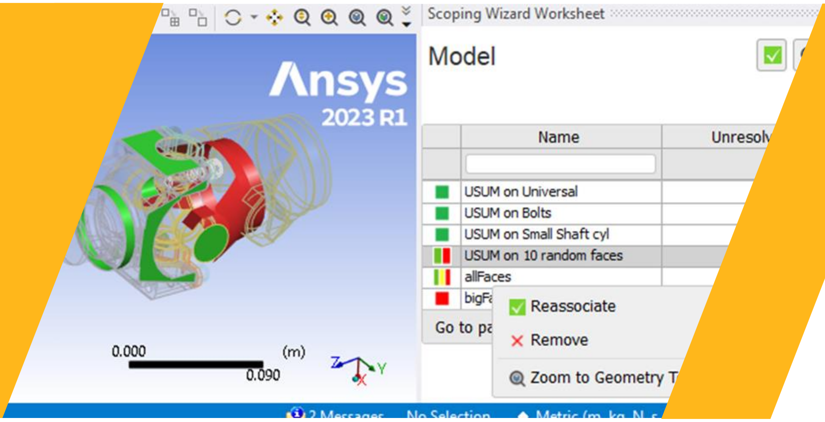
Release 2023 R1 Highlights
Ansys Mechanical

Ansys

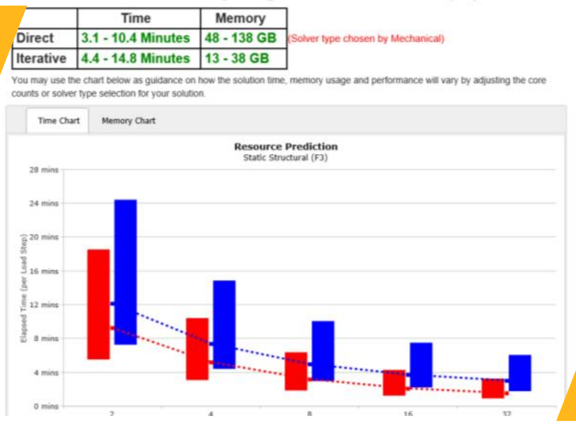
Table of Contents

Enabling More Efficient and Accurate FEA Simulations



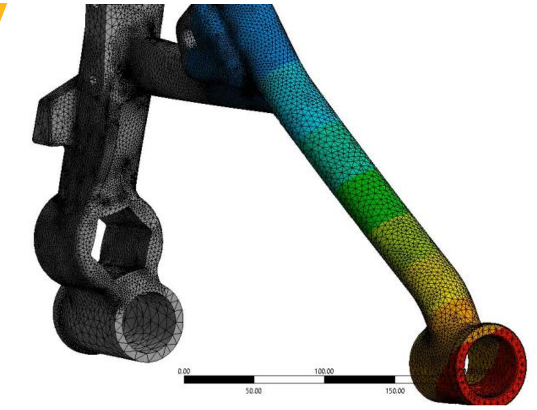
Easing the pain of geometry updates with Geometry Based Associativity

- ✓ Modify a CAD model, without losing the associativity of the model's features after setup
- ✓ Using geometry-based re-associativity with the Scoping Wizard detects and reestablishes scoping



Leveraging AI/ML to Predict Resources Required for Mechanical Simulations

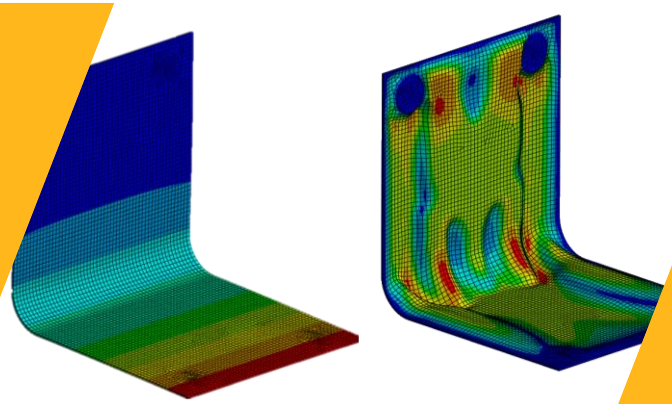
- ✓ Gain insight into the computational resources required to solve an Ansys Mechanical simulation
- ✓ Leverages AI/ML to analyze millions of APIP data points from previously solved models
- ✓ Provides estimated values for total computational solve time and memory usage based on model size and cores used

