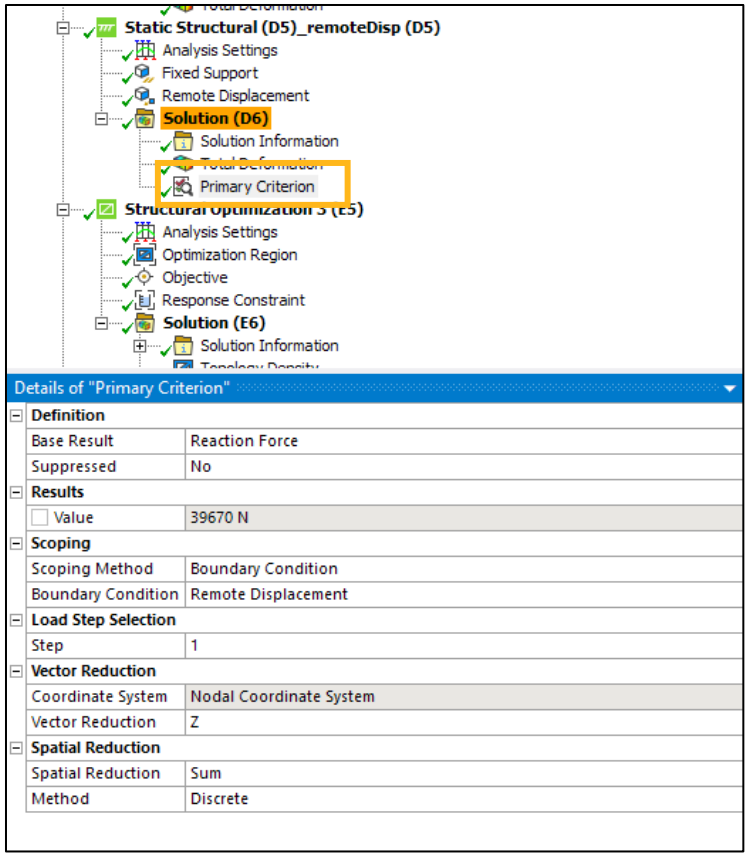
Since 2021 R1, a new capability has been introduced in a bid to create user-defined criterion in the upstream Static Linear Analysis system. This criterion (primary/composite) can then be consumed as objective or constraint in the downstream Structural Optimization

This feature has been extended:

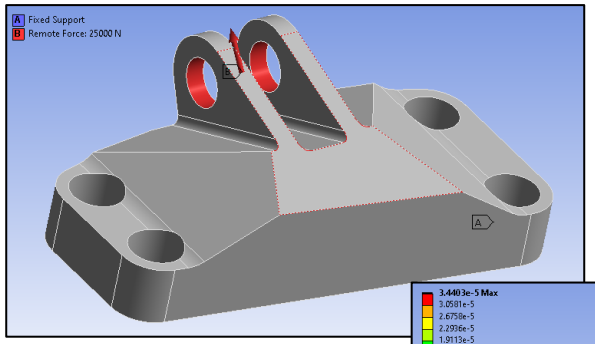
- New base result: displacement, **rotation**, reaction-force or **reaction-moment**
- New scoping: **remote point** or support of boundary condition (eg **remote-force, moment, remote-displacement**)

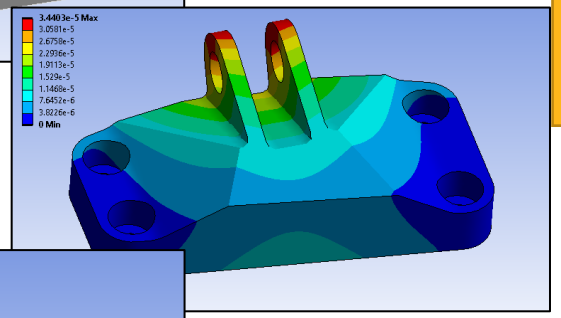
UDC has also been introduced in **Modal-Analysis**:

- **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



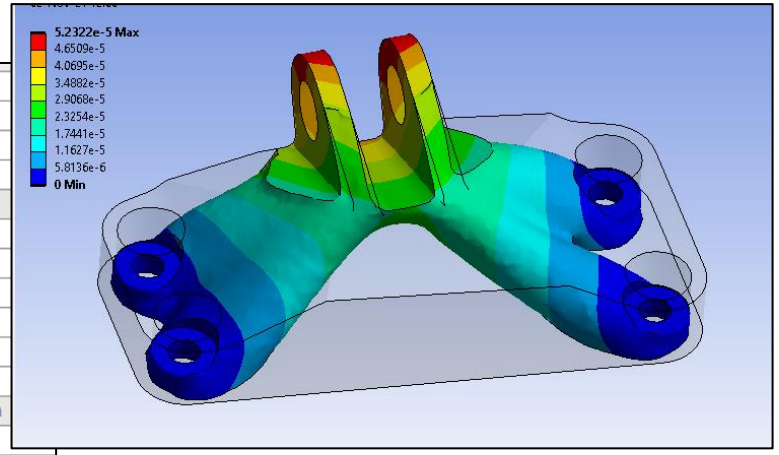
Example #1-2



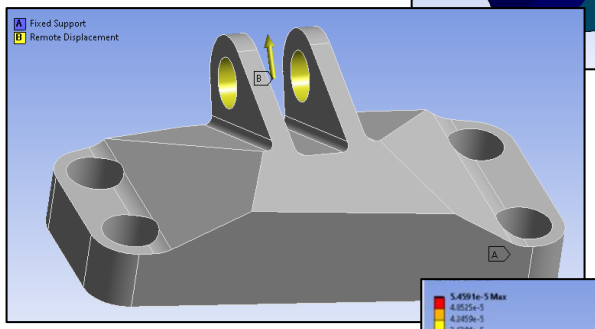


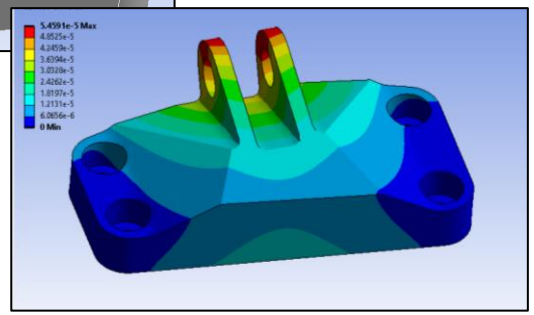
$$\begin{cases} \min\limits_{\Omega}\left(\textit{primary criterion}\right) \\ vol < 40\% \end{cases}$$

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
Constant	
Value	0.
Vector Reduction	Z
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete



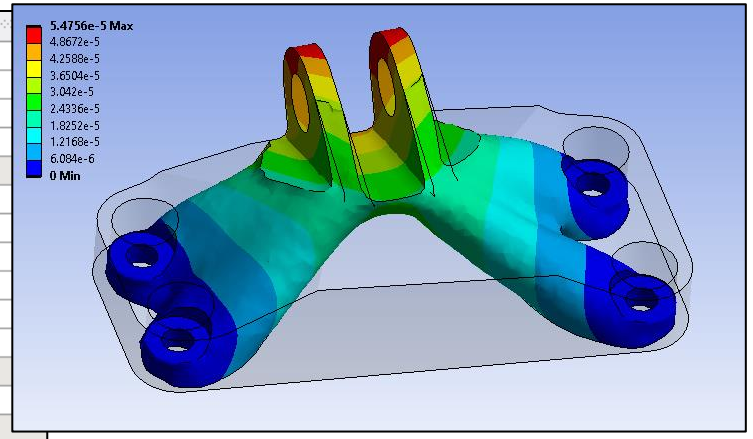
z-disp of the remote-force support





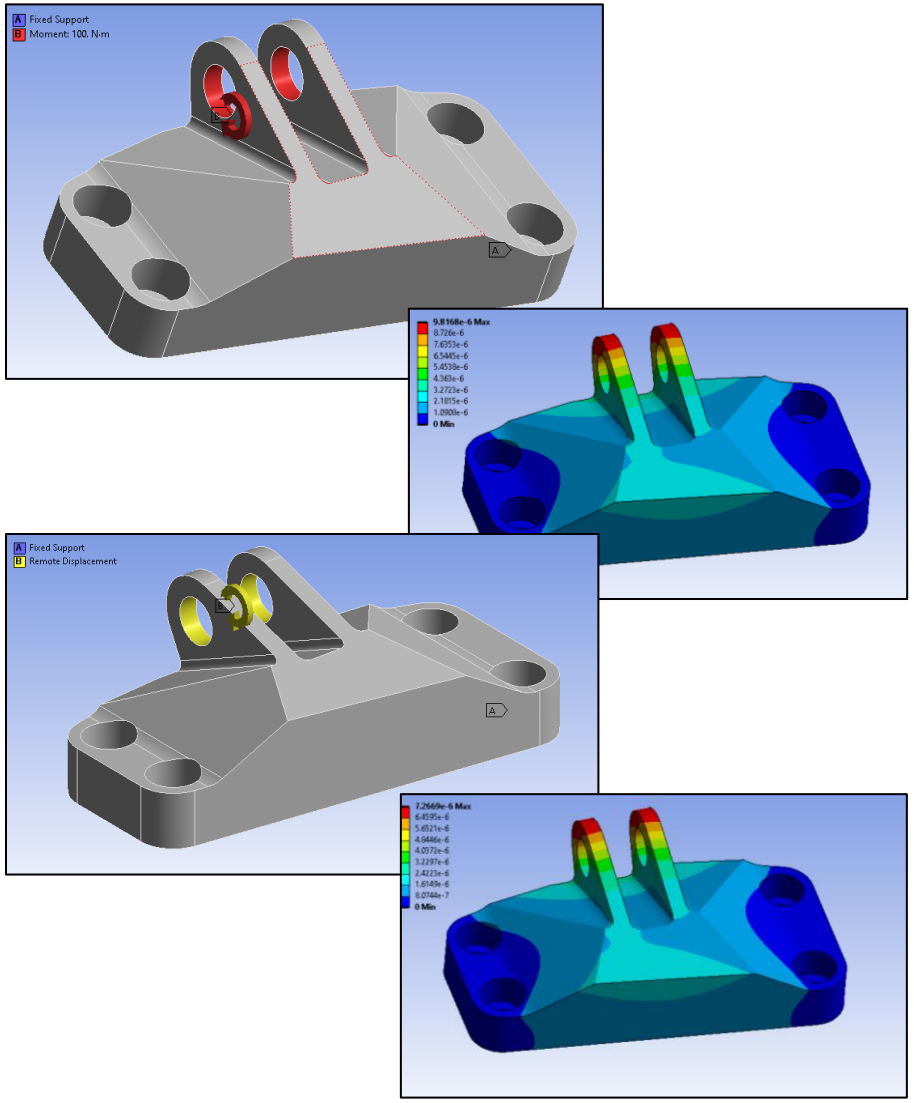
$$\begin{cases} \max\limits_{\Omega}\left(\textit{primary criterion}\right) \\ vol < 40\% \end{cases}$$

Definition	
Base Result	Reaction Force
Suppressed	No
Results	
Value	39670 N
Scoping	
Scoping Method	Boundary Condition
Boundary Condition	Remote Displacement
Selection	
	1
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

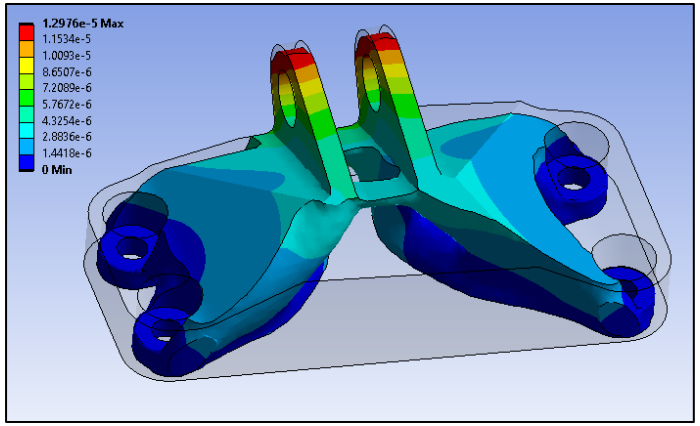
Example #3-4



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

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	
[illegible]	1
[illegible] Reduction	
[illegible] Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete

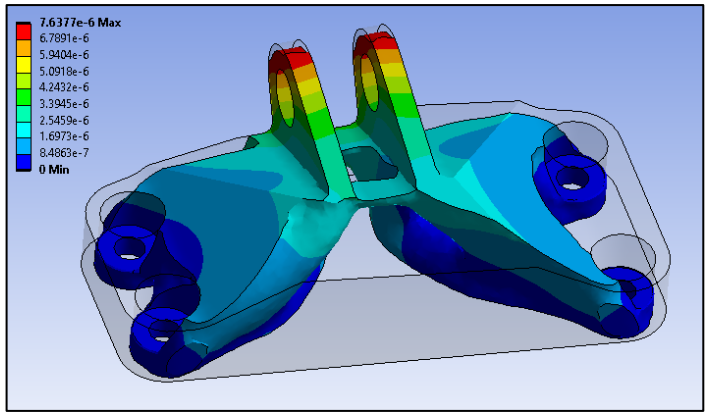


x-rotation of the remote-moment support

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

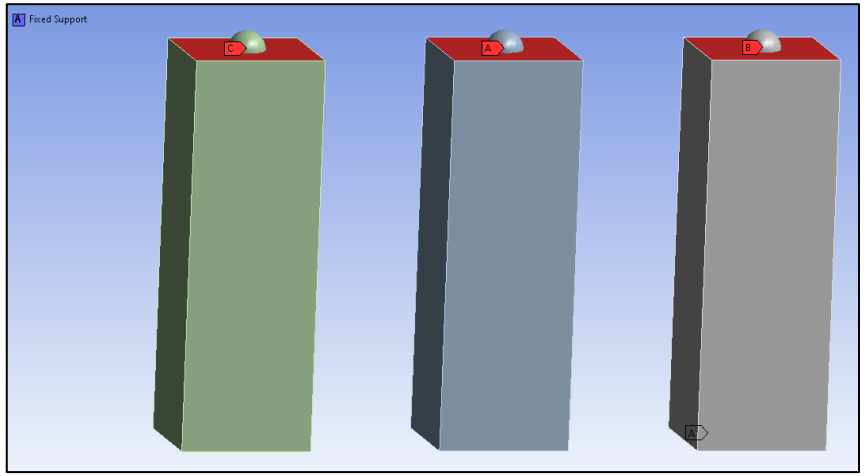
Details of "Primary Criterion"

Definition	
Base Result	Reaction Moment
Suppressed	No
Results	
Value	-74.025 N·m
Scoping	
[illegible] Method	Boundary Condition
[illegible] Condition	Remote Displacement
[illegible] Selection	
[illegible]	1
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Sum
Method	Discrete

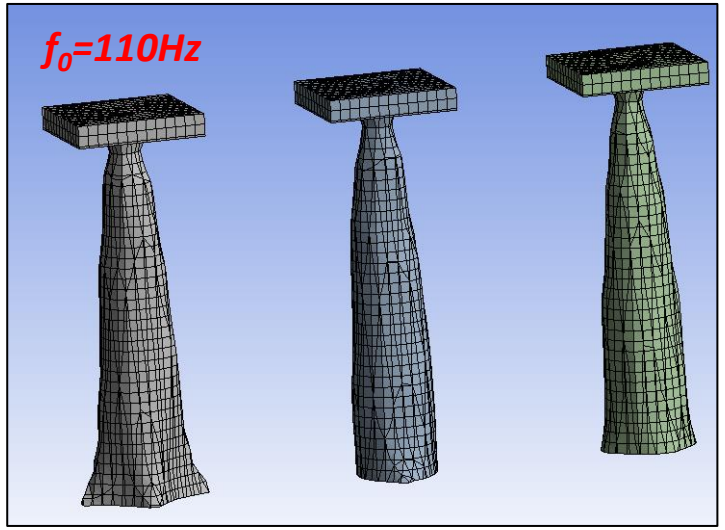


x-reaction-moment
at the remote-displacement support

UDC Modal: Robust Frequency Example



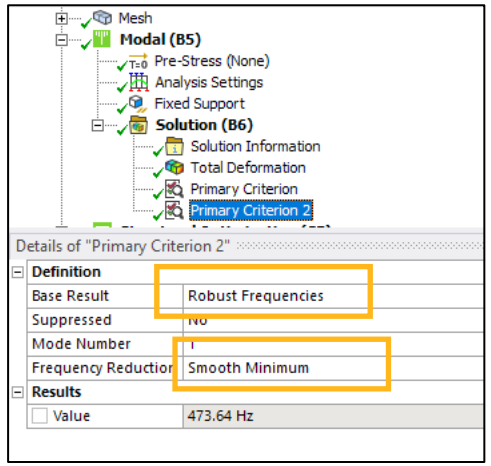
$$optim\#1 \quad \begin{cases} \max\limits_{\Omega} f_0 & [\text{single freq}] \\ mass \le 25\% \end{cases}$$

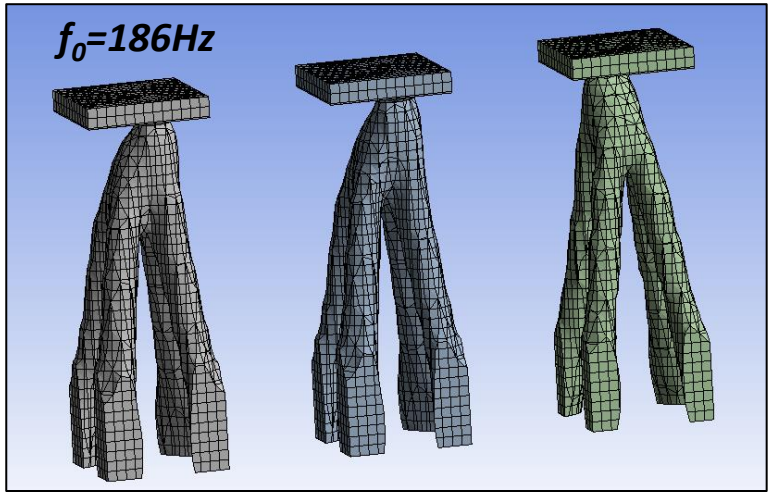


+70% better

Better performance thanks to the robust-frequency criterion

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





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

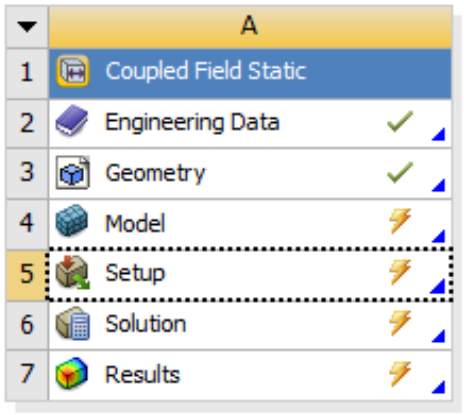
Notes

- 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

Linear Dynamics

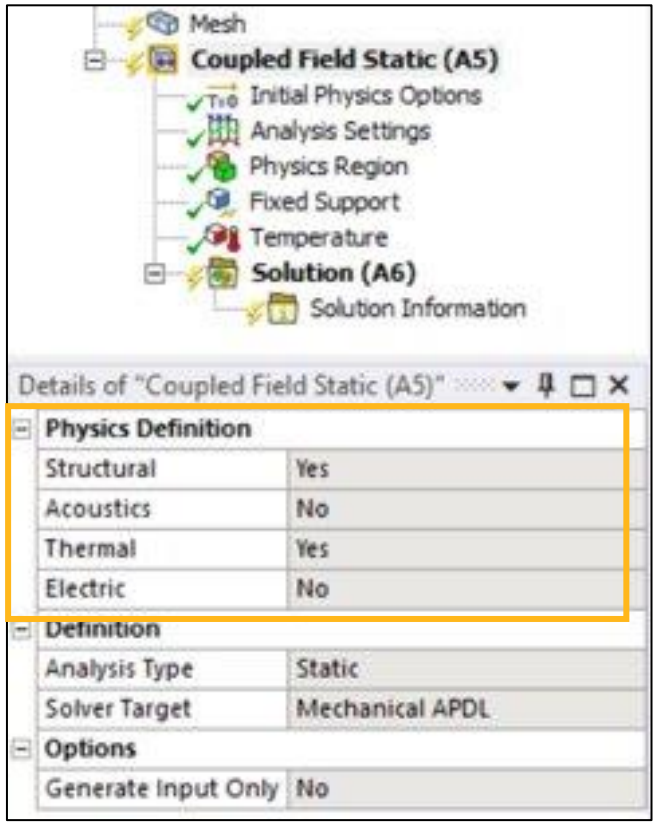
Coupled Field Static Analysis

- Piezoelectric coupling and Acoustics physics is supported for Coupled Field Static analysis



Coupled Field Static

	Property	Value
2	General	
3	Component ID	Setup
4	Directory Name	SYS
5	Update Condition Parameter (Beta)	None
6	Notes	
7	Notes	
8	Used Licenses	
9	Last Update Used Licenses	
10	System Information	
11	Physics	Multiphysics
12	Analysis	Static
13	Solver	Mechanical APDL
14	Physics	
15	Structural	☑
16	Acoustics	☐
17	Thermal	☑
18	Electric	☐



Physics Region and Analysis Settings

- PML settings are supported for respective Acoustics physics in the Physics region object
- Voltage and Charge convergence supported on Analysis settings
  - Program Controlled
  - On
  - Remove

Details of "Physics Region"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Structural	No
Acoustics	Yes
Thermal	No
Electric	No
Suppressed	No
Acoustic Domain Definition	
Artificially Matched Layers	Off
Element Morphing	Program Controlled
Advanced Settings	
Reference Pressure	2.e-005 Pa
Reference Static Pressure	1.0133e+005 Pa
Fluid Behavior	Compressible

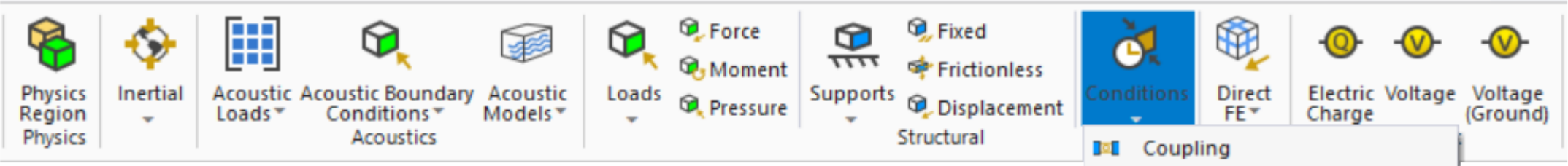
Coupled Field Static (A5)
- Initial Physics Options
- Analysis Settings
- Physics Region

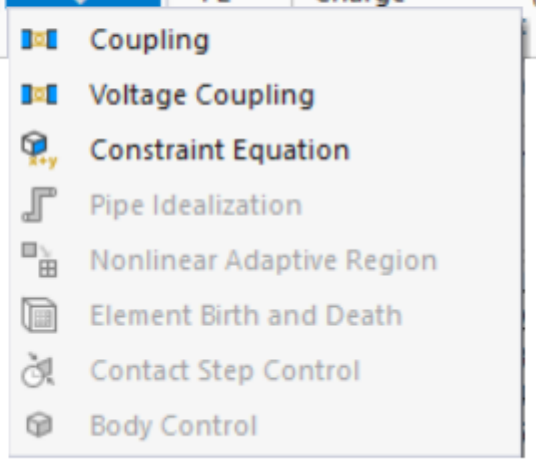
Details of "Analysis Settings"

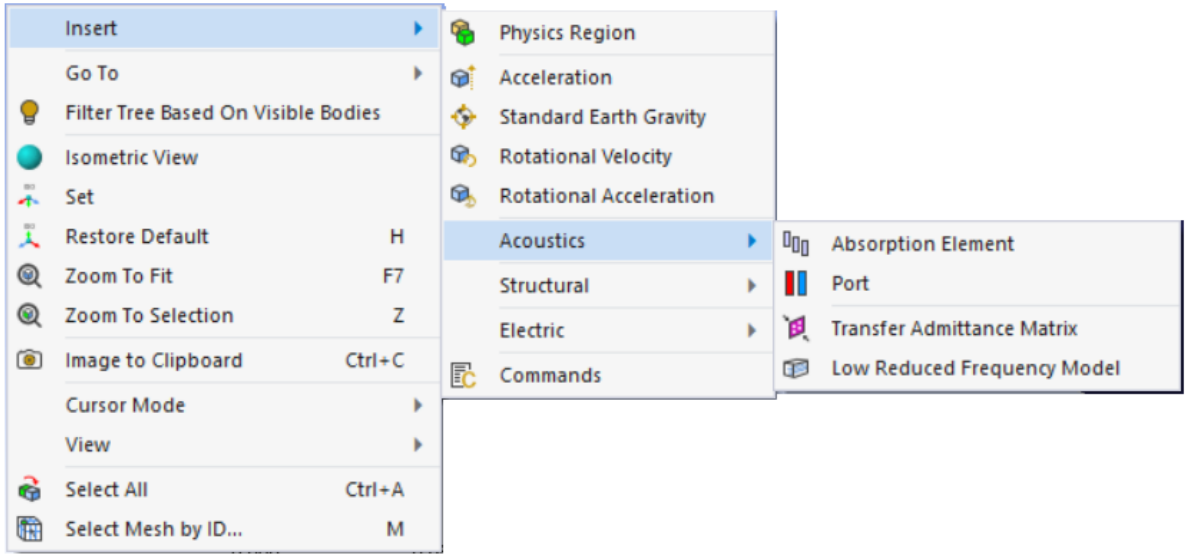
Step Controls	
Solver Controls	
Restart Controls	
Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled
Voltage Convergence	Program Controlled
Charge Convergence	Program Controlled
Advanced	
Output Controls	
Damping Controls	
Ignore Acoustic Damping	Yes
Analysis Data Management	

Boundary Conditions

- Electric and Acoustics boundary conditions are supported

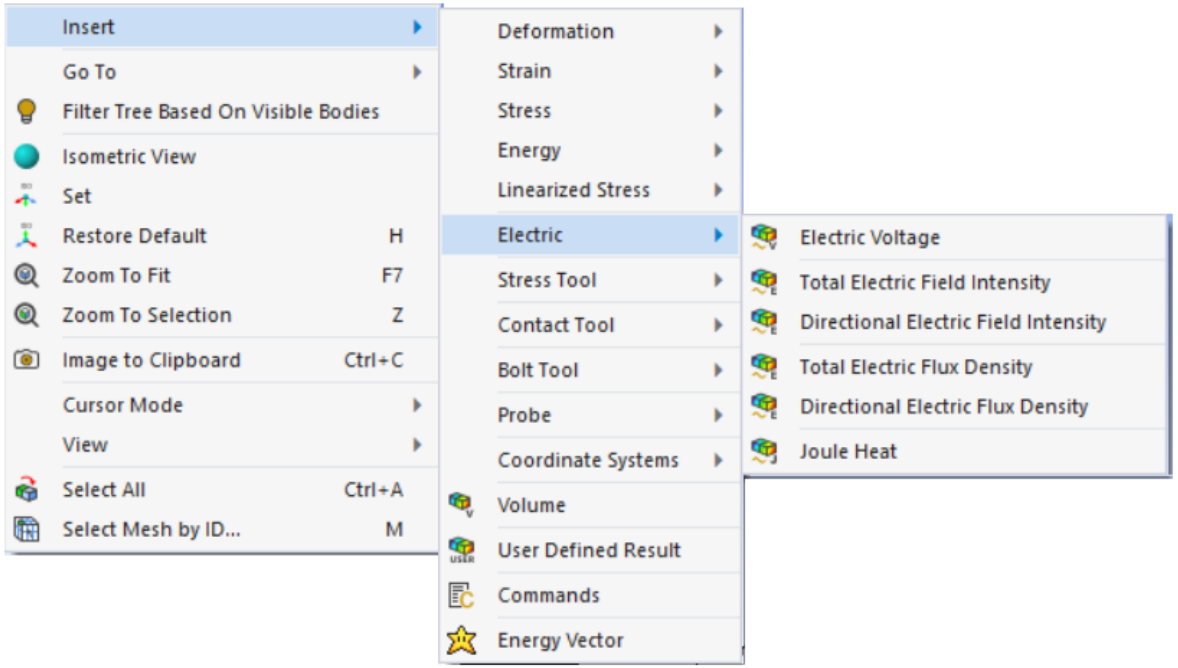


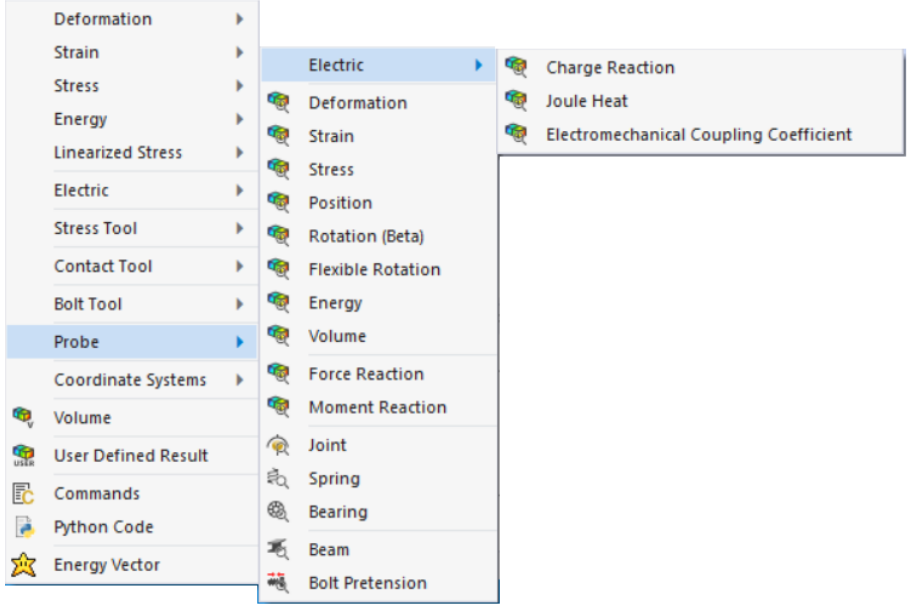




Results

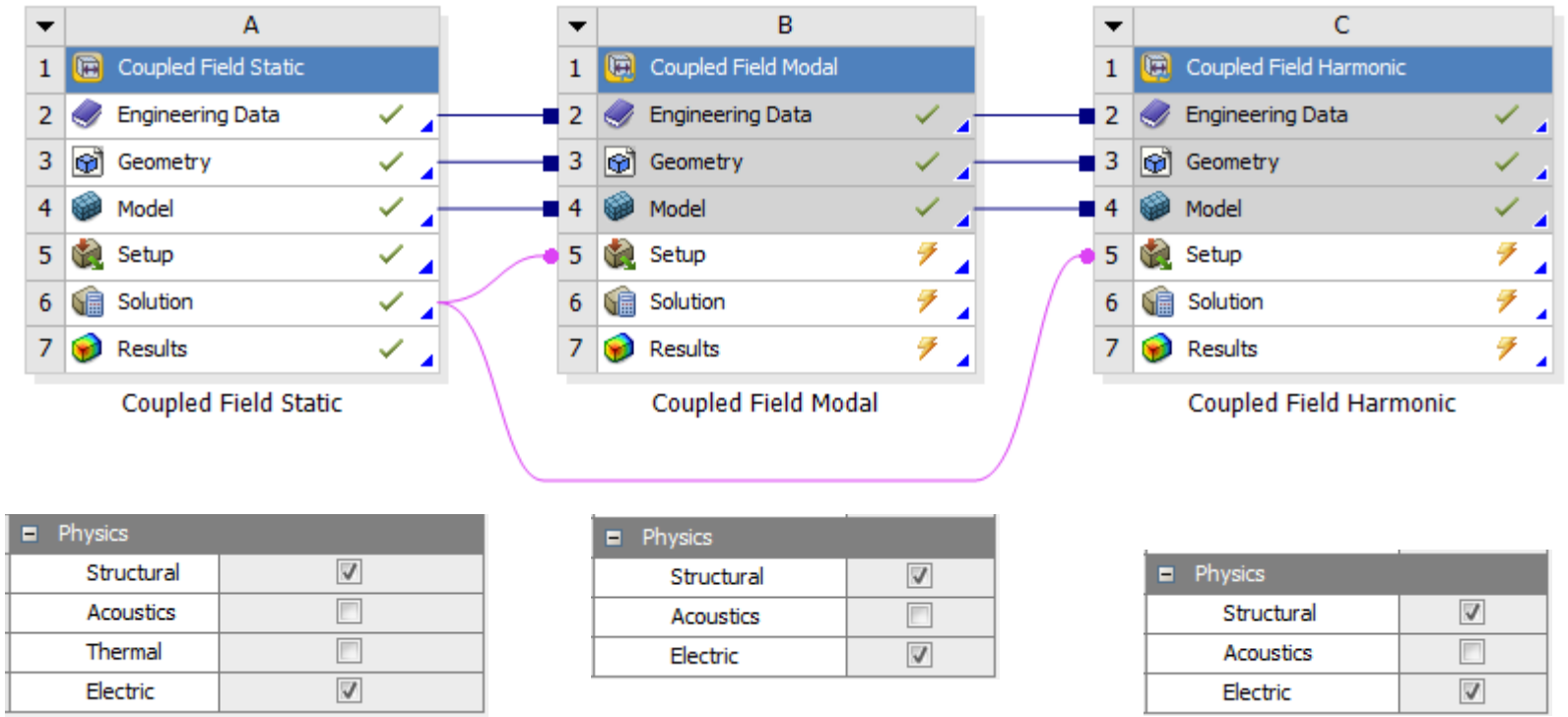
- Electric results and probes are supported





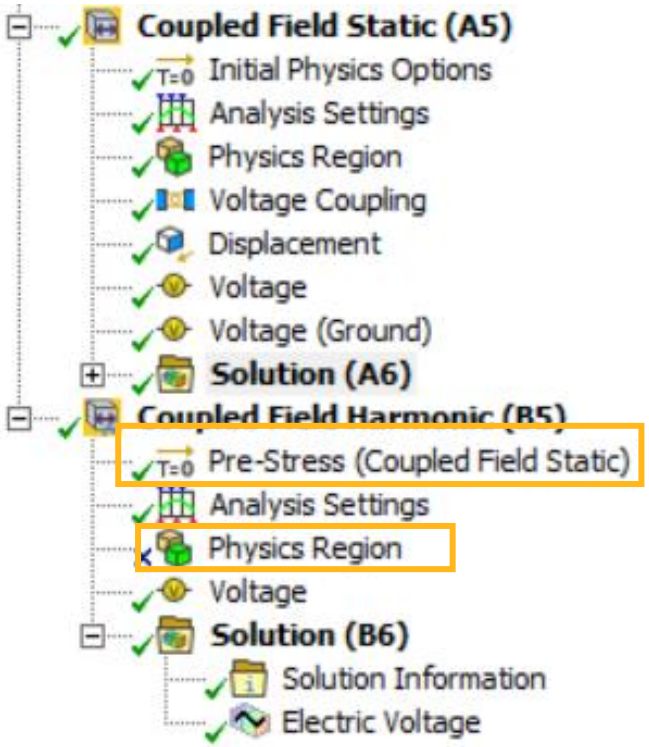
Pre-stress Coupled Field Analysis Enhancements

- Pre-stressed Coupled Field Modal and Pre-stress Coupled Field Harmonic (full harmonic) is supported by linking to upstream a Coupled Field Static analysis



Pre-stress Coupled Field Enhancements

- The physics combination which can be performed in Pres-stress Coupled Field analysis are:
  - Structural – Acoustics
  - Structural – Electric with Piezoelectric coupling
  - Acoustics with Piezoelectric coupling
- Physics region specified in the upstream coupled field static analysis will be automatically selected on downstream linked environment.
- Thermal physics selection in the coupled field static analysis will not support pre-stress workflow
- Linking on Coupled Field Modal and Harmonic can be done by selecting the pre-stress environment on the Initial conditions

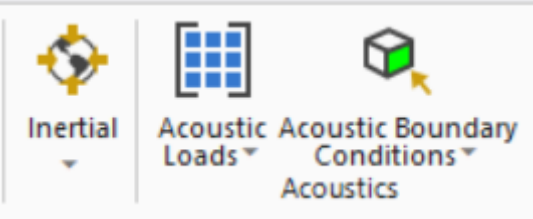


Boundary Conditions and Results for Pre-Stressed Analysis

Pre-stressed Coupled Field analysis supports these boundary conditions

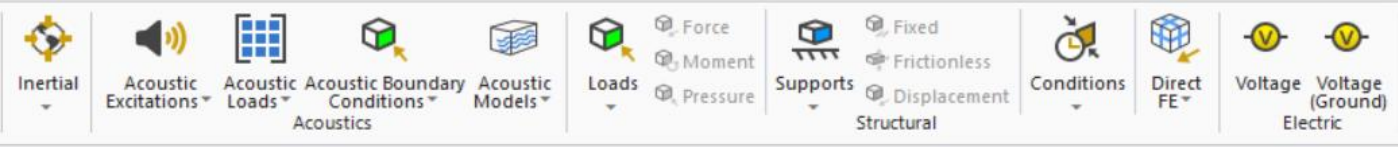
Pre-stress Coupled Field Modal

- Acoustic Boundary conditions



Pre-stress Coupled Field Harmonic

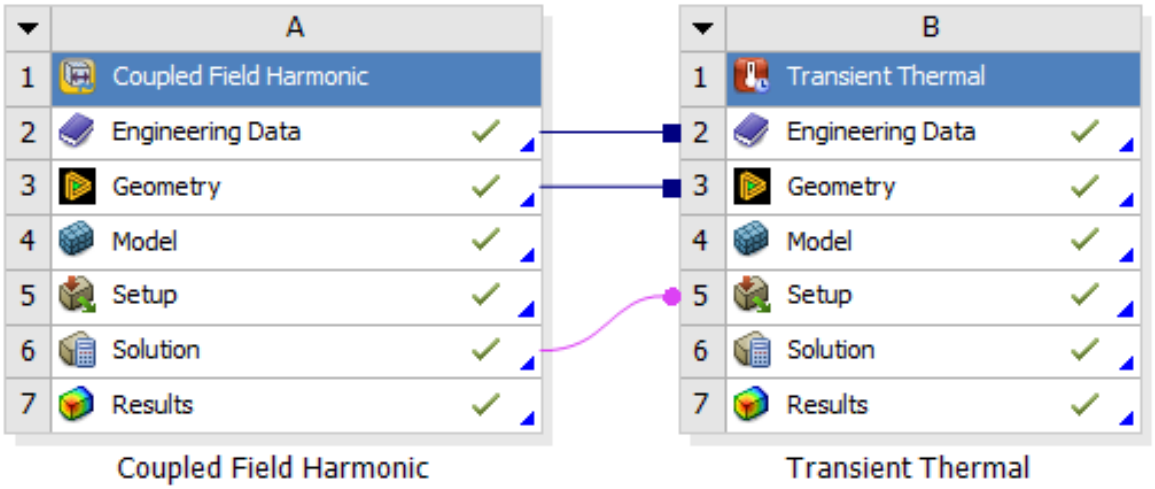
- Inertial Load, FE Loads
- Acoustic Boundary conditions
- Electric: Voltage and Voltage Coupling

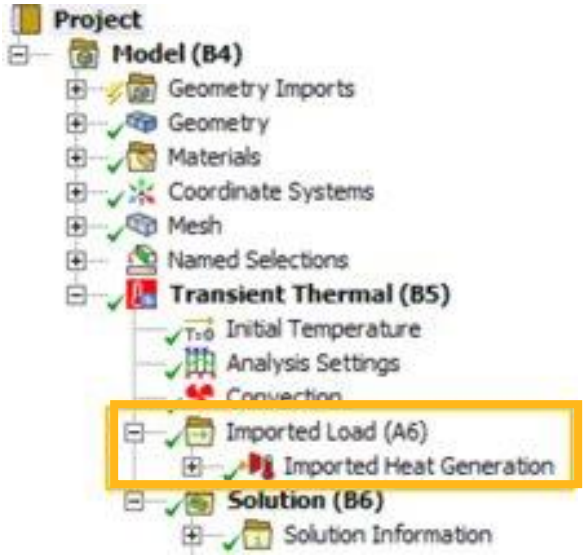


Results: All results supported for standalone analysis is applicable for pre-stress case

Imported Heat Generation Load from Coupled Field Harmonic

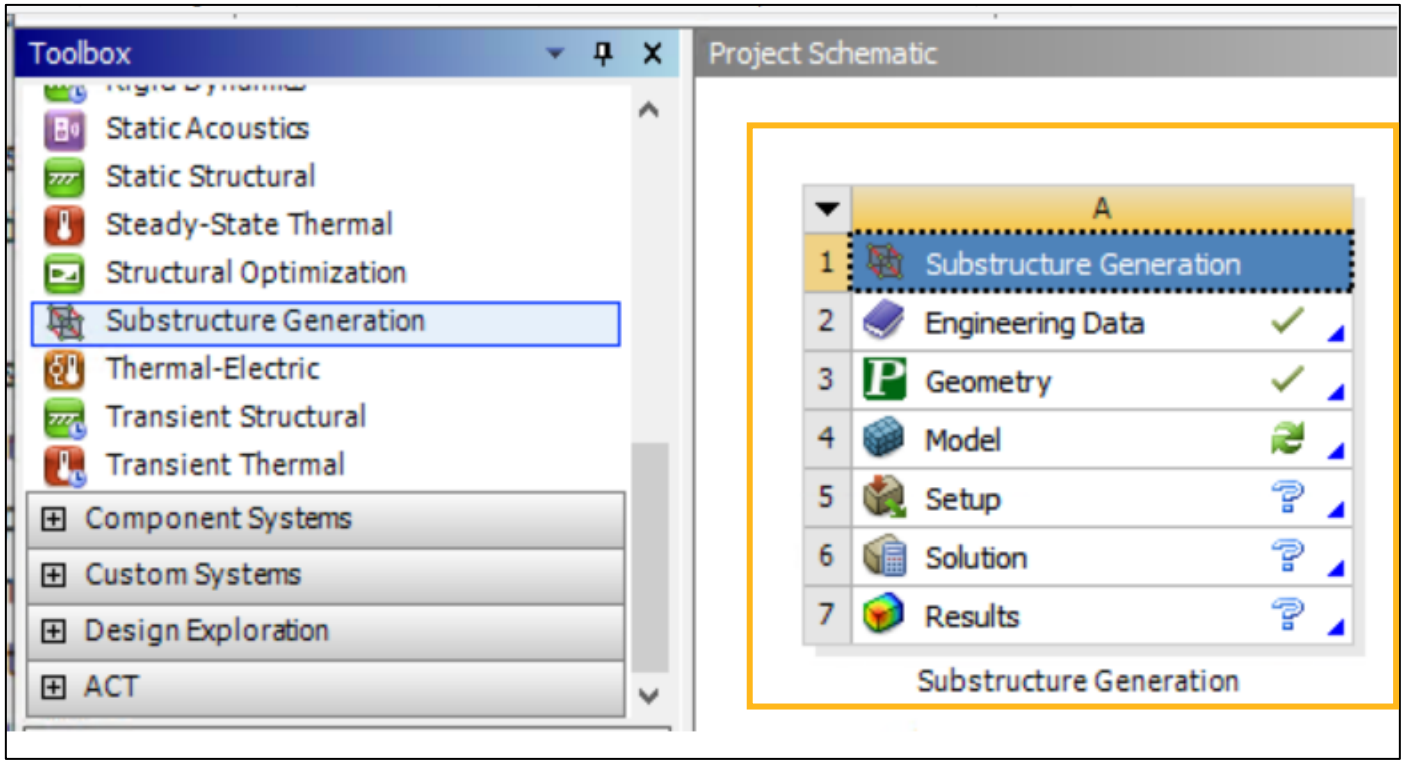
- Loss due to damping in the upstream Coupled Field Harmonic analysis can be imported as Heat Generation load in Transient Thermal analysis by linking the solution cell of Coupled Field Harmonic to setup of Transient Thermal analysis
- The losses are only considered from the coupled structural-electric bodies with Piezoelectric coupling and is applicable for dissimilar meshes. The source frequencies is split over equal time intervals in the transient thermal analysis when All option is selected from Worksheet of Imported Heat generation object

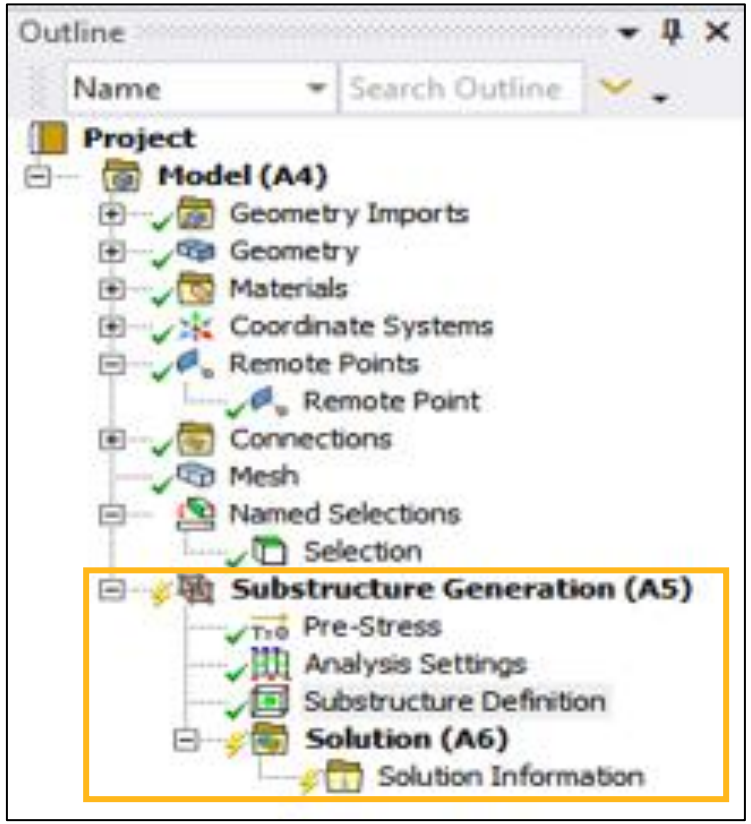




Substructure Generation Analysis

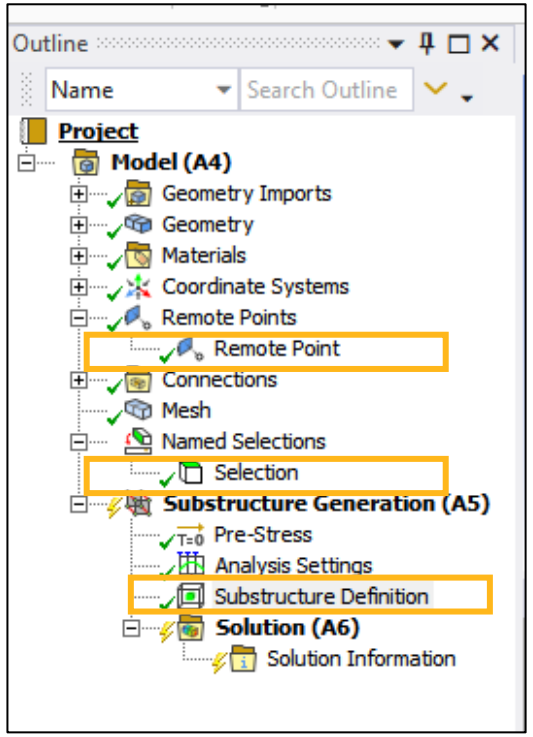
- Substructure Generation analysis is supported to condense the finite elements of your entire model using Component Mode Synthesis methodolgy into one “superelement”. These superelements can then be composed using Bottom-up Susbstructuring approach for use pass in another mechanical session

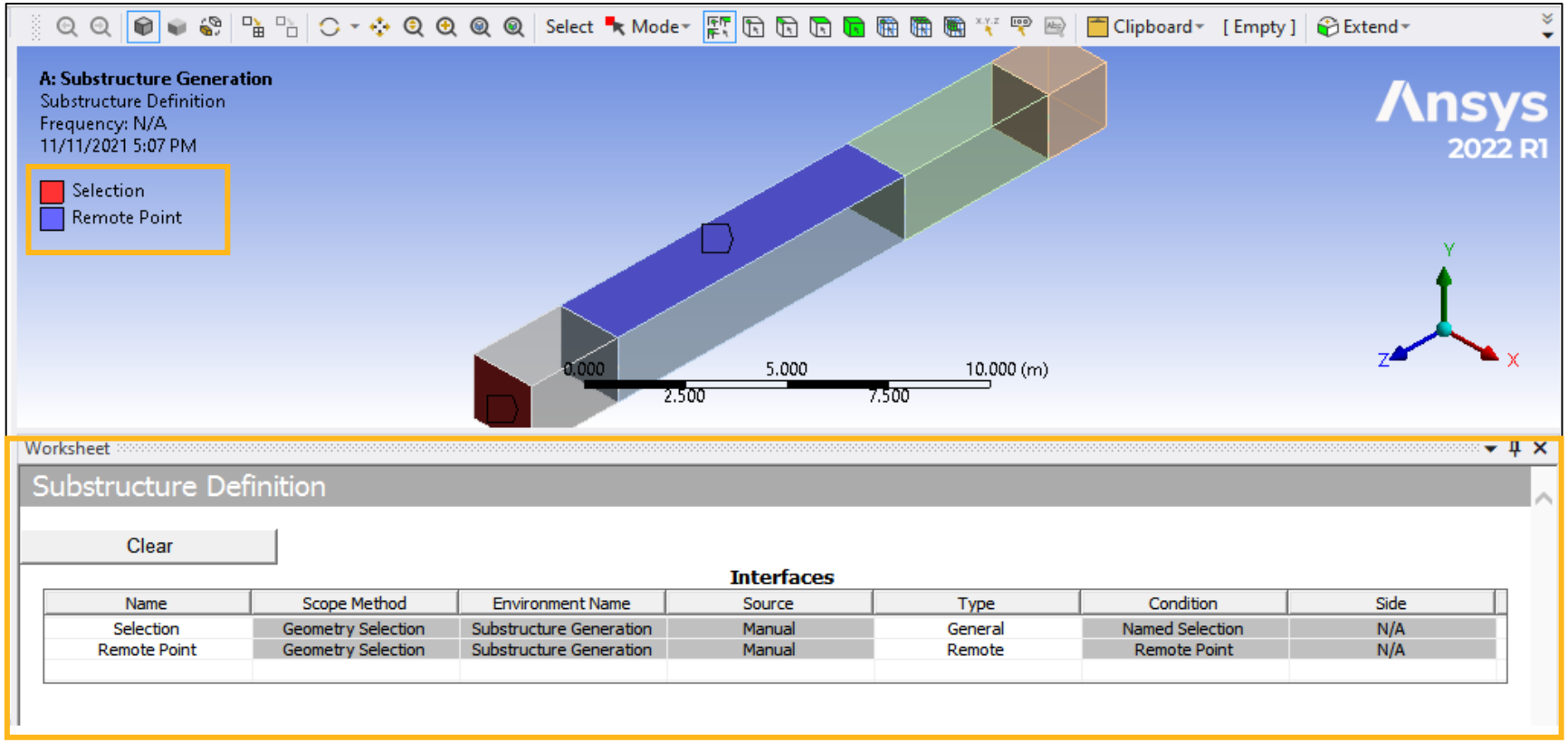




Substructure Generation

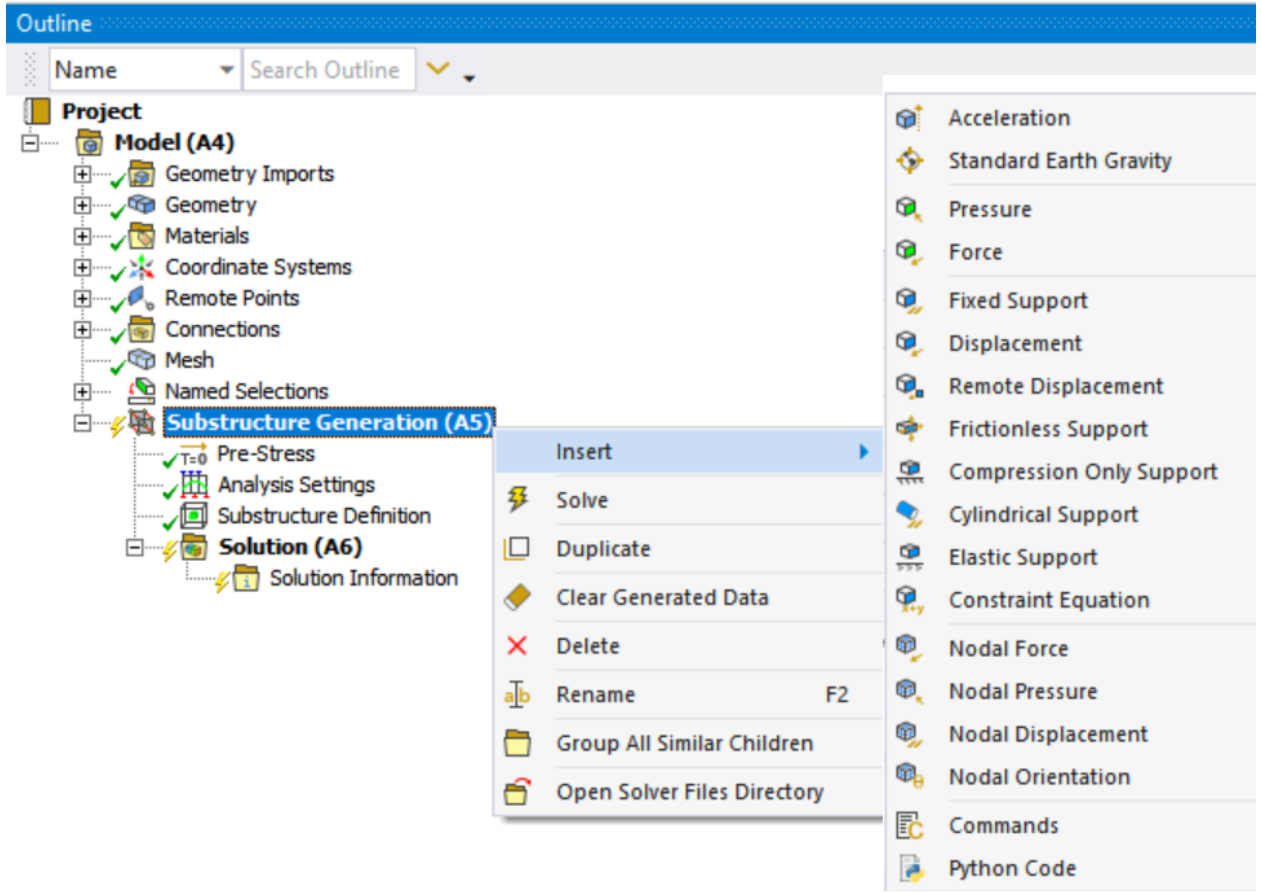
- Master degrees of freedom is specified using Remote point and Named selection added as Interface on Substructure Definition Worksheet. The user can drag and drop these objects on worksheet to add it as Interface





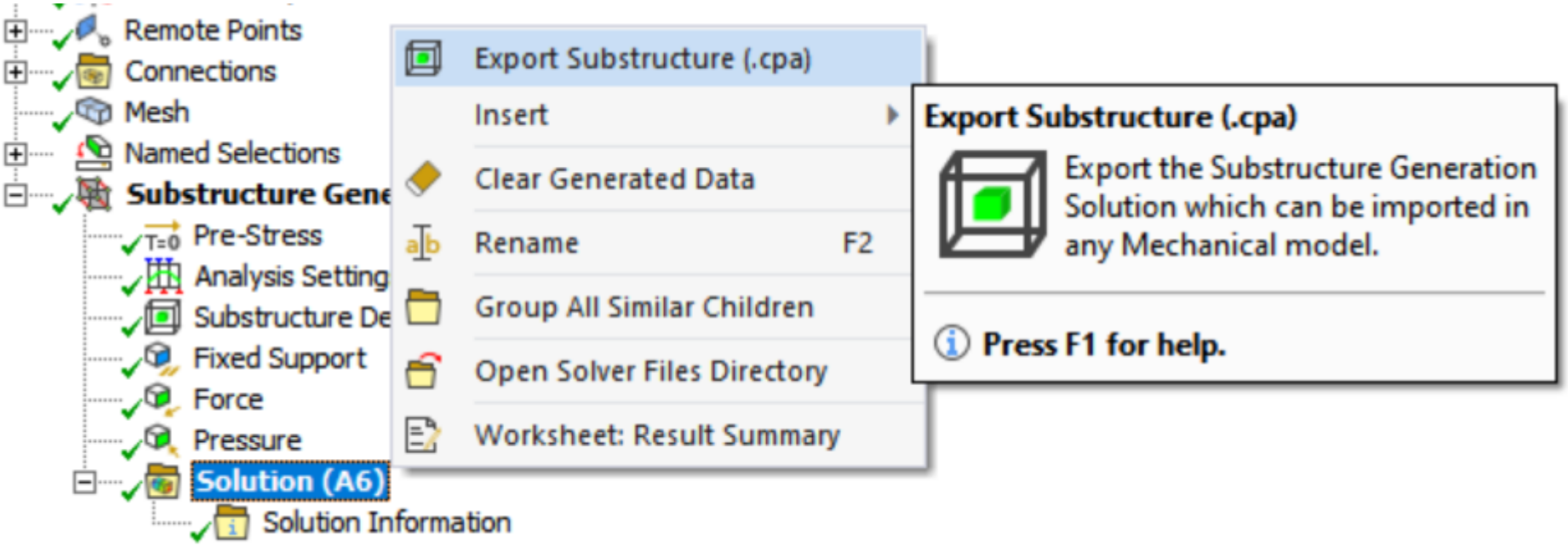
Substructure Generation

- Supports constant loads and boundary conditions.
- Solving the analysis will generate individual load vector for each load applied in the Substructure Generation analysis. These load vectors can then be applied in the use pass of MSUP harmonic analysis



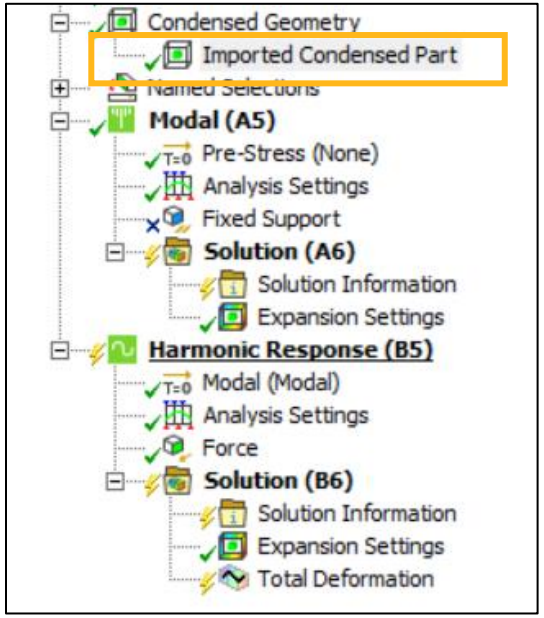
Substructure Generation

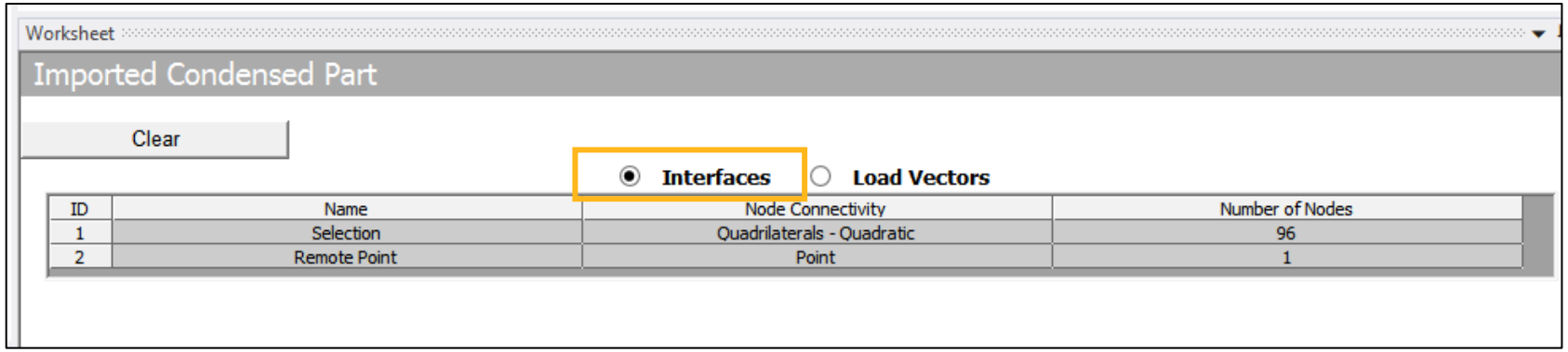
- These superelements can then be exported using Export Substructure (.CPA) option



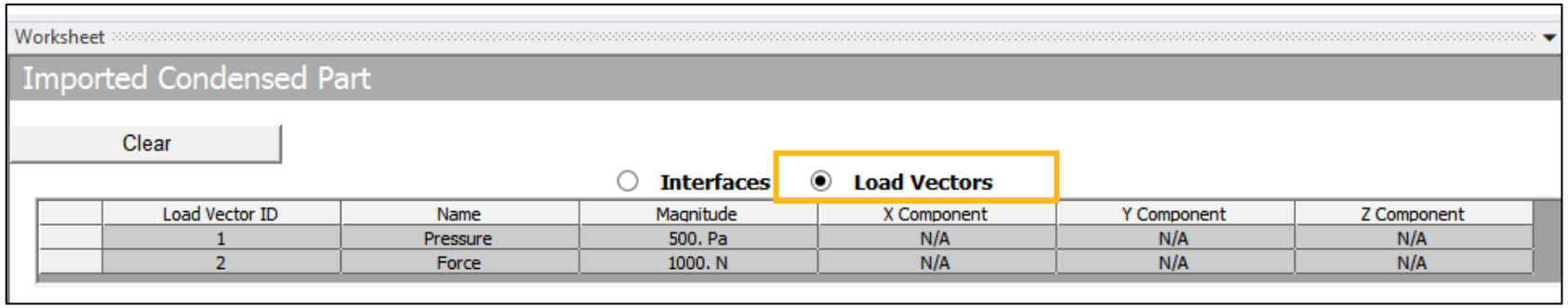
Substructure Generation

- These superelements can then be imported using Imported Condensed Part and will display the interfaces and load vectors in the Worksheet. These superelements will then be used for use pass in Modal, MSUP harmonic , Response Spectrum and Random Vibration analysis. The load vectors will be applied as unit loads only for MSUP harmonic analysis





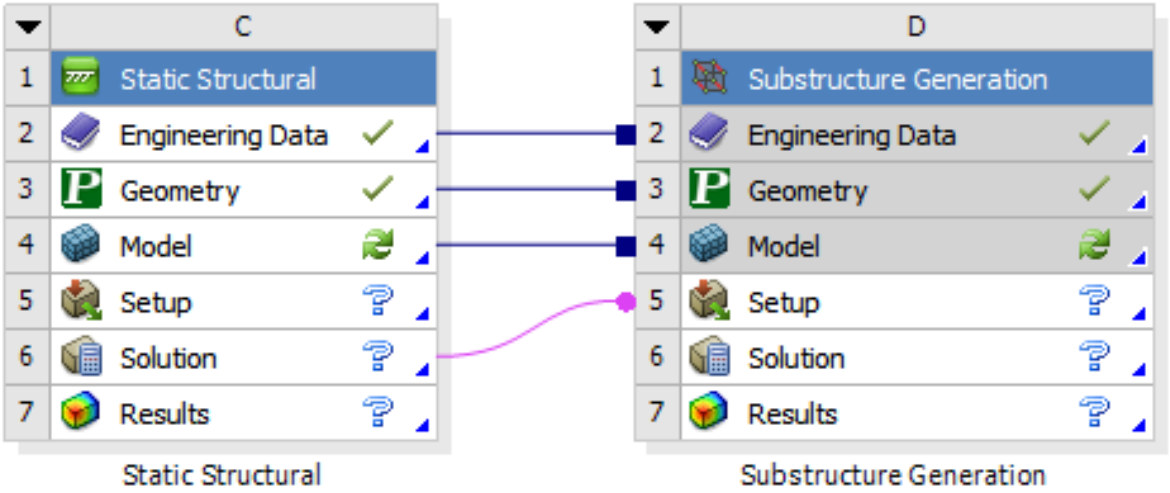
ID	Name	Node Connectivity	Number of Nodes
1	Selection	Quadrilaterals - Quadratic	96
2	Remote Point	Point	1



Load Vector ID	Name	Magnitude	X Component	Y Component	Z Component
1	Pressure	500. Pa	N/A	N/A	N/A
2	Force	1000. N	N/A	N/A	N/A

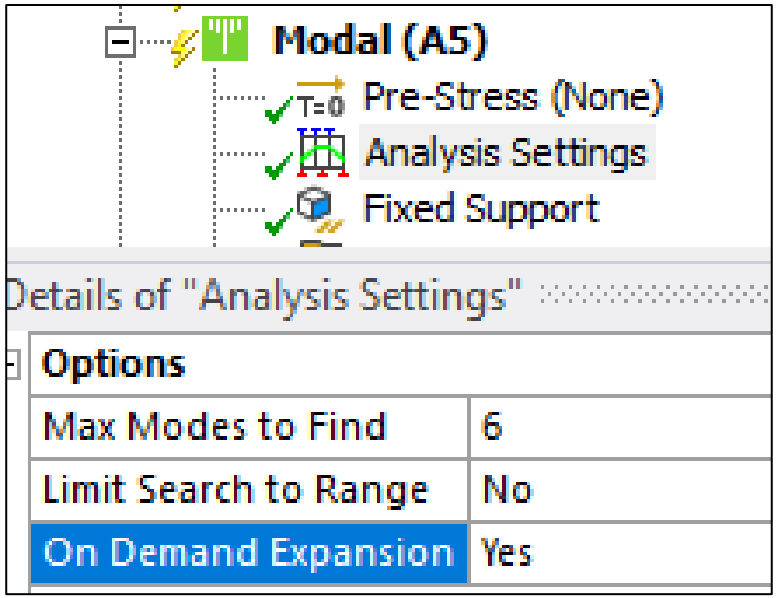
Substructure Generation

- Prestress effects can be included in the Substructure Generation analysis by linking it to upstream Static Structural analysis
- When linked to Static Structural analysis, the first load vector in the Substructure Generation analysis will be included from the pre-stress effects
- Substructure analysis is only supported for MAPDL solver and is supported in Beta mode when linked with External model



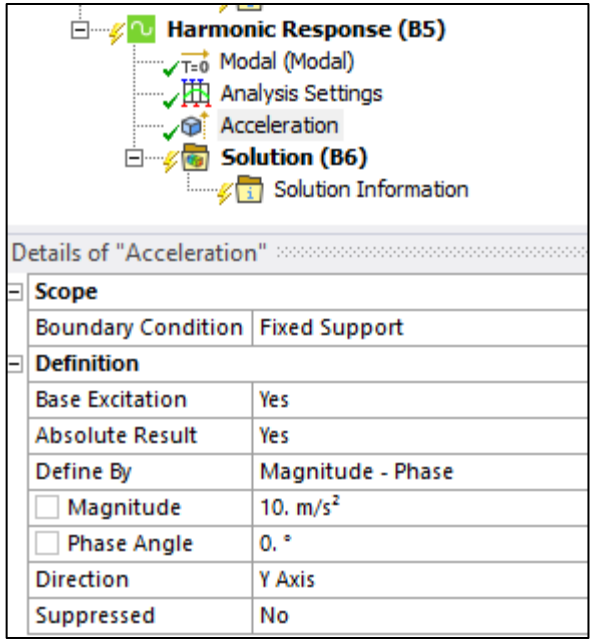
On Demand Expansion for Top-down Substructuring

- On Demand Expansion option is available in Modal analysis which when selected will evaluate Condensed Part results (Top-down Substructuring) “on the fly” and expansion pass through MAPDL solver is not needed for deformation results on the Condensed Parts

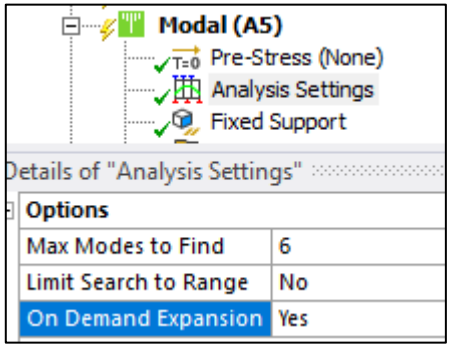


Mode Superposition Analysis Enhancements

- Enforced Motion with On Demand Expansion: Base Excitation (Relative & Absolute) can be used with On Demand Expansion



- On Demand Expansion option is available in Modal analysis allowing to post-process results using distributed files (dividing overall solution directory size by 2) and to evaluate Condensed Part results “on the fly” (Expansion Pass files not generated)



Mode Superposition Analysis Enhancements

- Analyses using On Demand Expansion now only retains the solution files required for post-processing which allows to global reduce the solution directory size

- A preference has been added to avoid storing displacement in the rst file using On Demand Expansion which can significantly reduce rst files size

Options (Modal, Harmonic and Transient Mode Superposition)	
On Demand Expansion	No
On Demand Mode Shapes	Yes

file0.rst	7,552 KB
file1.rst	7,872 KB

→

file0.rst	3,840 KB
file1.rst	4,160 KB

2021 R2

- ds.dat
- ds_file.xml
- dummy.dat
- file.db
- file.mlv
- file.mode
- file.rfrq
- file0.err
- file0.esav
- file0.full
- file0.log
- file0.mlv
- file0.mode
- file0.rfrq
- file0.rst
- file1.err
- file1.esav
- file1.full
- file1.mlv
- file1.mode
- file1.out
- file1.rfrq
- file1.rst
- MatML.xml
- parm.xml
- solve.out

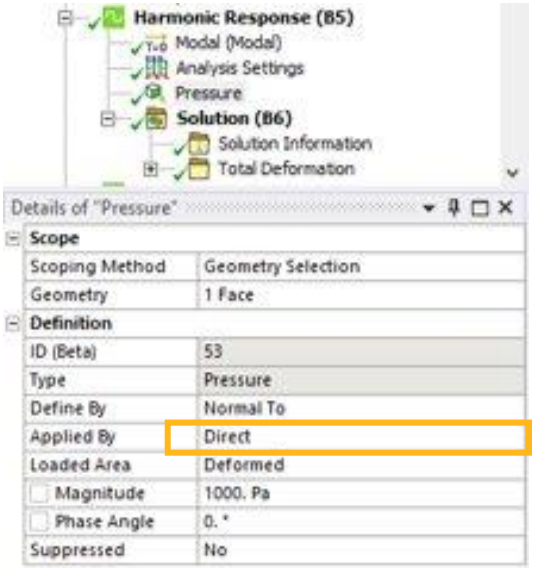
→

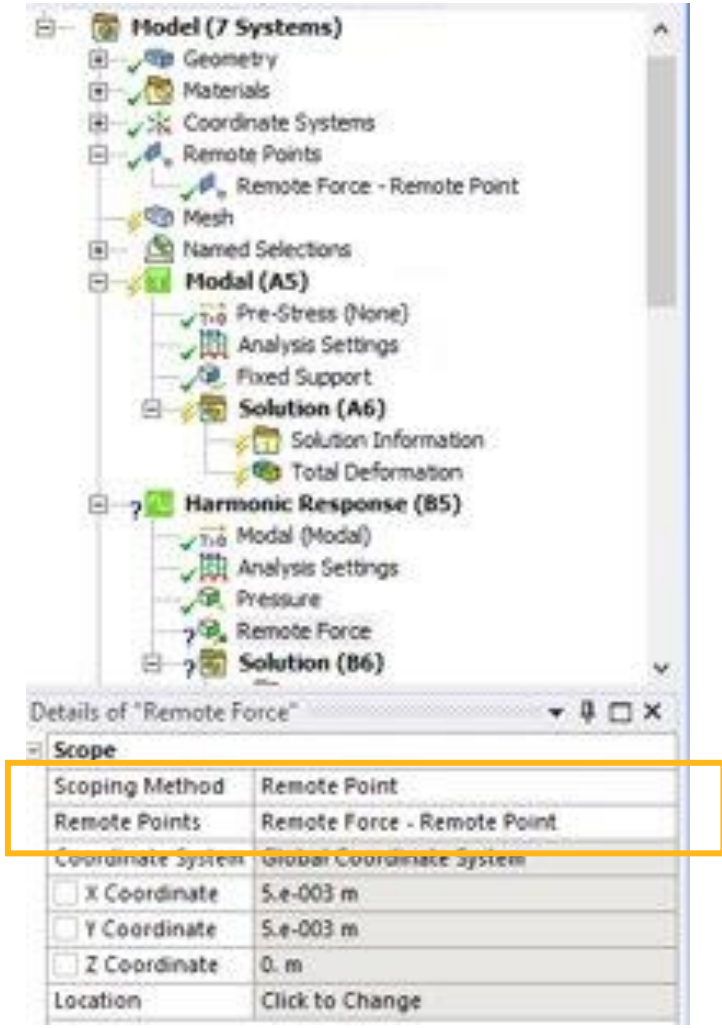
2022 R1

- ds.dat
- file.rfrq
- file0.err
- file0.mode
- file0.rst
- file1.mode
- file1.rst
- MatML.xml
- solve.out

Mode Superposition Analysis Enhancements

- MSUP analysis will use the Pressure and Force load default Applied by option of Direct and this will not create surface effect elements
- Remote force and moment when created will promote to remote point by default when no analysis is solved in the session and location is selected before creation



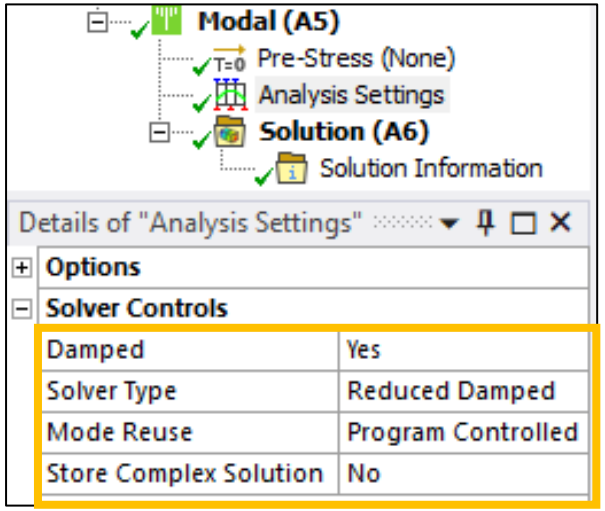


Material Damping losses in MSUP Harmonic Analysis

- In a linked Harmonic Response (MSUP) analysis when upstream Modal analysis uses Reduced Damped solver type and has Store Complex Solution property set to No
- Then, the losses associated with Material Dependent Damping under Engineering Data will be contributed by **Constant Structural Damping Coefficient** (MP,DMPS)
- For a Damped Modal analysis, the application now sends the Material-Dependent Structural Damping Coefficient (**MP,DMPS**) instead of Material-Dependent Damping Ratio (**MP,DMPR**)

Properties of Outline Row 3: Structural Steel

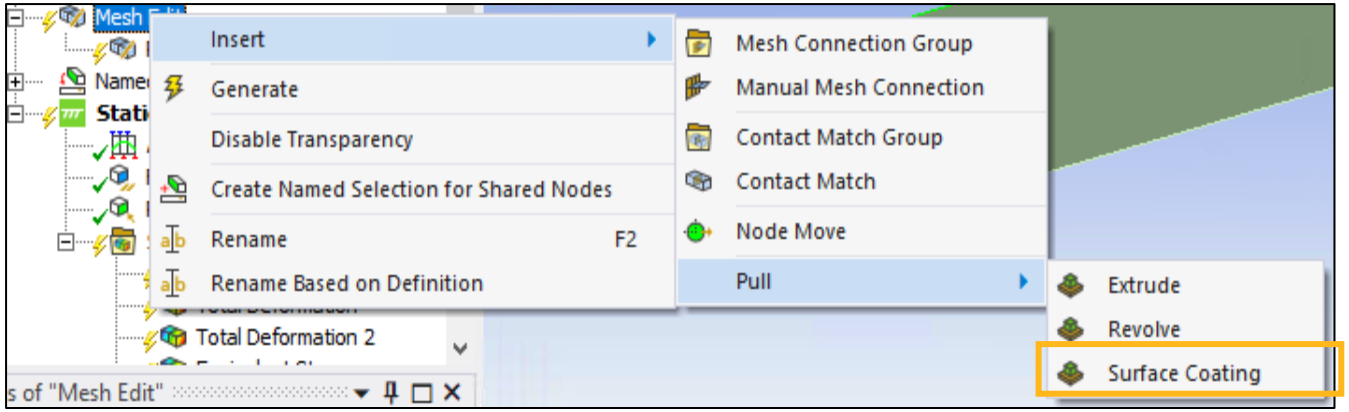
	A	B
1	Property	Value
2	Material Field Variables	Table
3	Density	7850
4	Isotropic Secant Coefficient of Thermal Expansion	
6	Material Dependent Damping	
7	Damping Ratio	0.25
8	Constant Structural Damping Coefficient	= 0.5

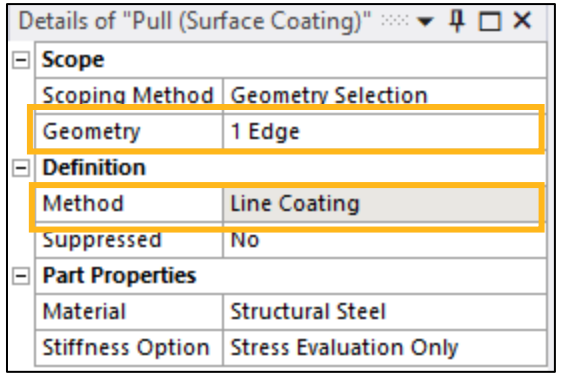



```
/com,********** Send Materials *******
Temperature = 'TEMP' ! Temperature
MP,DENS,1,7850, ! kg m^-3
MP,ALPX,1,1.2e-05,  ! C^-1
MP,C,1,434, ! J kg^-1 C^-1
MP,KXX,1,60.5,  ! W m^-1 C^-1
MP,RSVX,1,1.7e-07,  ! kg m^3 A^-2 s^-3
MP,EX,1,200000000000,   ! Pa
MP,NUXY,1,0.3,
MP,MURX,1,10000,
MP,DMPS,1,0.5,
```

Line Coatings Using Pull

- Surface Coating can be added under Mesh edit folder
- Pull method set to **Line Coating** when scoped to edge of surface body
  - In 2D analysis: Scope to edge of surface body.
  - In 3D analysis: Scope to edge of surface body with **Dimension** set to **2D**
- Scoping multiple line coatings on same edge is supported
- Supported only for analyses that use Mechanical APDL solver. i.e., Solver Target property on environment set to Mechanical APDL.





Line Coatings Using Pull

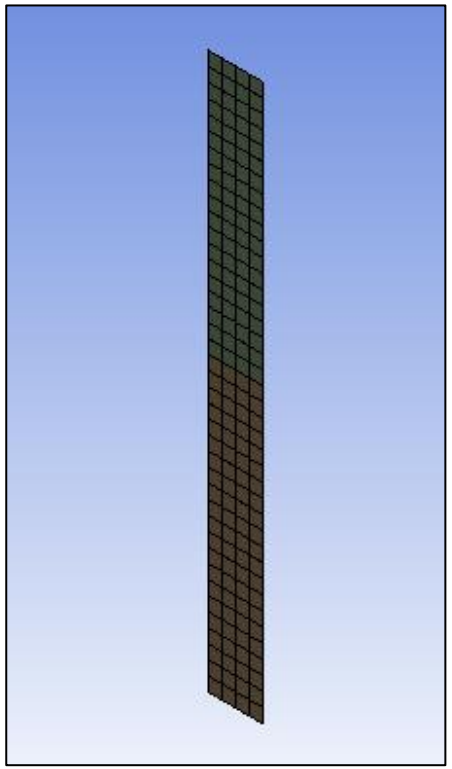
- Upon generating Pull, a line body is added in the tree
- Properties for this body are like a 3D surface body
- Model Type is set to Axisymmetric Shell (Read-Only)
- Send Shell 208/209 definition for these line coatings
- **Thickness** property definition is written to the input file when Stiffness Option is set to "Membrane and Bending" or "Membrane Only“
- The **Offset Type** property input is written to input file only when Stiffness Option is set to "Membrane and Bending"
- When Stress and Strain results scoped to Pull generated line coating bodies and the Position property is set to **Top/Bottom**, the contours displayed in Mechanical only correspond to the bottom layer.

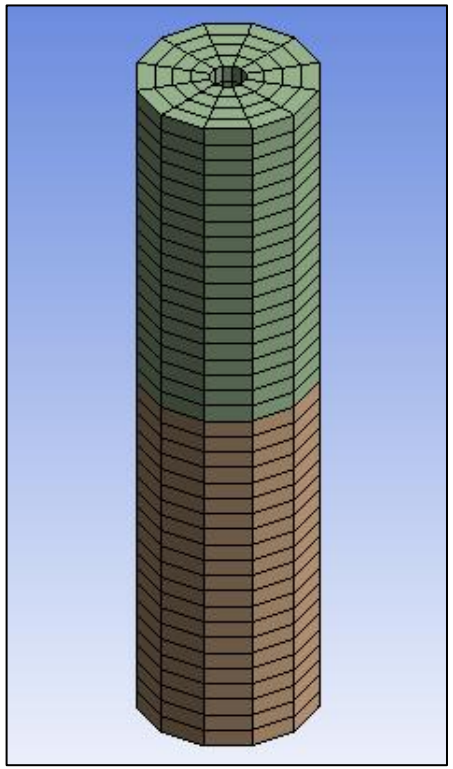
Details of "Pull (Surface Coating)"

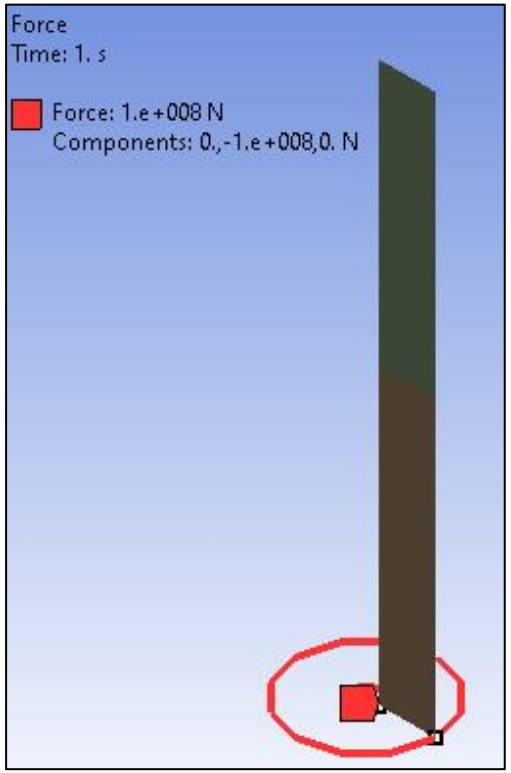
Graphics Properties	
Definition	
Suppressed	No
Model Type	Axisymmetric Shell
Stiffness Behavior	Flexible
Stiffness Option	Stress Evaluation Only
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	0. m
Offset Type	Bottom
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	0. m
Length Y	0.1 m
Properties	
Length	0.1 m
Statistics	
Nodes	11
Elements	5
Mesh Metric	None

General Axisymmetric Enhancements

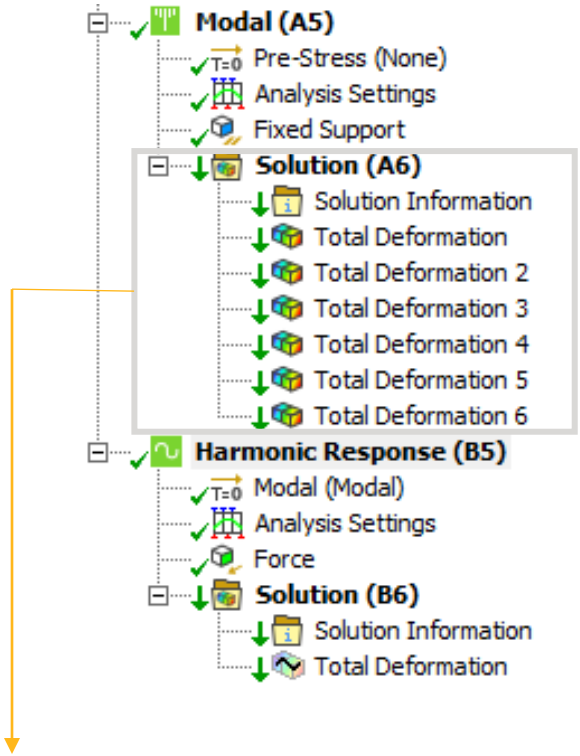
- General Axisymmetric mesh generation on an imported base mesh
- Force load scoping on edges and vertices of general Axisymmetric bodies. This force load behaves as a Nodal Force
- Bolt Pretension load allowed on General Axisymmetric bodies under Beta







Workflows



If performed on the same server, the upstream (Modal) results don't have to be downloaded to submit the downstream (Harmonic) analysis

1. Both analyses are solved on DCS:

- Upstream system result files, required for the downstream analysis, are internally referred in the cloud storage for supported analyses, thus saving time in downloading/uploading files.
- The linked systems can be submitted individually or together by pressing solve on the most downstream system.

2. Only the downstream analysis is solved on DCS:

- Files needed for the downstream analysis from the upstream system are automatically uploaded to the DCS server.

Analyses Supported

Without needing to download results of the upstream system, DCS currently supports:

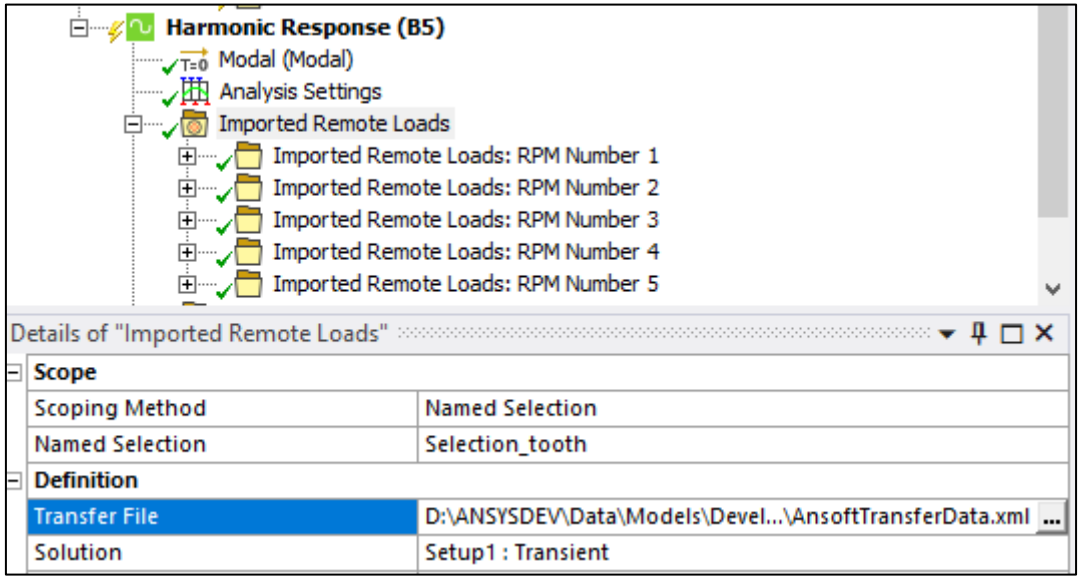
- Mode Superposition Harmonic Response, Transient, Random Vibration, Response Spectrum (including Prestress)
- Prestress Modal, Full Harmonic Response, Eigenvalue Buckling
- Prestress Coupled Field Modal, Coupled Field Harmonic
- Static Acoustics and Modal Acoustics, Harmonic Acoustics

Analyses in which upstream results are needed to be automatically downloaded:

- Downstream system having imported loads from upstream system such as Steady-State Thermal and Static Structural, and Electric Conduction and Steady-State Thermal systems
- Steady-State Thermal and Transient Thermal systems

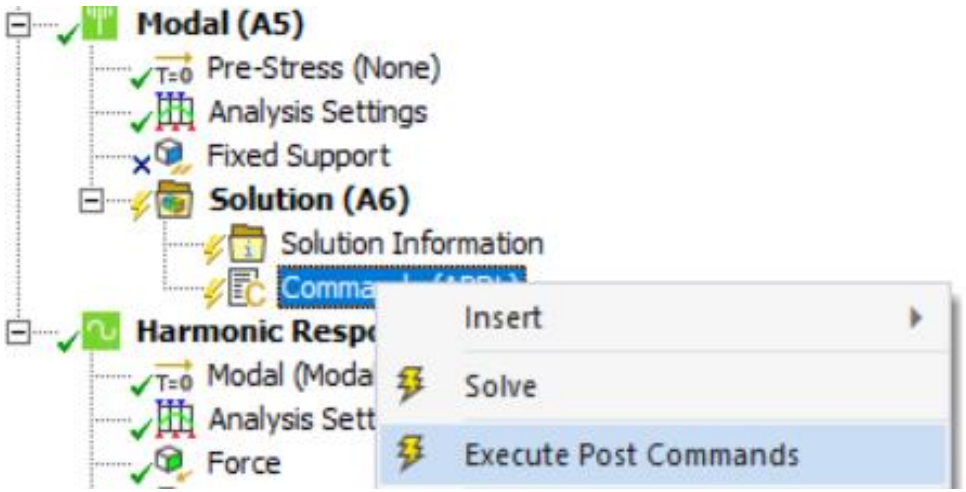
Maxwell: Mechanical Coupling

- Remote Forces / Moments can be generated from Maxwell exported file which eliminate the need of having both Maxwell and Mechanical models in the same Workbench project



Reinforcements and Execute Post Commands

- Reinforcements is now supported for Response Spectrum analysis
- Execute Post commands is supported to execute commands and extract information when the solution is partially solved or in a restart state



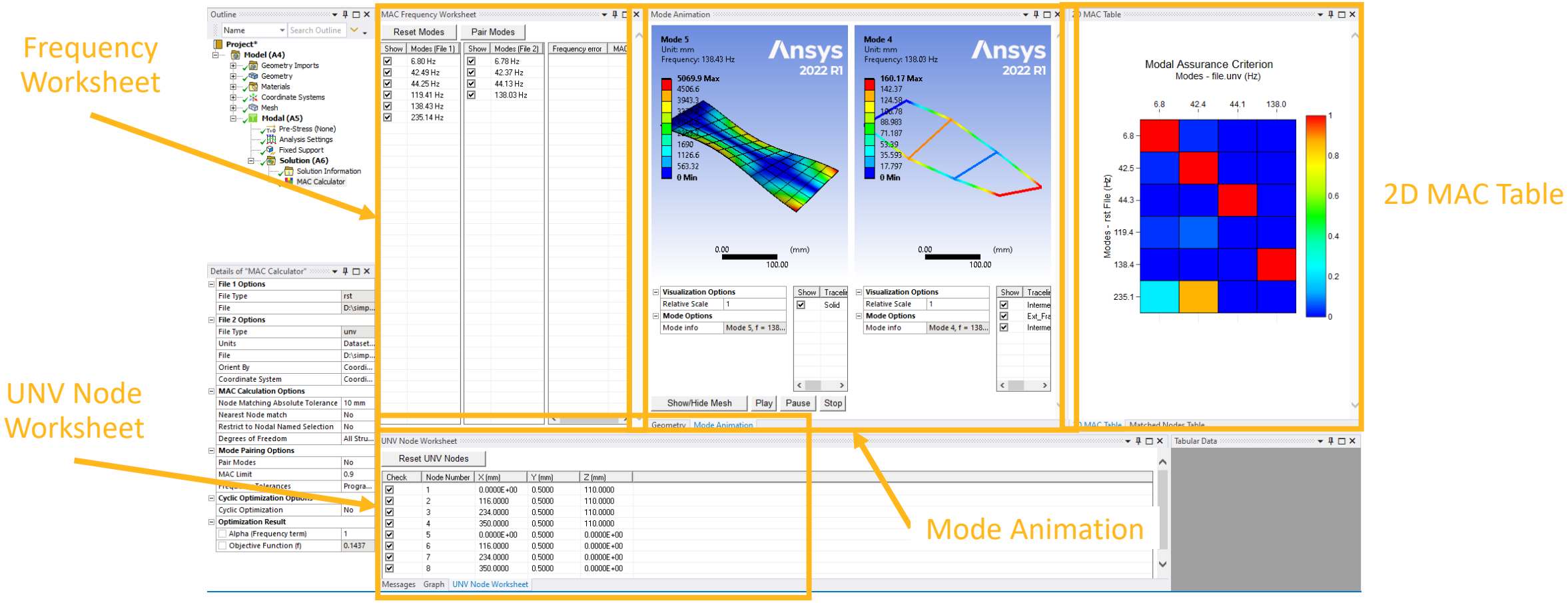
NVH Toolkit

NVH Toolkit: MAC Calculator

- The MAC Calculator result object computes the Modal Assurance Criterion (MAC) between a Modal Analysis (rst file) and a test (unv file). It allows you to:
  - Mode selection and flip to order them
  - Mode pairing to identify matches
  - Node selection and position tuning of unv nodes
  - UNV model orientation (Coord. System, Rigid Body Transformation or 3 Node Alignment)
  - Cyclic Optimization specific workflow
  - Interactive MAC Table
  - Interactive side-by-side Mode Animation

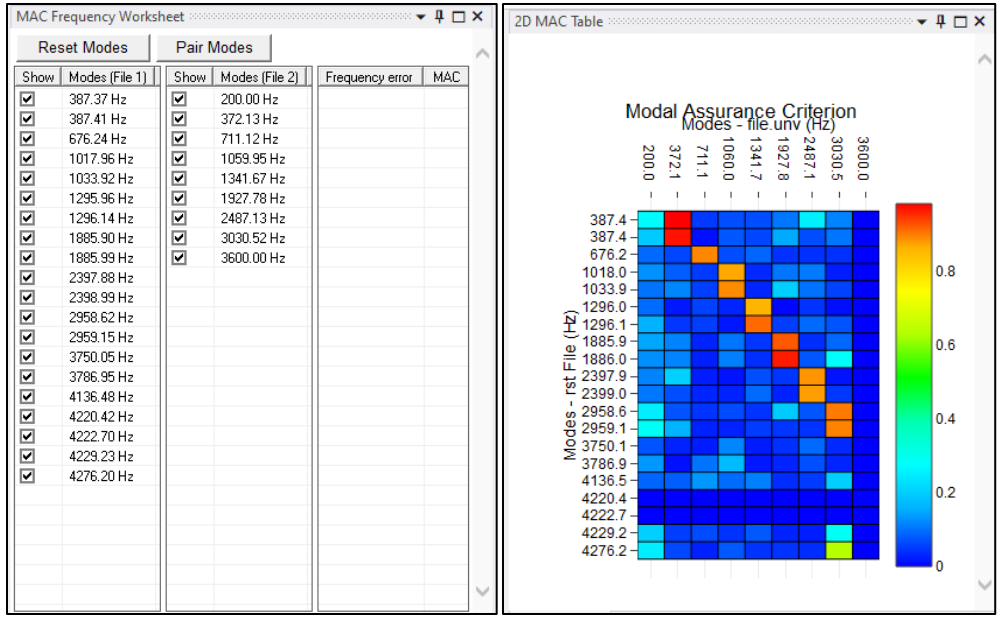
NVH Toolkit: MAC Calculator – Comprehensive UI

- The UI has been tailored to provide enhance all the preprocessing and postprocessing capabilities:

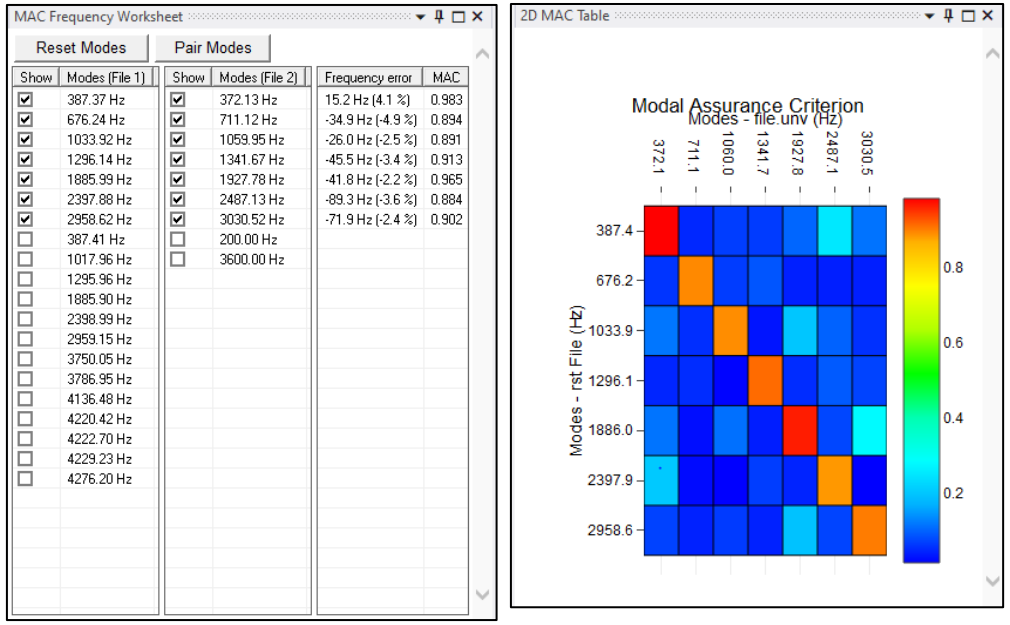


NVH Toolkit: MAC Calculator – Mode pairing

- Automatic mode pairing identifies rst-unv mode matches based on MAC and Frequency values:



Before pairing



After pairing

NVH Toolkit: MAC Calculator – UNV Node tuning

- The selected nodes are highlighted in green in the Geometry pane
- Coordinates are editable in the UNV Node Worksheet
- The UNV Model can be oriented in three ways:
  - By use of a Coordinate System already defined in the model
  - By entering the 3 displacements and 3 rotations that fully define a Rigid Body Transformation
  - By matching 3 rst and unv nodes

UNV Node Worksheet

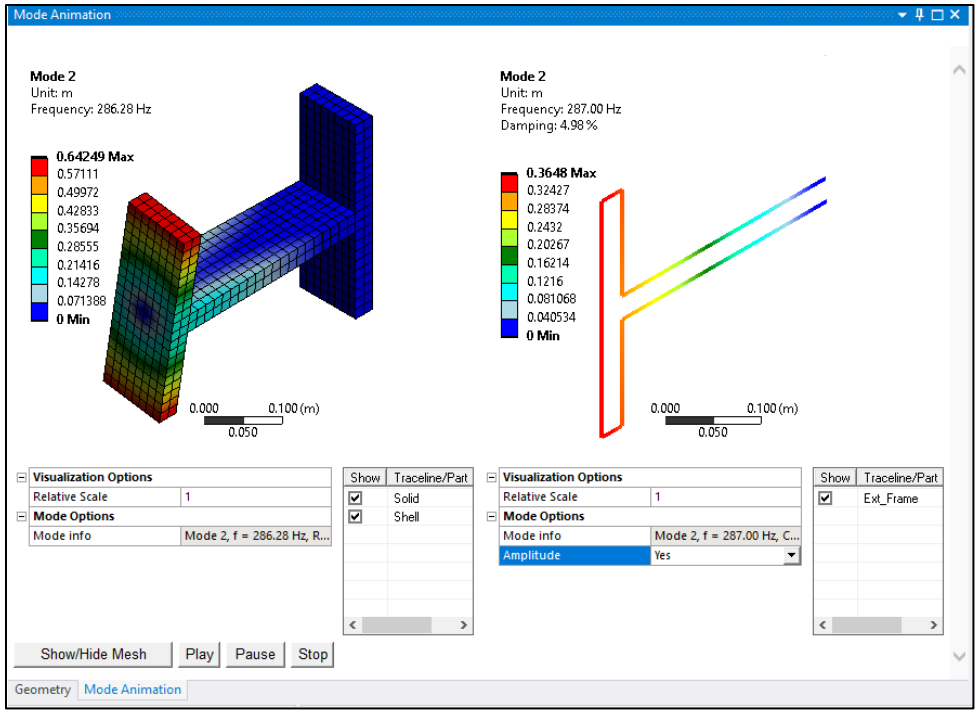
Reset UNV Nodes

Check	Node Number	X (mm)	Y (mm)	Z (mm)
☑	1	0.0000E+00	0.5000	110.0000
☑	2	116.0000	0.5000	110.0000
☑	3	234.0000	0.5000	110.0000
☑	4	350.0000	0.5	110.0000
☑	5	0.0000E+00	0.5000	0.0000E+00
☑	6	116.0000	0.5000	0.0000E+00
☑	7	234.0000	0.5000	0.0000E+00
☑	8	350.0000	0.5000	0.0000E+00

File 2 Options	
File Type	unv
Units	Dataset 164 (UNV File)
File	D:\h \...
Orient By	Coordinate System
Coordinate System	Coordinate System Rigid Body Transformation 3 Node Alignment
MAC Calculation Options	

NVH Toolkit: MAC Calculator – Mode Animation

- If a square in the 2D MAC Table is clicked, both Modes are Animated in a side-by-side window
- The user has the ability to control the graphical windows (rotate, pane) and to interact with the animation (Play/Pause/Stop)



NVH Toolkit: Stress/Strain Recovery

- The Stress/Strain recovery results computes elastic Stress/Strain from the Modal Superposition of the results of a Modal Analysis according to a Modal Coordinates File (MCF)
- The user inputs the MCF and selects its units and the normalization method (Mass or Unity) employed to derived the Modal Coordinates
- The MCF can be indexed by either Time or Frequency

Geometry	
Scoping Method	Geometry Selection
Geometry	1 Face
rst Options	
rst File	D:\ ...
MCF Options	
MCF File	D:\ ...
Skip Rows	10
Normalization	Mass Normalization
Units	Metric (mm-ton)
Extract Options	
By	Frequency
Display Frequency	400 Hz
Amplitude	No
Sweeping Phase	0 °
Stress Properties	
Type	Equivalent (von-Mises) Stress
Shell Layer	Top

MAPDL Contacts

Dual Shape Function Based Contact Formulation

- The dual shape function-based algorithm is state of the art for projection-based contact
  - It reduces solution time and memory usage (performance improvement)
- Reduces dependent terms for each contact constraint:
  - Gap function/slip from dual mortar contact are closer to local geometric gap/slip at contact nodes
  - It is more efficient and remedy potential over-constraints.