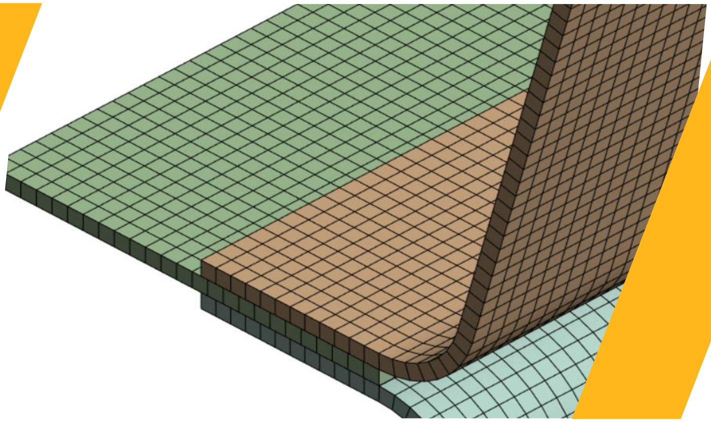


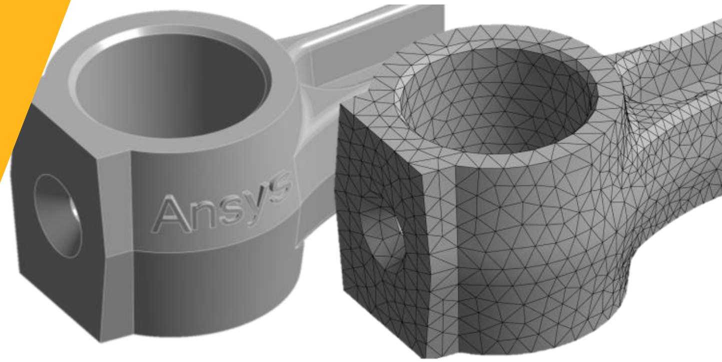
Improving Accuracy and Efficiency through Geometry-based Enhancements

- ✓ Improved simulation accuracy on complex models and parts utilizing geometry preserving adaptivity (GPAD) that automatically refines a mesh based on the initial geometry
- ✓ Eliminates the need for an over-refined mesh or advanced user knowledge

Enabling More Efficient and Accurate FEA Simulations







Optimize Frame and Shell Structures through Topography Optimization

- ✓ Optimize frame and shell structures using a new topography optimization method
- ✓ Improves the structural durability of the component while minimizing mass
- ✓ Eliminates unwanted scenarios like increased noise and vibration

Efficiently Setup Contacts for Complex Models

- ✓ An improved and simplified setup method now enables contact creation for complex models in less time
- ✓ Removes the need for duplicate contact setup and book-keeping of top/bottom faces of sheet bodies
- ✓ Includes models like contact between adhesives and metal sheet components common in Body in White (BIW) durability models.

Generate a High-quality Mesh

- ✓ Improvements will help to generate a higher-quality mesh that meets your criteria the first time
- ✓ Tetrahedral meshes for drop test simulations and feature suppression
- ✓ Swept hexahedral meshes and meshes for welds and shells
- ✓ Overall general usability improvements