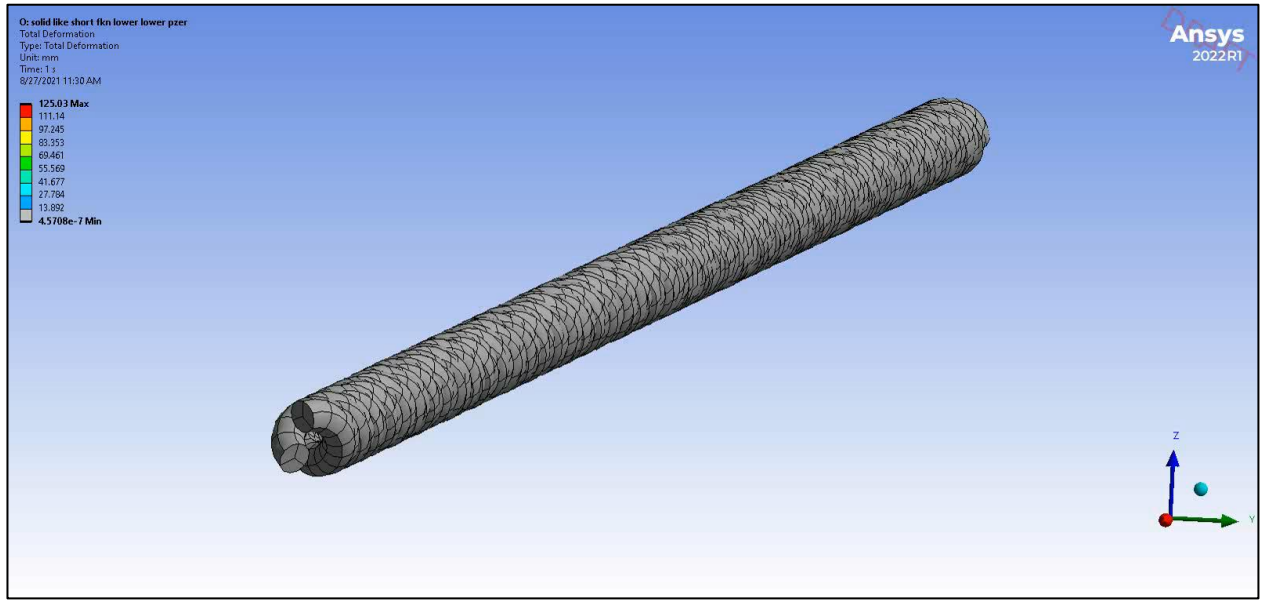
	Total Iteration	Max. Memory used	Max. Memory allocated	Run Time
Standard projection	69	89.1 GB	121.1 GB	12637 s
Dual shape function	68	56.7 GB	84.8 GB	5904 s (2.1x)

Three contact pairs Total Dofs 1,792,867, Augmented Lagrange method
Running with DMP 32 CPUs

New Adaptive Small Sliding Option

- Small sliding is assumed during substep. The nodal connectivity of the contact element reformed in the beginning of each substep and remains unchanged during iterations
- Bisection is performed if a large sliding distance is detected within a substep
- It can handle situations when contact pairs are initially in far-field and then come into contact
- The adaptive small-sliding logic often improves robustness/performance compared to the finite sliding logic and can also improve solution accuracy compared to the small sliding logic

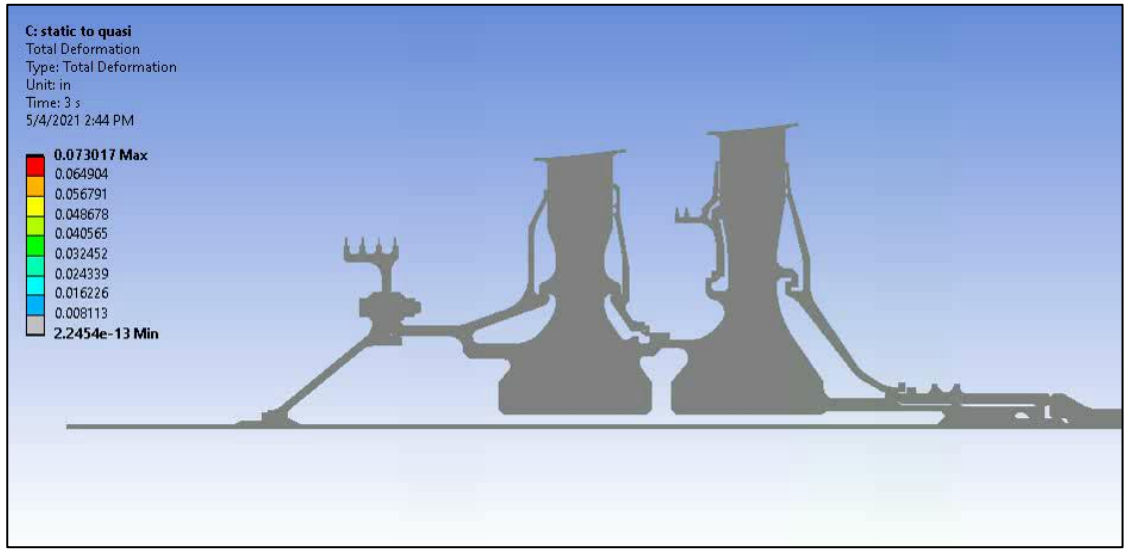
	Total Iterations	Elapsed Time
Pure small sliding	Fail at 4.5% loading	
Finite sliding	450	3 h 13 m
Adaptive small sliding	383	1 h 10 m (3x)



Contact Robustness Improvements

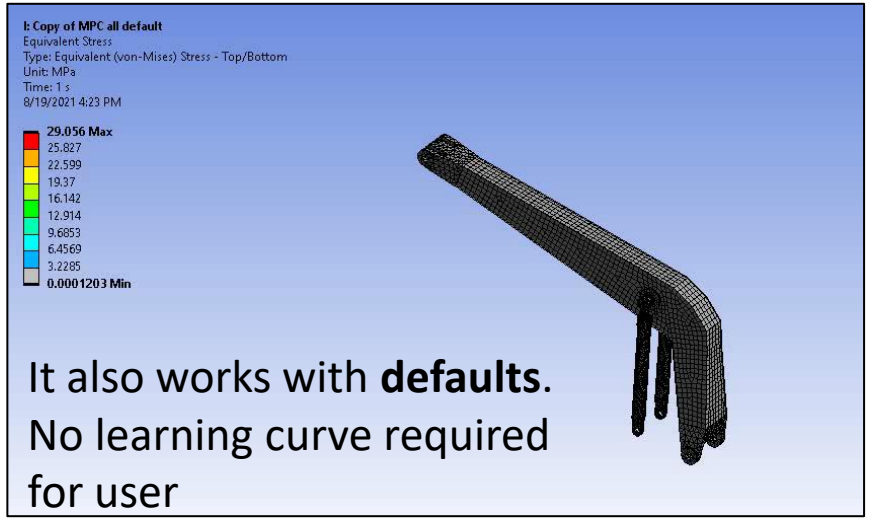
- Revise pinball radius for consistency between small and large deflection
  - Initial contact properties/results in Mechanical contact tools will be the same during solution
- Improve local contact searching to prevent potential spurious contact
  - Go over local target segments inside pinball to find the closest segment
- Improve solution for interference fits
  - Automatically turning on/off based on initial contact status/penetration
- Introduce cut-off contact stiffness for exponential relationship which no longer depends on the initial pressure

	Total Iterations	Elapsed Time
2021 R2	606	1621 s
2022 R1	514	1543 s



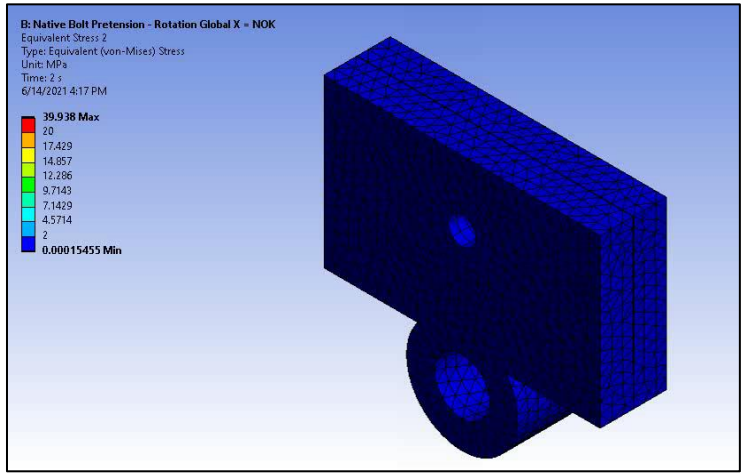
Robustness and Accuracy Improvements for Shell Assemblies

- Motivation: to address an issue of stress-free rotation for a shell assembly model
- Enhance SHSD command:
  - Supports Line contact element (177) defined as shell edge
  - Supports gap/penetration close
- MPC is built based on large rotation based RBE3 constraints
  - Shell to shell, Shell to solid-surface, shell-edge to shell-edge



Stressman model

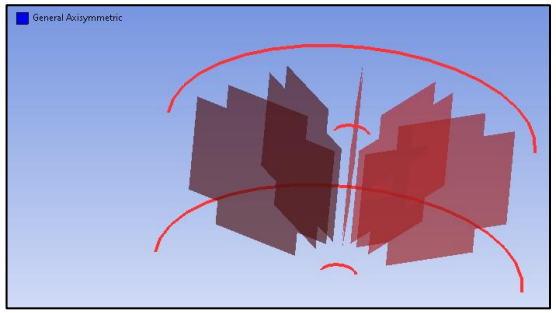
Method	Max Stress (MPA)
Bonded default formula (AL)	8064
Shared Topology	26
MPC Bonded (21R2)	Not solved
Bonded using beams	30
MPC bonded (22R2), all default	29

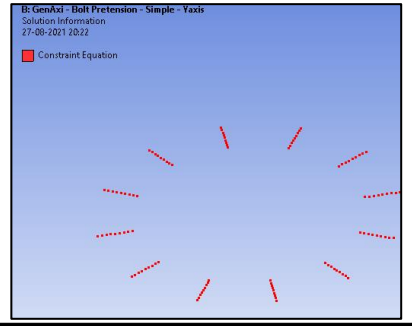


RBE3 type of constraint is also be applied to solid-surf to solid-surf

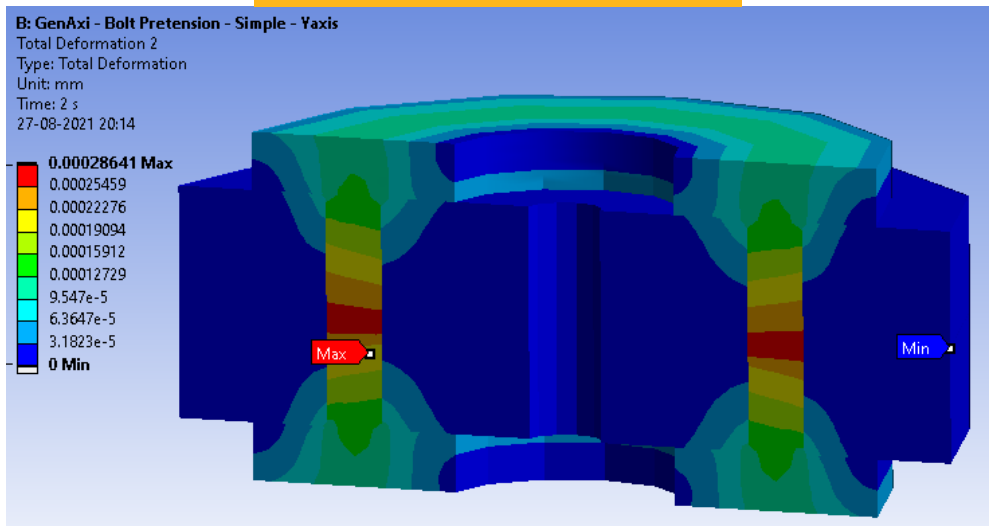
Bolt pretension is Now Supported with General Axisymmetric Elements

- PSMESH command is supported for general axisymmetric elements. It cuts the bolt and generate the pretension elements between all Fourier nodes.
- SLOAD command is also supported for general axisymmetric elements.
- Support adjustment and incremental load as well.
- Use Case: Apply preloads for part assemblies of gas turbines, automotive applications

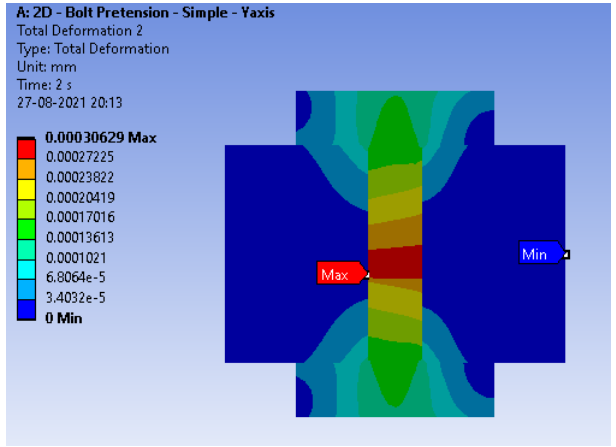




General Axisymmetric



2D Axisymmetric

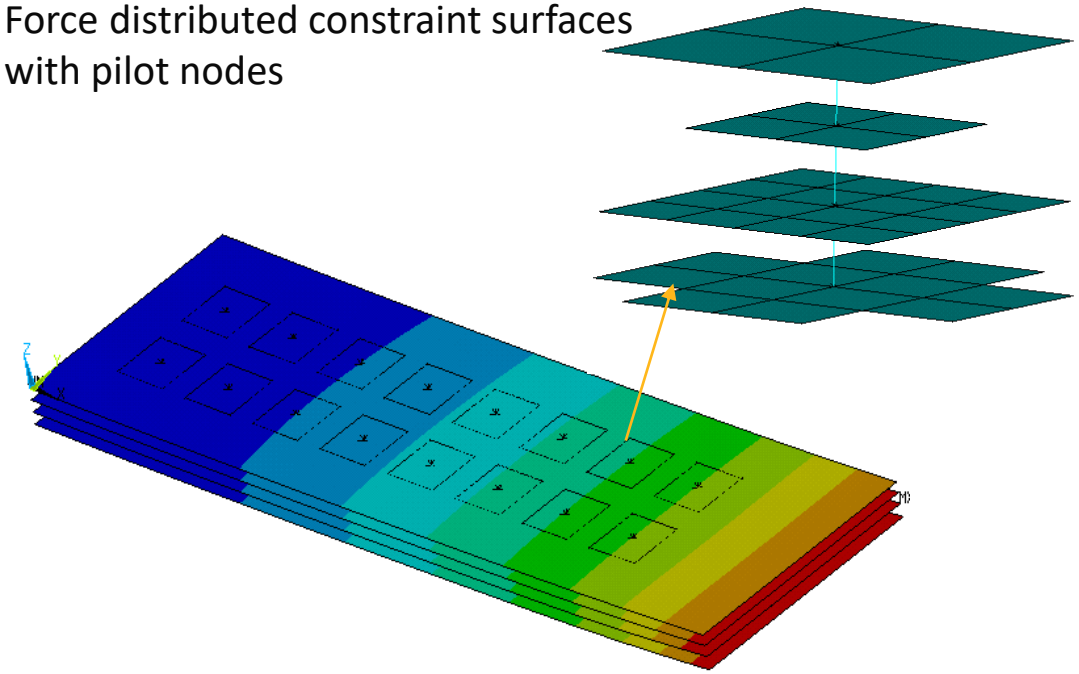


General axisymmetric results are similar as 2D axisymmetric or 3D solid

Mesh Independent Spot Weld Enhancements

- SWGEN/SWADD command:
  - Node-to-surface (175-170) MPC contact pairs are replaced by force distributed constraints with pilot nodes (174-170) to support large rotation framework and to make it more robust.
  - Spot weld generation is 2 times faster

Force distributed constraint surfaces with pilot nodes

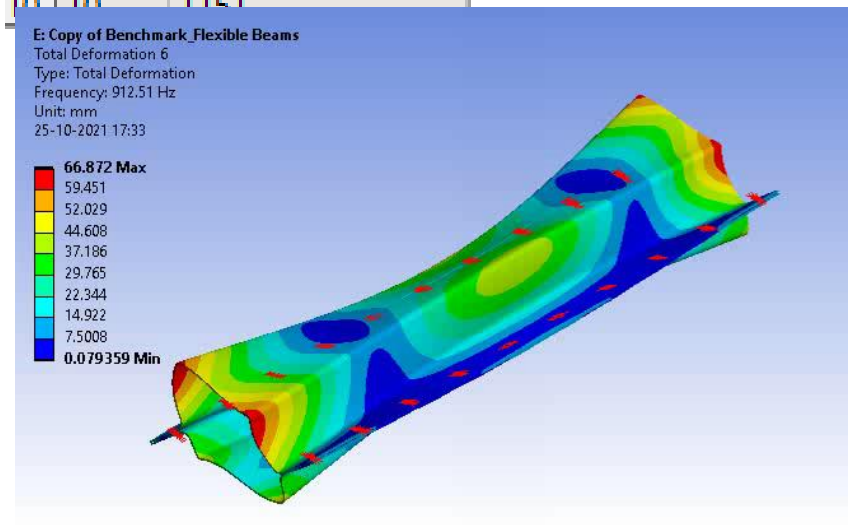


Mode shapes and frequencies are very close to reference results with new spot weld configuration

	Mode	Frequency [Hz]
1	1.	474.41
2	2.	482.49
3	3.	537.25
4	4.	538.37
5	5.	771.37
6	6.	912.51
7	7.	917.97
8	8.	1077.
9	9.	1106.2
10	10.	1151.

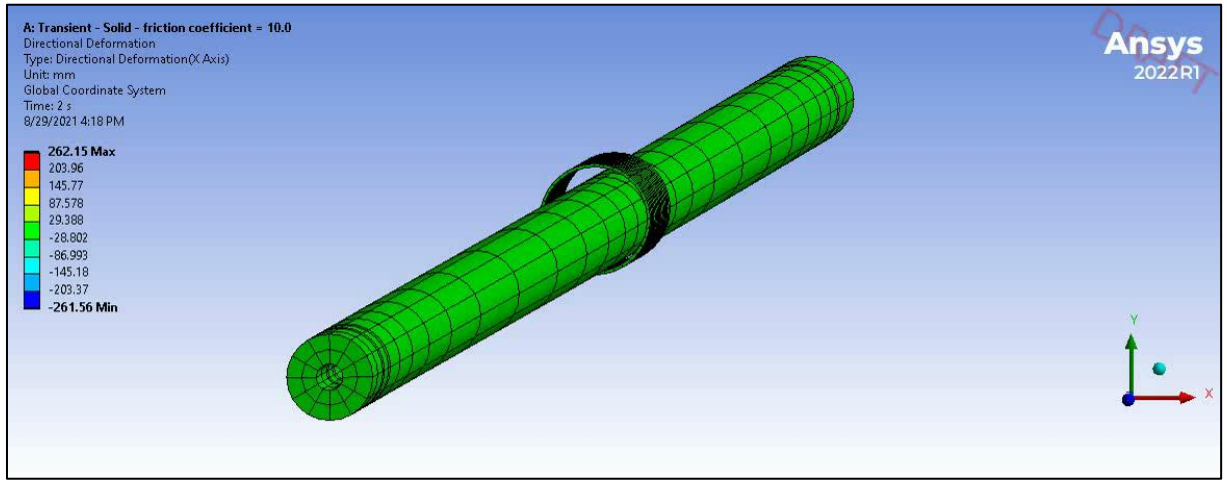
DH1 updating: natural frequency error

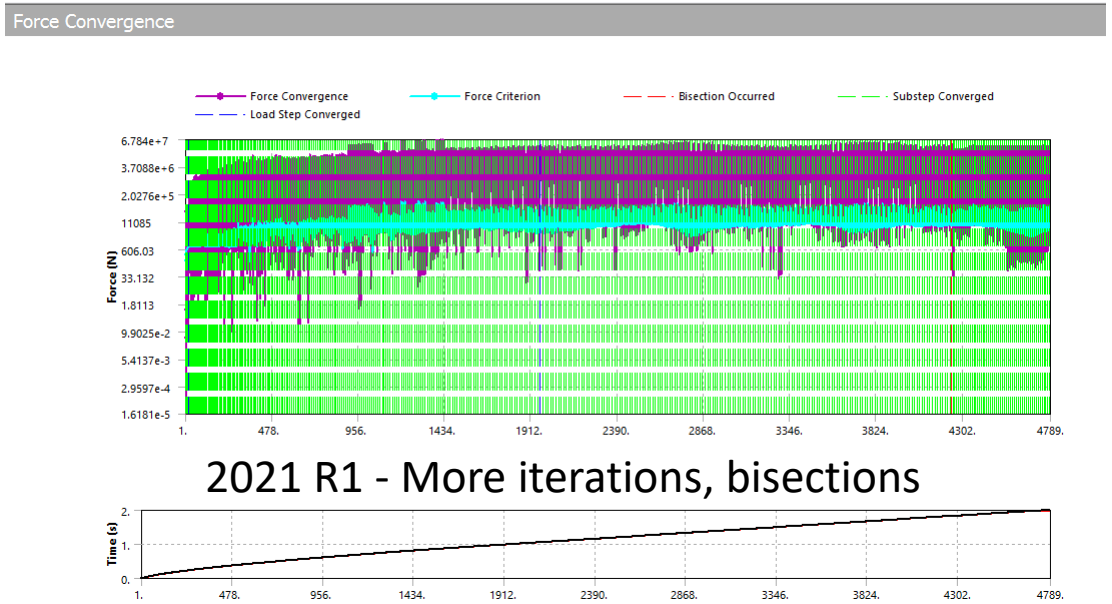
Mode	Description	Experimental (Hz)
1	A1	474.6
2	A2	497.3
3	B2	525.4
4	B1	536.3
5	BOTTOM	768.4
6	TOP	899.1
7	B3	913.3
8	FLANGES	1060.0
9	C3	1110.0
10	C1	1170.0
11	2ND BEND.	1190.0



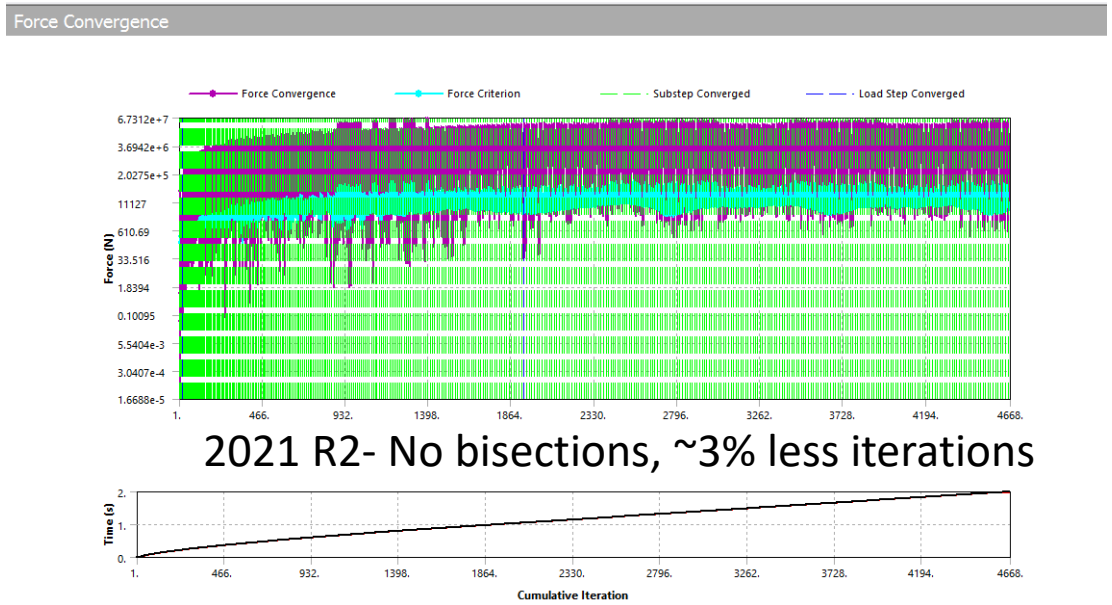
Transient Dynamic Accuracy Improvements: HHT Algorithm

- HHT algorithm heuristics are improved. Stiffness matrix more consistent
- Start of new load step continues in HHT (used to go back to Newmark for 1st substep)
- Restart continues with HHT (used to go back to Newmark for 1st substep)





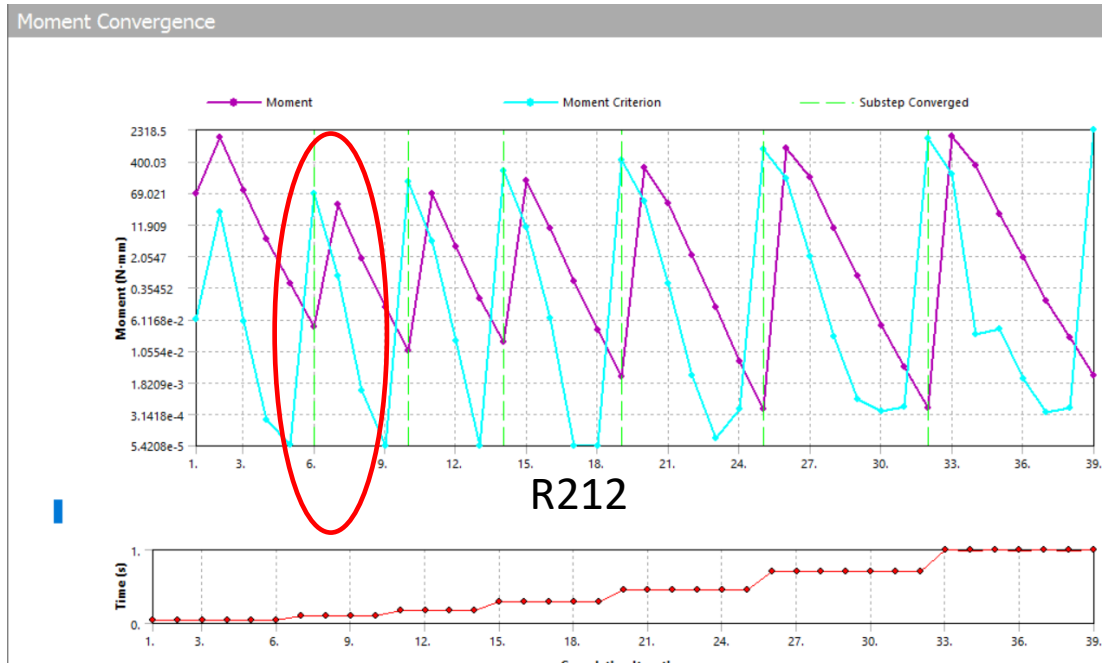
2021 R1 - More iterations, bisections

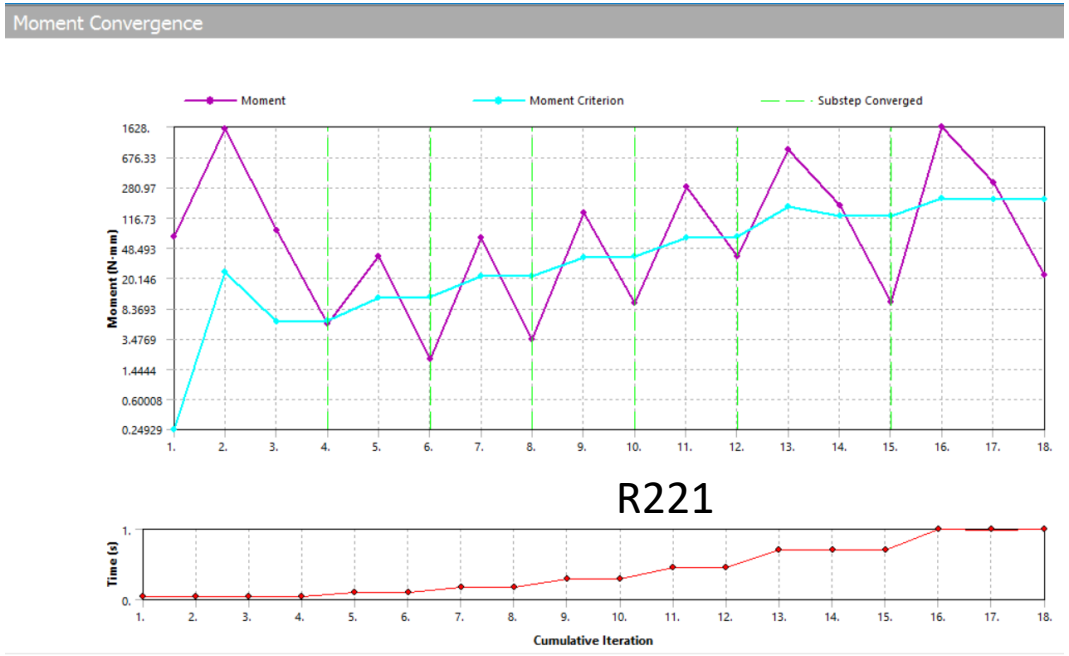


2021 R2- No bisections, ~3% less iterations

Moment Convergence Reference Calculation (Robustness Improvement)

- Improved minimum reference calculation when Moment convergence reference is small
- Minimum reference based on mesh size and bounding box and force reference value results in smooth and accurate convergence





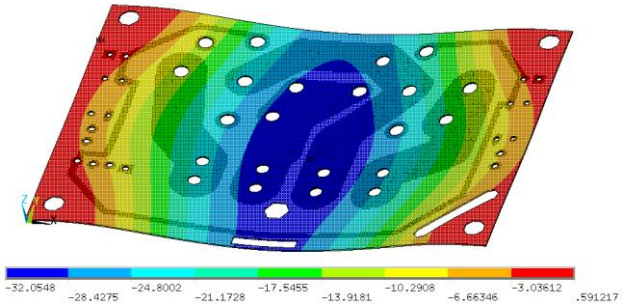
The sudden increase in moment convergence reference in 2021 R2 due to heuristics for “small values”

MAPDL Elements

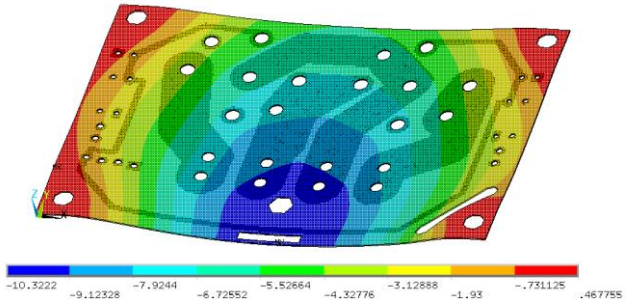
Bending Stiffness for Smeared Reinforcement

- Greatly improves the solution accuracy with 3D smeared reinforcing (REINF) models
- Eliminates the need to use multiple REINF layers to capture the bending stiffness
- Enhances the REINF modeling usability in the new PCP/Chip simulation workflow

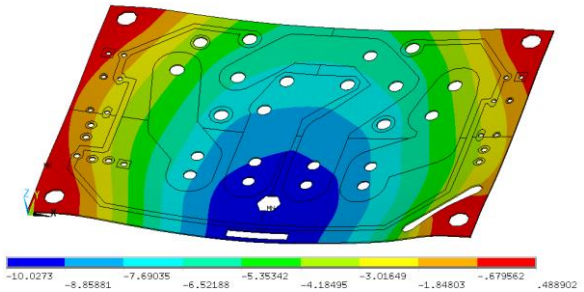
Accurate simulation results: REINF with bending vs. full 3D model



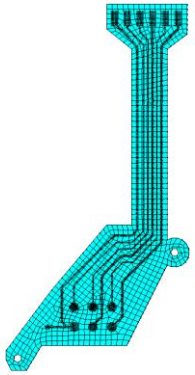
Reinforcing w/o bending stiffness

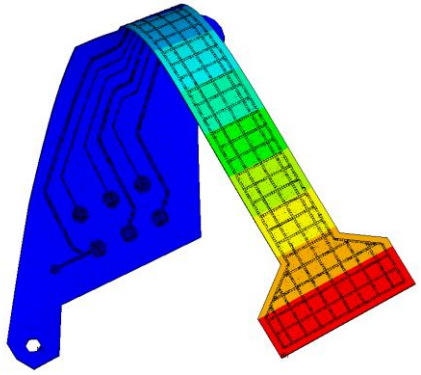


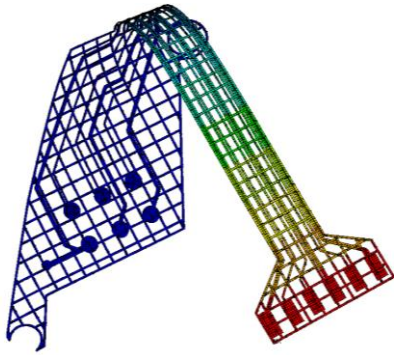
Reinforcing with bending stiffness



Full 3D model (trace modeling)



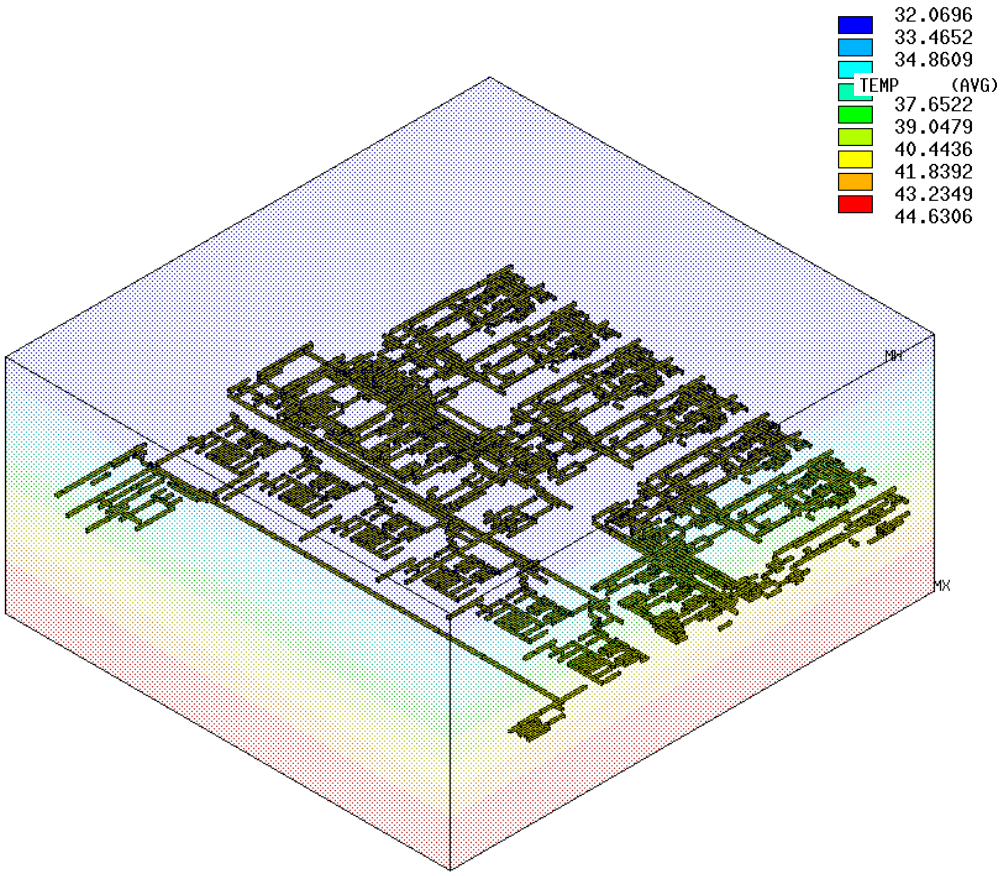




Robust and accurate simulation of flexible PCBs under large deformation

Reinforcing Performance Enhancements

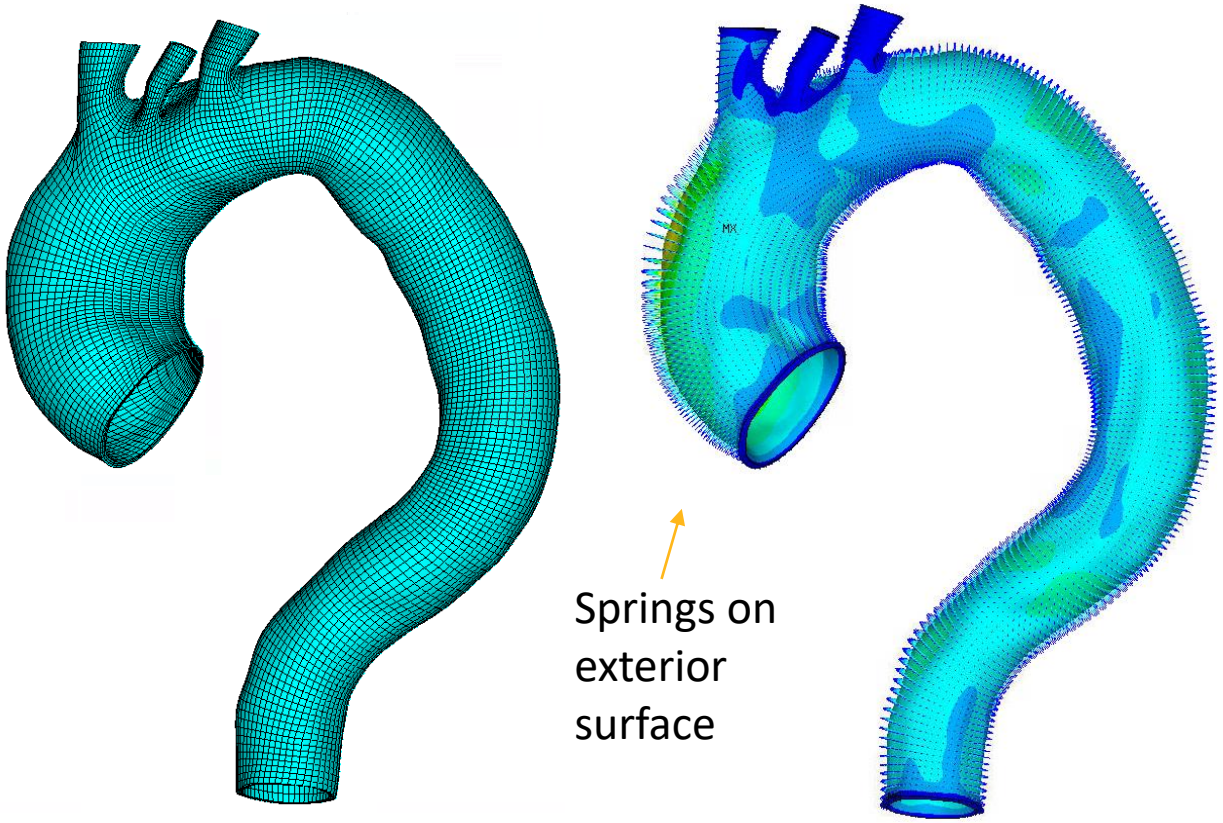
- Motivated by the requirements to account for large models (full PCB and chip models)
- Improved performance in pre-processing
  - Allows large number of reinforcing members in one base element
  - Reduces time needed for load mapping
- Improved solution efficiency
- Improved performance in post-processing
  - Significantly reduced time for querying min/max member results
  - Improved inter-member result smoothing



Inverse Solving with Uni-axial Stiffness Elements

- Motivated by the requirements from biomedical simulation to derive stress free configuration.
- LINK180 and COMBIN39 are now supported in inverse analysis
- Support user-defined nonlinear springs
- Capable of forward simulation with new loading after inverse solving

Inverse analysis of a thoracic aorta. Boundary conditions modeled with spring elements



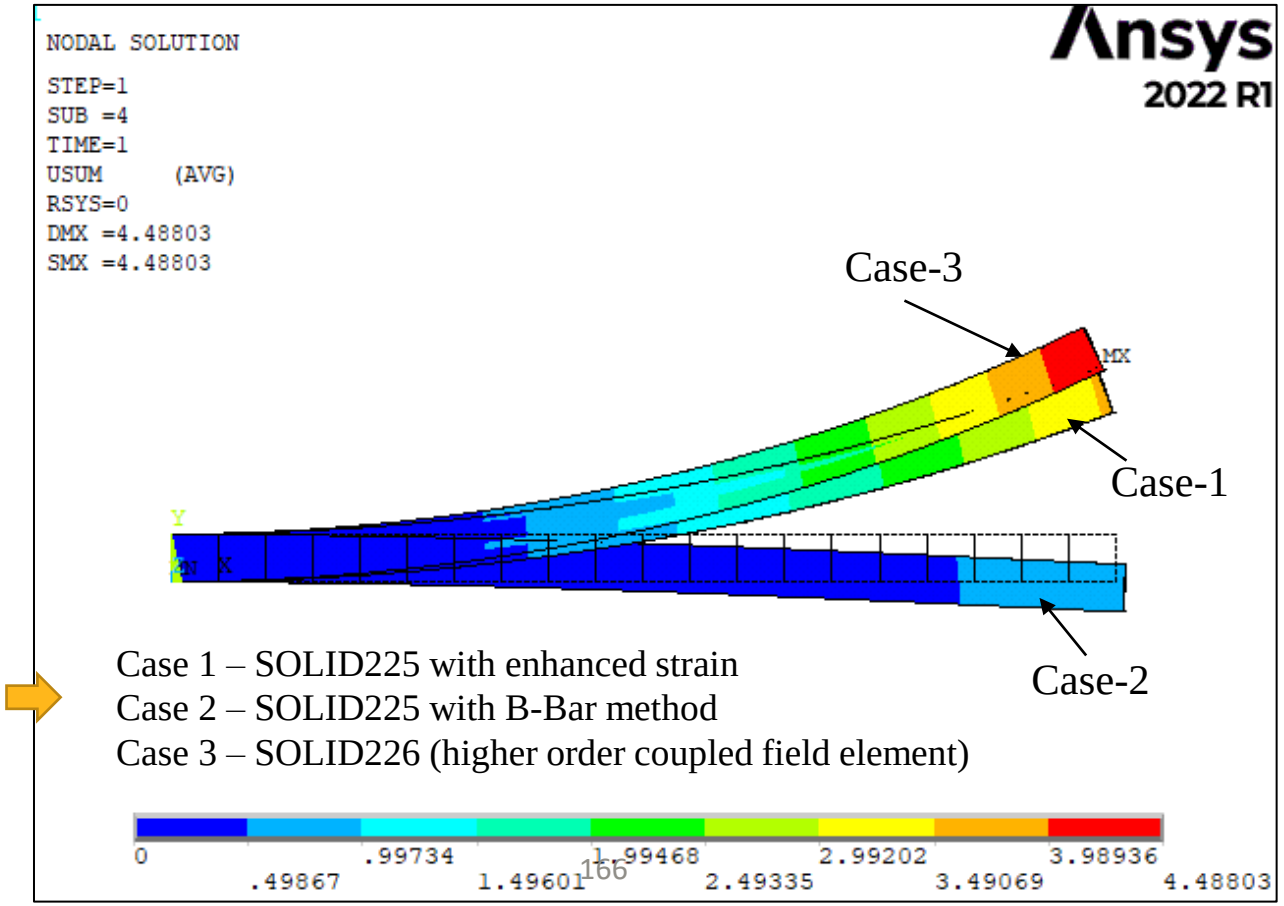
Input geometry under load

Recovered stress free state

Enhanced Strain Formulation for PLANE222 and SOLID225 in Structural-Thermal Analysis

- New enhanced strain formulation improves solution accuracy of lower order coupled-field elements (PLANE222 and SOLID225) for structural-thermal analysis
- Requires less elements to capture the accurate response of bending-dominated problems
- Alleviates volumetric locking issues associated incompressible materials

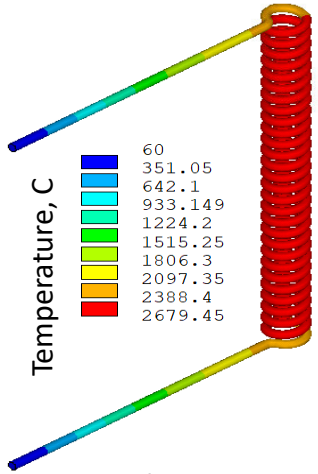
A cantilever beam under varying thermal load. Significant accuracy improvement with enhanced strain SOLID225

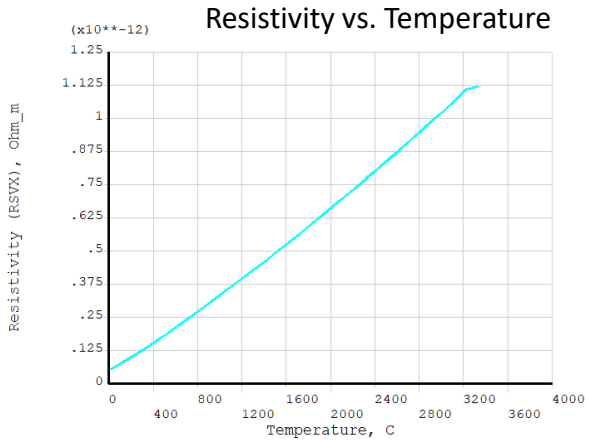


Enhanced Robustness of Coupled-Field Analyses

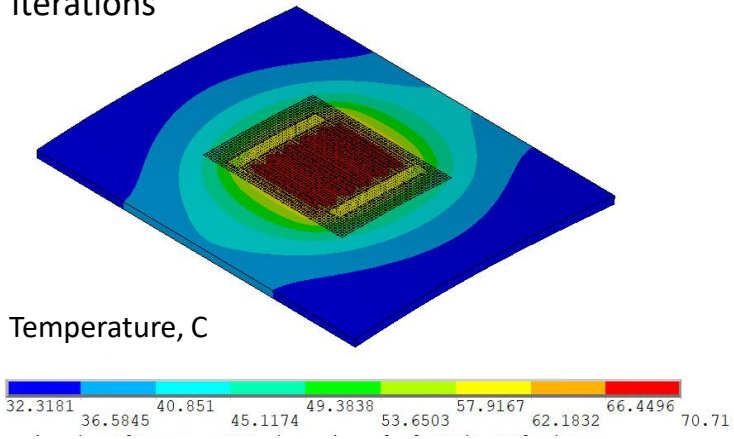
- Consistent element stiffness matrices to account for nonlinear heat generation rates and electrical resistivity
- Greatly reduces number of nonlinear iterations. Overcomes previously non-converging problems
- Modified elements:
  - PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227

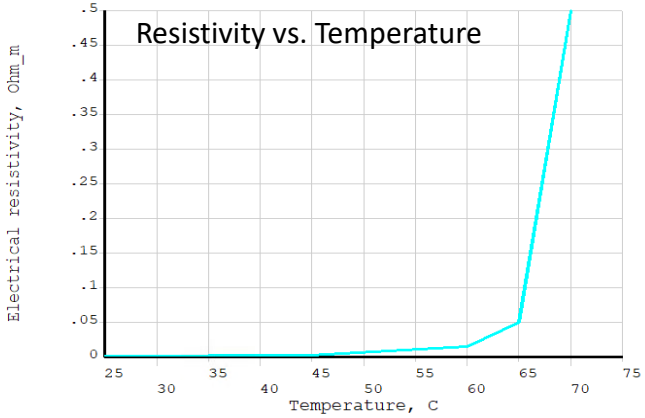
Steady-State Heating of a Tungsten Coil. Solution failed to converge previously. Now solved successfully.





Transient analysis with car seat heating elements, converged with much fewer number of iterations

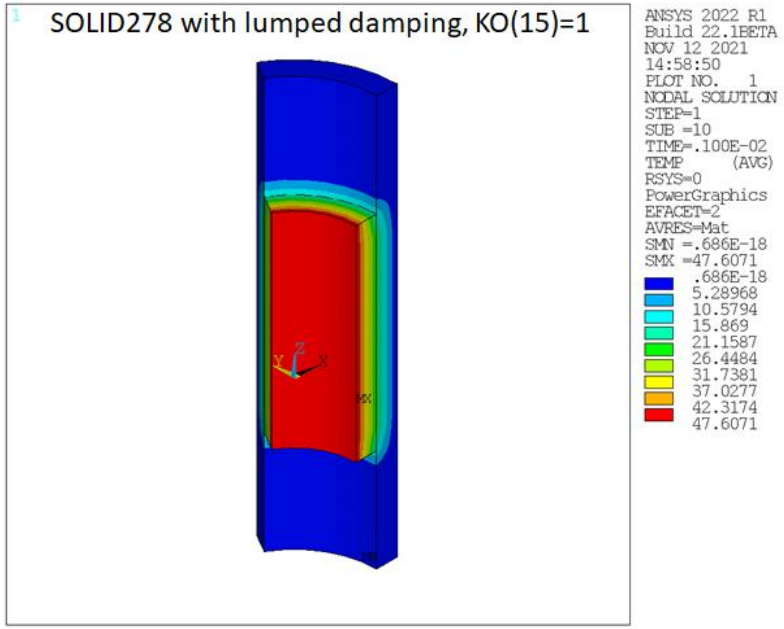


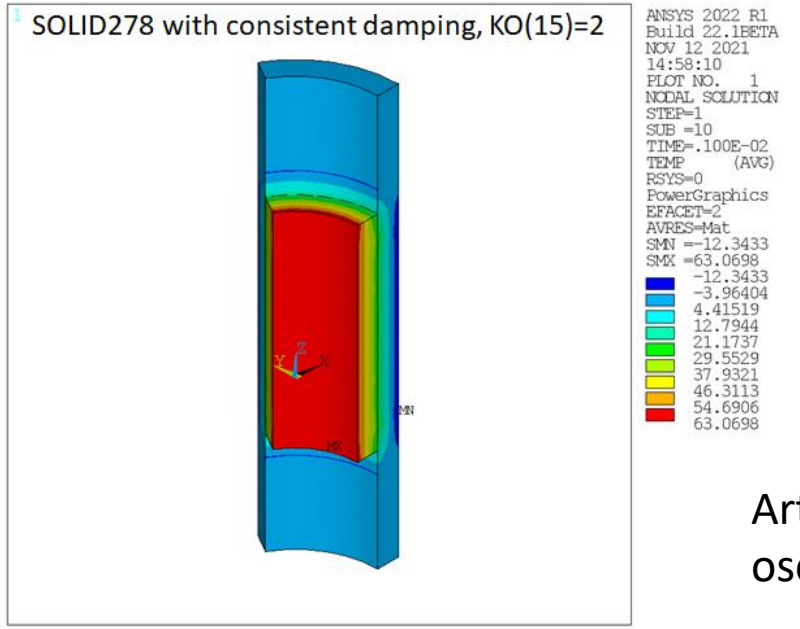


Lumped / Consistent Matrix Options for Thermal Elements

- Oscillations in thermal solution can be controlled with lumped/consistent form of damping (specific heat) matrix and convection matrix.
- A new option is now available to choose these options in current technology thermal elements
- With these settings user has better control on which regions should have lumped or consistent matrices.

Correct TEMP distribution



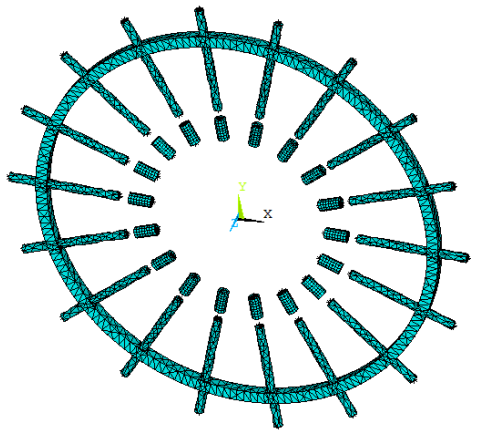


Artificial TEMP oscillation

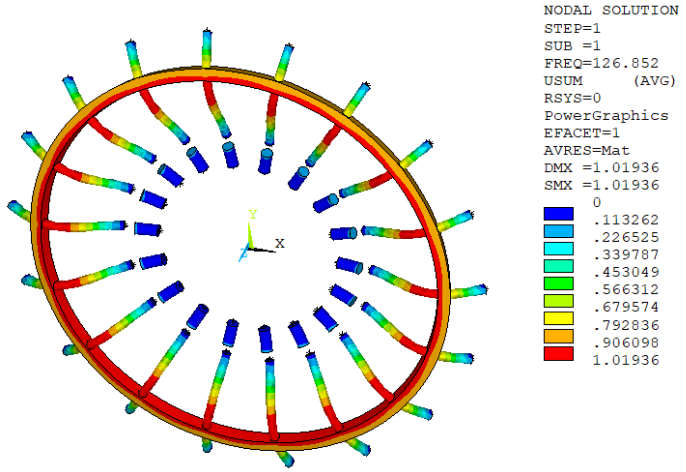
Transient thermal analysis of two concentric cylinders. The non-physical oscillation in temperature result is suppressed with a proper damping matrix option

Joint Element Enhancements

- Motivated by the issue of overconstraint that affects the Lagrange Multiplier method approach, a penalty-based approach to joint elements was implemented
- This method also avoids the introduction of any new solution variables unlike the Lagrange Multiplier method
- The penalty-based joint elements are now supported in modal-based and linear perturbation procedures
- Additionally, the penalty-based spherical joint element now supports locks, stops, and friction



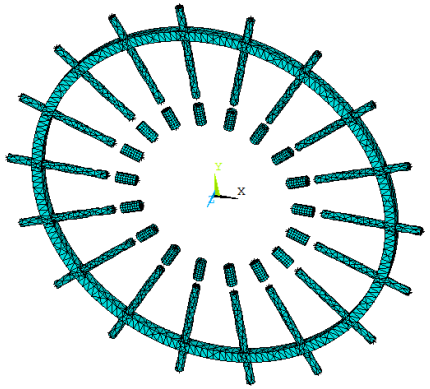
Cyclic Symmetric Boat Wheel Model



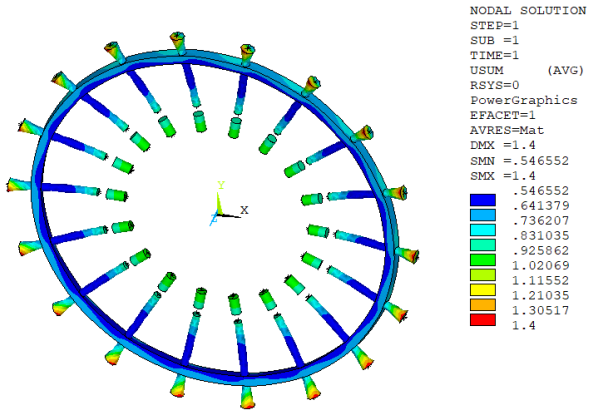
First Mode Shape

Demonstration Example: Linear Perturbation Analysis of a Cyclic Symmetric Boat Wheel

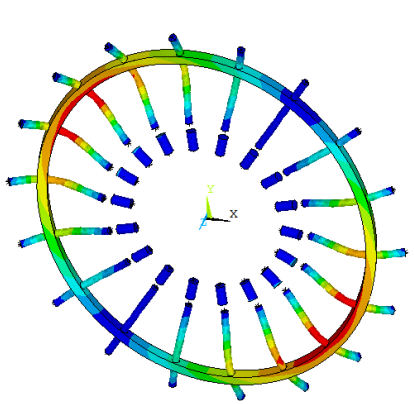
Model: Each spoke has two cylindrical parts connected by a MPC184 revolute joint with linear stiffness specified.
Loading:
Static load step: Radial displacements, axial displacements and surface pressure on outer cylinder
Linear Perturbation step: Modal analysis

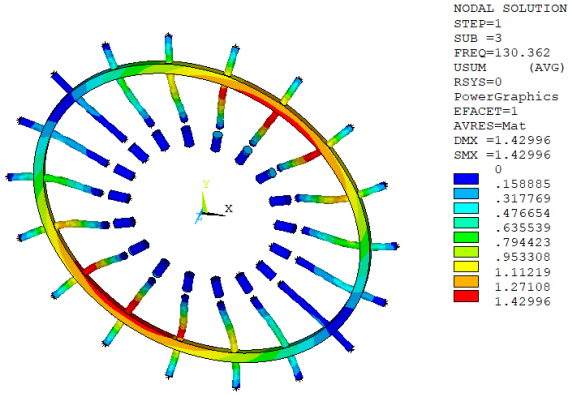


Geometry



Deformation at the end of static load step





2nd and 3rd eigen-mode shapes

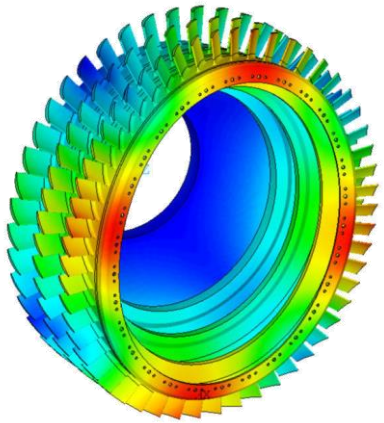
LM vs Penalty Method

SET	TIME/FREQ	
	KO(2)1: Penalty	KO(2)0: LM
1	126.85	126.89
2	130.36	130.4
3	130.36	130.4
4	140.96	140.99
5	140.96	140.99

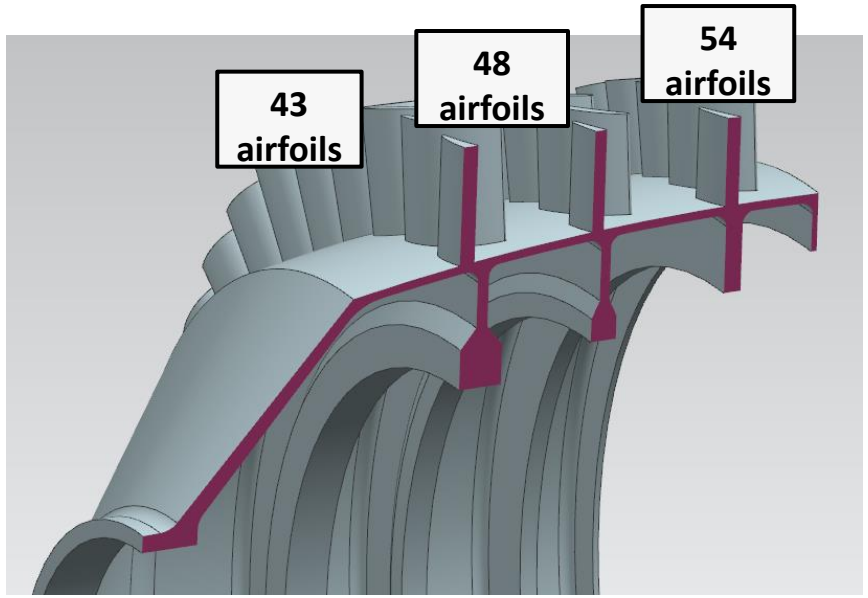
MAPDL Linear Dynamics

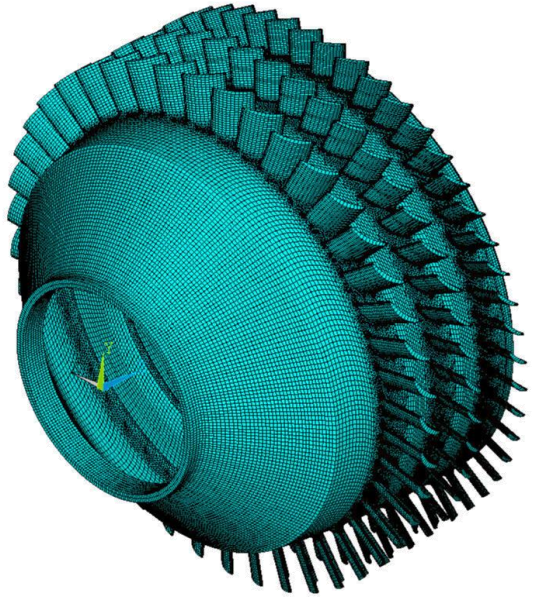
Multistage Cyclic Symmetry

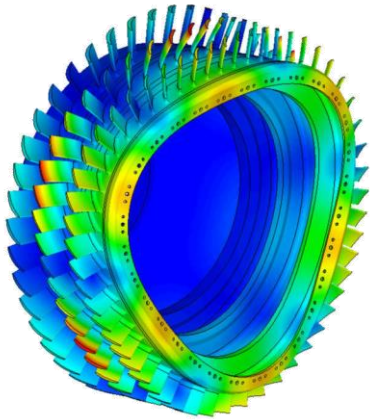
- Multistage cyclic symmetry involves combining distinct cyclic structures (stages) that do not have the same sector count
- This reduced model uses only cyclic sectors and special constraints at the stage interfaces
- This results in a dramatic reductions in model size, simulation time, and file storage



2 nodal diameter mode



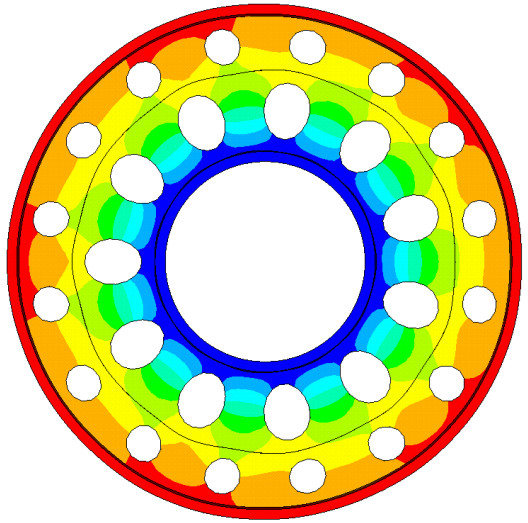




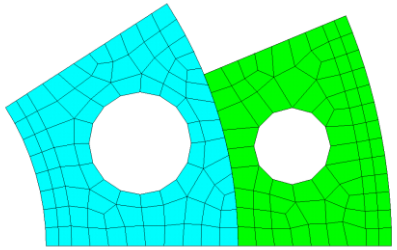
3 nodal diameter mode

Multiharmonic Multistage Cyclic Symmetry

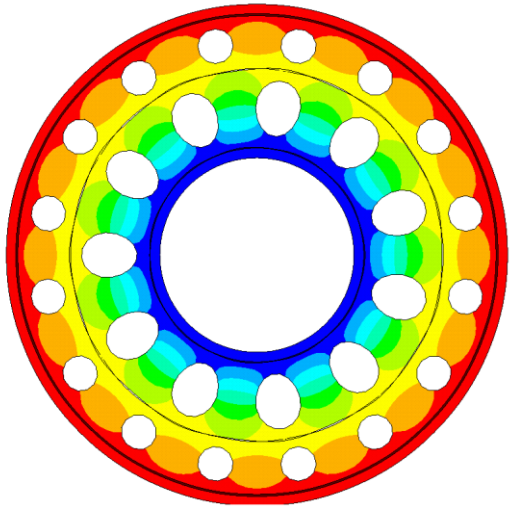
- Because there are different sector counts, multiple cyclic harmonics may contribute for each stage
- Multiple harmonics can be added to improve solution accuracy
- Results for each harmonic can be viewed individually to gain valuable engineering insight into contributions of the overall solution and how loading may affect the structure
- Harmonics can be automatically combined to achieve the expected physical solution



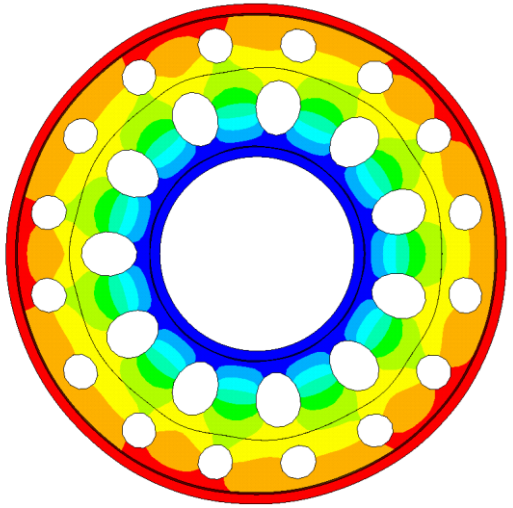
Reference Static Analysis:
Static analysis under rotational load



Multistage:
Static Analysis
HI = 0
Reduced Accuracy



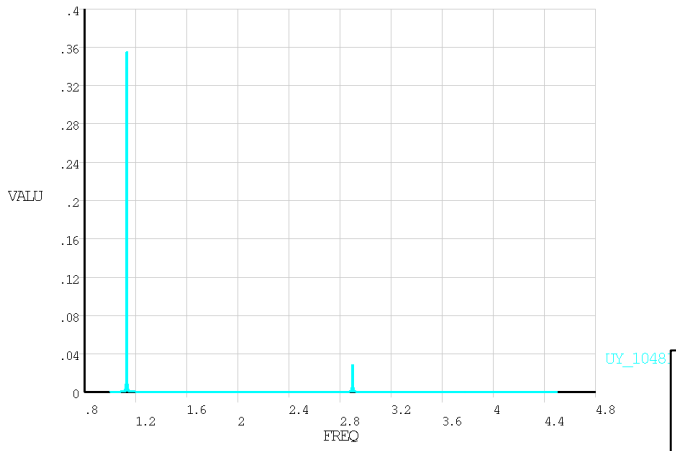
Multistage:
Static Analysis
HI = 0 + HI = 5
High Accuracy

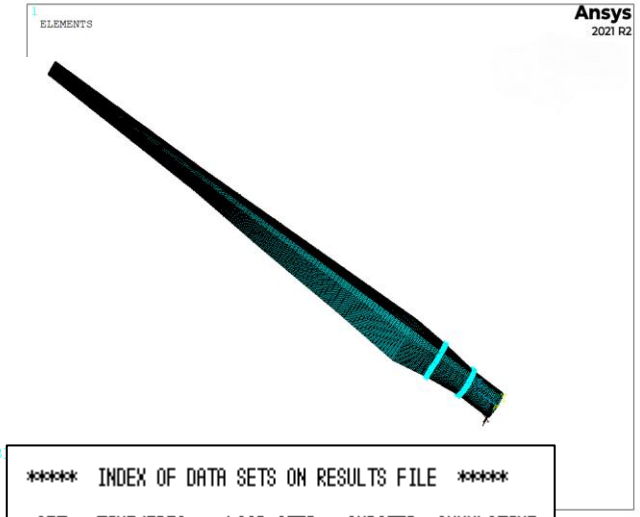


Solution Comparison Using the Frequency Response Function (FRF) Correlation Method: RSTMAC

- FRF correlation analysis between experimental (.unv file) and FE generated FRF at each frequency point is supported.
- Two criteria:
  - Cross-signature assurance criterion (CSAC),
  - Cross-signature scale factor (CSF).
- CSAC is a measure of the shape correlation, an adaptation of MAC for the frequency domain.
- CSF is a measure of the difference in amplitude.

FRF graph at one of the measurement points





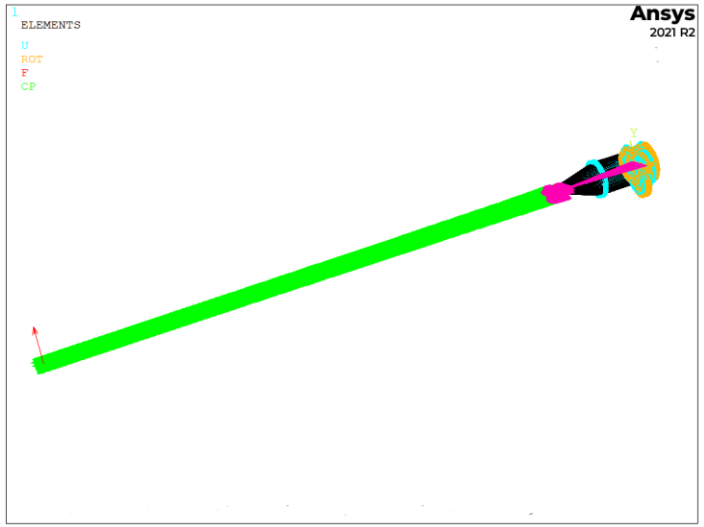
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.25969	1	1	1
2	0.95582	1	2	2
3	1.1308	1	3	3
4	2.8981	1	4	4
5	3.7871	1	5	5
6	4.9242	1	6	6
7	5.9346	1	7	7
8	6.4791	1	8	8
9	7.0171	1	9	9
10	7.4022	1	10	10

Solution Comparison Using the Frequency Response Function (FRF) Correlation Method: RSTMAC

- This feature is enhancing the RSTMAC procedure and can be applied using FRF option on MACOPT command: **MACOPT,FRF**

- Results are presented for a wind turbine blade excited at its end and measurements are stored for different nodes in unv file format.




```
****** Frequency Response Function correlation ******
       Solutions are complex
       Number of substeps:   37

       Frequency:      CSAC        CSF
            1.000      1.000      1.000
            1.088      1.000      1.000
            1.175      1.000      1.000
            1.263      1.000      1.000
            1.350      1.000      1.000
            1.438      1.000      1.000
            1.525      1.000      1.000
            1.613      1.000      1.000
            1.700      1.000      1.000
            1.788      1.000      1.000
            1.875      1.000      1.000
            1.963      1.000      1.000
            2.050      1.000      1.000
            2.138      1.000      1.000
            2.225      1.000      1.000
            2.313      1.000      1.000
            2.400      1.000      1.000
            2.488      1.000      1.000
            2.575      1.000      1.000
            2.663      1.000      1.000
            2.750      1.000      1.000
            2.838      1.000      1.000
            2.925      1.000      1.000
            3.013      1.000      1.000
            3.100      1.000      1.000
            3.188      1.000      1.000
            3.275      1.000      1.000
            3.363      1.000      1.000
            3.450      1.000      1.000
            3.538      1.000      1.000
            3.625      1.000      1.000
            3.713      1.000      1.000
            3.800      1.000      1.000
            3.888      1.000      1.000
            3.975      1.000      1.000
            4.063      1.000      1.000
            4.150      1.000      1.000
```

MAPDL Materials

Hyperelasticity with Embedded Fibers

- Isotropic hyperelastic materials can be augmented with embedded fiber terms
  - For example: $W = W_{iso} + \boxed{\frac{c_1}{2c_2}\left\{\exp\left[c_2\left(\bar{I}_4 - 1\right)^2\right] - 1\right\}}$ → Embedded fiber strain energy
  - Up to 5 fibers can be added to the isotropic model
  - The fibers can have different tension and compression properties

Tension → $$W_{fib} = \frac{c_1^t}{2c_2^t}\left\{\exp\left[c_2^t\left(\bar{I}_4 - 1\right)^2\right] - 1\right\}$$

Compression → $$W_{fib} = \frac{c_1^c}{2c_2^c}\left\{\exp\left[c_2^c\left(\bar{I}_4 - 1\right)^2\right] - 1\right\}$$

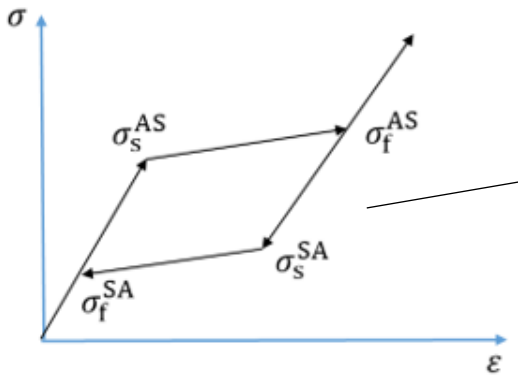
For typical fiber behavior, define the properties so the fibers only support tension:

$$c_1^t \neq 0, \quad c_2^t \neq 0, \quad c_1^c = 0, \quad c_2^c = 0$$

Current Ansys SMA Material Models: TB,SMA

SMA model for superelasticity: TB,SMA,,,,**SUPE**

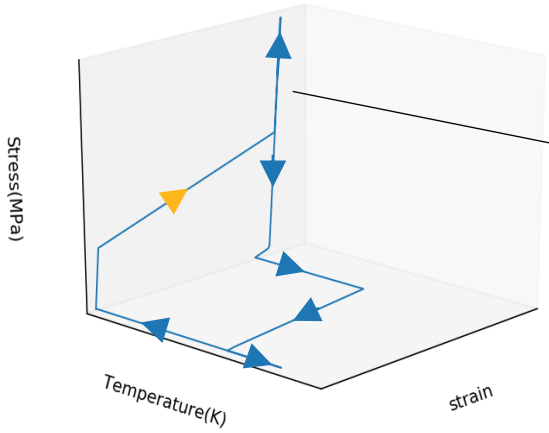
```
MP,EX,1,60000.0
MP,NUXY,1,0.36
Define SMA material properties
TB,SMA,1,,,SUPE
TBDATA,1, 520, 600, 300, 200, 0.07, 0.0
```



no shape memory effect

SMA model with shape memory Effect: TB,SMA,,,,**MEFF**

```
MP,EX,1,60000.0
MP,NUXY,1,0.36
Define SMA material properties
TB,SMA,1,,,MEFF
TBDATA,1,1000, 223, 50, 2.1, 0.04, 45000
TBDATA,7,0.05
```



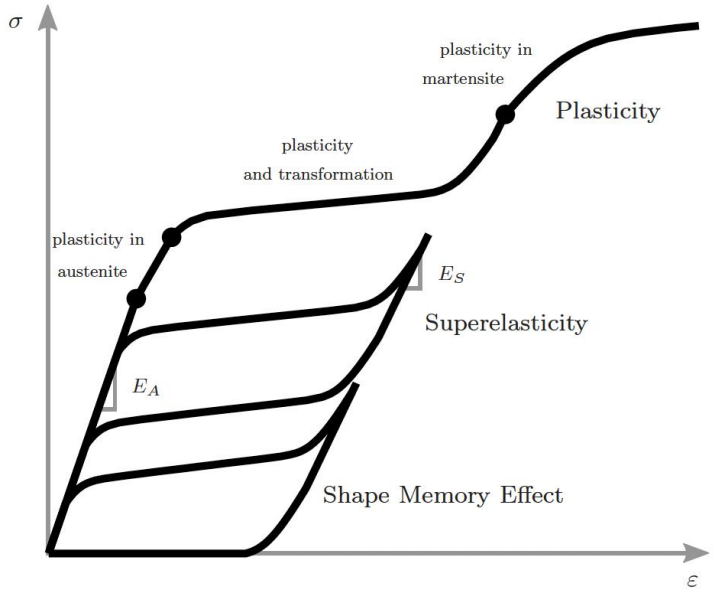
no plasticity

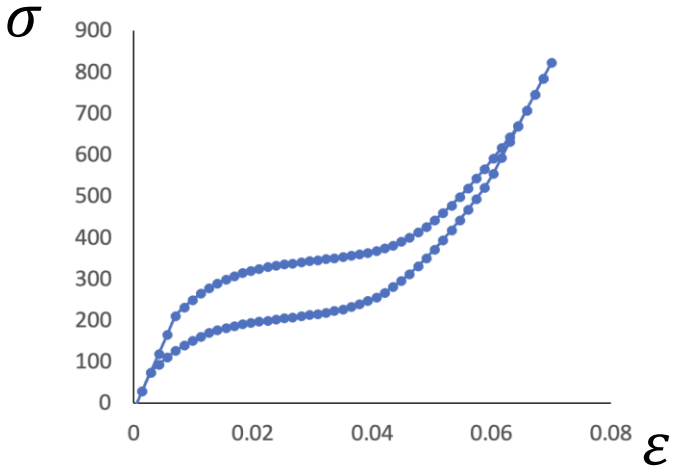
New SMA Model

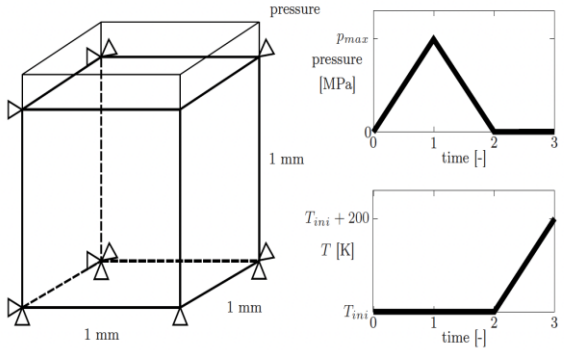
Table 1: Comparison of SMA models based on main features. Note that E is Young's modulus and TC is tension-compression.

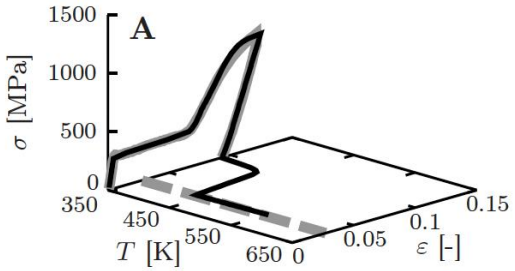
Feature	Present Work	Ansys SUPE	Ansys MEFF	Abaqus
Superelasticity	✓	✓	✓	✓
SME	✓	X	✓	X
Plasticity	✓	X	X	✓
Reorientation	✓	✓	✓	✓
Phase dependent E	✓	✓	✓	✓
TC asymmetry	X	✓	✓	✓
Plasticity in austenite	✓	X	X	X
Simultaneous plasticity and transformation	✓	X	X	X
Smooth transformation	✓	X	X	X

main features of the present SMA model







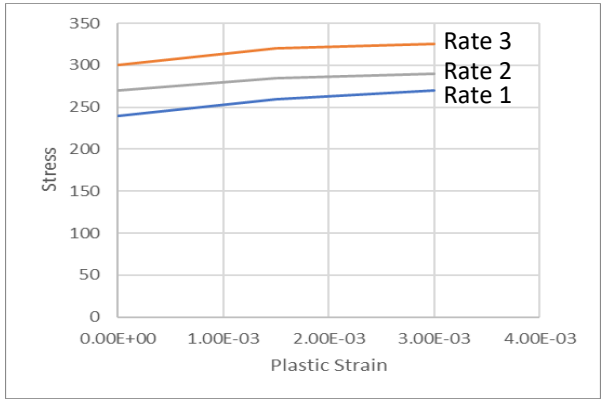


Plastic-Strain-Rate-dependent Isotropic Hardening

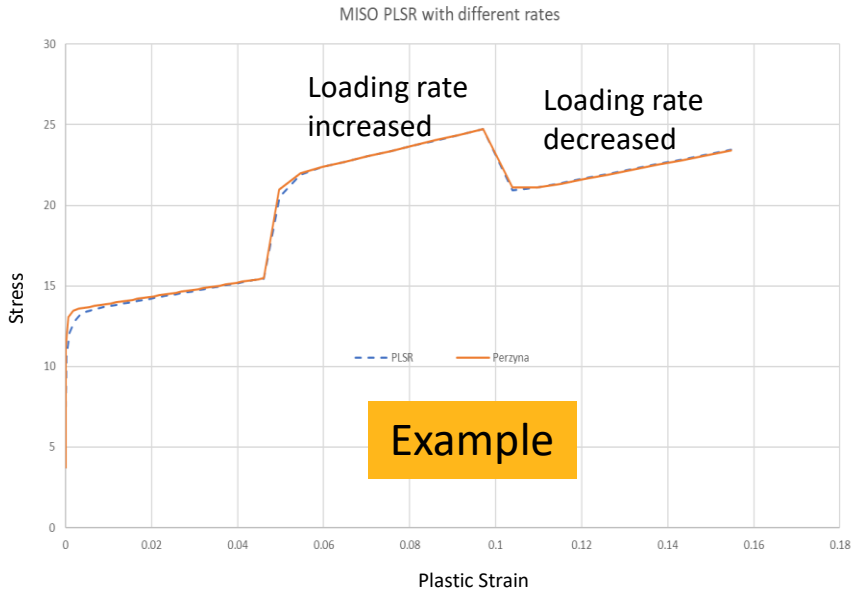
- Direct entry of isotropic hardening data as a function of equivalent plastic strain rate
  - Define tabular data at various plastic strain rates using TBFIELD,PLSR
- Applicable to all isotropic hardening models
  - Bilinear isotropic hardening (TB,PLAS,,,,BISO)
  - Multilinear isotropic hardening (TB,PLAS,,,,MISO)
  - Nonlinear isotropic hardening (TB,NLISO,,,,VOCE/POWER)
- Consistent tangent considers plastic strain rate term:

$$\frac{\partial \sigma}{\partial \varepsilon^{pl}} = \frac{\partial \sigma}{\partial \varepsilon^{pl}} + \frac{\partial \sigma}{\partial \dot{\varepsilon}^{pl}} \frac{\partial \dot{\varepsilon}^{pl}}{\partial \varepsilon^{pl}} = \frac{\partial \sigma}{\partial \varepsilon^{pl}} + \boxed{\frac{1}{\partial t} \frac{\partial \sigma}{\partial \dot{\varepsilon}^{pl}}}$$

- The direct entry method can be used as an alternative to traditional rate-dependent models
  - Allows users to directly use experimental data at different rates
  - Avoids fitting of rate-dependent material model parameters



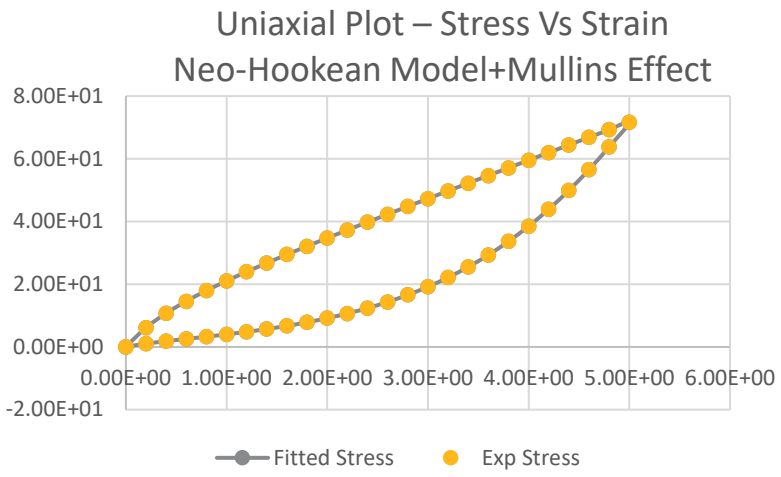

```
TB,PLAS,1,,,MISO
TBFIELD,PLSR,0.
TBPT,DEFI,0.0,15.0
TBPT…

TBFIELD,PLSR,rate2
TBPT,DEFI,0.0,18.624
TBPT…

TBFIELD,PLSR,rate3
TBPT,DEFI,0.0,20.483
TBPT…
```

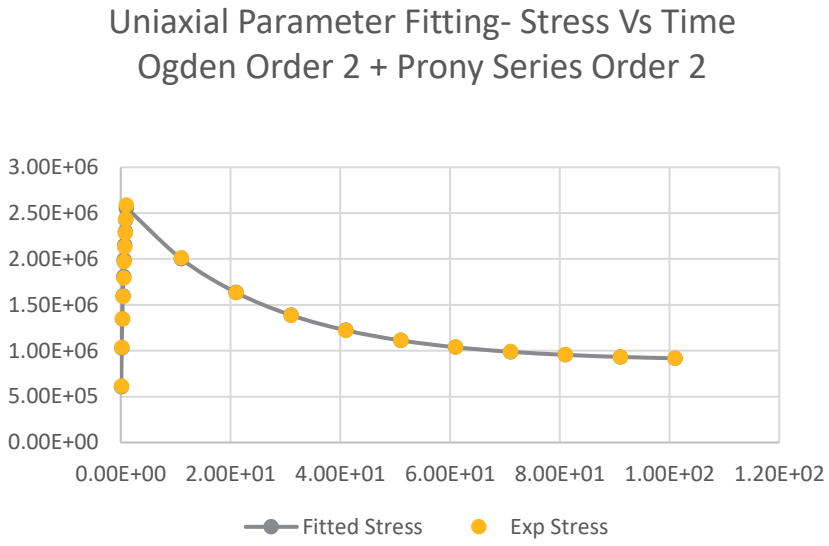


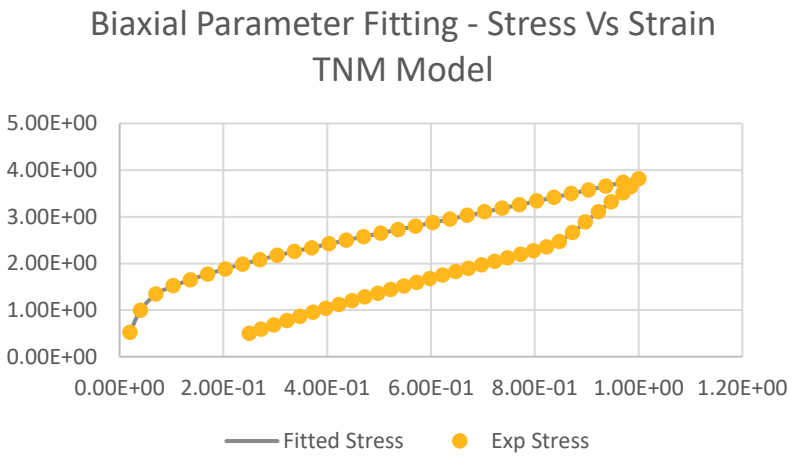
HyperElastic Parameter Fitting with AML

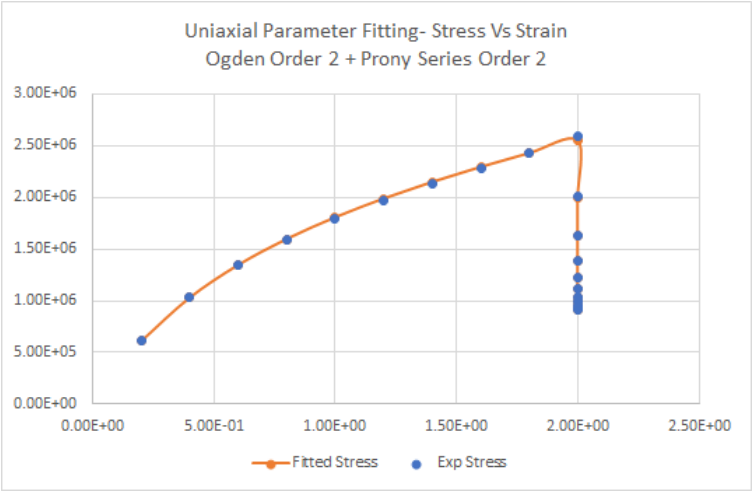
- Parameter Fitting has been expanded to support hyperelasticity and the following listed combinations
  - Hyperelasticity
  - Hyperelasticity with Mullins effect
  - Hyperelasticity with Prony Series
- Experimental Types
  - Uniaxial ; Biaxial; Pure Shear, Simple Shear and Volumetric
- Individual material models supported
  - Mooney Rivlin, Polynomial, Yeoh, Arruda Boyce, Gent, Ogden Foam, Blatz-Ko, NeoHookean, TNM Model(support extended to all exp types), Extended Tube, Bergstrom-Boyce
  - Mullins Effect
  - Prony Series (Shear and Bulk Options)
- MAPDL GUI support via AML for previously supported hyperelastic models in the older GUI.
- Plotting available via TBFPLOT macro for all models

HyperElastic Parameter Fitting with AML









MAPDL Fracture

Stress-Ratio Based Fatigue Crack Closure Models

SMART – fatigue crack growth

Crack growth rate model
(Paris/Tabular fatigue law)

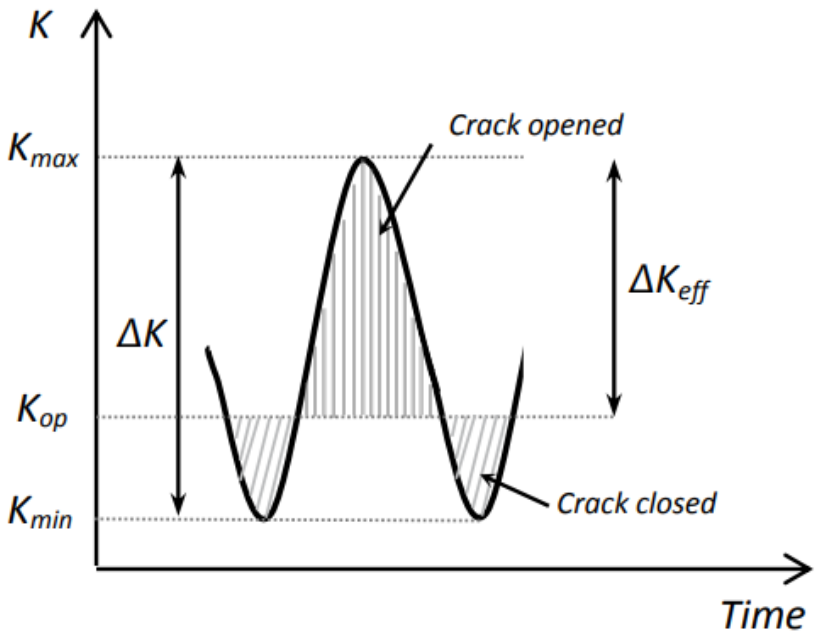
+

Crack-closure model
[optional]

Measure of crack-closure

$$U = \frac{\Delta K_{eff}}{\Delta K}$$

$$\frac{da}{dN} = g(\Delta K_{eff}) = g(U\Delta K)$$

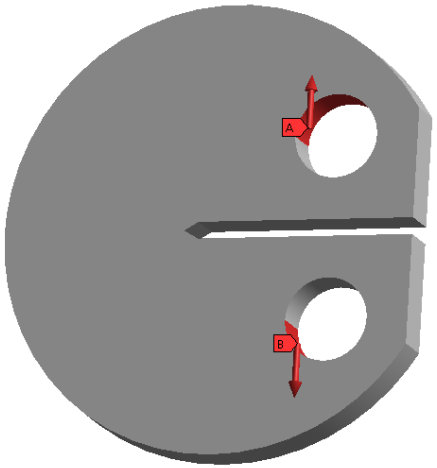


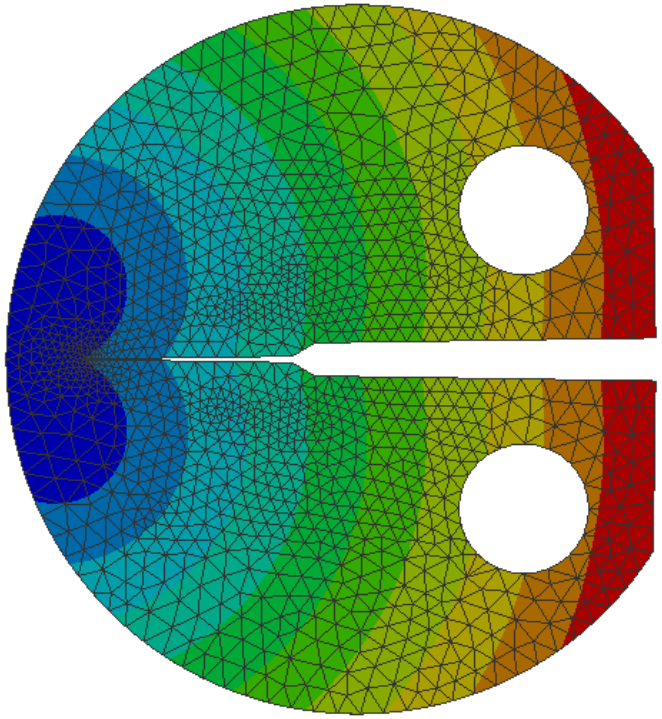
$$R = \frac{K_{\min}}{K_{\max}}$$ stress ratio

Fatigue crack closure models	
Elber function	$U = 0.5 + 0.4R$
Schijve function	$U = 0.55 + 0.33R + 0.12R^2$
Newman function	$U = \frac{1 - f(R)}{1 - R}$ *f(R)* is Newman crack opening function
Polynomial function	$U = A_0 + A_1R + .. + A_nR^n$

Example: Paris Law Combined with Crack Closure Effect

Disk-shaped CT specimen





P = 5kN

Stress ratio **R = 0.2**

Paris law

(C=1.3E-10, m=2.08)

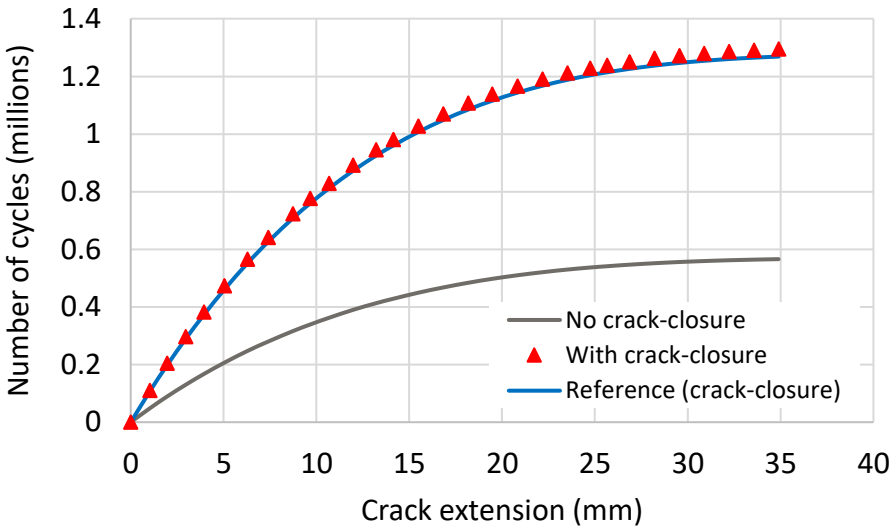
+

Polynomial crack-closure function

(A_0=0.6, A_1=0.35, A_2=0.05)

$$\frac{da}{dN} = C\left(U\Delta K\right)^m$$

$$U = 0.6 + 0.35R + 0.05R^2$$



Stress-ratio dependent fatigue crack growth law

SMART – fatigue crack growth

New field variable - SRAT (stress ratio)

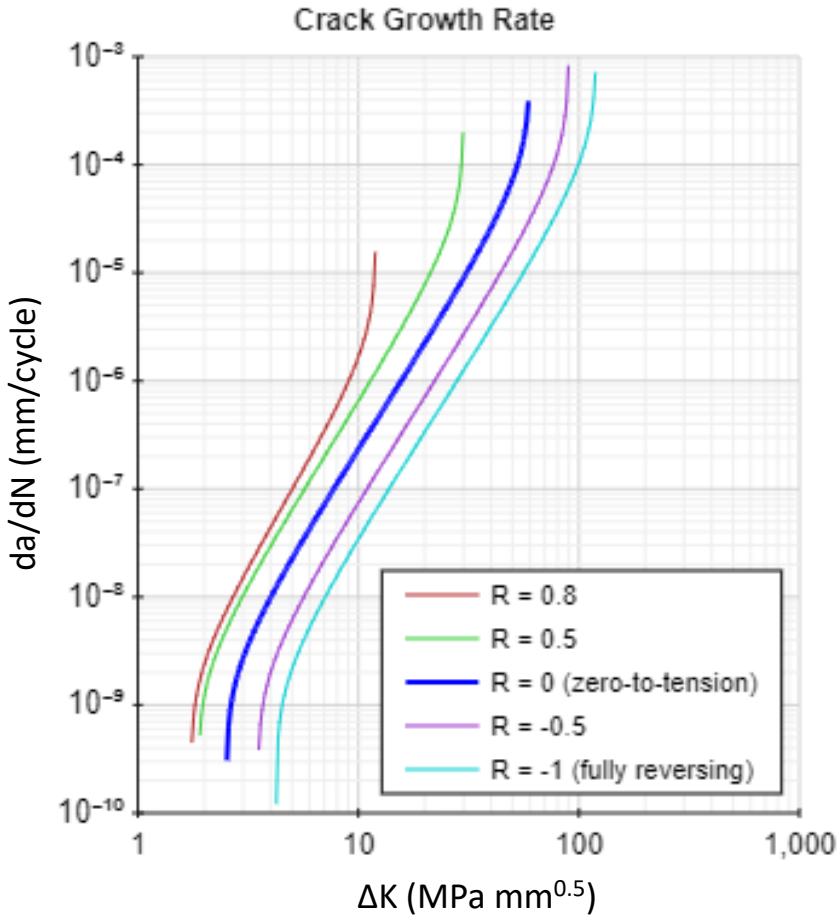
- useful to specify stress ratio R dependent parameters in a crack-growth rate model
- supported with Paris' law (PARIS) and tabular fatigue law (TFDK) options

TB,CGCR,*matID*,,,PARIS

TBFIELD,**SRAT**,R_1
TBDATA,1,C_1,m_1
..
..
TBFIELD,**SRAT**,R_n
TBDATA,1,C_n,m_n

TB,CGCR,*matID*,,,TFDK

TBFIELD,**SRAT**,R_1
TBPT,,DK_{11},$(da/dN)_{11}$
TBPT,,DK_{12},$(da/dN)_{12}$
..
..
TBFIELD,**SRAT**,R_n
TBPT,,DK_{n1},$(da/dN)_{n1}$
TBPT,,DK_{n2},$(da/dN)_{n2}$
..