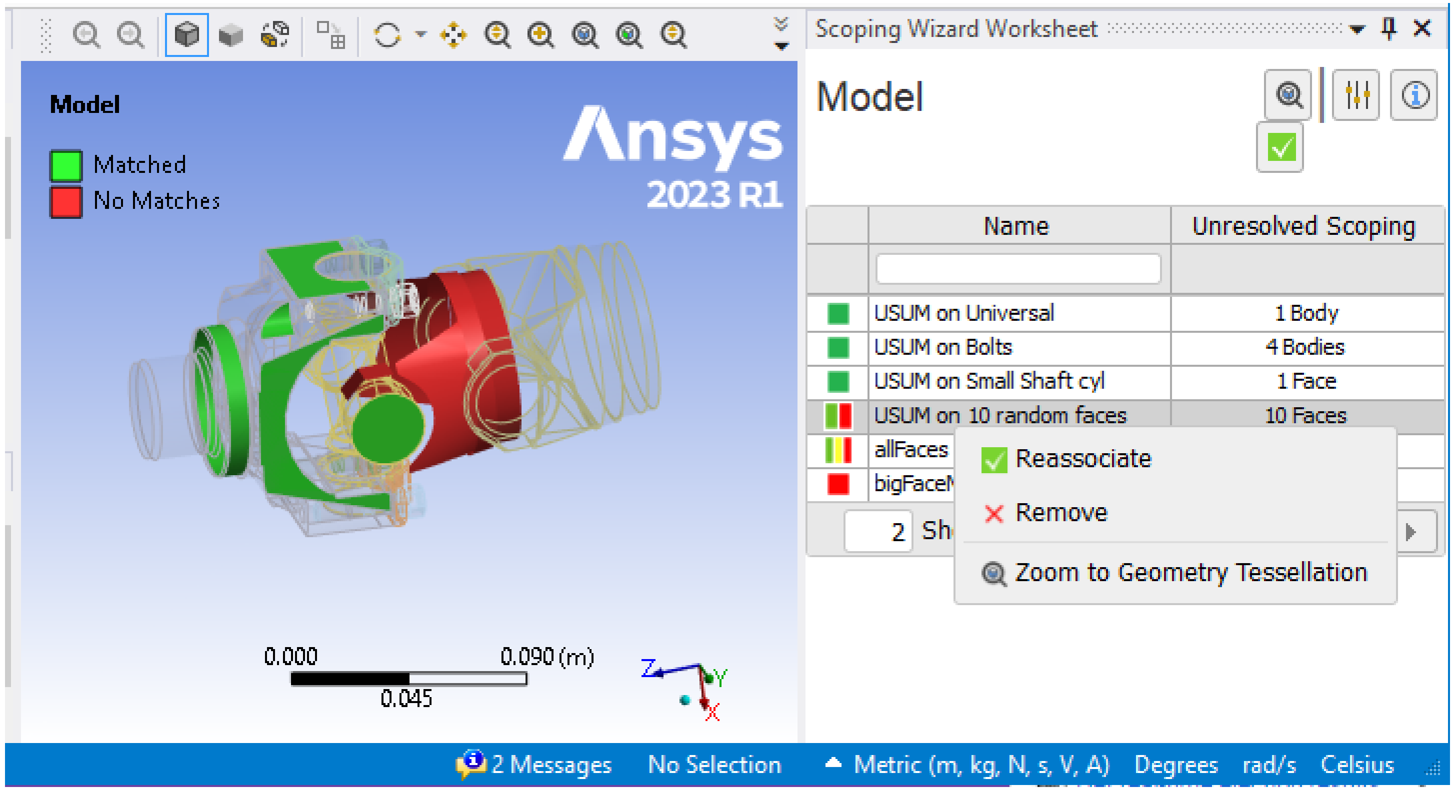
Mechanical

Automated Rescoping Using Geometry Based Associativity (GBA)