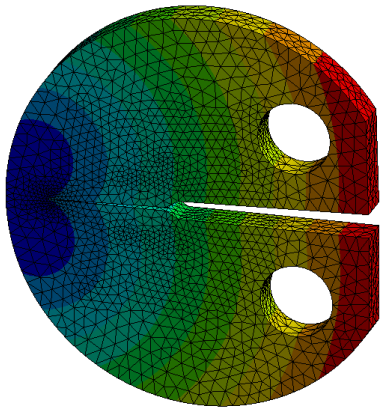


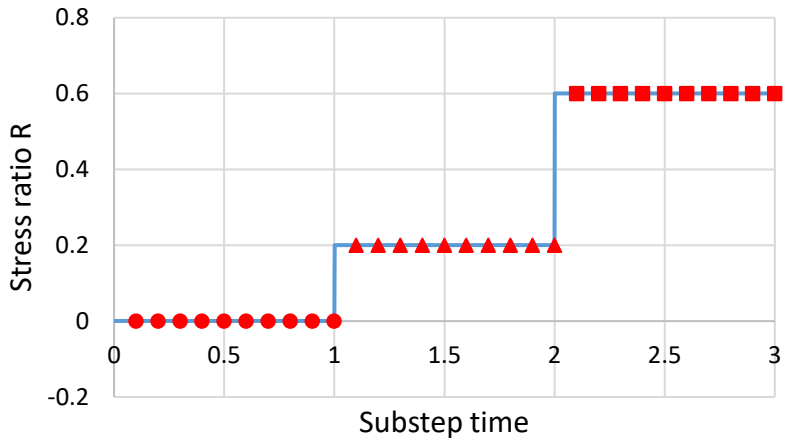
Fatigue crack-growth curves for a typical alloy at different stress ratios (R)

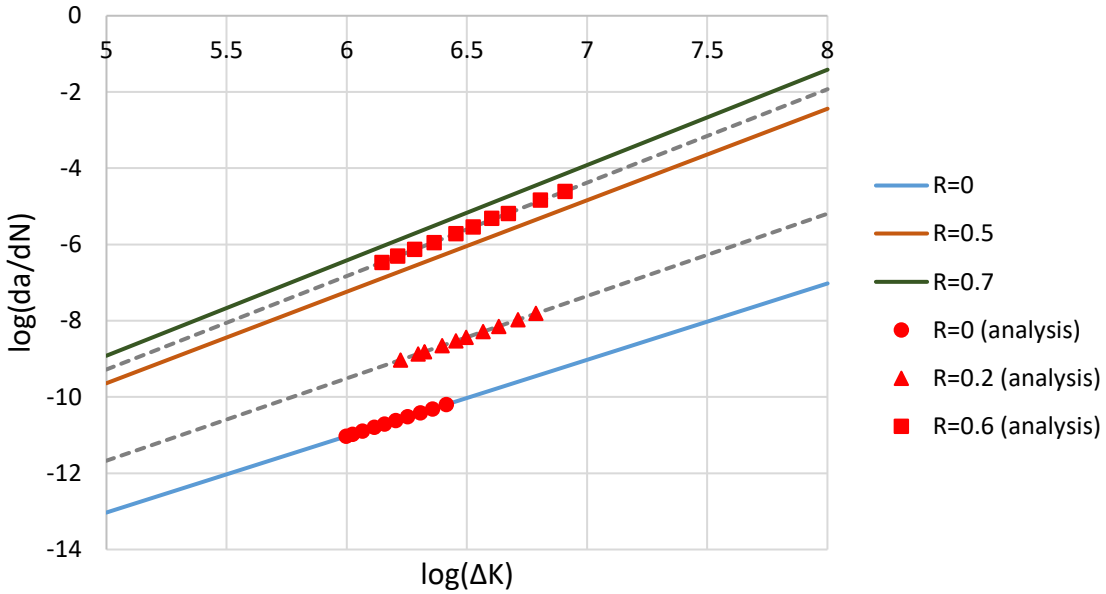
Example: Stress-Ratio Dependent Fatigue Crack Growth Law

R dependent Paris' law

R	C	m
0	1E-10	2
0.5	4E-10	2.4
0.7	5E-10	2.5







Interpolation at intermediate R

Temperature/Time Dependent Fracture Criterion for Static Crack Growth

KIC (temp, time)
JIC (temp, time)

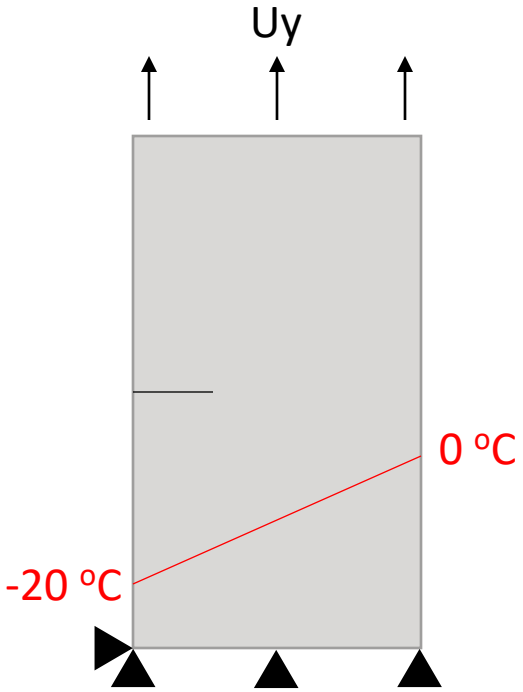
SMART – static crack growth

- Fracture criterion
  - KIC – critical stress-intensity-factor
  - JIC – critical J-integral
- 2022 R1 allows users to specify **KIC or JIC as function of temperature and/or time**
- Useful for crack-growth analysis, such as
  - non-uniform temperature field, T(x,y,z)
  - fracture toughness varying with time (e.g., due to corrosion)
  - temperature varying with time, T(t)

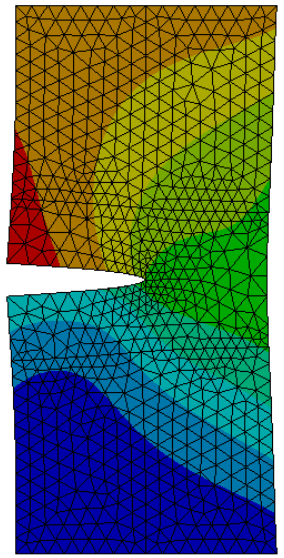
TB,CGCR,*matID*,,,KIC
TBFIELD,**TEMP** (or **TIME**),val_1
TBDATA,1,KIC_1
..
TBFIELD,**TEMP** (or **TIME**),val_n
TBDATA,1,KIC_n

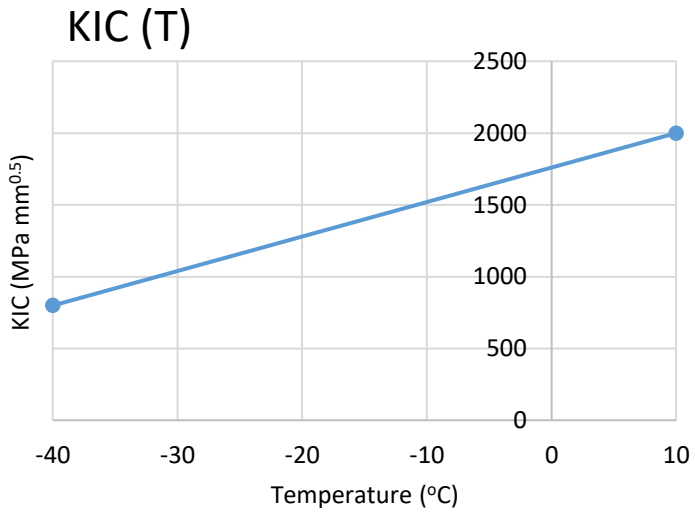
TB,CGCR,*matID*,,,JIC
TBFIELD,**TEMP** (or **TIME**),val_1
TBDATA,1,JIC_1
..
TBFIELD,**TEMP** (or **TIME**),val_n
TBDATA,1,JIC_n

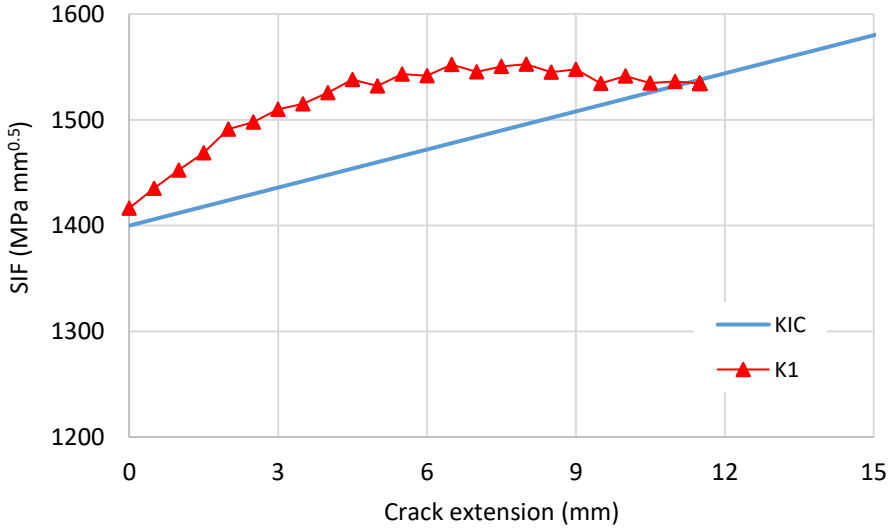
Example: Temperature-Dependent Fracture Criterion



a_0 = 10 mm
W = 40 mm
Uy = 0.075 mm





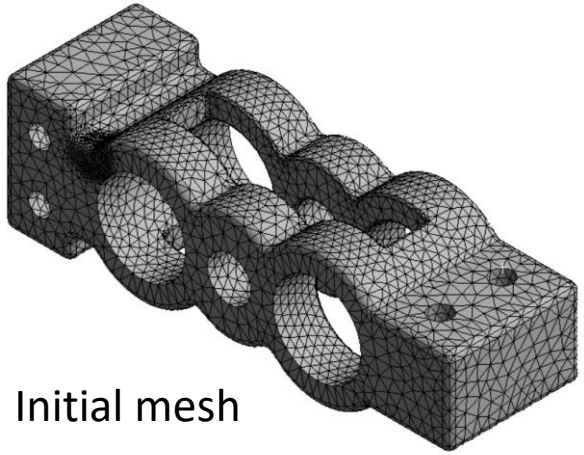


- Spatially-varying temperature field
- Temperature-dependent KIC
- Static crack growth stops when KIC exceeds K1

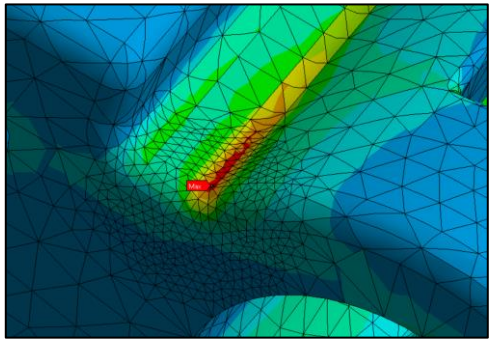
Adaptive Crack Initiation/Insertion: Bracket Model

Automatic crack initiation based on principal stress criterion

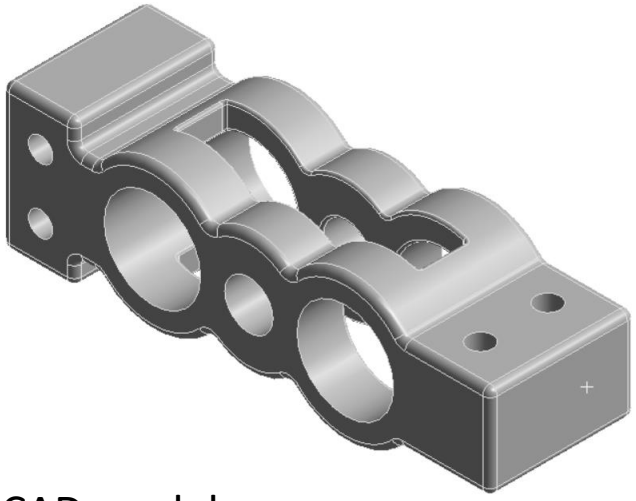
- Predefined elliptical crack will be inserted when the criterion is reached
- Subsequent crack growth follow SMART procedure



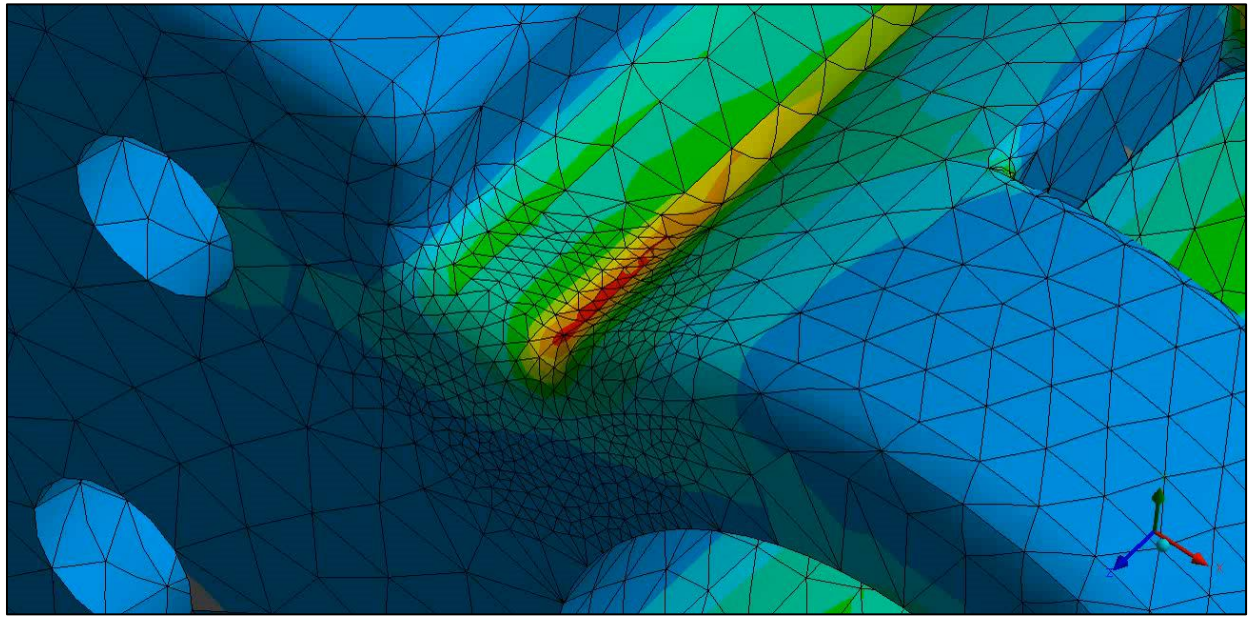
Initial mesh



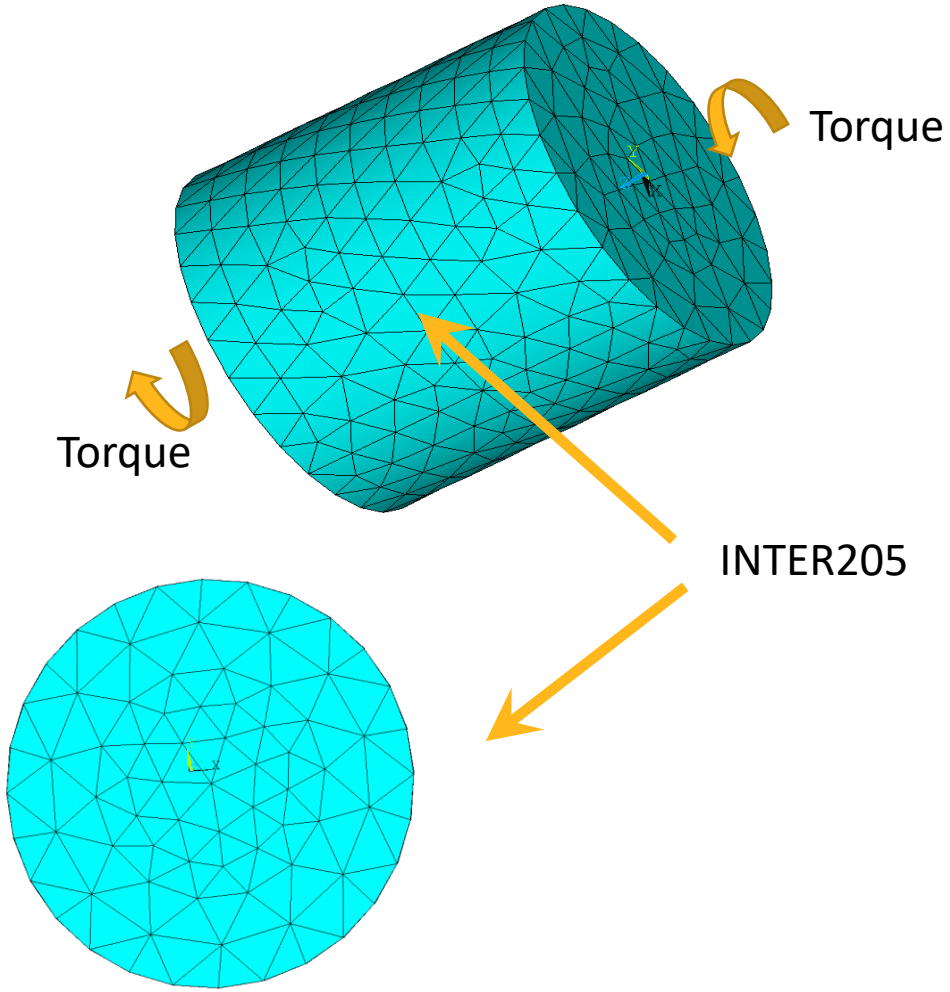
The high-risk zone



CAD model

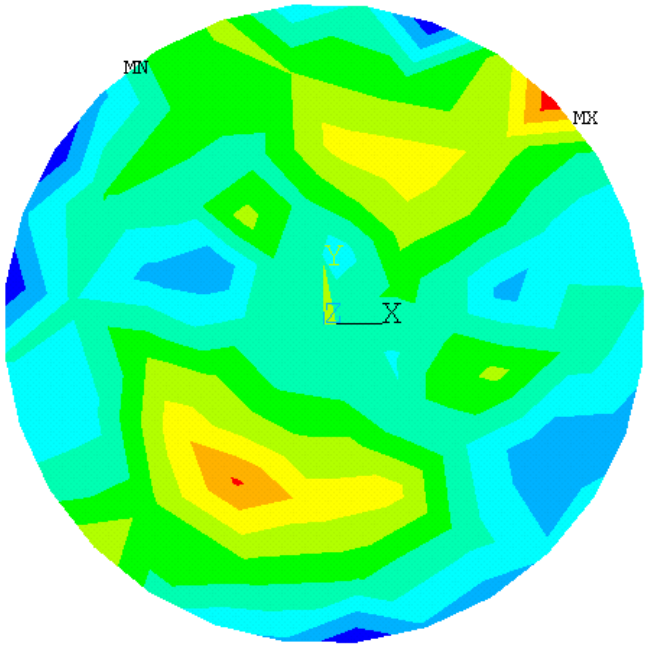


ESYS for Interface Elements: INTER204 and INTER205

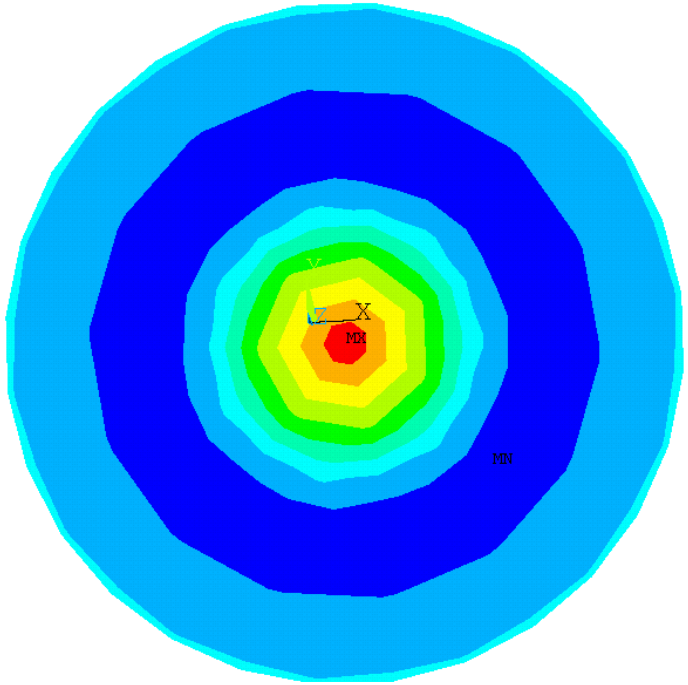


Two cylinders connected by INTER205 elements

Top view of INTER205 elements



(a) without ESYS



(b) with ESYS

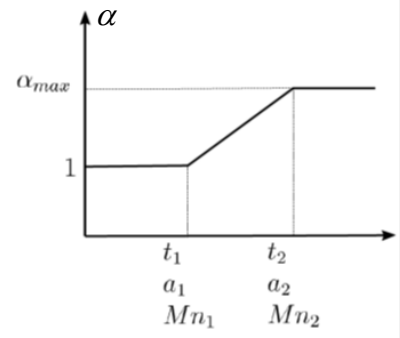
Display of shear traction of INTER205 calculated with and w/o ESYS

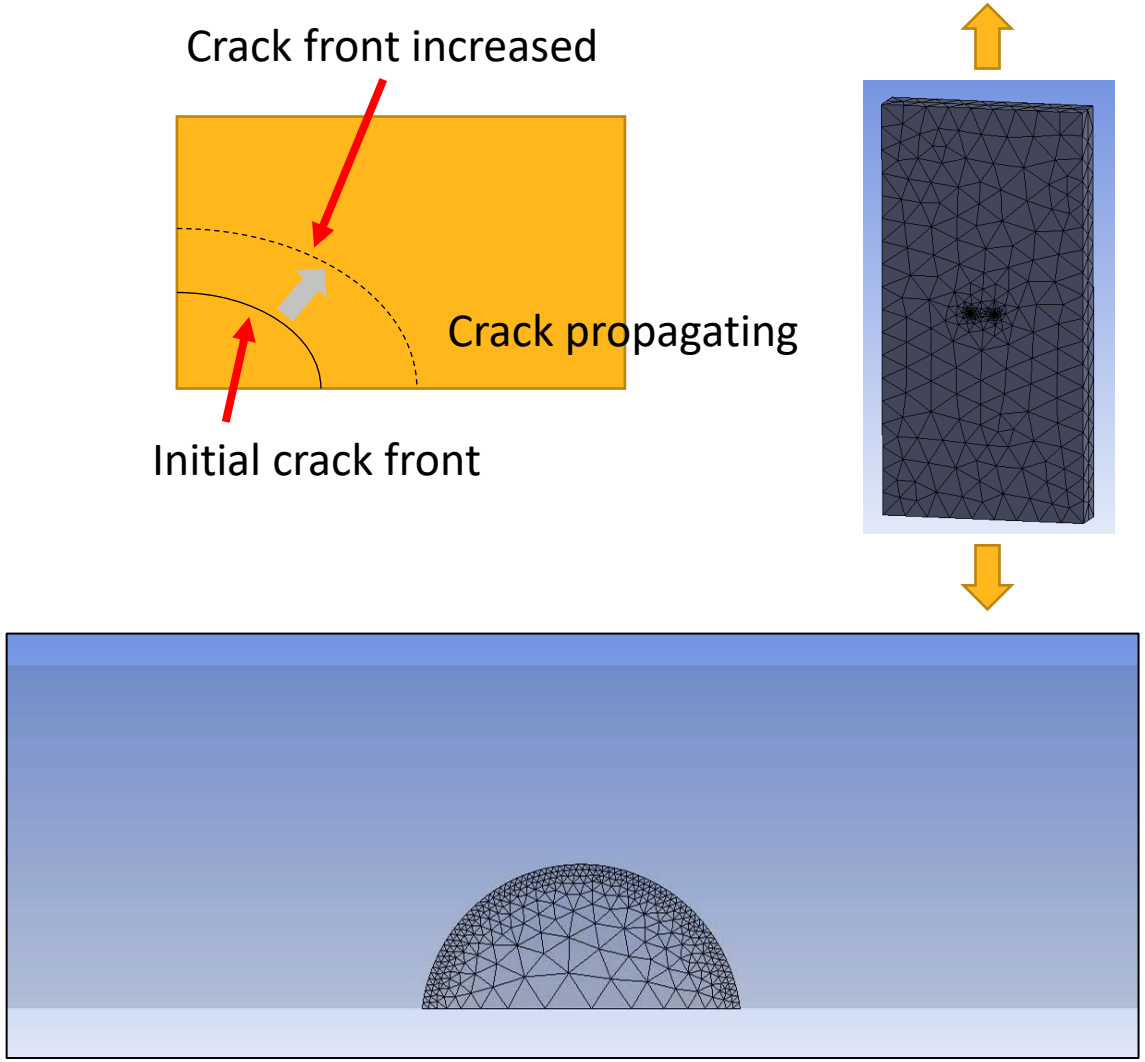
Dynamic Crack Extension Size Control

SMART crack growth

- Allow to change crack front element size and crack extension size along with crack propagating
- Useful for crack-growth analysis, such as
  - reducing the model size by increasing crack front element size with increased crack front as a result of crack growth
  - increase crack extension increment with crack growing deep into structure components – reduce the solution time to reach the desired crack extension

$$\Delta a = \alpha \cdot Esize_{ref}$$



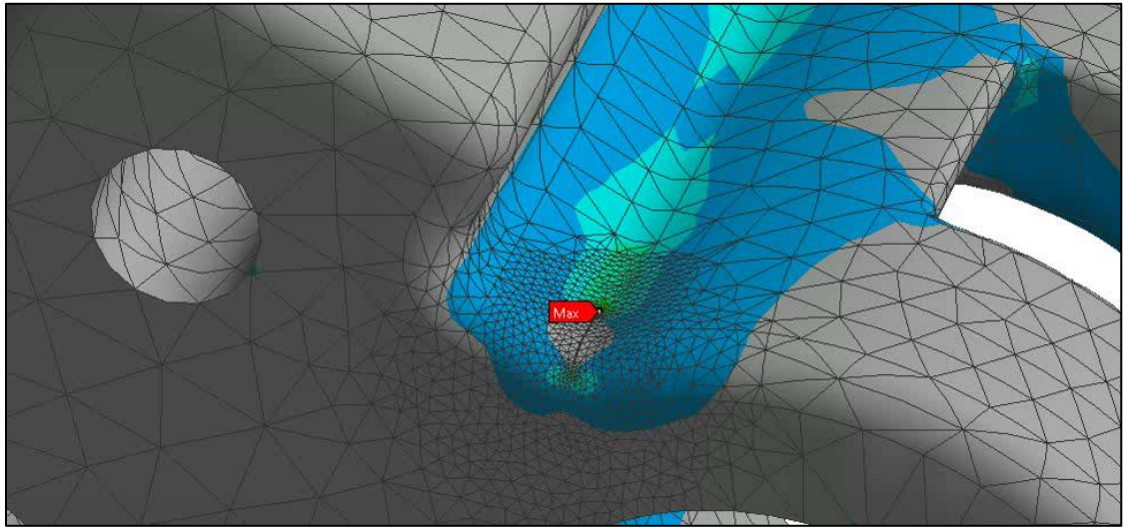


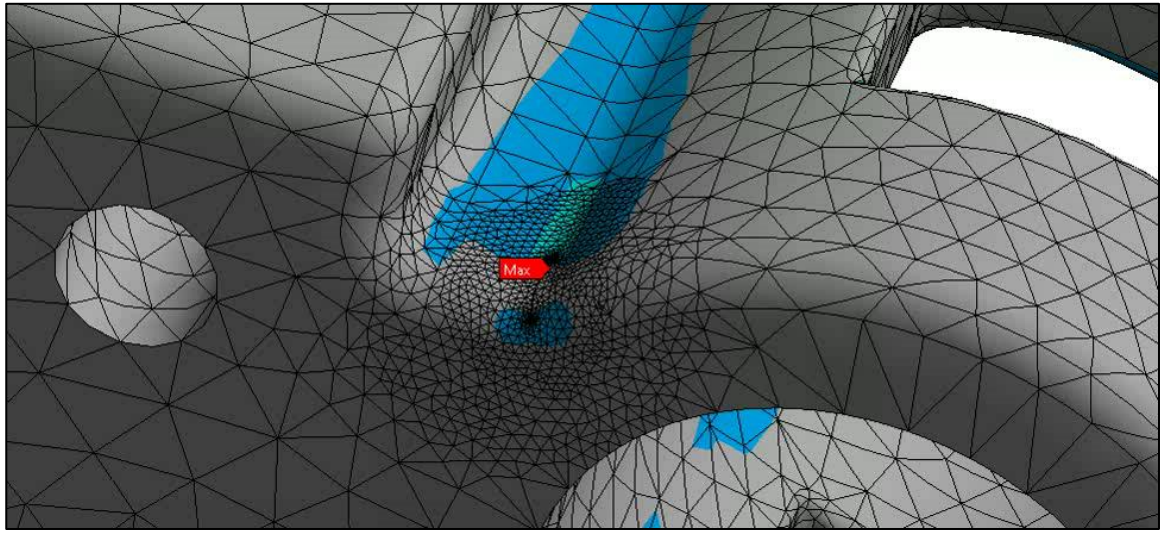
Dynamic Crack Extension Size Control (Solution Comparison)

40 sub-steps solution
Total crack increment = 12.87 mm
Duration of the solution = 1922 sec

VS

40 sub-steps solution
Total crack increment = 1.52 mm
Duration of the solution = 4150 sec





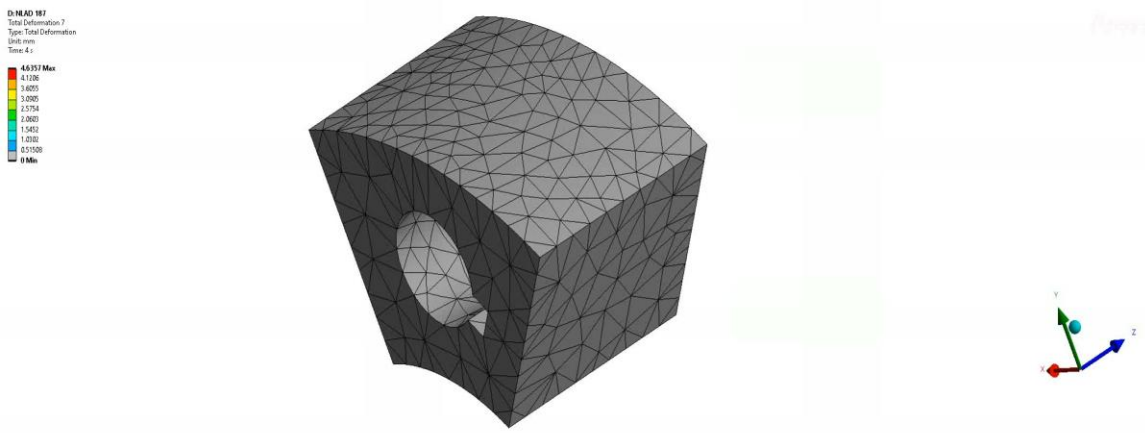
With a multiplier factor of 20 after 40 remeshing

MAPDL Nonlinear Adaptivity (NLAD)

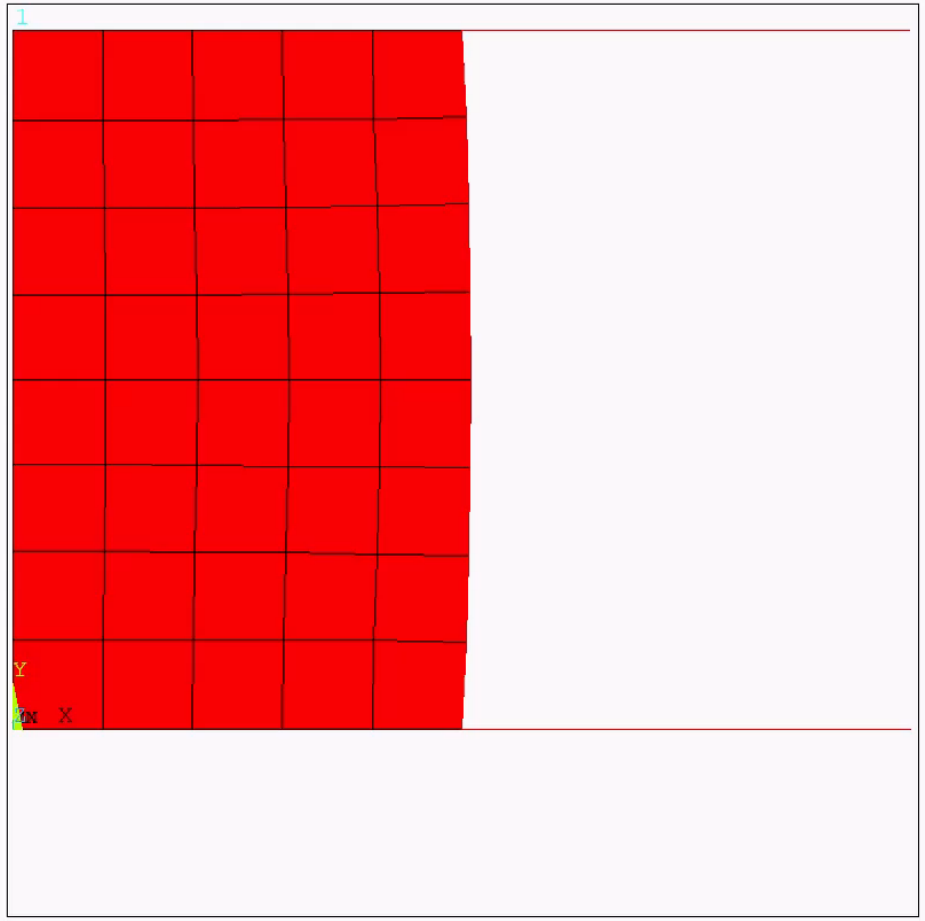
Ansys

Loads and Constraints on Initial Mesh with Multiple Load Steps for Nonlinear Adaptivity Analysis

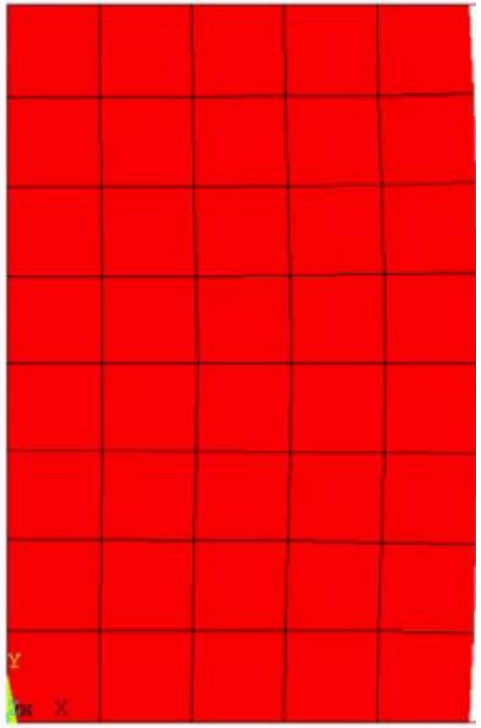
- Motivated by the challenges to apply proper load and boundary conditions on successive mesh configurations in a nonlinear adaptivity framework
- Extended to support pressure load
- Provide a feasible and user-friend workflow
- Enhanced the capability to handle large deformation nonlinear analysis
  - All supported load and boundary conditions
- Robust and accurate load transfer from initial mesh to current mesh even for large deformations



Demonstration Example: Initial Mesh-Based Body Temperature with Large Deformation

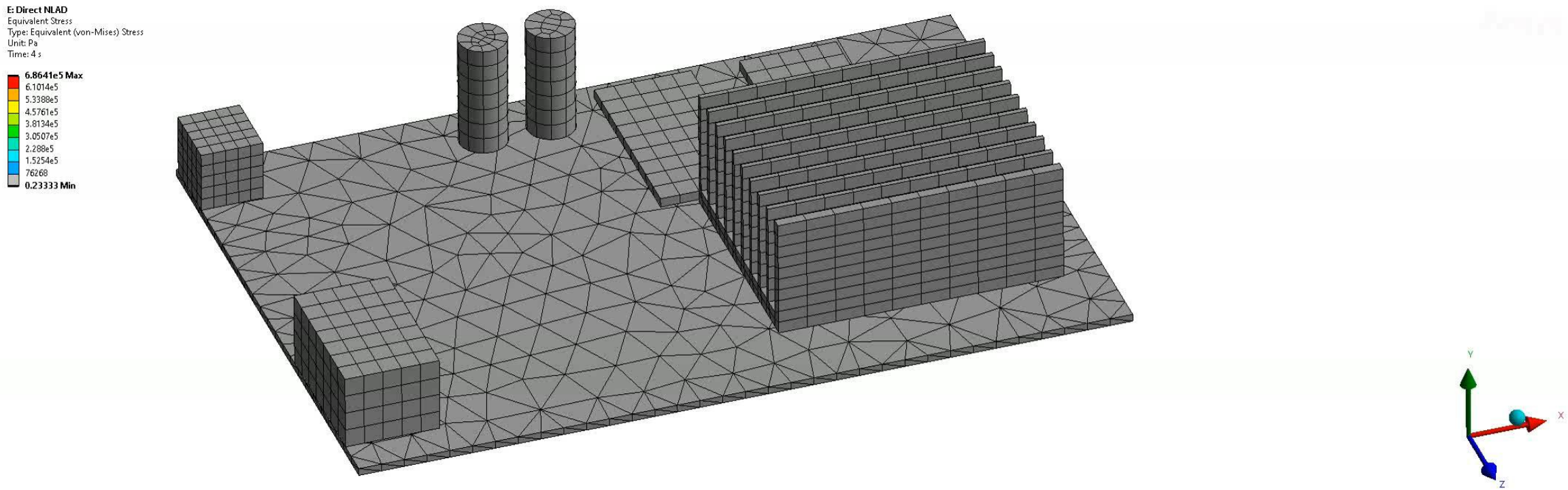


ANSYS 2022 R1
Build 22.1BETA
ELEMENT SOLUTION
STEP=1
SUB =1
TIME=.041667
BFETEMP (NOAVG)
RSYS=0
PowerGraphics
EFACET=1
DMX =.213333
SMN =301.625
SMX =301.625



Body temperatures are still applied on initial mesh after large deformation and remeshing

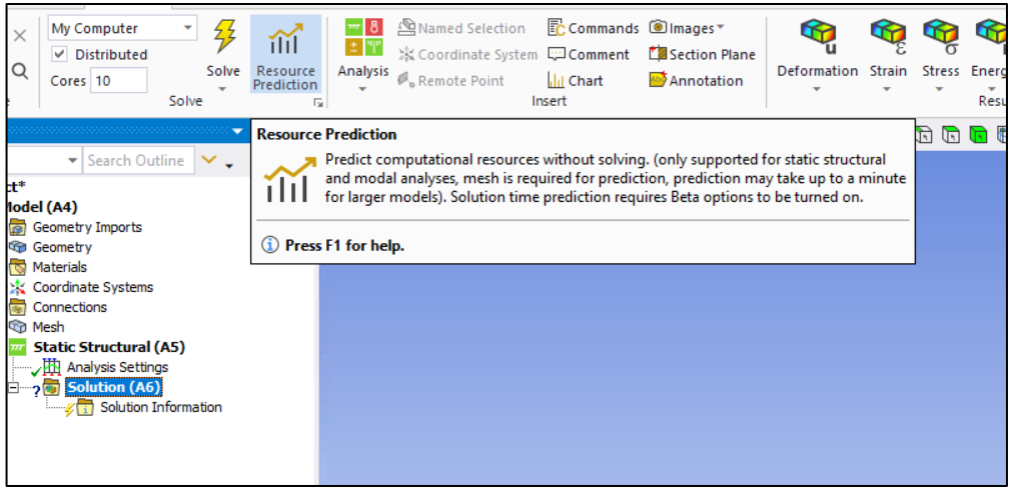
Demonstration Example: Initial Mesh-Based Pressure Loads

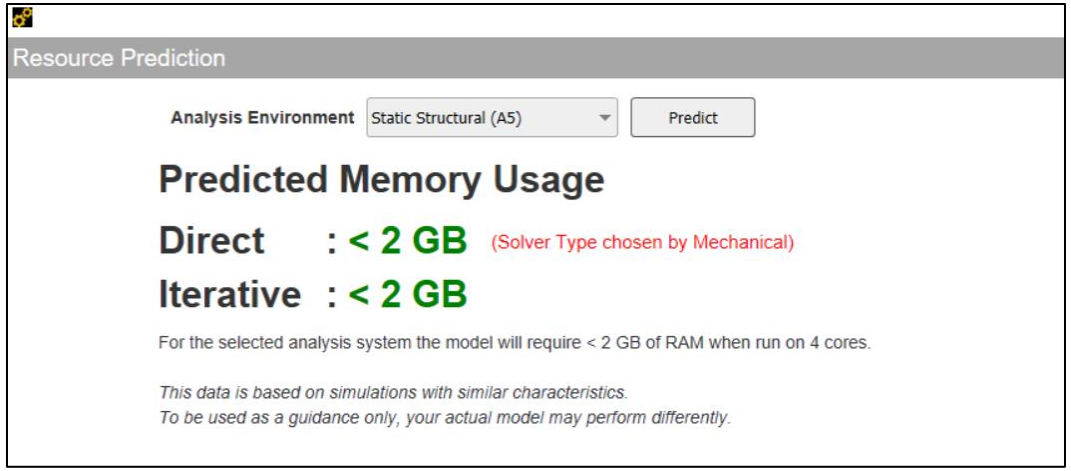


MAPDL Solver

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 <#>

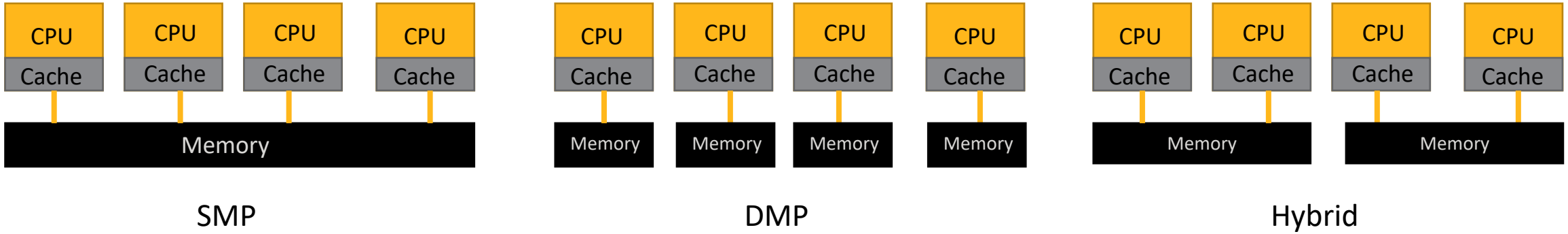
- **SMP** → -smp & -np N to specify using "N" OpenMP threads
- **DMP** → -np N to specify using "N" MPI processes
- **Hybrid** → *-np N* to specify using "N" MPI processes
  -nt M to specify using "M" OpenMP threads per process during SOLVE command

Total core count = N 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

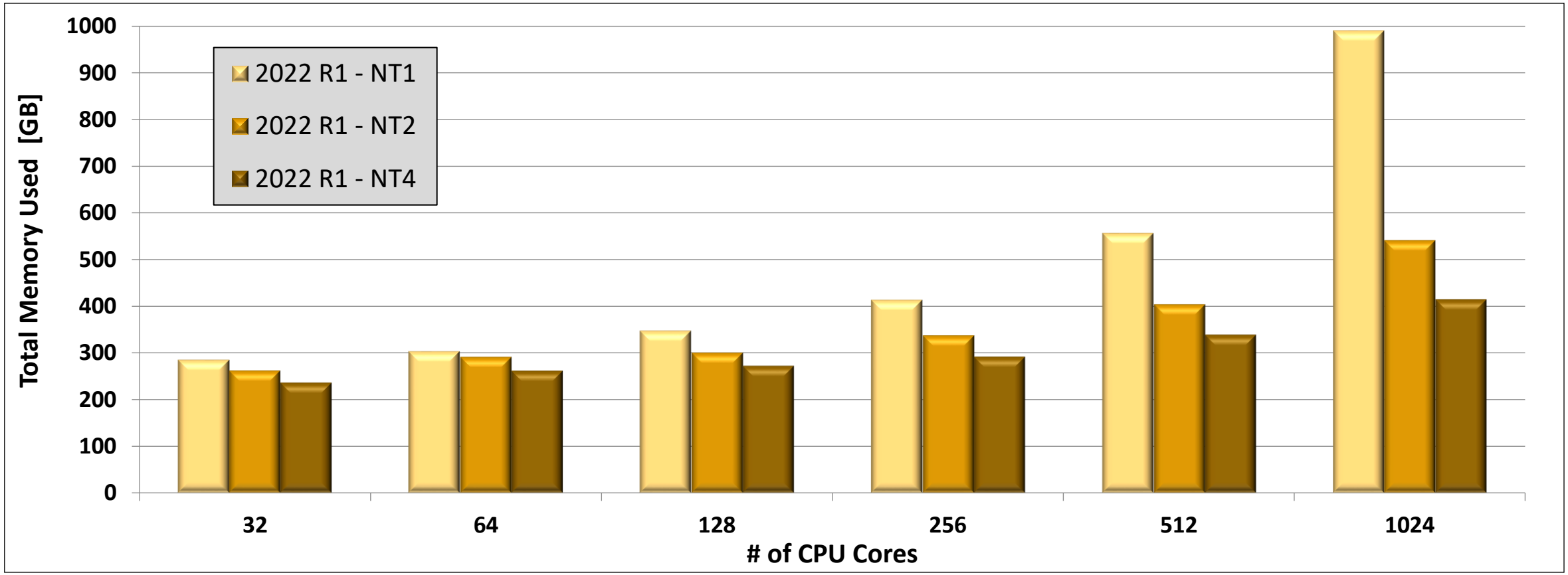
Distributed Ansys Enhancements

- 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

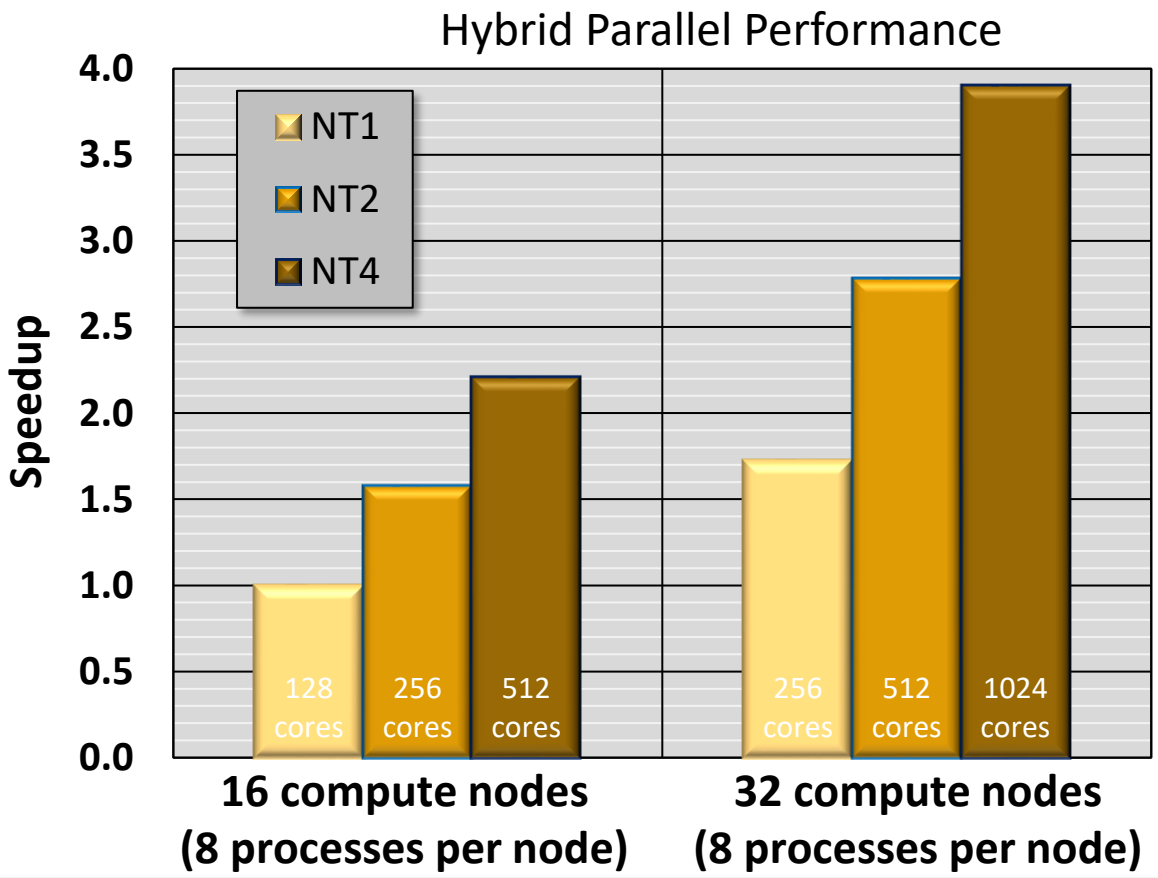
- Hybrid parallel reduces total memory requirements (Engine benchmark)



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



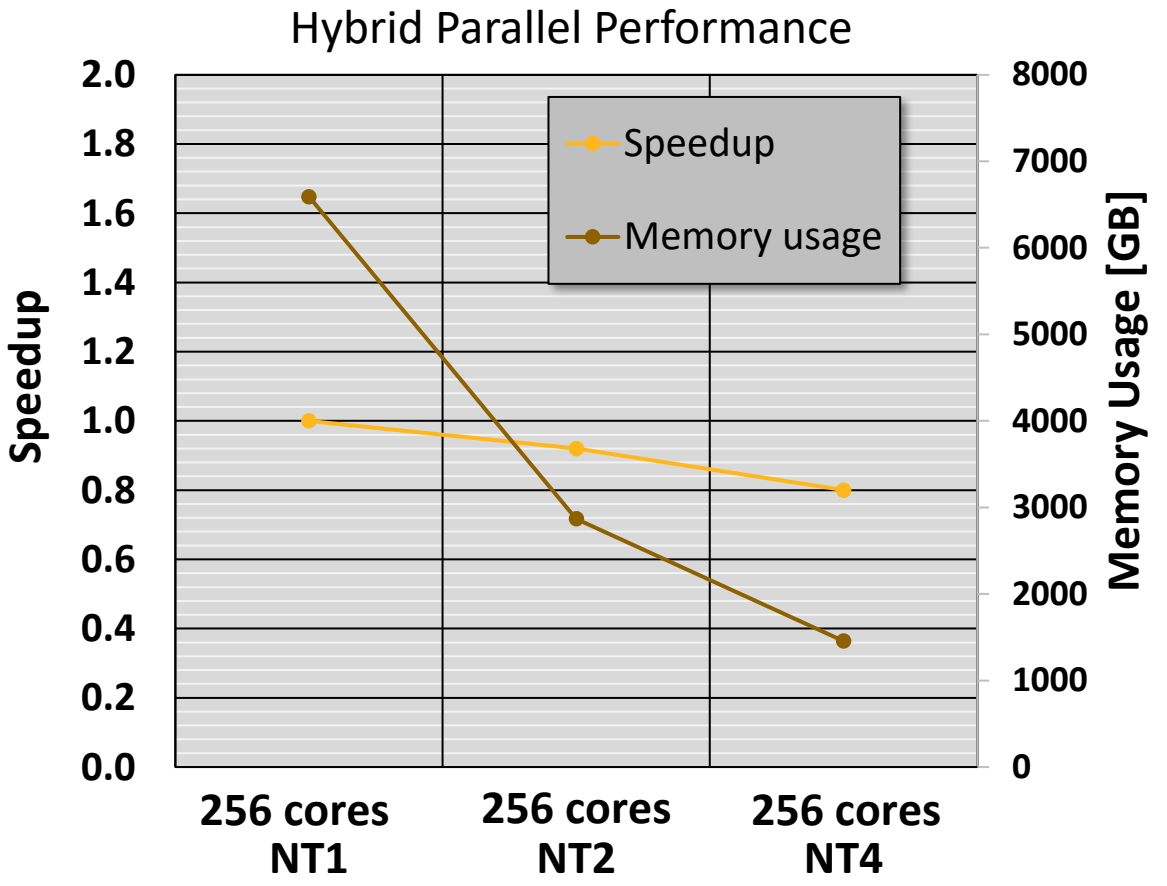
More cores used per compute node

Faster performance with constant memory requirement

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

- **MPI library support**
  - Upgraded to Intel MPI 2019 Update 10 on Windows
  - Intel MPI 2018 Update 3 support is unchanged at this release on Linux (**default**)
  - Support has been added for Intel MPI 2019 Update 12 on Linux (-mpi intelmpi2019)
  - Open MPI v4.0.5 support on Linux is unchanged at this release
  - Microsoft MPI v10.0 support on Windows is unchanged at this release

Distributed Ansys Enhancements

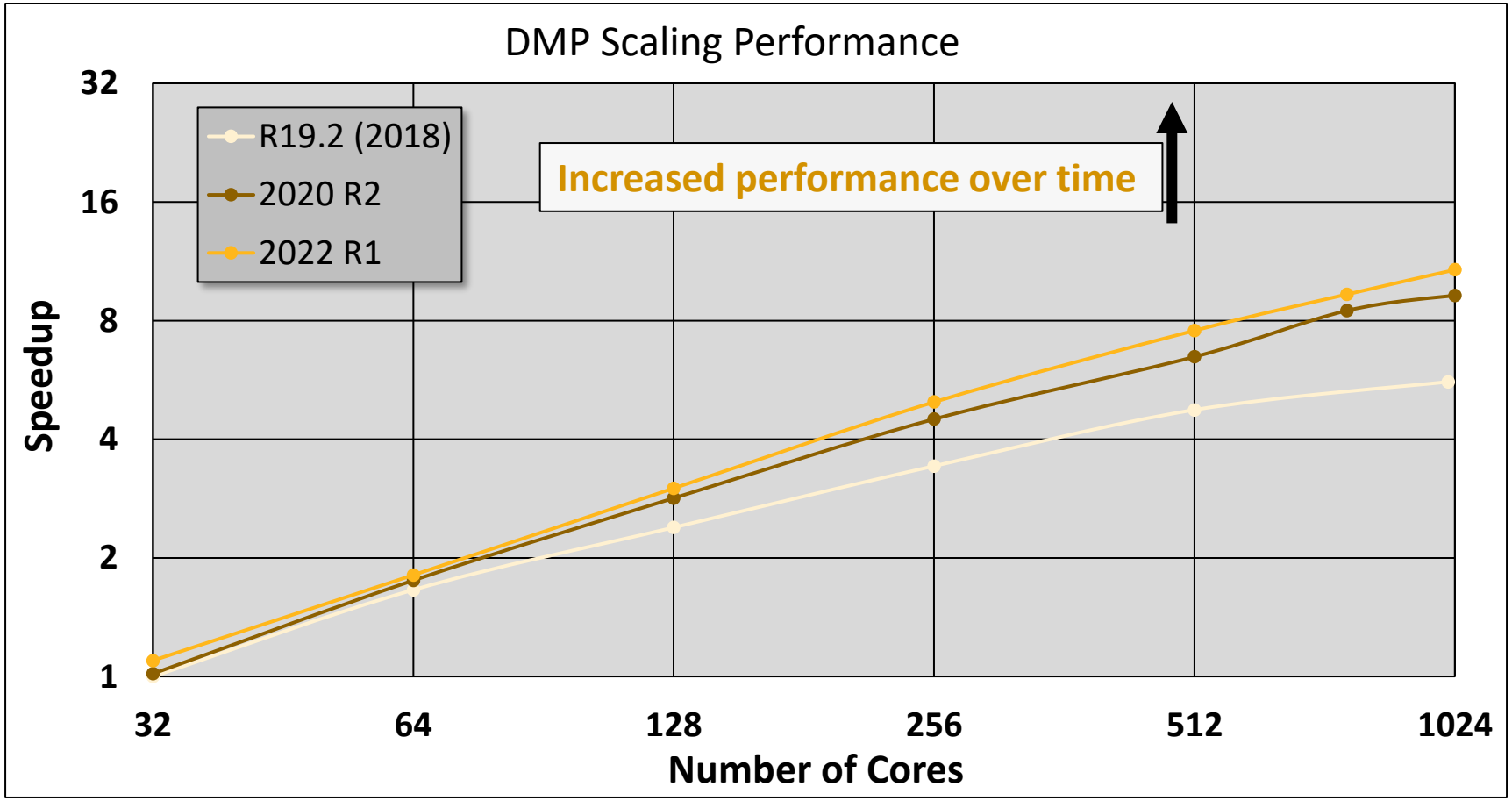
- **Improved scaling**
  - Significantly faster performance and reduced memory & I/O requirements when constraint and/or coupling equations are present in the model
    - Assembled matrix (.full) file contains more efficient data storage
    - Mostly affects sparse direct solver and related eigensolvers (Block Lanczos, Subspace, etc.)

    - Symmetric matrices supported in 2021 R2
    - **Unsymmetric matrices supported in 2022 R1**

Distributed Ansys Enhancements

- Improved scaling at higher core counts

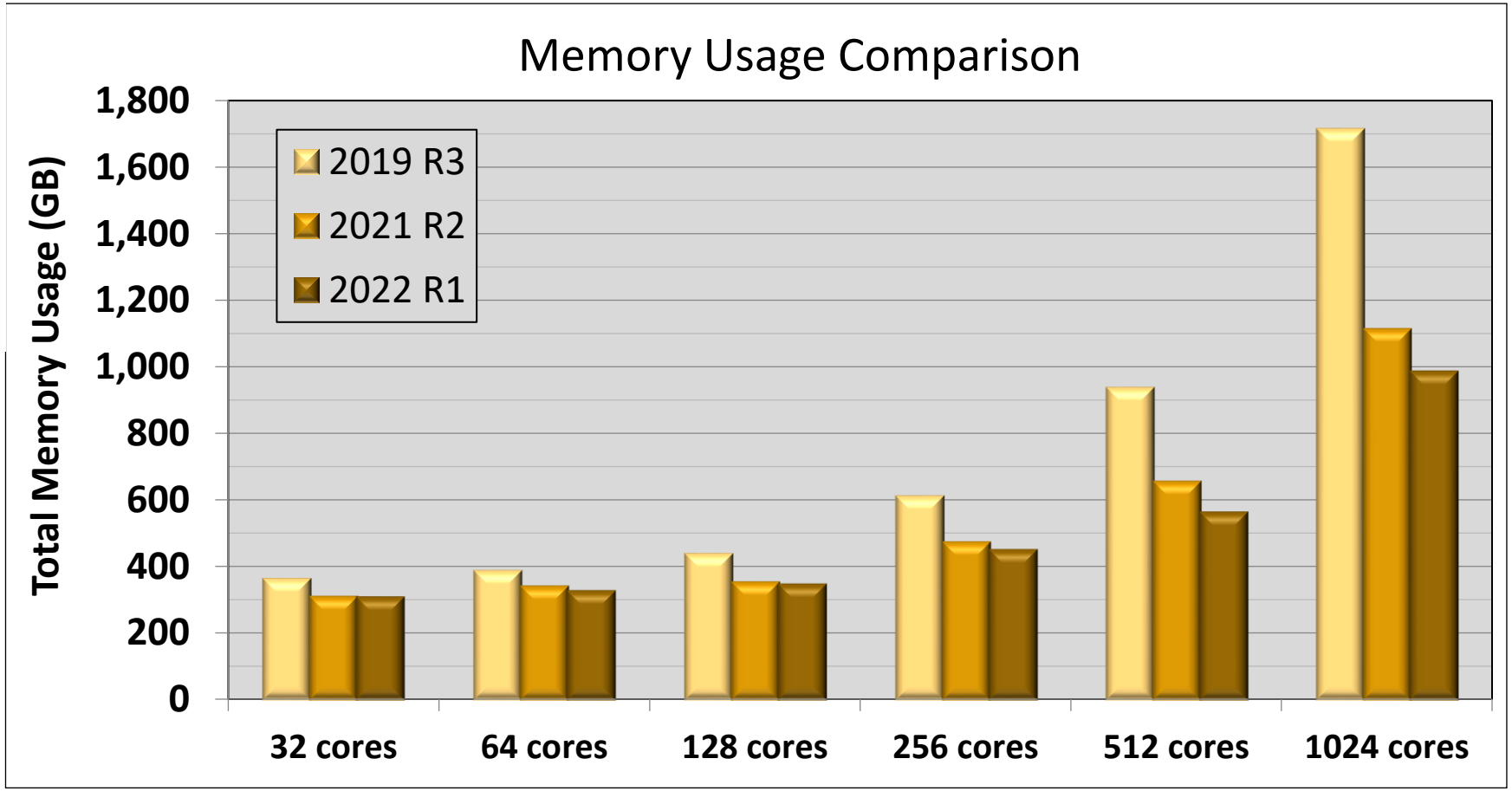
- 5.6 million DOF; sparse solver
- Nonlinear static analysis involving contact, constraint equations, unsymmetric matrices
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6



Distributed Ansys Enhancements

- Reduced memory usage at higher core counts

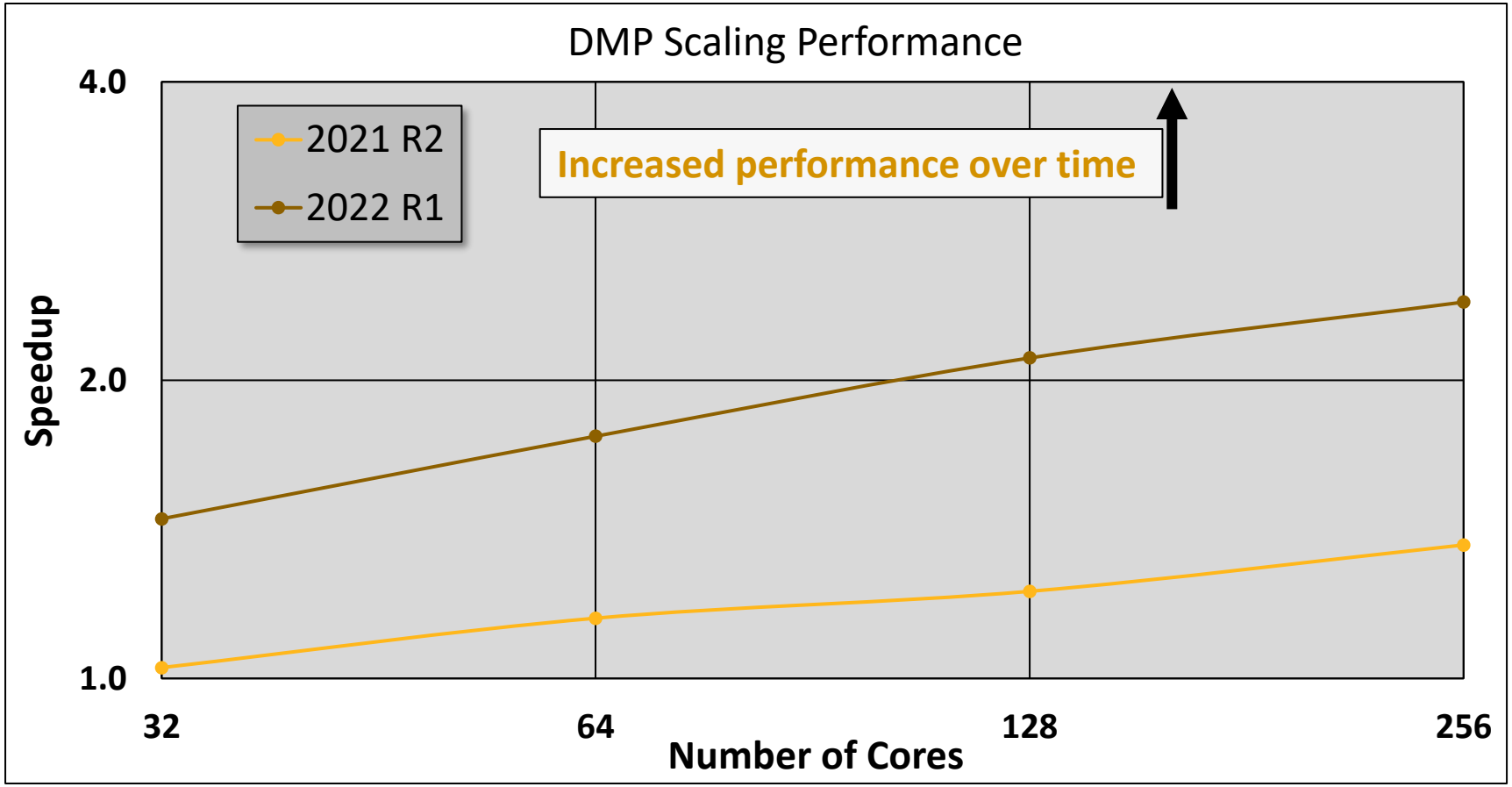
- 5.6 million DOF; sparse solver
- Nonlinear static analysis involving contact, constraint equations, unsymmetric matrices
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6



Distributed Ansys Enhancements

- Improved scaling at higher core counts

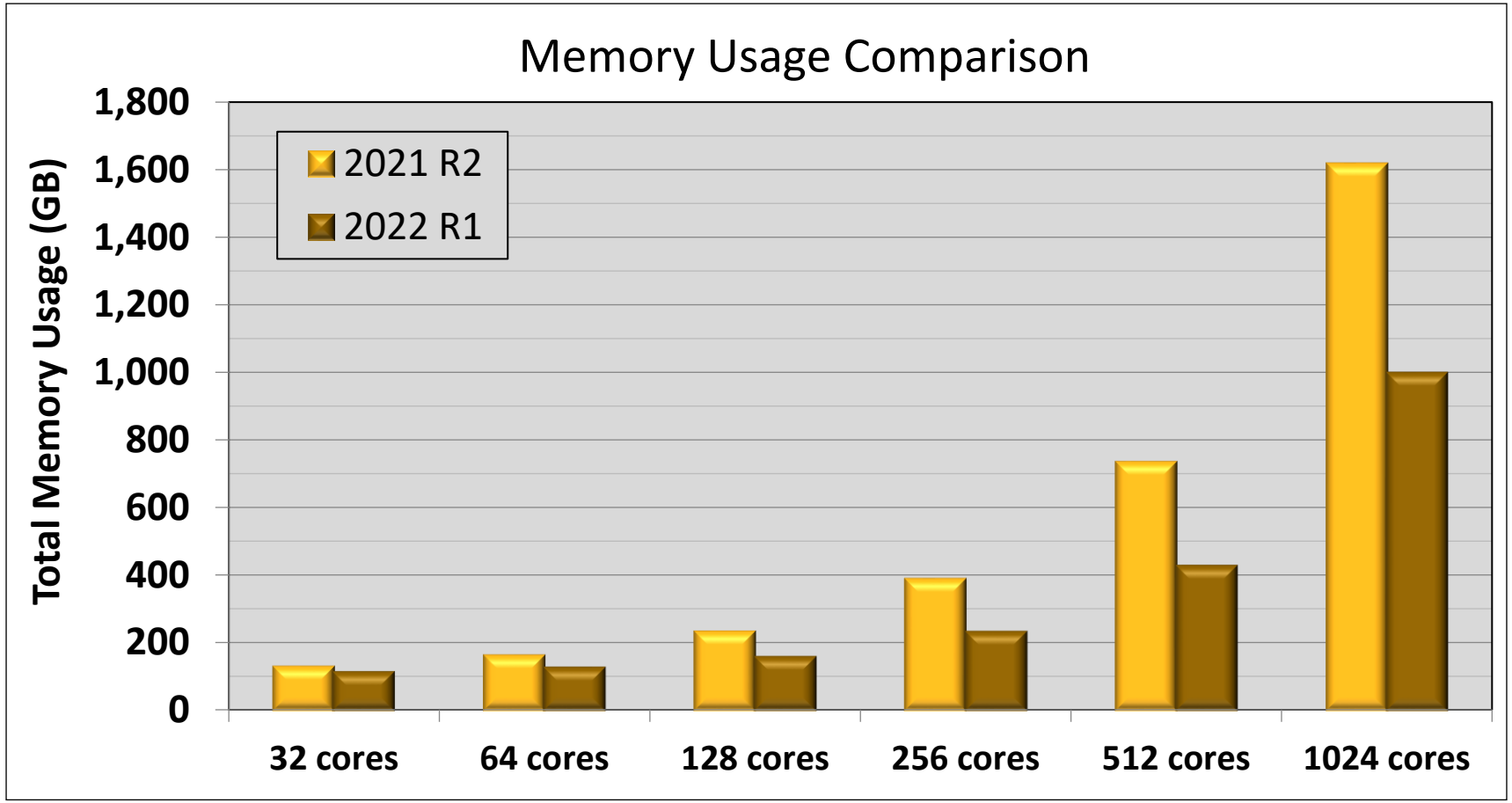
- 3.5 million DOF; UNSYM eigensolver
- Modal analysis requesting 200 mode shapes
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6



Distributed Ansys Enhancements

- Reduced memory usage at higher core counts

- 3.5 million DOF; UNSYM eigensolver
- Modal analysis requesting 200 mode shapes
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, Mellanox Infiniband, CentOS 7.6



Miscellaneous Enhancements

- PCG solver support added for MPC184 joint elements using penalty formulation
- Improved over-constraint detection for user-defined constraint equations

Ansys Aqwa

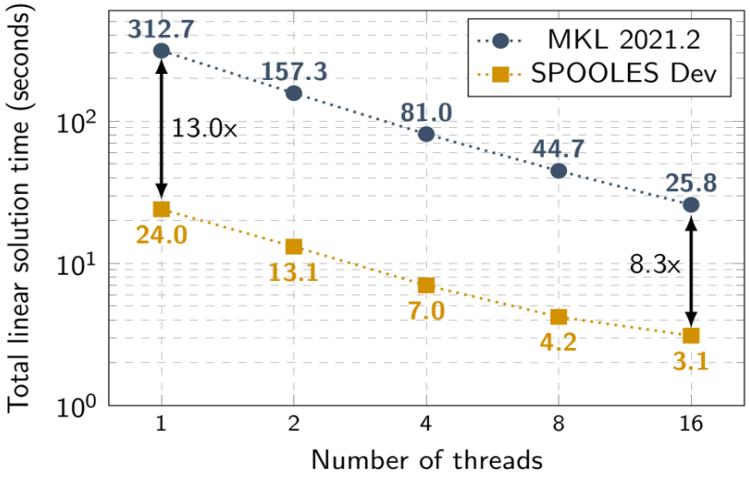
Integrated system for hydrodynamic, mooring and structural analysis of marine structures

Ansys

Block Low-Rank Iterative Solver in Hydrodynamic Diffraction Analysis

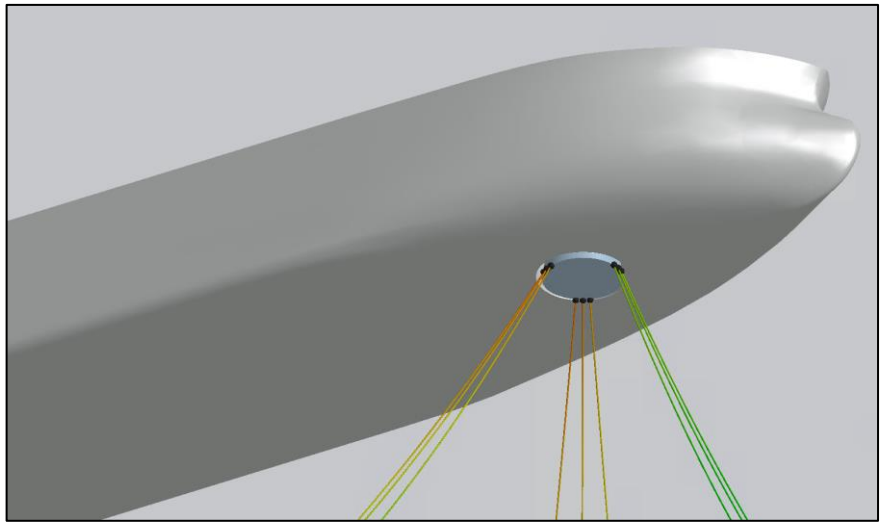
- Integrate the Block Low-Rank Generalized Minimal Residual solver of SPOOLES library for solving the large-scale complex full matrix equation for the source strength distribution
- Replaces the Intel MKL solver
- OpenMP parallel calculation, including multithreading to the assembly process
- New solver is faster and requires less memory while keeping the same accuracy (6 digits)
- Integrated in both Windows and Linux Aqwa versions
- For a single ship model with 22000 diffracting elements and 41 wave directions:

  10x faster and 5.5x memory reduction for equation solving

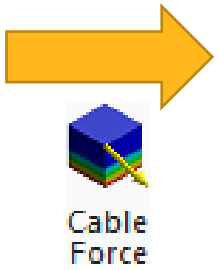


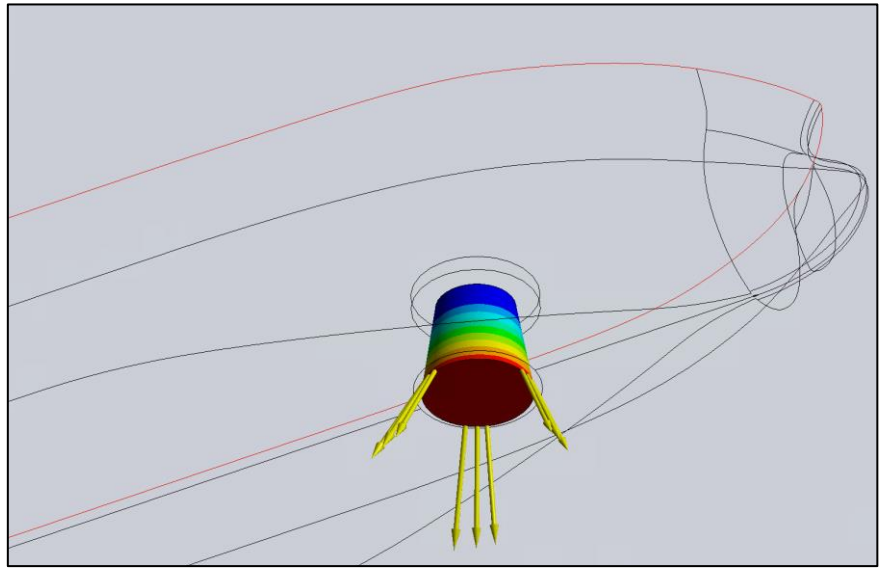
Time Domain Hydrodynamic Load Mapping to Static Structural for Cables, Tethers and Joints

- Complementary to Hydrodynamic Pressure mapping in Mechanical
- Previously, pressure mapping was for freely-floating structures only
- In Release 2022 R1 we can now transfer instantaneous forces on cable attachments, and forces/moments on tether or joint attachments, as well as pressure on surfaces
- This provides a more complete description of the hydrodynamic loading for FE analysis



Hydrodynamic model



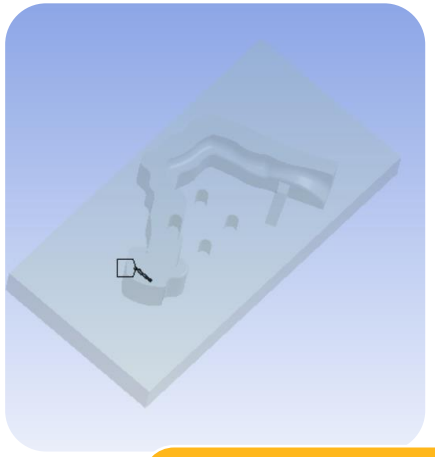


Structural model

Workbench Additive

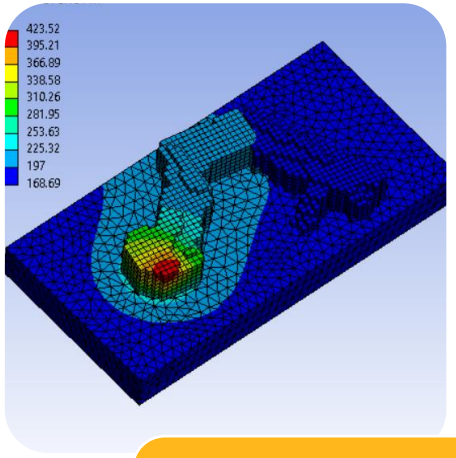
Directed Energy Deposition (DED) Simulation: Full Release

- Process simulation in Mechanical Workbench for Directed energy process (DED)
- Predict the macro-level temperature-level distortions and stresses in parts to prevent build failures and provide trend data for improving designs for additive manufacturing including part orientation and part build order.



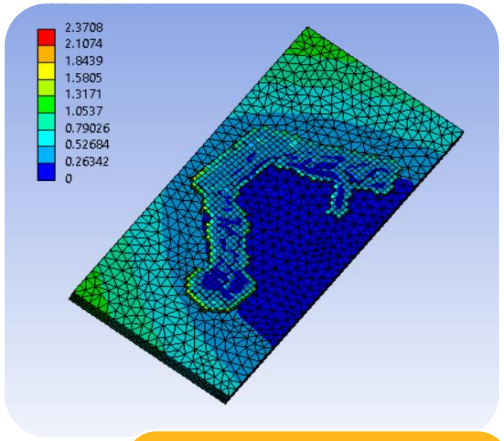
Welds (Element clustering) generation

- Manual approach
- G-Code approach



Transient thermal analysis

- Thermal history
- Process parameters, e.g. deposition rate



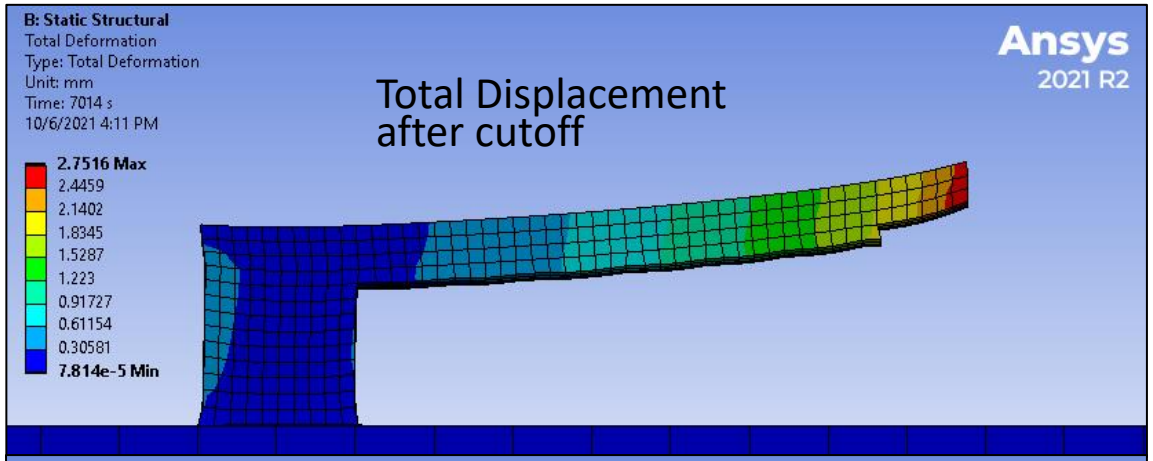
Static structural analysis

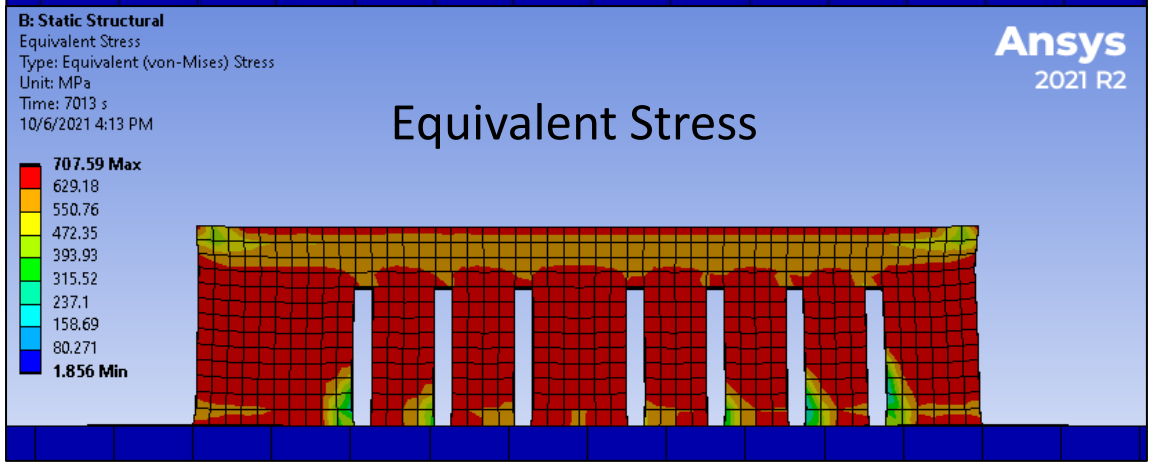
- Stresses, strains, distortions
- Cutoff

Improved Simplified Heat Treatment (Relaxation Temperature)

- Zero out only plastic strain
- No heat treatment for base plate

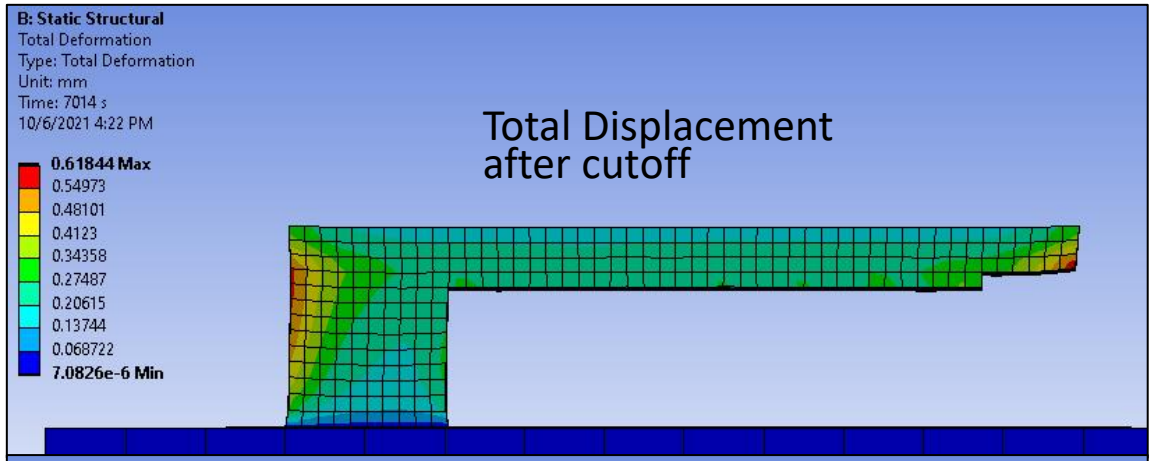
2021 R2

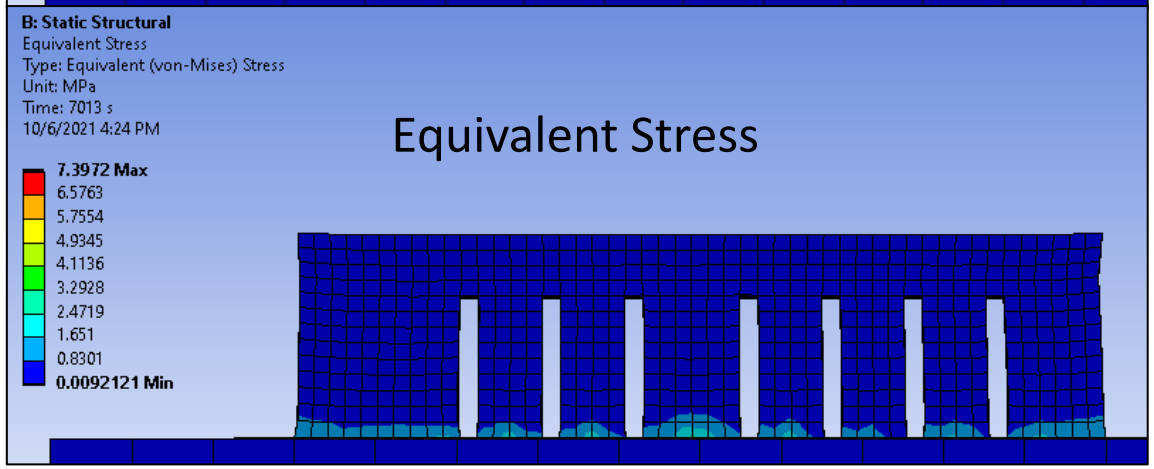




- Zero out elastic strain, plastic strain, stress
- Heat treatment for base plate

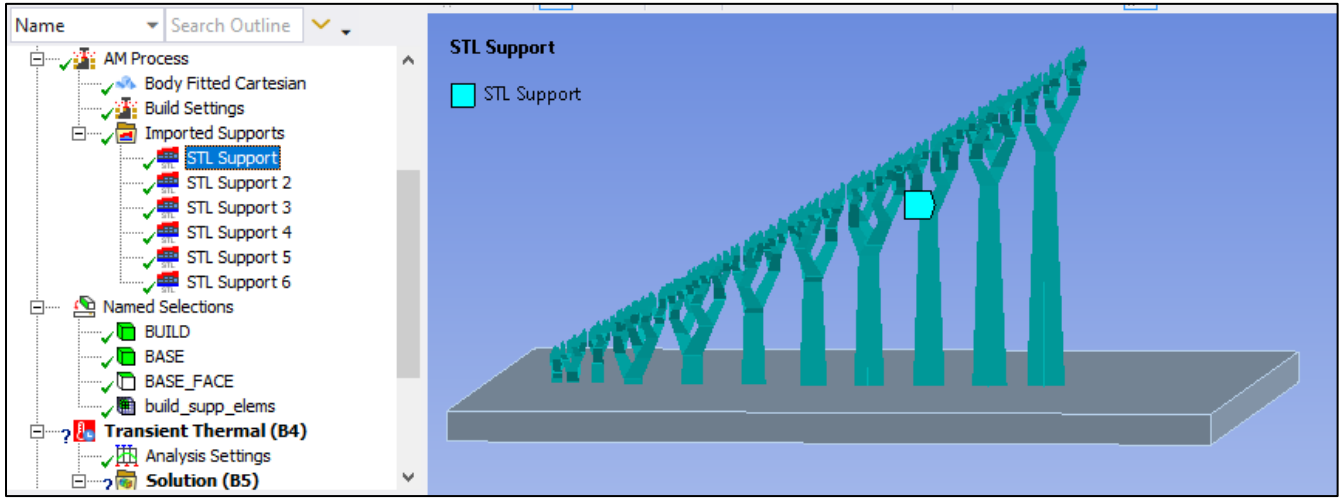
2022 R1





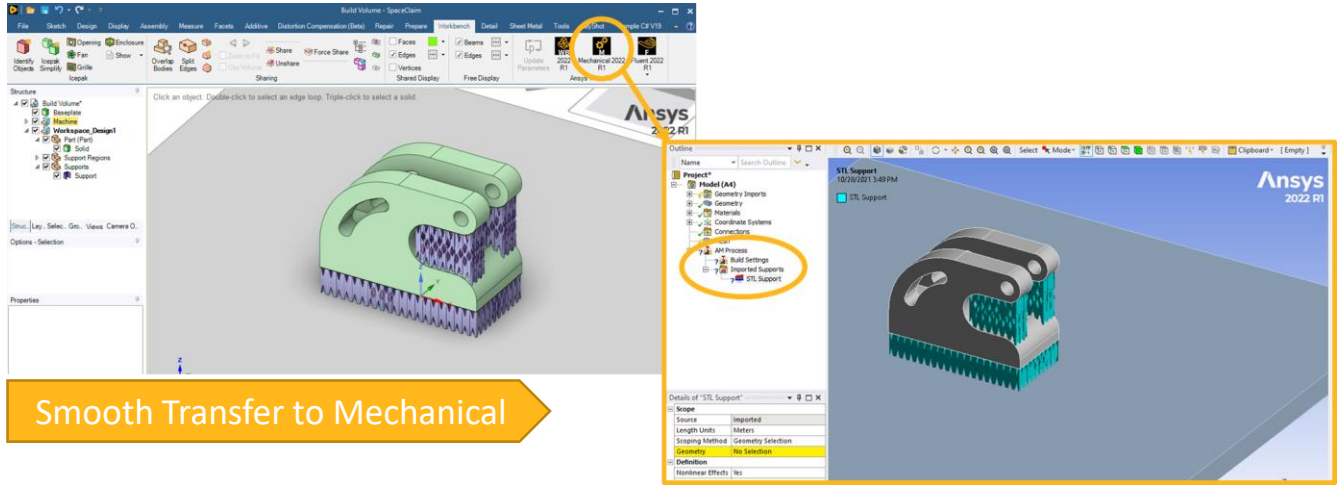
Import of Additive Supports from CAD

- Supports created using the Additive Prep SpaceClaim add-in are now automatically imported into Mechanical
- This removes the need to export supports and manually scope to the files in Mechanical
- This also simplifies the workflow when transitioning to Mechanical from Additive Prep to perform an AM simulation

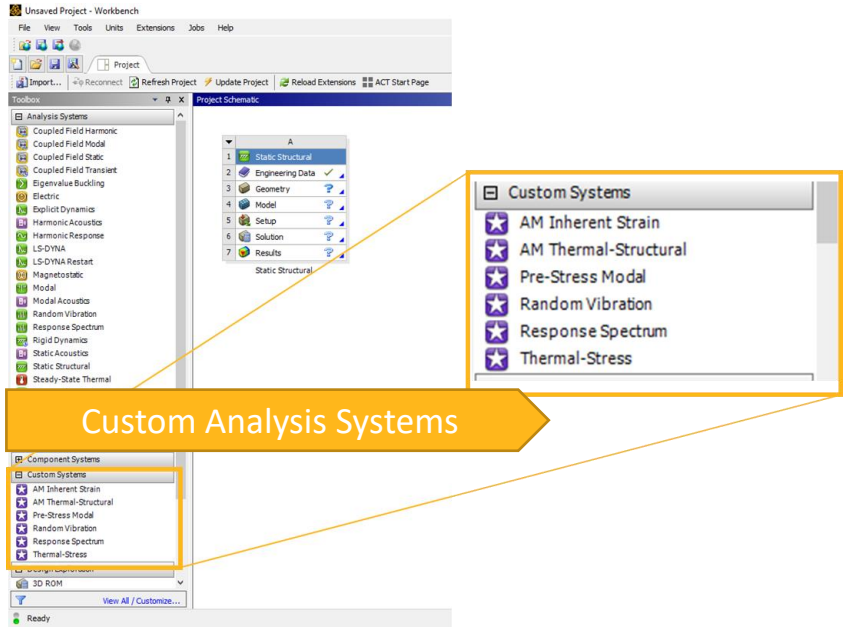


2022 R1 Additive Enhancements

- Smoother transfer of supports from Additive Prep to Workbench/Mechanical
- New Material!
  - 17-4PH in Additive Science (Stainless Steel)
- Custom Additive Manufacturing (AM) Analysis Systems
  - Custom systems automatically load AM sample materials and adds AM Process object in Mechanical
- Parallelization: 2D Microstructure solver in Additive Science
  - Fully threaded all the core component of the solver
- Improvements to MAPDL



Smooth Transfer to Mechanical



Custom Analysis Systems

Explicit Dynamics

Multi-Material Euler Exposed in Mechanical for Explicit Dynamics

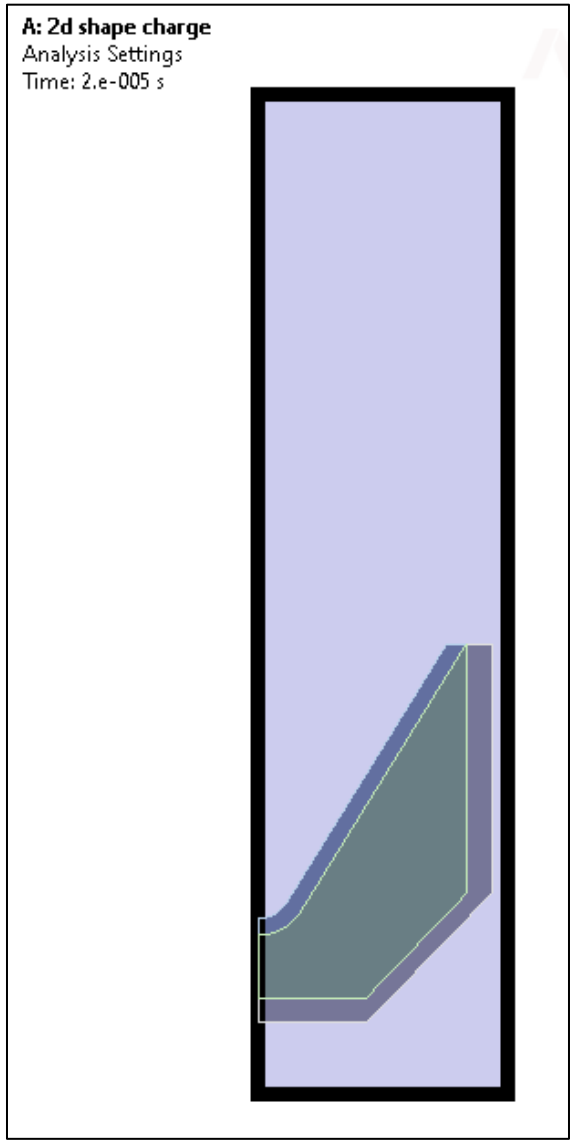
- A virtual Multi-Material Eulerian (MME) domain is created in any 2d analysis if any of the bodies have the reference frame set to Eulerian (Virtual)
- Euler-Lagrange coupling is enabled automatically

Details of "Surface Body"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Reference Frame	Eulerian (Virtual)
Material	
Assignment	STEEL 1006
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	0

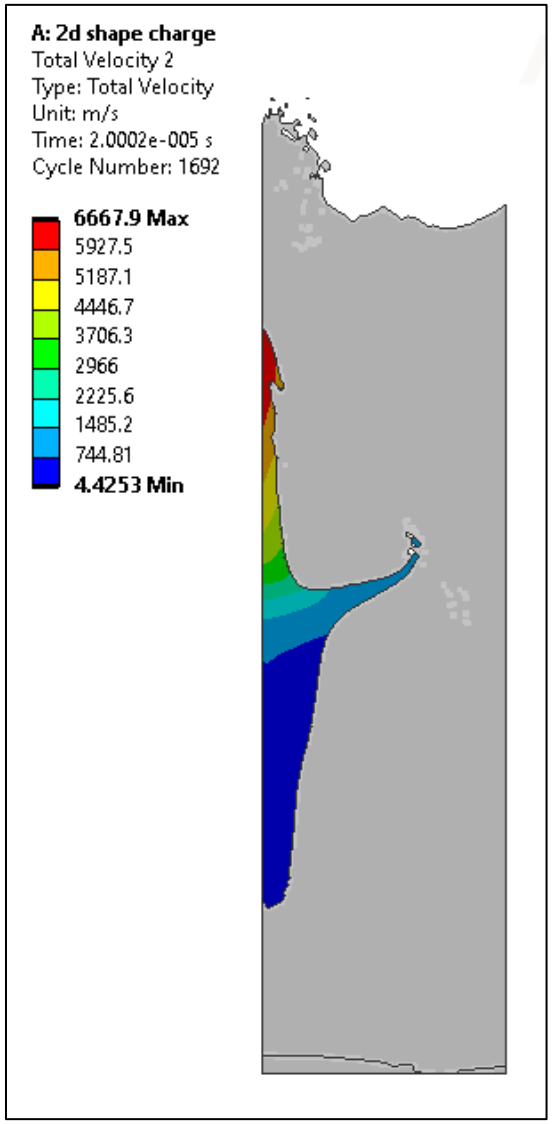
Details of "Analysis Settings"

Type	Custom
Step Controls	
Solver Controls	
Euler Domain Controls	
Domain Size Definition	Manual
Display Euler Domain	Yes
Minimum X Coordinate	0. m
Minimum Y Coordinate	-4.e-002 m
X Dimension	3.e-002 m
Y Dimension	0.12 m
Domain Resolution Definition	Cells per Component
Number of cells in X	120
Number of cells in Y	480
Lower X Face	Rigid
Lower Y Face	Flow Out
Upper X Face	Flow Out
Upper Y Face	Flow Out
Euler Tracking	By Body
Damping Controls	



Multi-Material Euler Exposed in Mechanical for Explicit Dynamics

- Enables the simulation of extreme deformation events such as fluid or gas flow without suffering from high mesh distortion or tangling which may occur when using a Lagrangian mesh
- Allows for significant speed-up of solution times for analyses that can be considered as 2d axisymmetric or 2d plane strain compared to an equivalent 3d MME analysis

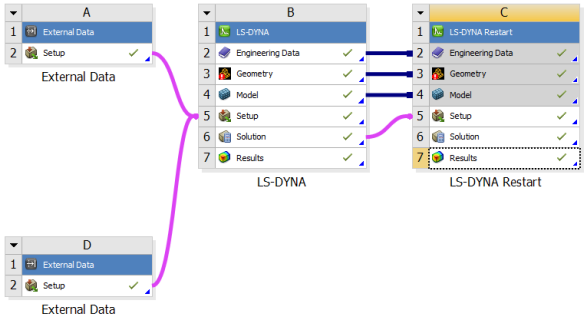


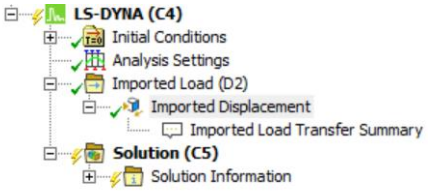
Workbench LS-DYNA

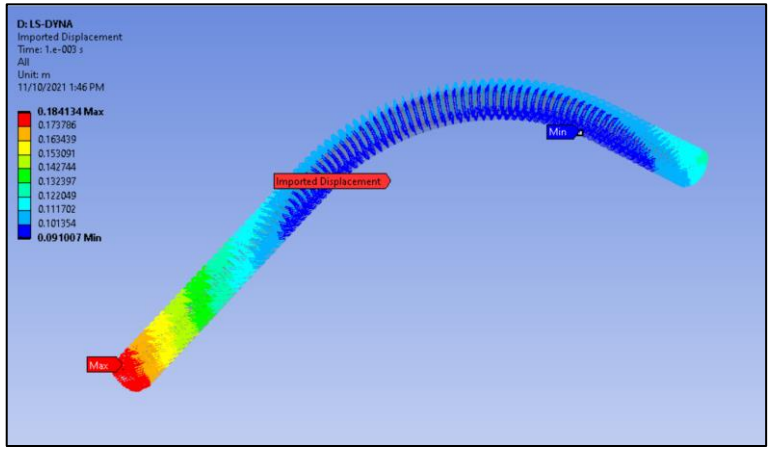
Imported Displacement

- LS-DYNA now supports Imported Displacement, allowing you to setup the model in non-trivial ways
- The mesh can either be initialized towards a final mesh, allowing the solver to prestress the model, or it can be used to specify the reference geometry in foams applications, if the user has the deformed mesh instead
- The solver will automatically calculate the stresses at the beginning of the calculation
- Create an external data system and import a file containing displacements defined by coordinates or mesh reference
- Link to an LS-DYNA system in the project schematic
- The Imported Load folder is automatically added and Imported Displacement is available for selection in the context menu along with the previously supported Imported Pressure
- Includes the standard options for mapping and display

	A	B	C	D	E
1	Column	Data Type	Data Unit	Data Identifier	Combined Identifier
2	A	X Coordinate	m		File1
3	B	Y Coordinate	m		File1
4	C	Z Coordinate	m		File1
5	D	Displacement	mm	Displacement1	File1:Displacement1



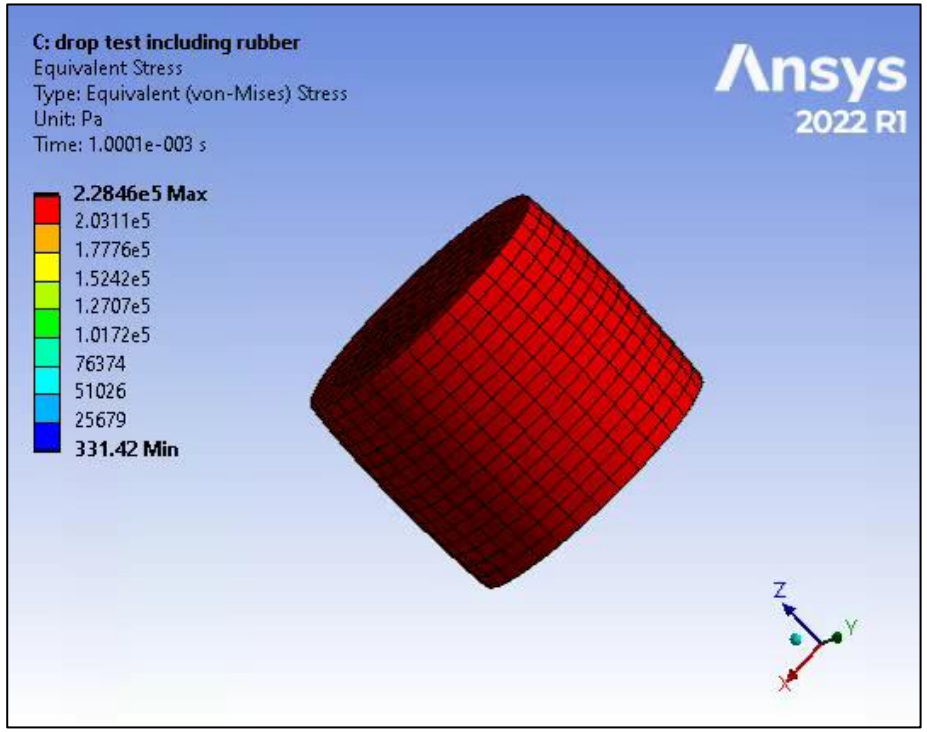




Graphics Controls	
By	Active Row
Active Row	1
Component	All
Display Source Points	Off
Display Source Point Ids	Off
Settings	
Mapping Control	Manual
Mapping	Profile Preserving
Weighting	Direct Assignment
Legend Controls	
Legend Range	Program Controlled
Named Selection Creation	
Unmapped Nodes	Off
Mapped Nodes	Off