Ansys

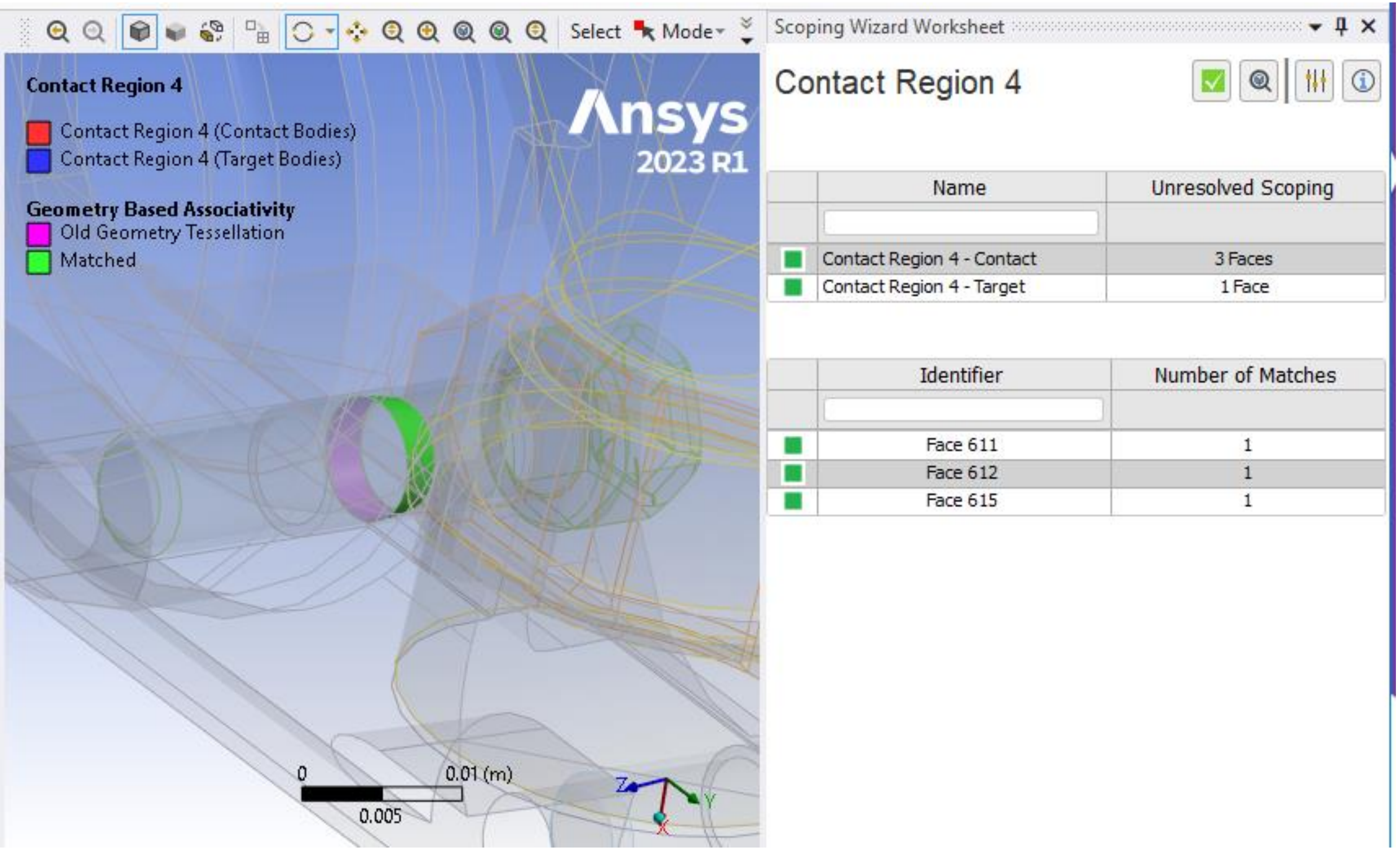
GBA: Scoping Wizard

- The Scoping Wizard is a tool that provides a list of the objects with missing scoping due to geometry updates. It provides the count of missing items, and the visual view of the previous scoping.
- Scoping Wizard for Model object in Tree:
  - Shows a list of all objects with unresolved scoping.
  - Context menu options in the wizard allow the user reassociate the matched scoping, remove the object from the Wizard view, or zoom in the graphics for a closer look.
  - State icons show in the first column
    - Green shows items that have an exact one to one match for each reference.
    - Red shows items that has no matches
    - Yellow shows items that have multiple-matches.