Imported Displacement

- Two options are available:
  - **Boundary Prescribed Final Geometry** allows to move the scoped nodes from their initial positions to the final geometry along a straight-line trajectory. The geometry can be prestressed in a nontrivial way.
  - Loading can be applied in a series of Stepped or Ramped sections
  - A single displacement file is allowed for each component in the load table
  - **Initial Foam Reference Geometry**, which is only valid when used with foams materials. The mesh in Mechanical is the deformed geometry. The external file contains the coordinates of the reference (undeformed) geometry. The stresses are initialized by the solver
  - A single row is allowed in the load table
  - Tabular Loading is not applicable in this case. The solver determines the rate at which the displacement is applied
  - Reference field in material is automatically selected when the Imported Displacement is scoped to the relevant body. When using a material defined using a command snippet the reference field is not modified



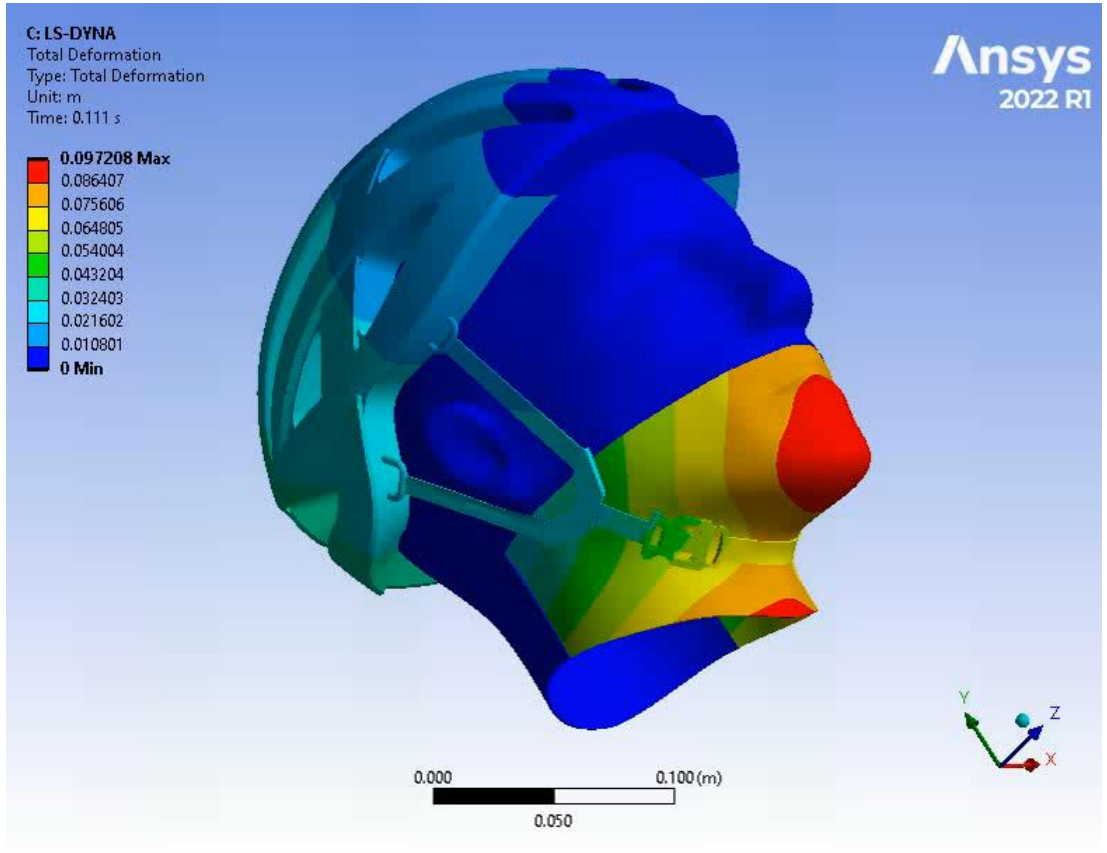
Details of "Imported Displacement"

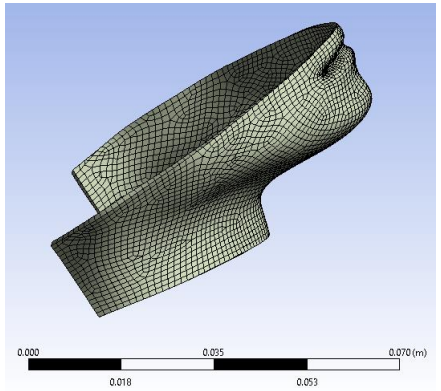
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Type	Imported Displacement
Displacement Type	Boundary Prescribed Final Geometry
Tabular Loading	Ramped
Suppressed	No
Override Constraints	No
Coordinate System	Source Coordinate System

	X Component (m)	Y Component (m)	Z Component (m)	Analysis Time (s)	Scale
1	File1:Displacement1	File1:Displacement2	File1:Displacement3	0.005	0.25
2	File1:Displacement1	File1:Displacement2	File1:Displacement3	0.001	1
*					

Imported Displacement – Boundary Prescribed Final Geometry

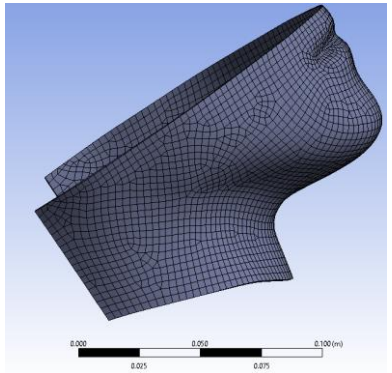
- The displacements between a scaled version of the chin and the full size geometry are calculated and imported via an External Data system
- An imported displacement load is used to apply the displacement from the initial to the final configuration pulling the helmet into place and pre-stressing the chin straps



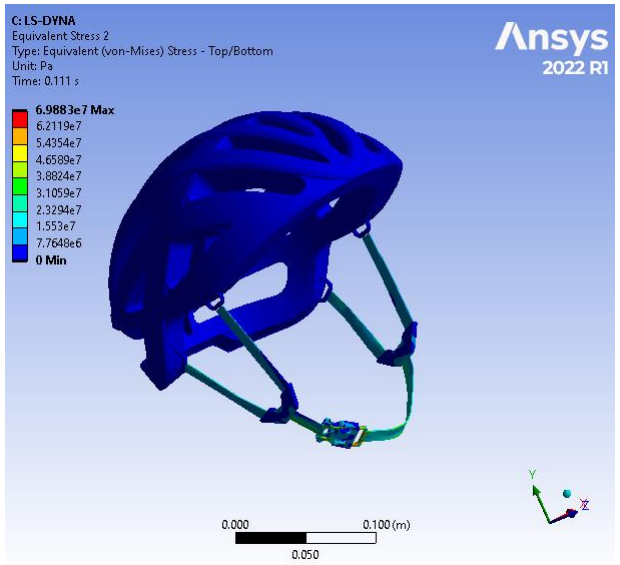


Initial Configuration

Apply Displacements

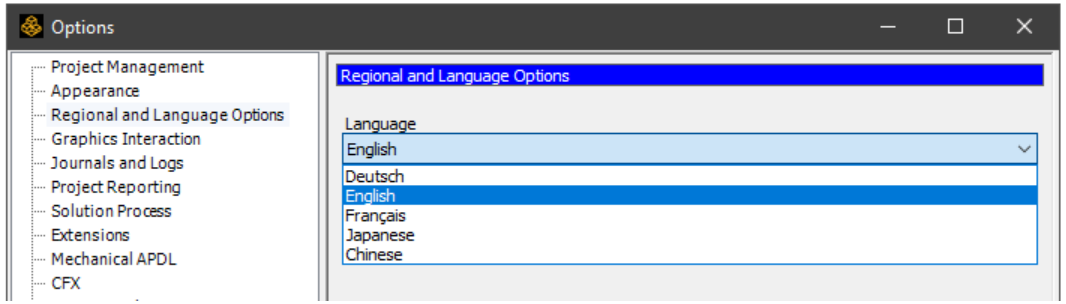


Final Configuration



Localization

- The LS-DYNA Worbench system now supports additional languages
- Enhancement covers items specific to the LS-DYNA analysis system; Analysis Settings, Load Objects, Toolbars



Language Selection

Détails de "Réglages de l'analyse"

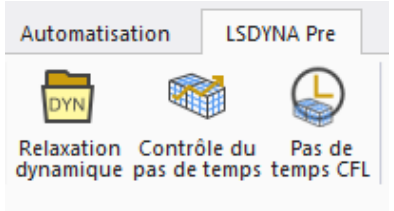
Contrôles de pas	
Heure de fin	0.001 s
Coefficient de sécurité de l'incrément de temps	0.9
Nombre de cycles maximal	10000000
Mise à l'échelle automatique de la masse	Non
Gestion du temps CPU et de la mémoire	
Allocation de Mémoire	Contrôlé par le programme
Nombre de CPUs	1
Type de traitement	Contrôlé par le programme
Contrôles de Solveur	
Type de solveur	Contrôlé par le programme
Précision du Solveur	Contrôlé par le programme
Système d'unités	nmm
Explicite Solution Seulement	Oui
Numérotation de Nœud Invariable	Désactivé
Mise à jour de la contrainte de second degré	Non
Version du solveur	Contrôlé par le programme

Analysis Settings

Détails de "Contrôle Hourglass"

Geometry	
Méthode de champ d'application	Sélection de géométrie
Géométrie	1 Corps
Définition	
Type Hourglass	Contrôlé par le programme
LS-DYNA ID	0
Coefficients	
Hourglass	0
Compression quadratique	0
Allongement Linéaire	0

Analysis Specific Loads



Analysis Specific Toolbars

Localization

Includes all 4 additional languages that currently available in Workbench, French, German, Chinese, and Japanese

Détails de "Réglages de l'analyse"

Contrôles de pas	
Heure de fin	0.001 s
Coefficient de sécurité de l'incrément de temps	0.9
Nombre de cycles maximal	10000000
Mise à l'échelle automatique de la masse	Non
Gestion du temps CPU et de la mémoire	
Allocation de Mémoire	Contrôlé par le programme
Nombre de CPUs	1
Type de traitement	Contrôlé par le programme
Contrôles de Solveur	
Type de solveur	Contrôlé par le programme
Précision du Solveur	Contrôlé par le programme
Système d'unités	nmm
Explicite Solution Seulement	Oui
Numérotation de Nœud Invariable	Désactivé
Mise à jour de la contrainte de second degré	Non
Version du solveur	Contrôlé par le programme
Vitesses Initiales	
Contrôles de l'amortissement	
Amortissement Global	Non
Contrôles Hourglass	
Type Hourglass	Contrôlé par le programme
LS-DYNA ID	0
Coefficient de Hourglass défaut	0.1
Contrôles ALE	
Contrôles de Liaison	
Formulation	Contrôlé par le programme
Contrôles des composites	
Contrôles de sortie	
Format de sortie	Contrôlé par le programme
Facteur d'échelle de taille du fichier binaire	70
Contraintes	Oui
Déformations	Non
Déformation plastique	Oui
Historique des Variables	Non
Calculer les résultats à	Contrôlé par le programme
Fichier de contrainte pour pièces flexibles	Non
Contrôles de Sortie de l'Historique des Temps	
Calculer les résultats à	Non
Gestion de données d'analyse	

French

Details von "Analyseeinstellungen"

Schrittsteuerungen	
Endzeit	0.001 s
Zeitschritt-Sicherheitsfaktor	0.9
Maximale Zyklenanzahl	10000000
Automatische Massenskalierung	Nein
CPU und Speicherverwaltung	
Speicherallokierung	Programmgesteuert
Anzahl CPUs	1
Verarbeitungstyp	Programmgesteuert
Solver-Steuerungen	
Solvertyp	Programmgesteuert
Solver-Genauigkeit	Programmgesteuert
Einheitensystem	nmm
Nur explizite Lösung	Ja
Invariante Knotennummerierung	Aus
Spannungsupdate zweiter Ordnung	Nein
Solver Version	Programmgesteuert
Anfangsgeschwindigkeiten	
Dämpfungssteuerungen	
Globale Dämpfung	Nein
Hourglass-Steuerungen	
Hourglass-Typ	Programmgesteuert
LS-DYNA-ID	0
Standard-Hourglass-Koeffizient	0.1
ALE Steuerung	
Gelenkkontrollen	
Formulierung	Programmgesteuert
Composite-Steuerungen	
Ausgabesteuerungen	
Ausgabeformat	Programmgesteuert
Skalierungsfaktor für Binärdateigröße	70
Spannung	Ja
Dehnung	Nein
Plastische Dehnung	Ja
Zeitverlauf-Variablen	Nein
Ergebnisse berechnen für	Programmgesteuert
Spannungsdatei für flexible Teile	Nein
Ausgabesteuerungen für den Zeitlichen Verlauf	
Ergebnisse berechnen für	Nein
Analysedatenverwaltung	

German

"分析设置"的详细信息

步骤控制	
结束时间	0.001 s
时步安全系数	0.9
最大周期数量	10000000
自动质量缩放	没有
CPU和内存管理	
内存分配	程序控制的
CPU数	1
处理类型	程序控制的
求解器控制	
求解器类型	程序控制的
求解器精度	程序控制的
单位系统	nmm
仅显式解	是
不变节点编号	关闭
二阶应力更新	没有
求解器版本	程序控制的
初始速度	
阻尼控制	
全局阻尼	没有
沙漏控制	
沙漏类型	程序控制的
LS-DYNA ID	0
默认沙漏系数	0.1
ALE控制	
连结控制	
公式	程序控制的
复合控制	
输出控制	
输出格式	程序控制的
二进制文件大小比例因子	70
应力	是
应变	没有
塑性应变	是
历史变量	没有
计算结果	程序控制的
柔性部件的应力文件	没有
时间历史输出控制	
计算结果	没有
分析数据管理	

Chinese

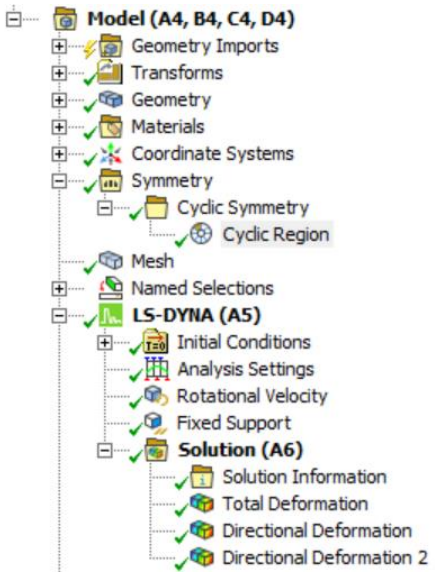
"解析設定" の詳細

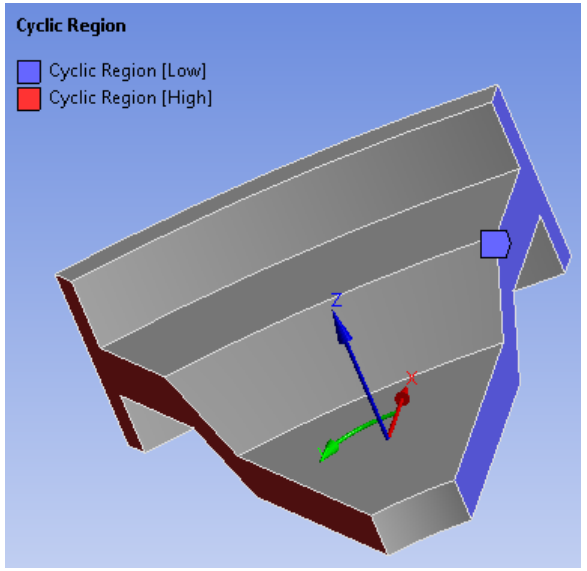
ステップコントロール	
終了時間	0.001 s
時間ステップ安全率	0.9
最大サイクル数	10000000
自動質量スケール	No
CPU およびメモリー管理	
メモリー割り当て	プログラムによるコントロール
CPU 数	1
処理タイプ	プログラムによるコントロール
ソルバーコントロール	
ソルバータイプ	プログラムによるコントロール
ソルバー精度	プログラムによるコントロール
単位系	nmm
陽解法解析のみ	Yes
不変節点番号	オフ
2 次応力の更新	No
ソルバーバージョン	プログラムによるコントロール
初期速度	
減衰コントロール	
グローバル減衰	No
アワーグラス制御	
アワーグラスタイプ	プログラムによるコントロール
LS-DYNA ID	0
デフォルトのアワーグラス係数	0.1
ALE コントロール	
ジョイントコントロール	
定式化	プログラムによるコントロール
複合材コントロール	
出力コントロール	
出力形式	プログラムによるコントロール
バイナリファイルサイズのスケールファクター	70
応力	Yes
ひずみ	No
塑性ひずみ	Yes
履歴変数	No
結果の計算点	プログラムによるコントロール
弾性体の応力ファイル	No
時刻歴出力コントロール	
結果の計算点	No
解析データ管理	

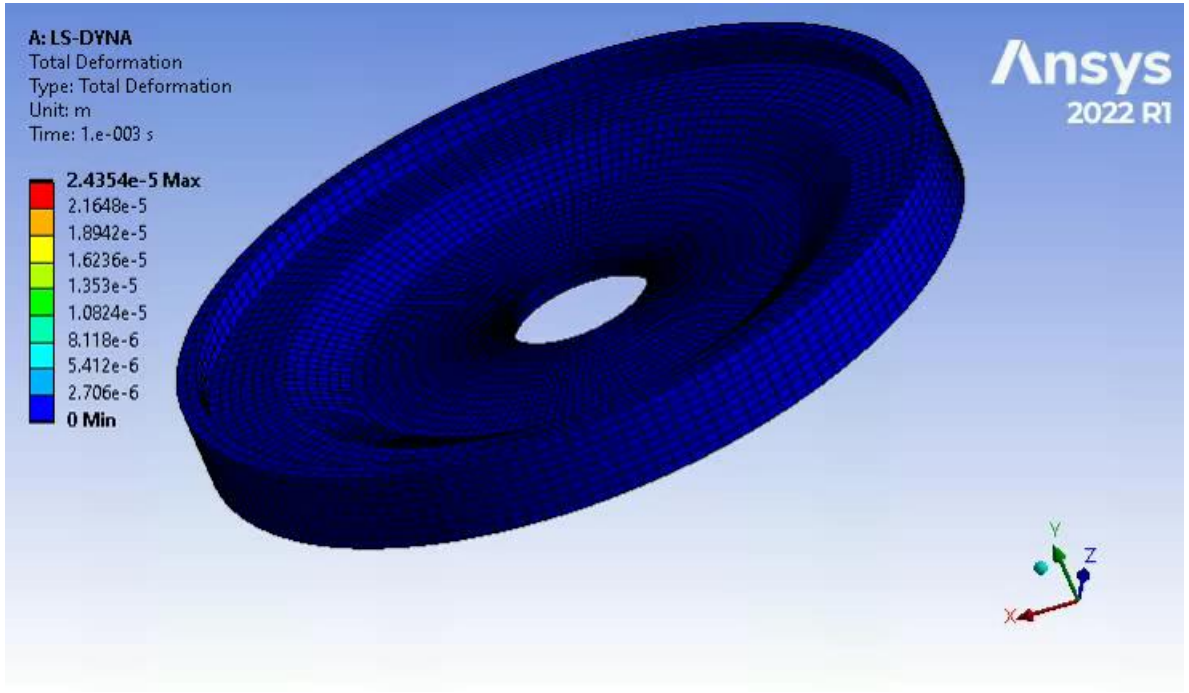
Japanese

Cyclic Symmetry

- Cyclic Symmetry is now supported in the LS-DYNA Workbench system and can be used in turbomachinery applications for faster running times
- Option to display the mesh and results with full symmetry (Beta)

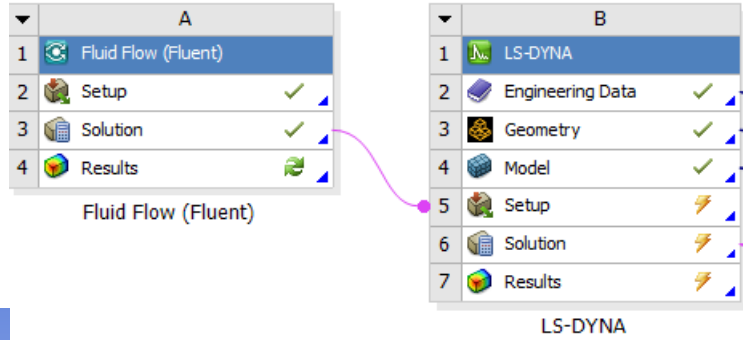


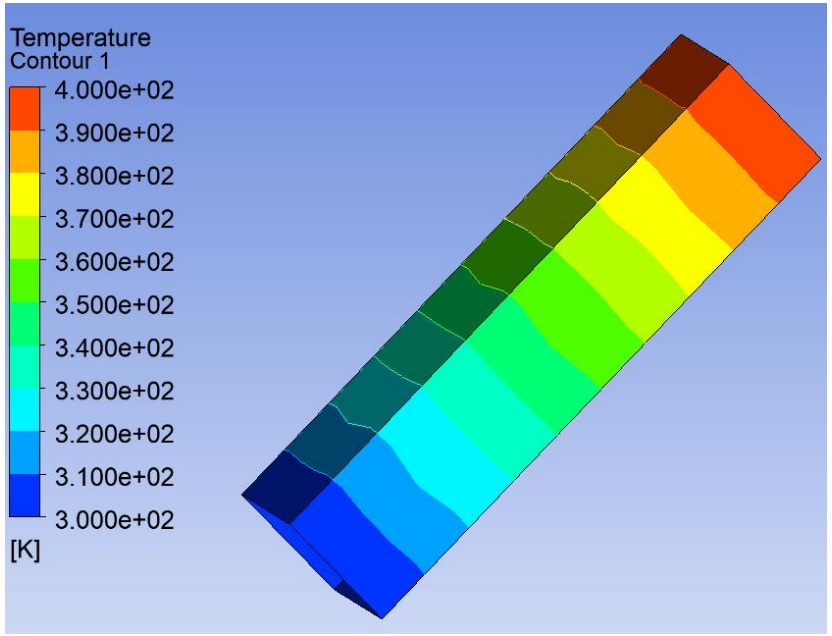




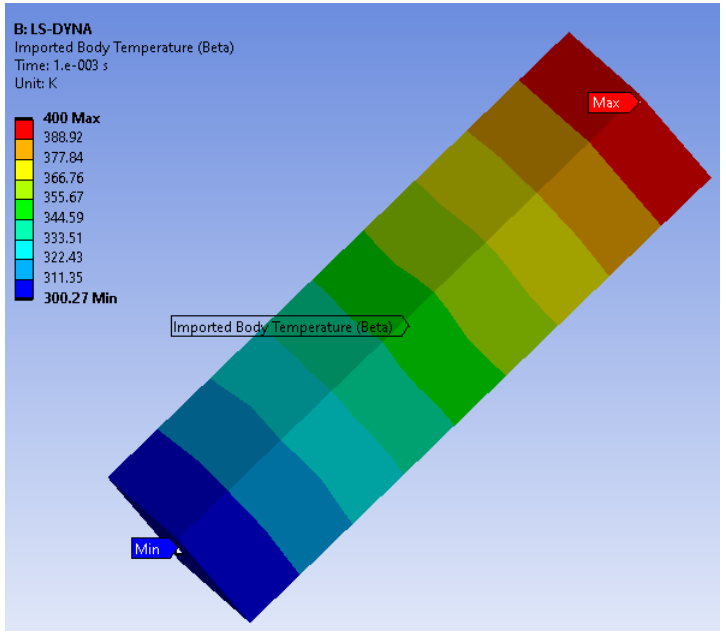
Fluent to LS-DYNA 1 Way Thermal Transfer

- Body Temperatures can be now imported from CFD calculations allowing to take in accurately temperatures effects in LS-DYNA simulations
- Link systems in the project schematic, imported load folder is added automatically, body temperature load can be inserted from the context menu
- Standard imported load features are available; Stepped\Ramped loading with the option to apply scale factor and offset

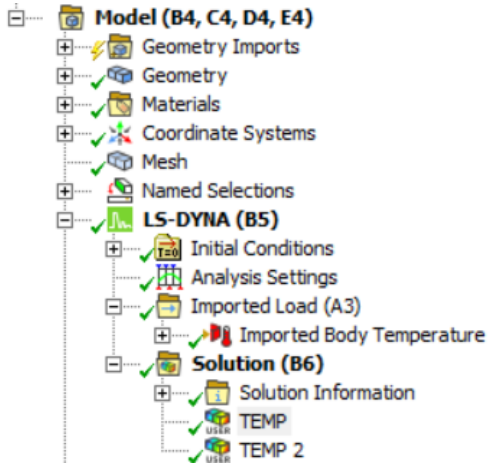




Fluent

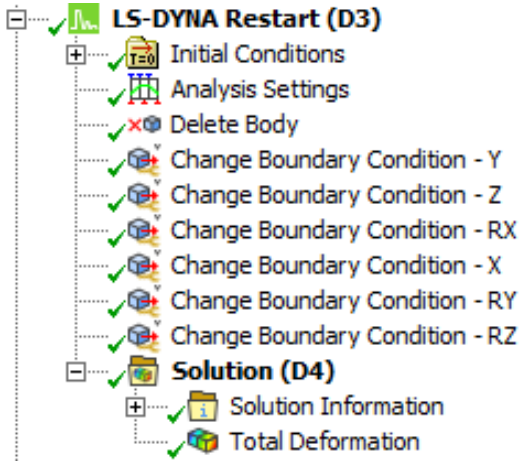


LS-Dyna



Restarts Improvements

- Displacements and Remote Displacements can be modified in restarts allowing to simulate complex movements
- Location method allows the selection of a boundary condition or a curve (added using the keyword manager)
- Each component of a boundary condition can be independently redefined by a curve

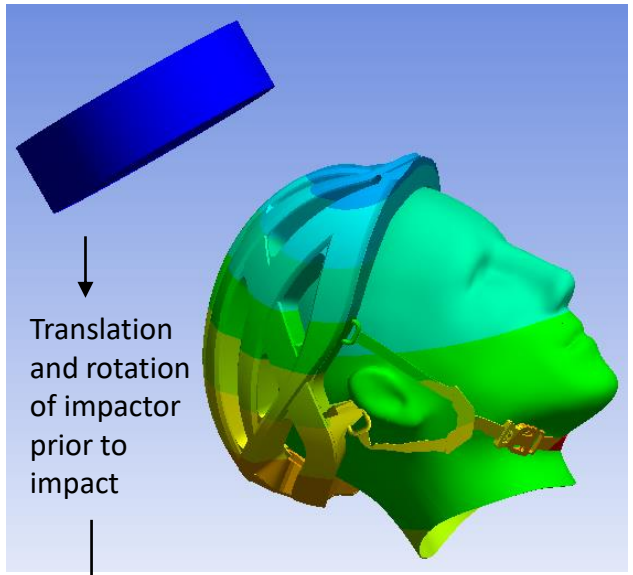


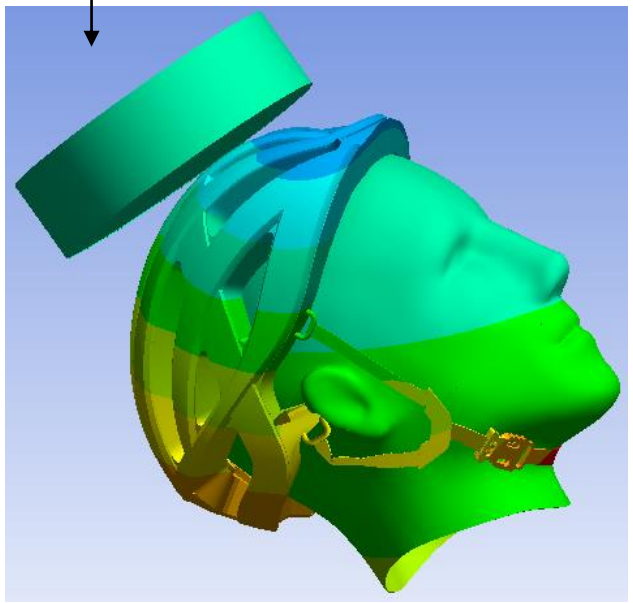
Details of "Change Boundary Condition 2"

Definition	
Location Method	Boundary Condition
Boundary Condition	Remote Displacement_Round_Impactor
Component	Y Component

Tabular Data

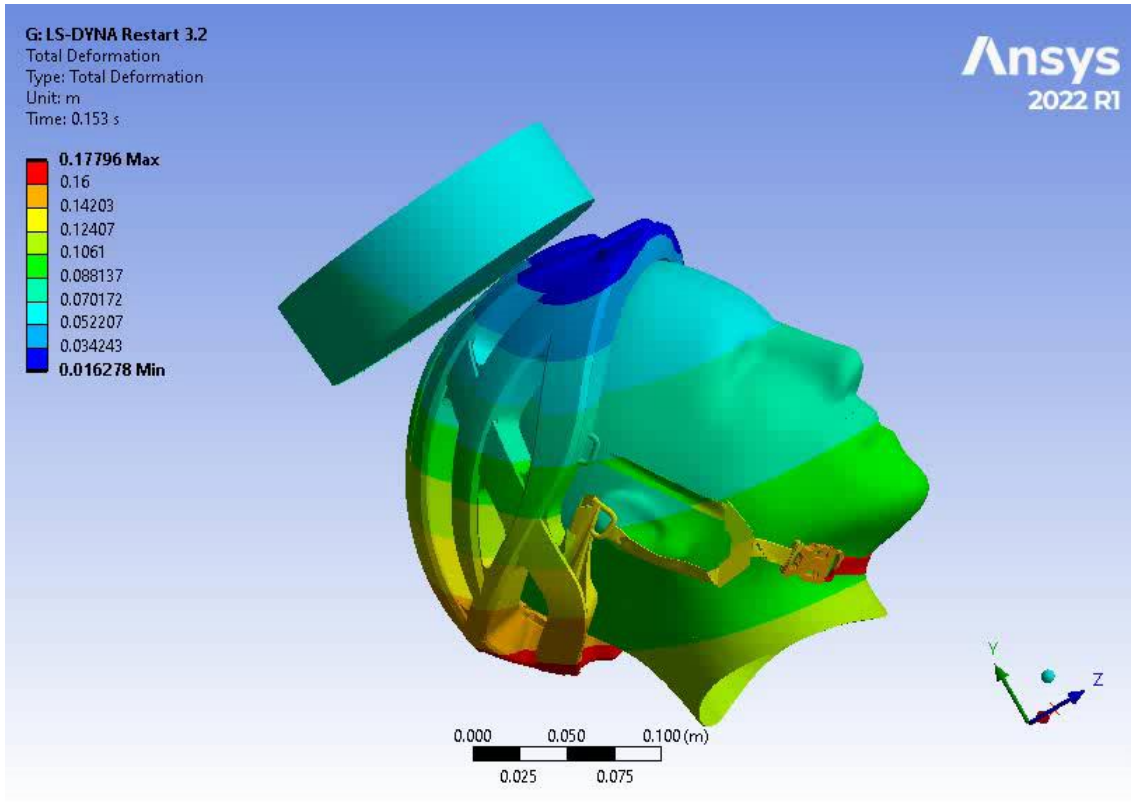
		Time [s]	Displacement [m]
1	1	0.	= 0.
2	1	0.132	0.
3	1	0.133	3.5e-002
4	N/A	3.	3.5e-002
5	N/A	4.	3.5e-002
*			

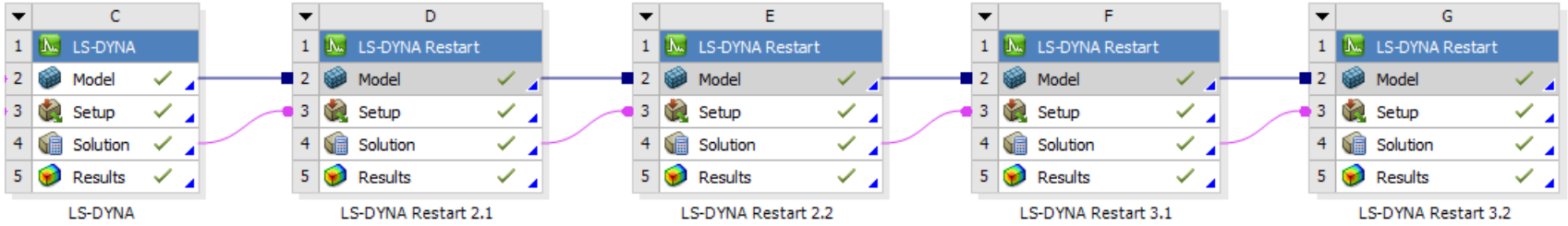




Restarts Improvements

- Multi-step impact analysis using a series of small restarts
  - Pre-Stress
  - Impactor 1 Positioning
  - Impact 1
  - Impactor 2 Positioning
  - Impact 2
- Uses the new Change Boundary Condition object along with the Delete Body and Change Velocity objects





Enhanced Material Support

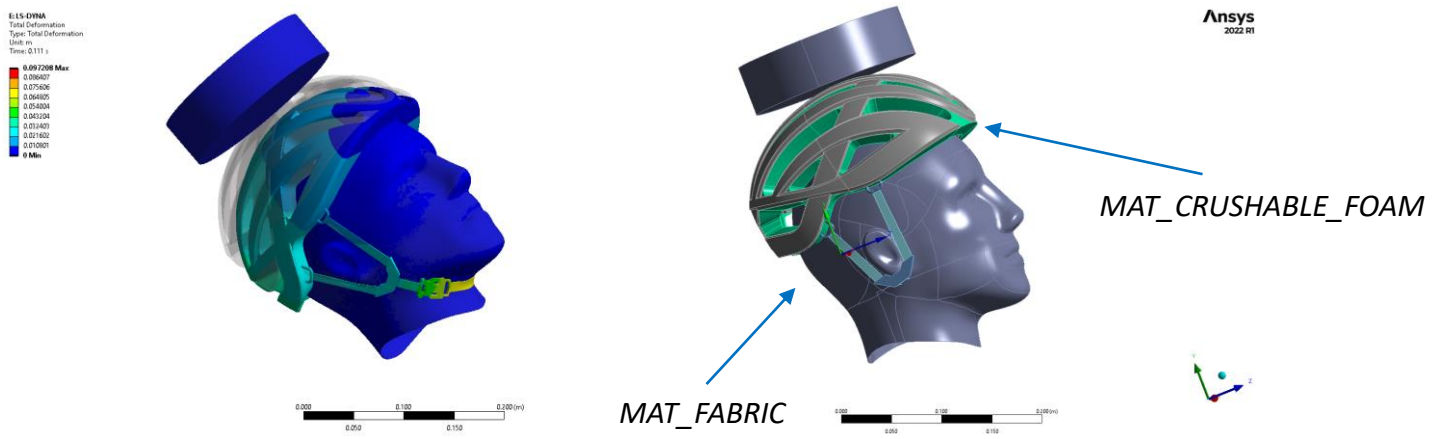
- 4 additional material models have been introduced for applications using Fabric\Foam
- Can be imported to Engineering Data from .k file using External Model
- These definitions generally follow the input card with the variable names added as a suffix

LSDYNA External Model - MAT

- *MAT_ELASTIC
- *MAT_ORTHOTROPIC_ELASTIC
- *MAT_ANISOTROPIC_ELASTIC
- *MAT_PLASTIC_KINEMATIC
- *MAT_BLATZ-KO_RUBBER
- *MAT_HIGH_EXPLOSIVE_BURN
- *MAT_NULL
- *MAT_JOHNSON_COOK
- *MAT_POWER_LAW_PLASTICITY
- *MAT_PIECEWISE_LINEAR_PLASTICITY
- *MAT_CRUSHABLE_FOAM
- *MAT_SIMPLIFIED_JOHNSON_COOK
- *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY
- *MAT_SIMPLIFIED_RUBBER/FOAM
- *MAT_BILKHU/DUBOIS_FOAM
- *MAT_FABRIC

Properties of Outline Row 8: MAT_CRUSHABLE_FOAM

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	70	kg m^-3		
4	Isotropic Elasticity				
5	Derive from	Young's Mod...			
6	Young's Modulus	2.5E+07	Pa		
7	Poisson's Ratio	0.2			
8	Bulk Modulus	1.3889E+07	Pa		
9	Shear Modulus	1.0417E+07	Pa		
10	*MAT_CRUSHABLE_FOAM				
11	Definition				
12	Tensile Stress cutoff, tsc	3.9E+05	Pa		
13	Rate Sensitivity via damping coefficient, damp	0			
14	Yield Stress versus Volumetric Strain, lcid	Tabular			
15	Scale	1			
16	Offset	0	Pa		



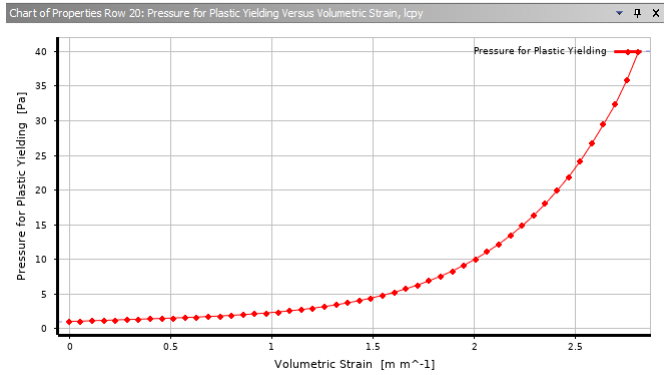
Enhanced Material Support

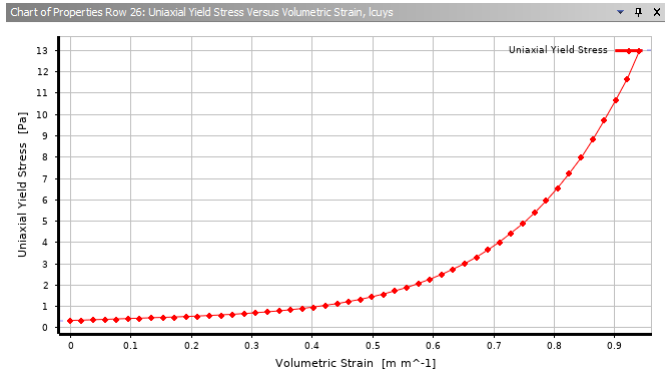
- Associated load curves are available
- Added by default or available under the additional tabular data when selecting the material
- Not all options are supported when there are optional cards

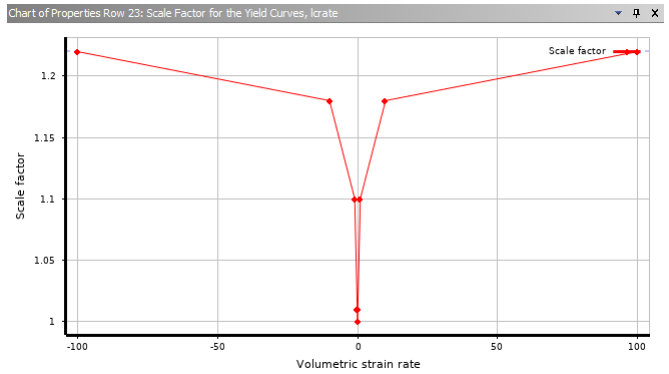
*MAT_BILKHU/DUBOIS_FOAM		
Definition		
Tensile Response	Volumetric Strain	
Viscous Damping Coefficient, vc	0	
Pressure Cutoff, pc	0	Pa
Variable Pressure Cutoff as a Fraction of Pressure Yield Value, vpc	0	Pa
Tension Cutoff for Uniaxial Tensile Stress, tsc	0	Pa
Variable Tension Cutoff as a Fraction of the Uniaxial Compressive Yield Strength, vtsc	0	Pa
Stiffness Coefficient for Contact Interface Stiffness, kcon	0	Pa
Number of Cycles to Determine the Average Volumetric Strain Rate, ncycle	0	
Pressure for Plastic Yielding Versus Volumetric Strain, lcpy	Tabular	
Uniaxial Yield Stress Versus Volumetric Strain, lcuys	Tabular	

Additional Tabular Data

Scale Factor for the Yield Curves

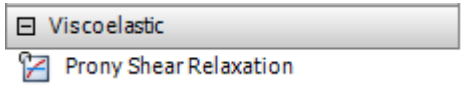






Enhanced Material Support: Prony Series

- Prony Series (Shear Relaxation) are supported for the following material properties, allowing to simulate time-dependent stress relaxation effects:
  - Mooney-Rivlin
  - Ogden
  - Yeoh
  - Polynomial
  - Arruda-Boyce
  - MAT_SIMPLIFIED_RUBBER/FOAM (new addition at 2022 R1)



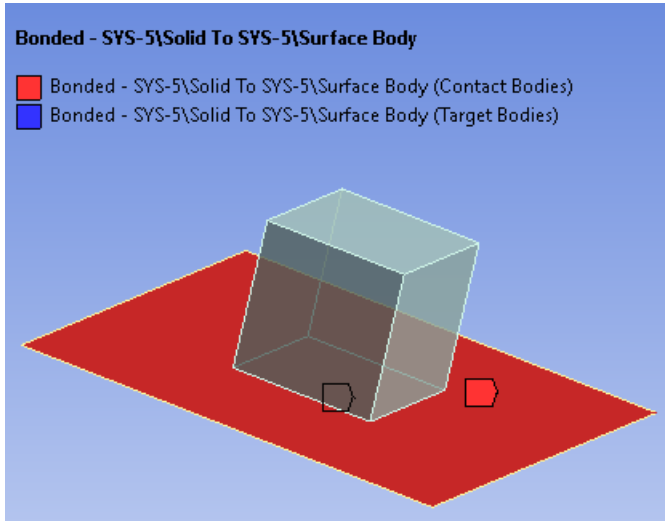
Properties of Outline Row 3: Admat_OD1

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	1000	kg m^-3		
4	Ogden 1st Order				
5	Material Constant MU1	23845	Pa		
6	Material Constant A1	2.1151			
7	Incompressibility Parameter D1	1.4429E-07	Pa^-1		
8	Prony Shear Relaxation	Tabular			
9	Number of Terms	12			
10	Relative Moduli(i): Scale	1			
11	Relative Moduli(i): Offset	0			
12	Relaxation Time(i): Scale	1			
13	Relaxation Time(i): Offset	0	s		

	B	C	D
1	Index i	Relative Moduli(i)	Relaxation Time(i) (s)
2	1	1000	0.0001
3	2	10000	0.00011
4	3	9000	0.002
5	4	8000	0.003
6	5	7000	0.004
7	6	6000	0.005
8	7	5000	0.006
9	8	4000	0.007
10	9	3000	0.008
11	10	2000	0.009
12	11	1000	0.00111
13	12	2050	0.0123

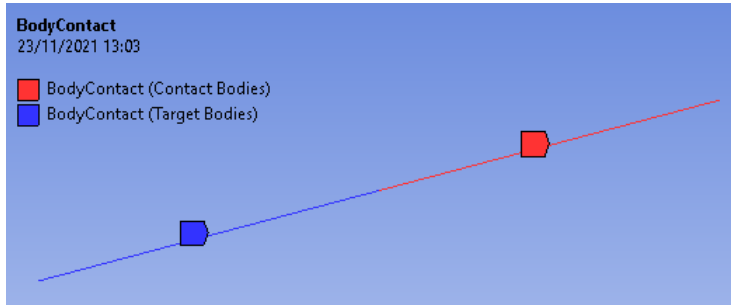
Contact Scoping

- Additional contact scoping options are available for LS-DYNA
- Contacts can now be scoped to Shell and Beam bodies in a 3D Analysis



Details of "BodyContact"

Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	1 Body
Contact Bodies	Beam2
Target Bodies	Beam1
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Trim Contact	Program Controlled
Maximum Offset	1.e-002 m
Breakable	No
Suppressed	No
Advanced	
Formulation	Program Controlled



Details of "Bonded - SYS-5\Solid To SYS-5\Surface Body"

Scope	
Scoping Method	Geometry Selection
Contact	1 Body
Target	1 Face
Contact Bodies	SYS-5\Surface Body
Target Bodies	SYS-5\Solid
Contact Shell Face	Program Controlled
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Maximum Offset	1.e-007 m
Breakable	No
Suppressed	No
Advanced	
Formulation	Program Controlled

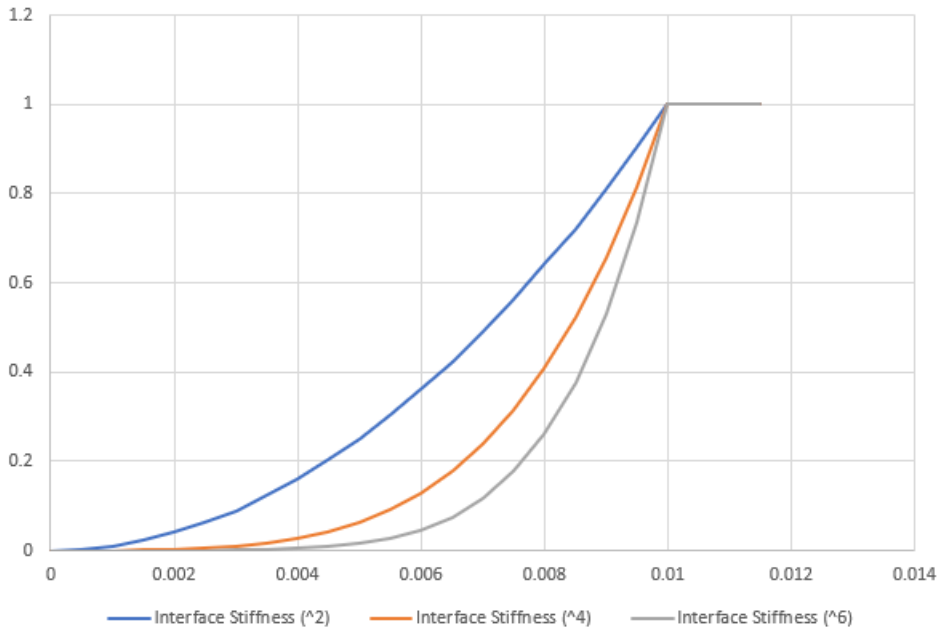
Interference Contact Properties

- New options added to give additional flexibility in contact stiffness curve definition
- End time for contact. Previously this just used analysis time
- Stiffness curve function exponent to vary the transition to peak stiffness

```
*DEFINE_CURVE_FUNCTION
$        ID       sidr
         3          0
min(1, (time / 0.01)**4)
```

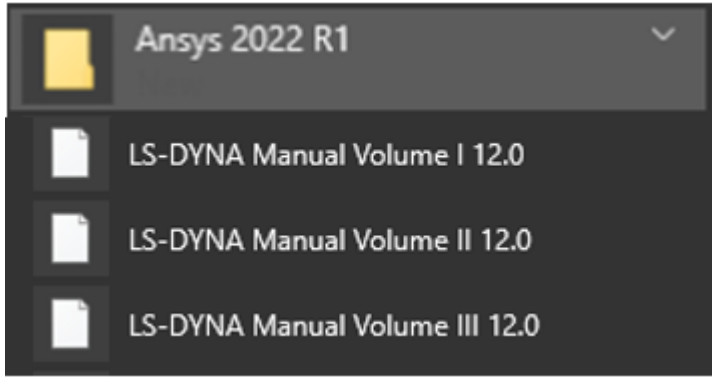
Details of "Contact Properties 2"

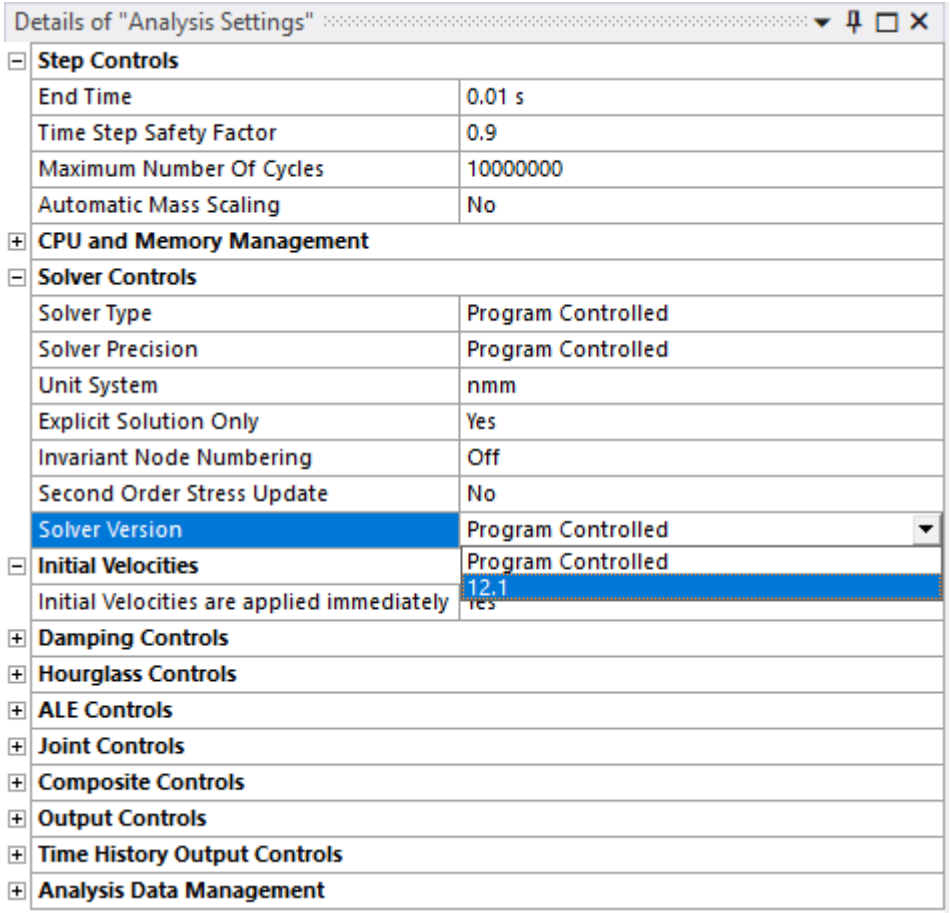
Definition	
Contact	Frictional - Multiple To Multiple (Interference)
Type	Interference
Formulation	SURFACE_TO_SURFACE_INTERFERENCE
Common Controls	
Birth Time	0 s
Death Time	0 s
Viscous Damping Coefficient	10
Contact Penalty Scale Factor	4
Target Penalty Scale Factor	4
Advanced Controls	
Optional Thickness for Contact Surface	0 m
Optional Thickness for Target Surface	0 m
Optional Solid Element Thickness	0 m
Soft Constraint Formulation	Program Controlled
Soft Constraint Scale Factor	0.1
Depth	5
Interference Controls	
Stiffness Scale factor At End of Dynamic Relaxation	1
Interference End Time	0.01 s
Stiffness Curve Exponent	4



New Solver Version

- Default solver version for 2022 R1 is 12.1
- Refer to the documentation for a summary of new developments \ improvements to stability and release notes for further information



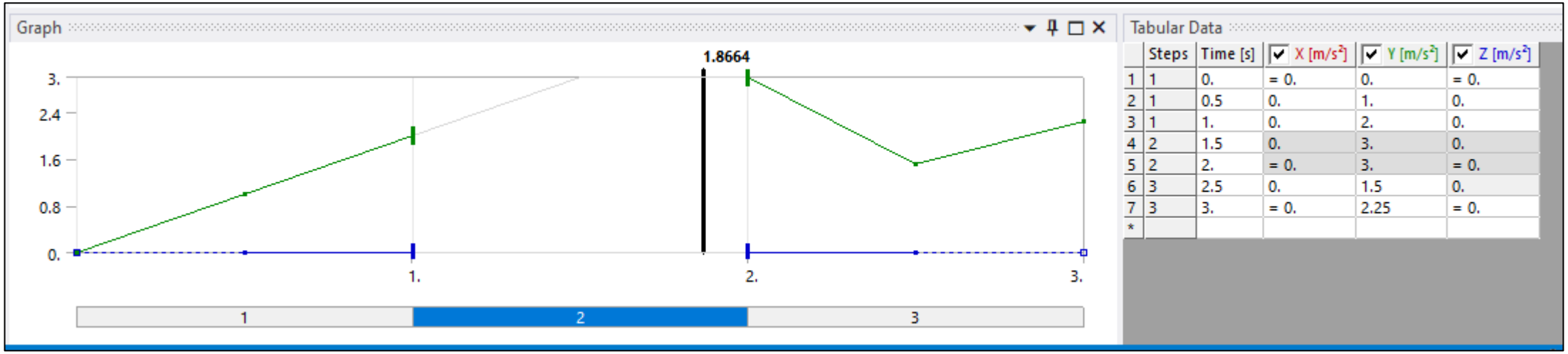


Rigid Body Dynamics

Time Varying Acceleration Load

Remove the RBD solver specific limitations on the Acceleration and Standard Earth Gravity loads. Now the definition of Acceleration or Standard Eart Gravity is fully consistent with the other solvers, particularly the MAPDL solver, allowing smooth transition from one solver to another.

- Allows time varying accelerations specified using functions or tabular data
- Allows activating/deactivating acceleration or Standard Earth Gravity per load steps



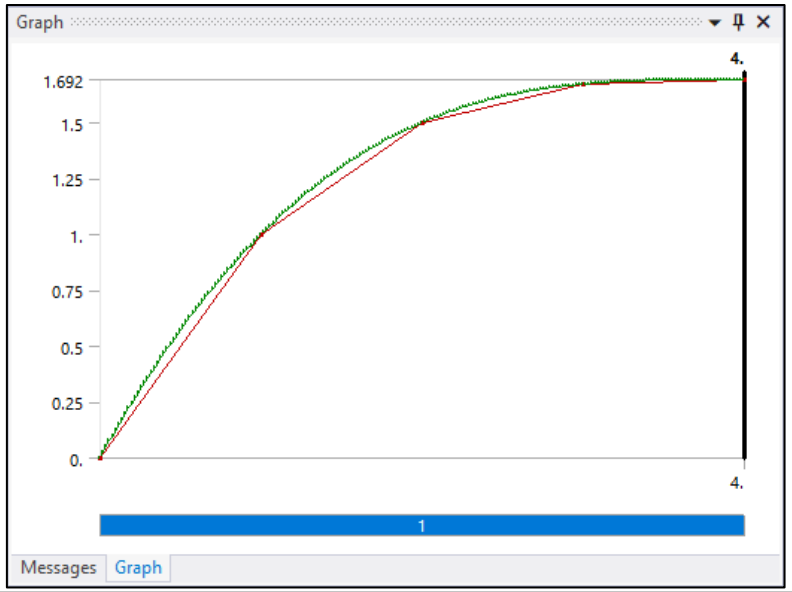
	Steps	Time [s]	X [m/s²]	Y [m/s²]	Z [m/s²]
1	1	0.	= 0.	0.	= 0.
2	1	0.5	0.	1.	0.
3	1	1.	0.	2.	0.
4	2	1.5	0.	3.	0.
5	2	2.	= 0.	3.	= 0.
6	3	2.5	0.	1.5	0.
7	3	3.	= 0.	2.25	= 0.
*					

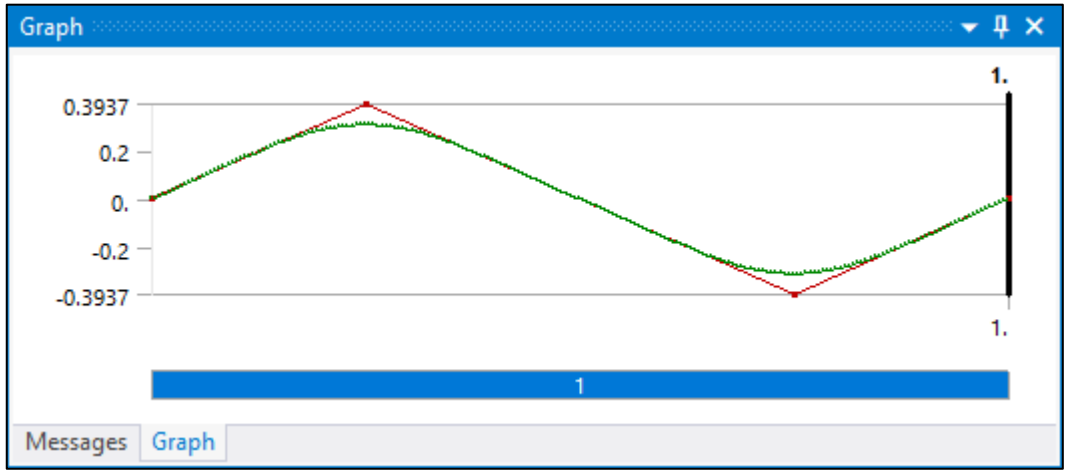
Display of Joint Condition Fitting

- In the RBD solver, imposed joint conditions (displacement, velocity, rotations, rotational velocities) given by tables are internally fitted

➔consequently, the results may differ from the piecewise curve display on the joint condition.

The results of the fitting is now overlaid on the joint condition curve.



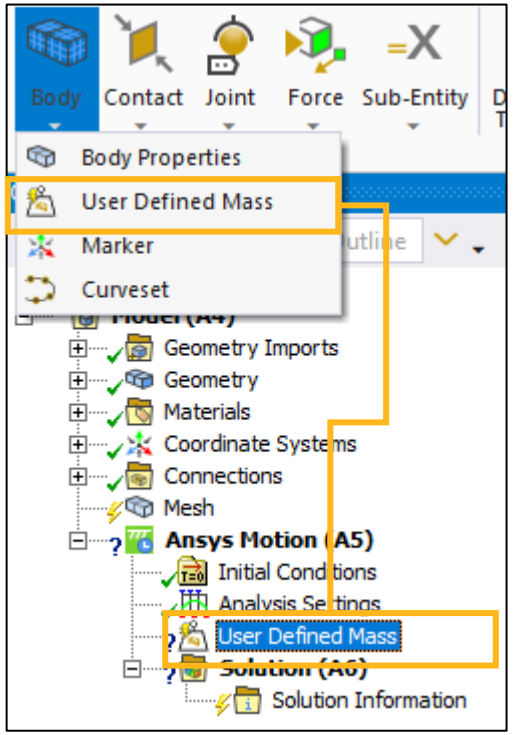


Motion Integration with Mechanical

Ansys

User Defined Mass

- Change Mass/Inertia property
  - A common practice in rigid dynamics analysis is to replace a complex geometry by a simpler one, but it is important to use the right geometry properties, thus User Defined Mass object allows to modify the Mass, Moments of Inertia and centroid of selected rigid bodies

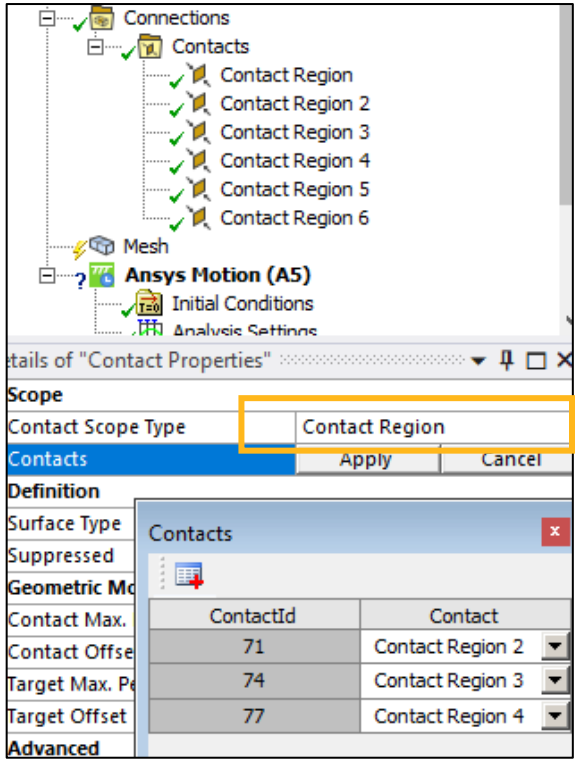


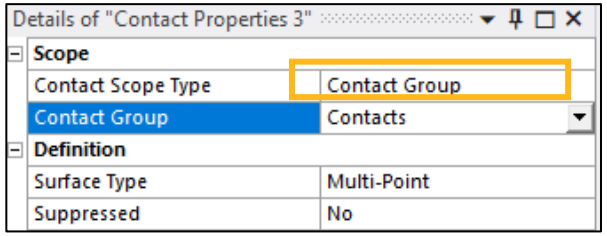
Details of "User Defined Mass"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Mass	9.8125 kg
Moment of Inertia Ip1	0.408854117927411 kg·m²
Moment of Inertia Ip2	0.408854069188203 kg·m²
Moment of Inertia Ip3	0.408854069188204 kg·m²
Centroid	Unchanged

Contact Group Scoping

- Improvement in selection Method
  - Contact Properties, Contact Friction Properties and Tie Properties objects are used to modify contact settings. For important number of contact regions, it was cumbersome to select all desired contacts to modify, thus now it is possible to modify all contacts of a given Connection Group improve a lot user friendliness



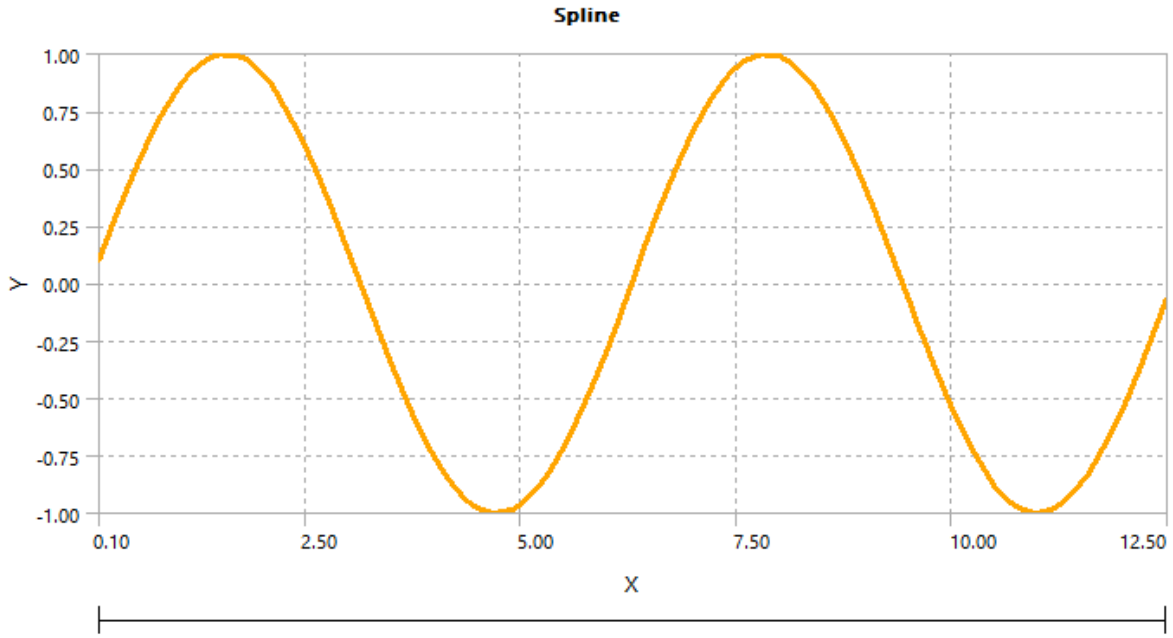


New – Contact group

Previous – Contact Region with tabular data

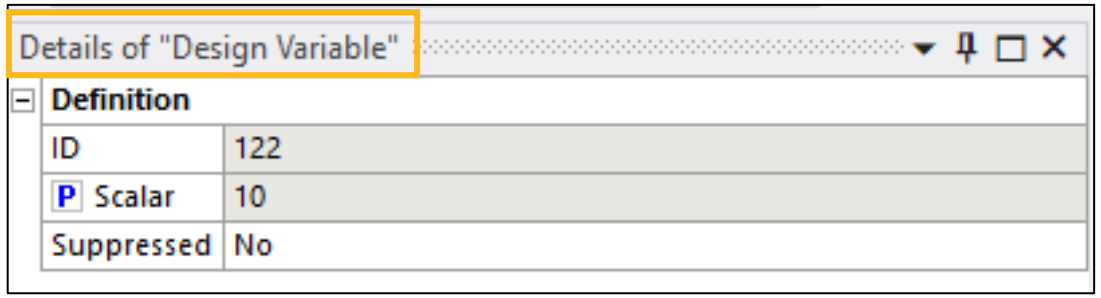
Function Previewer

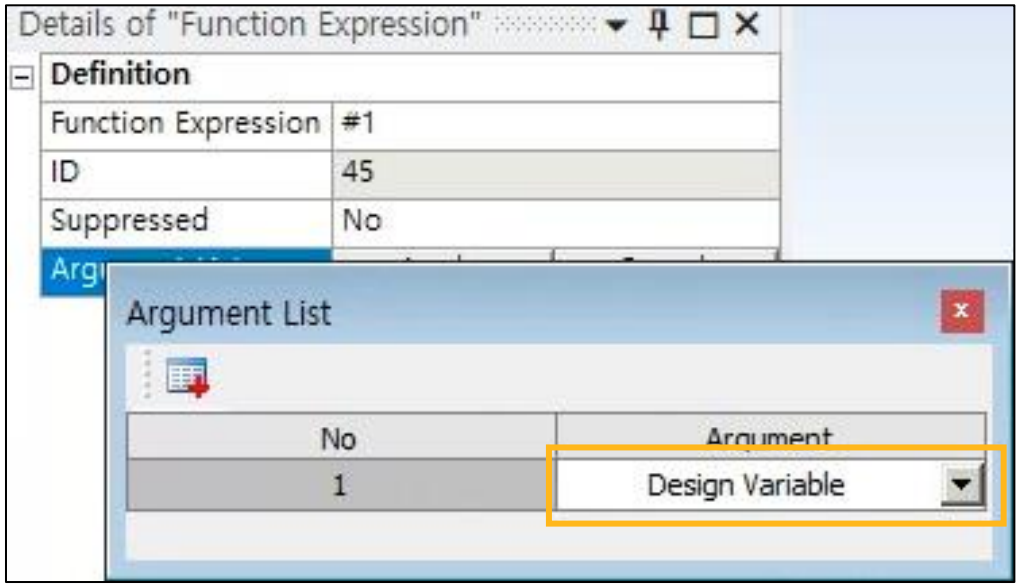
- Preview function expression
  - Evaluation of function expression expressed by Mathematical(sin, cos, tan and so on), Logical (If) and several constant value are available
  - Integration variables are not available and will be extended for STEP, LININT in next version



Design Variable

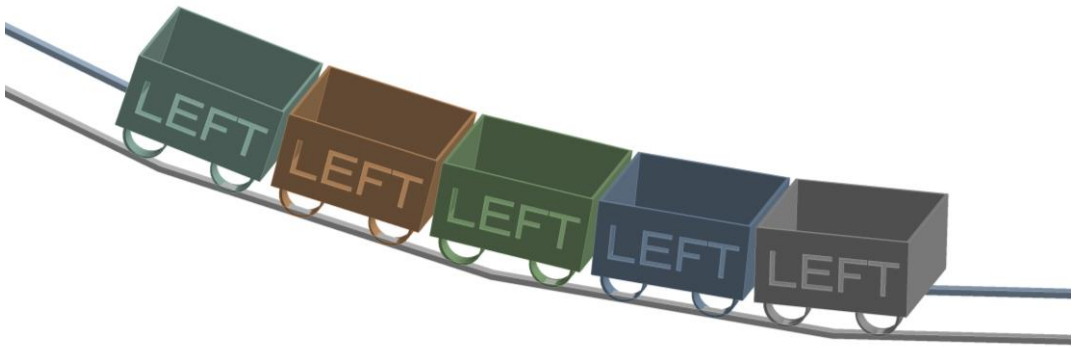
- Parametric study with design variable
  - A Design Variable(DV) object can be assigned to “Parameter Set”.
  - The DV can be used within Function Expression, and it enables to perform design experiments with complex loading conditions





Links: Path Follower

- A new way to assemble segment
  - Links toolkit enables you to assemble segment along with imaginary line passing “Path”. On the other hands, A Path Follower function enables you to place segments on the predefined Curvesets
  - It can be useful to build a roller coaster or train applications for placing cabin on the rail



Additional Improvements

- Extended Joint Friction
  - Joint Friction Properties has been extended to Translational and Cylindrical joints
  - Transition Velocity coefficient is now available

Details of "Joint Friction Properties 2"	
Scope	
Joint	Joint 2 [Translational]
Definition	
Static Friction Coefficient	0.5
Dynamic Friction Coefficient	0.3
Stiction Transition Velocity	0.0001 m/s
Transition Velocity Coefficient	1.5
Max Stiction Deformation	1E-05 m
Reaction Force	Use
Bending Moment	Use
Torsional Moment	Use
Friction Effect	Sliding And Stiction
Reaction Arm	0.001 m
Overlap Option	Constant
Initial Overlap	0.001 m
Pre Force	0 N

- FMI 2.0
  - FMI 2.0 is available. Interface Time Step and Model description are available

Details of "Co-Simulator"	
Definition	
Interface Type	FMI
FMI Version	2.0
Interface Time Step	0.01 s
Model Description	pendulum
Signal Input	Tabular Data
Signal Output	Tabular Data
Suppressed	No

Postprocessing Improvements

- Custom Result
  - Custom Result object allows you to evaluate output of Function expression and User Subroutine
  - Result values can also be used as output parameter

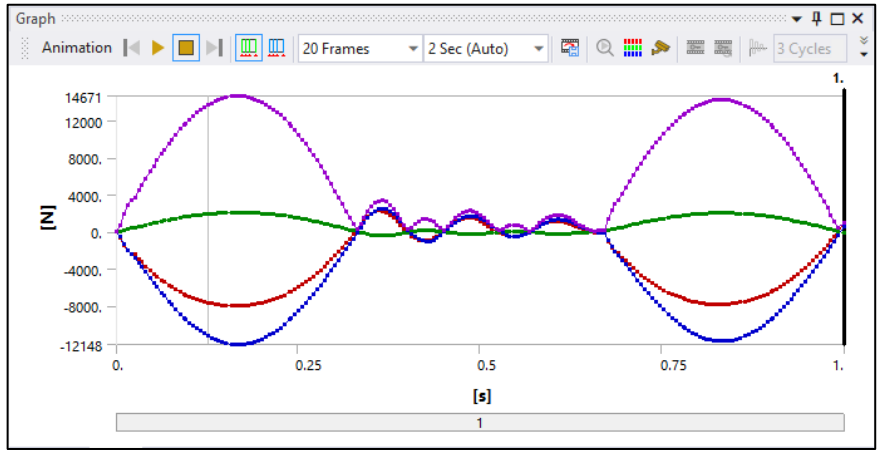
Details of "Custom Result"

Definition	
Type	Function Expression
Function Expression	output_Knuckle_right_dz
Unit Name	Length
Suppressed	No
Results	
Result	0.150153694 m
Maximum Value Over Time	
Result	0.242967413 m
Minimum Value Over Time	
Result	0.142779276 m

- Joint Probes
  - Joint Probe results can now be used to analyze joint results

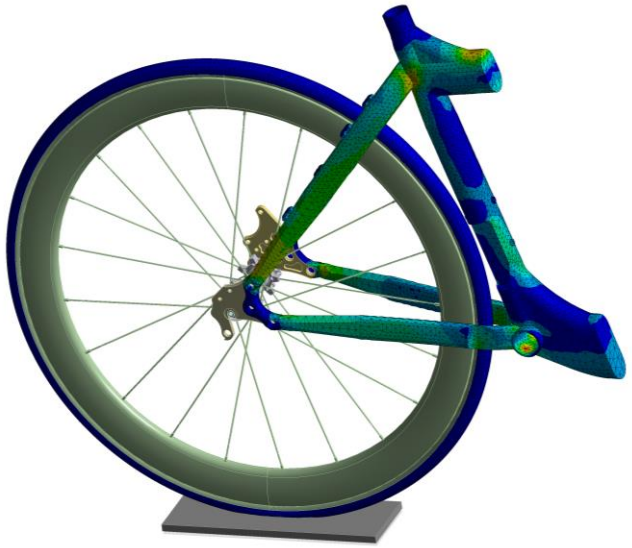
Details of "Joint Probe"

Definition	
Type	Joint Probe
Boundary Condition	BUSHING_05
Orientation Method	Joint Reference System
Suppressed	No
Options	
Result Type	Total Force
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
X Axis	2295.6 N
Y Axis	2091.3 N
Z Axis	2485.4 N
Total	14671 N
Minimum Value Over Time	
X Axis	-7961.4 N
Y Axis	-393.5 N
Z Axis	-12148 N
Total	0.11785 N



Postprocessing Improvements

- Nodal Averaged Stress & Strain
  - Stresses and strains are now evaluated either at the nodes or at the element
  - Thus, now Mechanical postprocessing is consistent with standalone Postprocessor



nCode DesignLife

Ansys

Vibration Fatigue, Direct Harmonic Case with PSD Loading

DesignLife Mechanical UI allows Direct Harmonic calculation with PSD Loading:

- 4 Analysis Types supported - Stress, Strain, Solid and Shell Seam Weld
- Single Loading Event and Multiple Loading Events
- Harmonic Environment selection
- PSD Cycle Counting Method: Lalanne, Dirlik, Narrow Band or Strainberg
- Table Definition: Input Frequency vs Values

Details of "Analysis Settings"

Definition	
Analysis Domain	Time based
Analysis Type	Time based / Frequency based
Scale Factor	1
Calculate Safety Factor	No
Number of Threads	2
Solver Directory	0
Analysis Data Management	

Details of "Loading Event"

Definition	
Domain	Frequency
Event	Loading Event
Type	PSD
Repeat Count	1
Add Load	
Add Load	Add

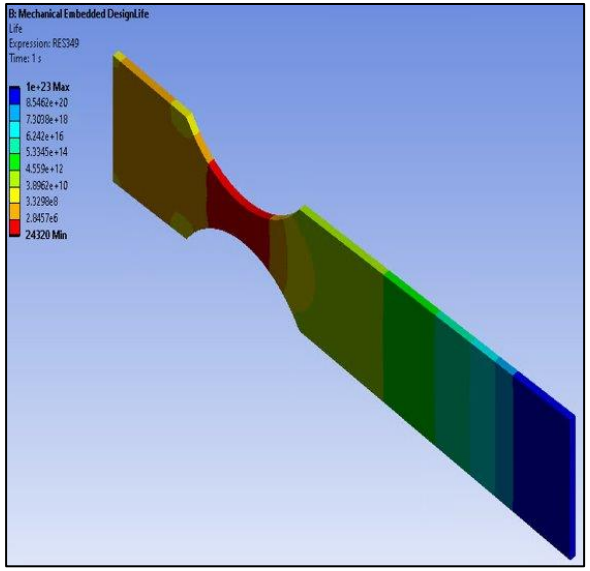
Details of "PSD Load"

Definition	
Environment	Harmonic Response
PSD Cycle Counting Method	Lalanne
Table Definition	Tabular Data
Use Static Load Case	Yes
Static Load Case Environment	Static Structural
Static Load Case Step	2

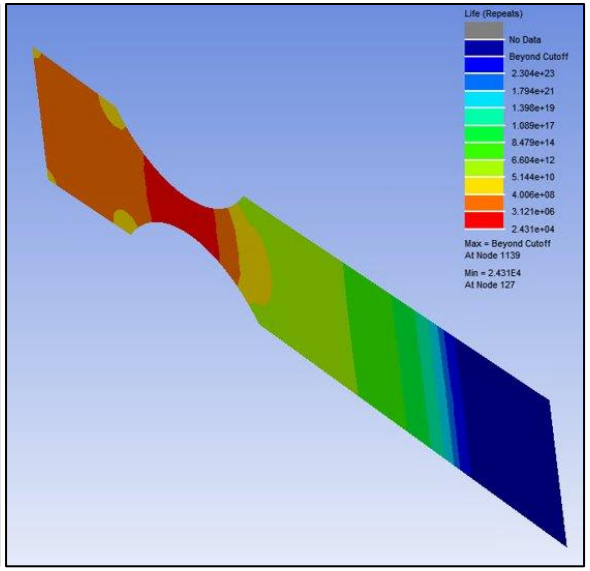
Vibration Fatigue, Direct Harmonic Case with PSD Loading for Stress Analysis Type

Life Result

Mechanical UI



Standalone



Overall – Result comparison

Mechanical UI

Node	Damage	RMS stress [MPa]	Irregularity factor	Largest Stress Cycle Amplitude [MPa]	Life [Repeats]
127	4.11E-05	60.09	0.741	173.2	2.4320E+04
192	3.94E-05	59.84	0.7415	172.5	2.5370E+04
126	3.53E-05	59.23	0.9286	165.8	2.8340E+04
193	3.45E-05	59.1	0.9284	165.4	2.8970E+04
128	3.42E-05	58.63	0.425	180.6	2.9240E+04
191	3.30E-05	58.48	0.4505	179	3.0310E+04
295	2.35E-05	56.77	0.7387	163.7	4.2590E+04
552	2.23E-05	56.5	0.8782	159.4	4.4890E+04

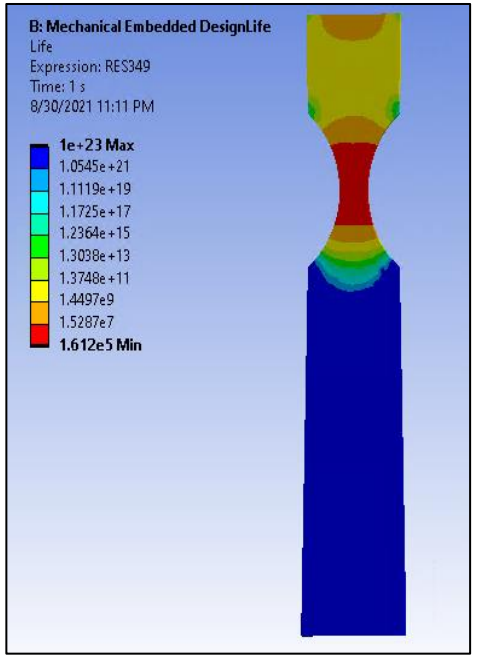
Standalone

Node	Damage	RMS stres	Irregularity factor	Largest Stress Cycle Amplitude	Life
127	4.11E-05	60.09	0.7237	173.7	2.4310E+04
192	3.94E-05	59.84	0.7289	172.8	2.5360E+04
126	3.52E-05	59.22	0.9372	165.6	2.8410E+04
193	3.45E-05	59.09	0.9349	165.3	2.9020E+04
128	3.42E-05	58.62	0.4273	180.5	2.9290E+04
191	3.32E-05	58.48	0.4392	179.5	3.0170E+04
295	2.35E-05	56.77	0.711	164.5	4.2530E+04
552	2.22E-05	56.5	0.8993	158.9	4.4970E+04

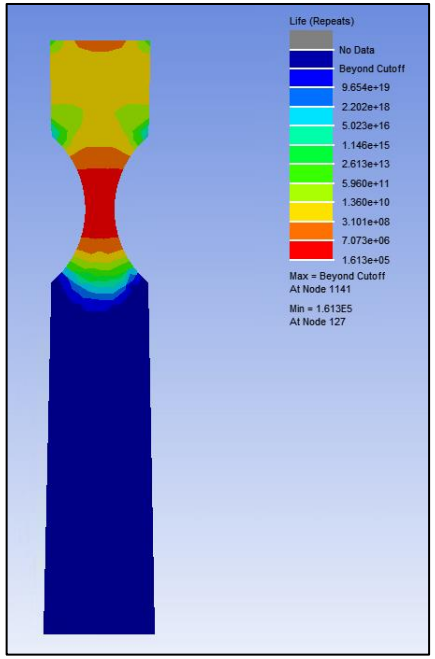
Vibration Fatigue, Direct Harmonic Case with PSD Loading for Strain Analysis Type

Life Result

Mechanical UI



Standalone



Overall – Result comparison

Mechanical UI

Node	Damage	RMS stress [Pa]	Mean stress [Pa]	Return Period to Exceed UTS [s]	Irregularity factor	Largest Sti	Life [Repeats]
127	6.2040E-06	5.86E+07	0	1.71E+05	0.7382	1.70E+08	1.6120E+05
192	5.9750E-06	5.83E+07	0	1.98E+05	0.739	1.69E+08	1.6740E+05
126	5.4000E-06	5.77E+07	0	3.56E+05	0.927	1.63E+08	1.8520E+05
128	5.3010E-06	5.72E+07	0	2.23E+05	0.4225	1.77E+08	1.8860E+05
193	5.2870E-06	5.76E+07	0	3.87E+05	0.9268	1.62E+08	1.8920E+05
191	5.1860E-06	5.70E+07	0	2.44E+05	0.4249	1.77E+08	1.9280E+05
295	3.8180E-06	5.55E+07	0	1.19E+06	0.7362	1.61E+08	2.6190E+05
552	3.6480E-06	5.52E+07	0	1.69E+06	0.8762	1.57E+08	2.7410E+05
397	3.6280E-06	5.52E+07	0	1.65E+06	0.8287	1.58E+08	2.7570E+05
296	3.4610E-06	5.49E+07	0	2.21E+06	0.9267	1.55E+08	2.8890E+05

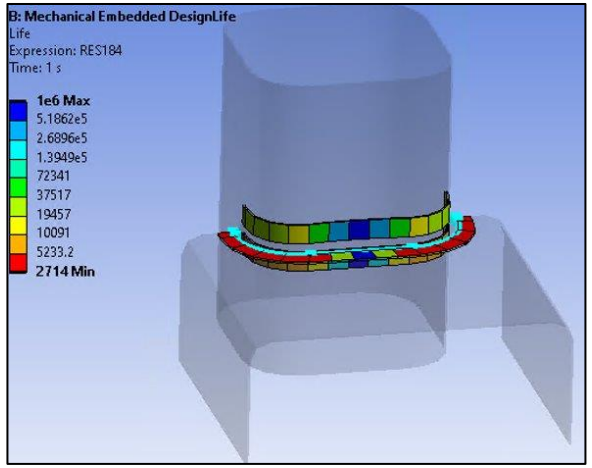
Standalone

NodeId	Damage	RMSStress	MeanStress	ReturnPeriodToExceedUTS	IrrFactor	LargestStressCycle	Life
127	6.198E-06	5.86E+07	0	1.74E+05	0.7499	1.70E+08	1.613E+05
192	5.967E-06	5.83E+07	0	2.04E+05	0.7597	1.69E+08	1.676E+05
126	5.386E-06	5.77E+07	0	3.62E+05	0.9369	1.62E+08	1.857E+05
193	5.275E-06	5.76E+07	0	3.93E+05	0.9352	1.62E+08	1.896E+05
128	5.264E-06	5.71E+07	0	2.31E+05	0.4351	1.77E+08	1.900E+05
191	5.150E-06	5.70E+07	0	2.52E+05	0.4375	1.76E+08	1.942E+05
295	3.817E-06	5.55E+07	0	1.20E+06	0.7378	1.61E+08	2.620E+05
552	3.639E-06	5.52E+07	0	1.75E+06	0.9056	1.56E+08	2.748E+05
397	3.626E-06	5.52E+07	0	1.65E+06	0.8311	1.58E+08	2.758E+05
296	3.454E-06	5.49E+07	0	2.24E+06	0.9347	1.55E+08	2.895E+05

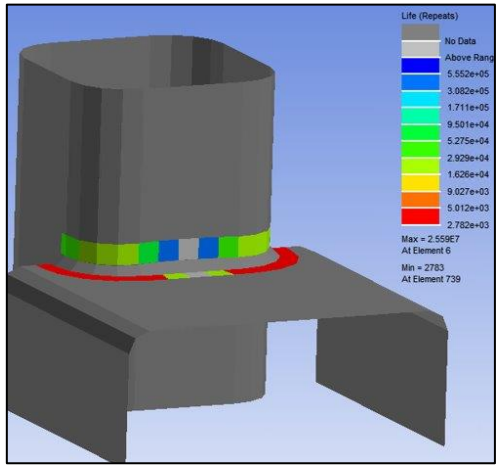
Vibration Fatigue, Direct Harmonic Case with PSD Loading for Shell Seam Weld Analysis Type

Life Result

Mechanical UI



Standalone



Overall – Result comparison

Mechanical UI

Element	Damage	RMS bending ratio	RMS stress [Pa]	Mean stress [Pa]	Irregularity factor	Largest Stress Cycle Amplitude [Pa]	Life [Repeats]
739	3.684E-04	0.02089	5.08E+07	0	0.8541	1.83E+08	2714
745	3.641E-04	0.02083	5.06E+07	0	0.854	1.82E+08	2747
728	3.629E-04	0.04665	4.80E+07	0	0.9623	1.73E+08	2756
729	3.588E-04	0.04701	4.79E+07	0	0.9624	1.73E+08	2787
748	3.208E-04	0.003562	4.66E+07	0	0.9604	1.68E+08	3117
736	3.182E-04	0.003561	4.65E+07	0	0.9604	1.67E+08	3143
390	2.751E-04	0.2171	4.61E+07	0	0.8128	1.68E+08	3634
737	2.665E-04	0.0477	4.48E+07	0	0.9444	1.61E+08	3752
747	2.663E-04	0.04748	4.48E+07	0	0.9446	1.61E+08	3755
300	2.606E-04	0.1987	4.52E+07	0	0.8145	1.64E+08	3838

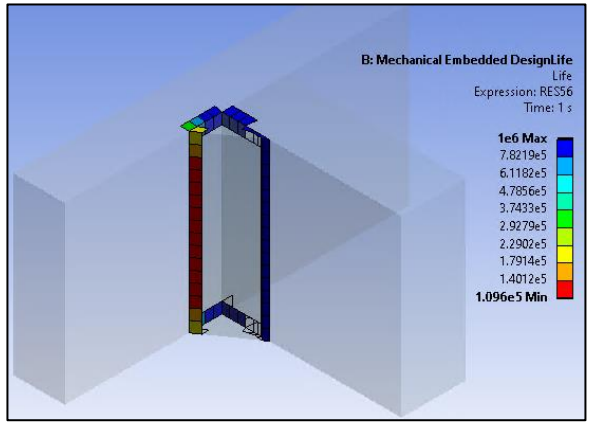
Standalone

ElementId	Damage	RMSBendingRatio	RMSStress	MeanStress	IrrFactor	LargestStressCycle	Life
739	3.59E-04	1.26E-04	5.04E+07	0	0.8542	1.82E+08	2783
728	3.58E-04	1.16E-04	4.78E+07	0	0.9643	1.73E+08	2794
745	3.55E-04	1.25E-04	5.02E+07	0	0.8541	1.81E+08	2816
729	3.54E-04	1.17E-04	4.76E+07	0	0.9644	1.72E+08	2827
748	3.20E-04	2.44E-05	4.65E+07	0	0.9612	1.68E+08	3129
736	3.17E-04	2.43E-05	4.64E+07	0	0.9612	1.67E+08	3155
390	2.72E-04	5.96E-04	4.59E+07	0	0.8132	1.67E+08	3682
737	2.67E-04	6.95E-05	4.49E+07	0	0.9449	1.61E+08	3745
747	2.67E-04	6.84E-05	4.49E+07	0	0.9451	1.61E+08	3748
300	2.57E-04	5.48E-04	4.49E+07	0	0.8149	1.63E+08	3889

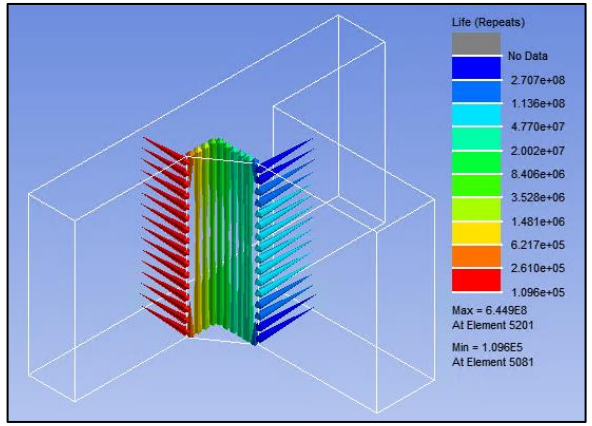
Vibration Fatigue, Direct Harmonic Case with PSD Loading for Solid Seam Weld Analysis Type

Life Result

Mechanical UI



Standalone



Overall – Result comparison

Mechanical UI

Element	Damage	RMS stress [Pa]	Irregularity factor	Largest Stress Cycle Amplitude [Pa]	Life [Repeats]
5081	9.1270E-06	1.37E+07	0.8841	5.03E+07	1.096E+05
5080	9.1270E-06	1.37E+07	0.8841	5.03E+07	1.096E+05
5082	9.0410E-06	1.36E+07	0.8841	5.01E+07	1.106E+05
5079	9.0410E-06	1.36E+07	0.8841	5.01E+07	1.106E+05
5083	8.8620E-06	1.35E+07	0.8841	4.98E+07	1.128E+05
5078	8.8620E-06	1.35E+07	0.8841	4.98E+07	1.128E+05
5084	8.5830E-06	1.34E+07	0.8841	4.93E+07	1.165E+05
5077	8.5830E-06	1.34E+07	0.8841	4.93E+07	1.165E+05
5085	8.1920E-06	1.32E+07	0.8841	4.85E+07	1.221E+05
5076	8.1920E-06	1.32E+07	0.8841	4.85E+07	1.221E+05

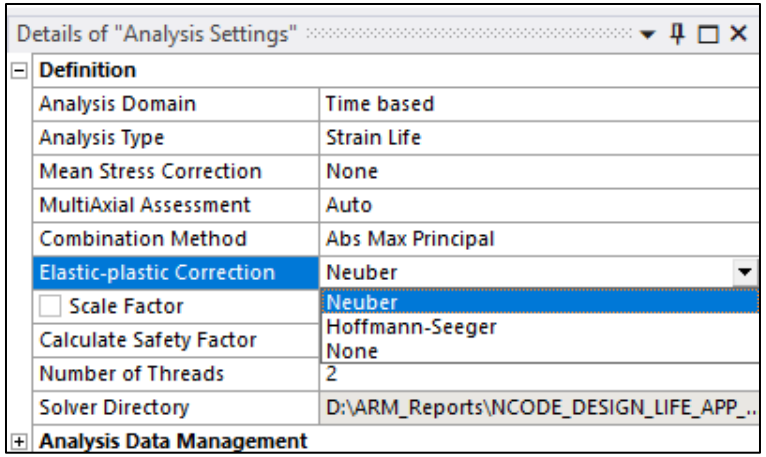
Standalone

ElementId	Damage	RMSStress	IrrFactor	LargestStressCycle	Life
5081	9.127E-06	1.37E+07	0.8841	5.03E+07	1.096E+05
5080	9.127E-06	1.37E+07	0.8841	5.03E+07	1.096E+05
5082	9.041E-06	1.36E+07	0.8841	5.01E+07	1.106E+05
5079	9.041E-06	1.36E+07	0.8841	5.01E+07	1.106E+05
5083	8.862E-06	1.35E+07	0.8841	4.98E+07	1.128E+05
5078	8.862E-06	1.35E+07	0.8841	4.98E+07	1.128E+05
5084	8.583E-06	1.34E+07	0.8841	4.93E+07	1.165E+05
5077	8.583E-06	1.34E+07	0.8841	4.93E+07	1.165E+05
5085	8.192E-06	1.32E+07	0.8841	4.85E+07	1.221E+05

Ansys nCode DesignLife Mechanical UI: EPC Options

Elastic-Plastic Correction:

- For Strain Life analysis, the Elastic-plastic Correction option is exposed
  - Choose between Neuber, Hoffmann-Seeger or None option
  - The None option should be used when plastic stress or strain are present
  - None option is only available for Mechanical Premium or Mechanical Enterprise license
- Analysis of rst files containing plastic results:
  - For Stress Life analysis, if the rst files used for loading contain plastic results, the user should check the results carefully. The extension notifies the user by issuing the warning message:
    - "FE contain plastic stress/strain results, which are invalid for SN fatigue. Please check results carefully."
  - For Strain Life analysis, if the rst files used for loading contain plastic results, the user should use None option or check results carefully. The extension notifies the user by issuing the warning message:
    - "FE contain plastic stress/strain results. Set the Elastic-Plastic correction to None for accurate results. This option is not available with PRO license. Please check results carefully."

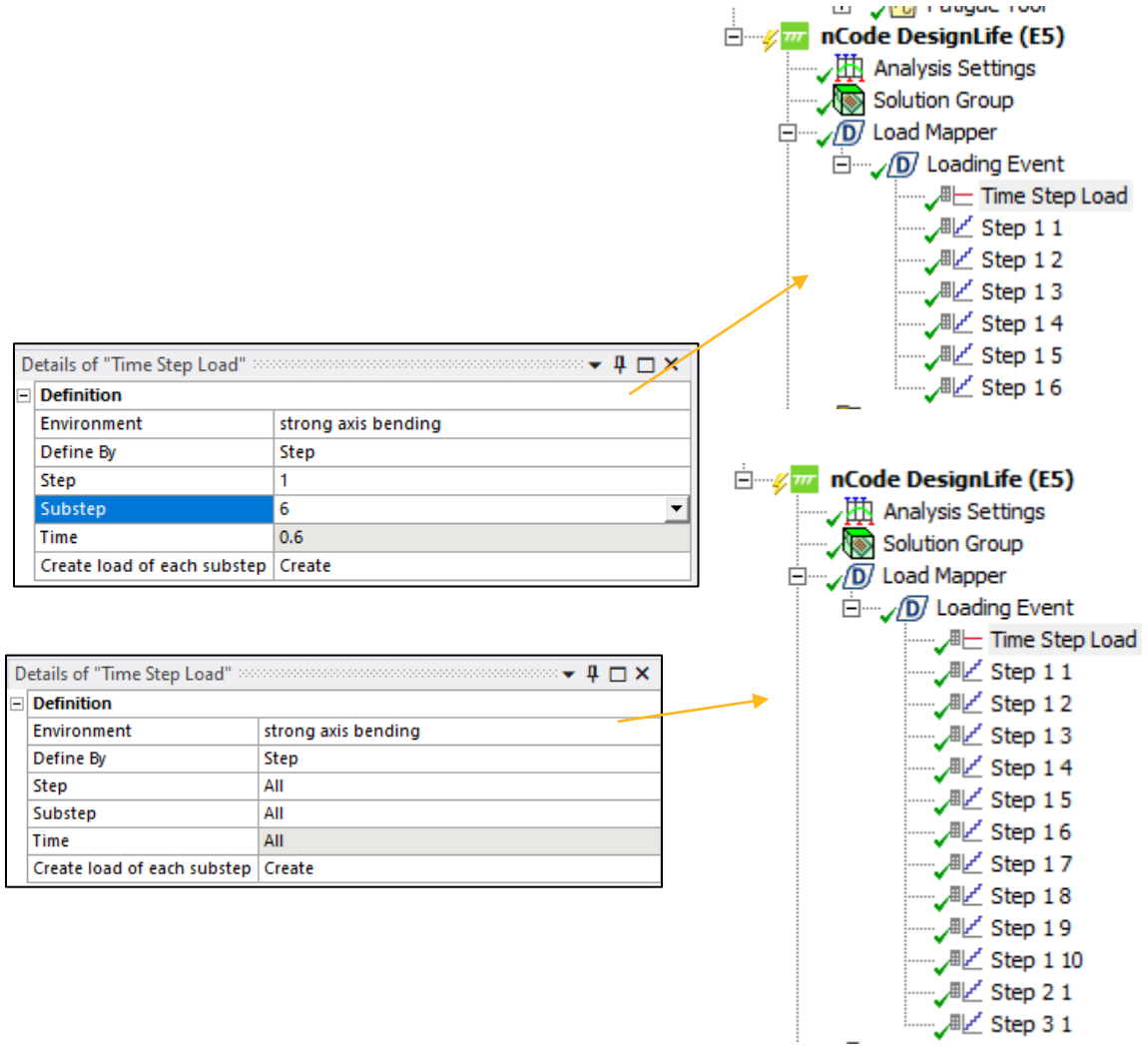


Ansys nCode DesignLife Mechanical UI: Load for all sub step

Time Step Load: Define the loading cases by Time or Step

1. If the load is defined by “Time”, select the desired time as loading conditions or select “All” to use full results
2. If the load is defined by “Step” the user will have the options:
   - ‘Step’, ‘Substep’ and ‘Create load for each substep’.
   - Set the Step and Substep desired for each loading condition

- To setup multiple loading conditions, use the “Create load of each substep” button. It creates a loading case per step defined and per substep, until the set substep
- If you want to create a loading case for every step and its substep, then select “All”, “All” and hit the button
- Once you have all the loading per step and substep you can unsuppress or delete the undesired ones



Ansys nCode DesignLife Mechanical UI: Effect of Stress Averaging

Results between DesignLife UI and Mechanical UI can be different due to stress averaging

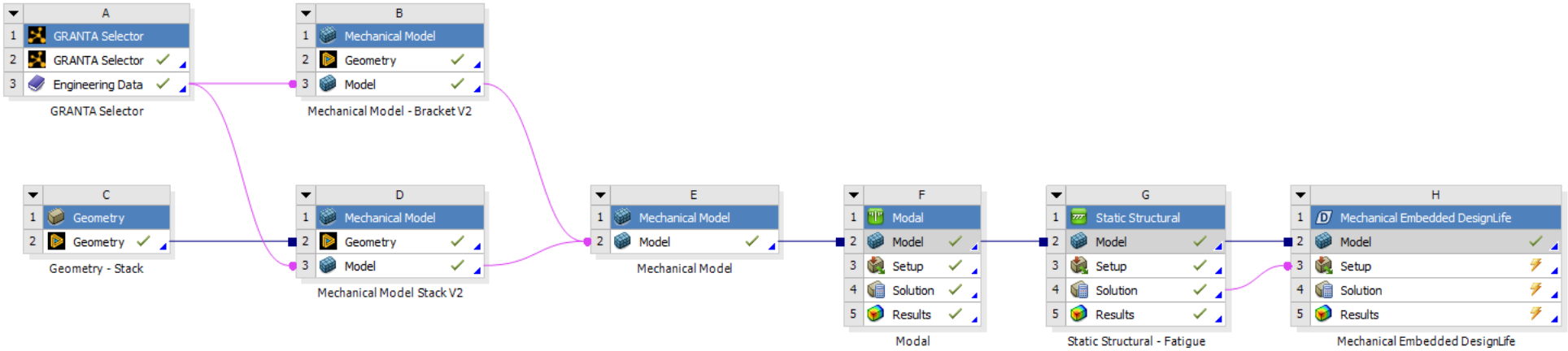
- DesignLife UI calculates the surface nodal stresses by averaging the stresses from all elements that share that node
- Mechanical UI only includes the stresses from elements that have a face on the surface

NOTES:

- For refined, well-shaped meshes, the effect of the nodal stress averaging method should be minimal
- For coarse tet meshes, the averaged nodal stresses can be affected by the averaging scheme. Thus, the calculated fatigue results may be different

Ansys nCode DesignLife Mechanical UI: Usability

- Non-compatible systems can be linked with nCode extension if they are not selected as loading events
- Improvements focusing on Performance, Robustness, Accuracy & Documentation



Ansys