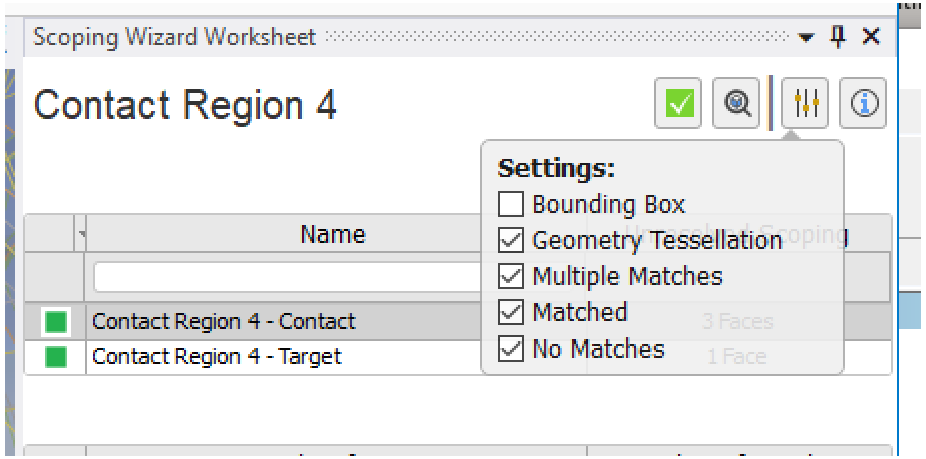
GBA: Scoping Wizard

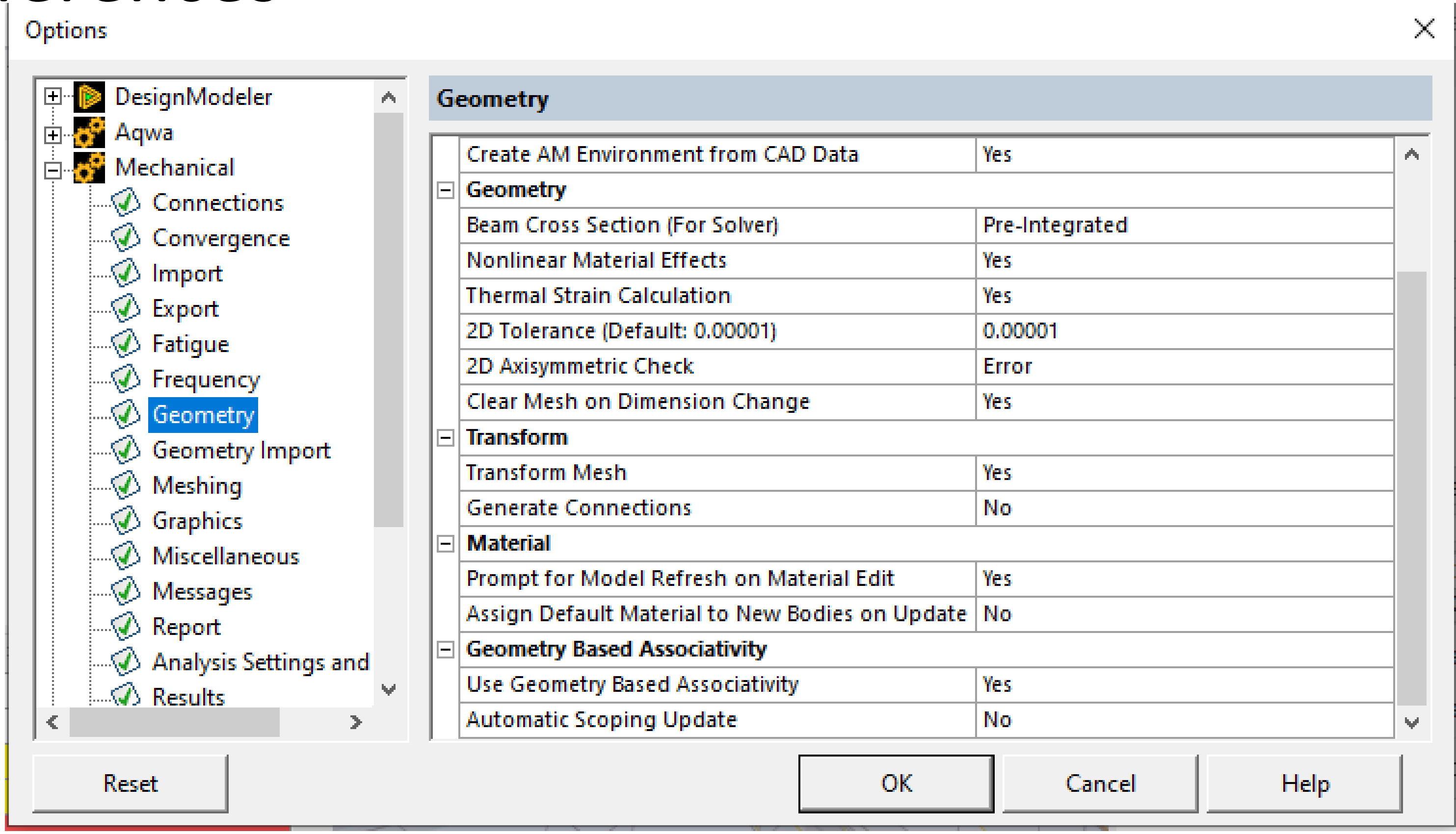
- Scoping Wizard for non-Model object
  - A second list will show, enabling the to user to visualize each reference that is missing.
  - When the user selects the "Reassociate" option, only those items with a green icon (one-to-one matches) will be relinked.
  - The old geometry tessellation will show with any potential matches.
  - Context menu options in the 2nd list allow the user to reassociate, remove or zoom to a specific geometric entity.



GBA: Scoping Wizard - Preferences

- Preferences exist in the Options panel to turn Geometry Based Associativity off/on and to make it automatic or user controlled.
- By default, GBA will be on, with automatic scoping turned off.
- Options also exist in the Scoping Wizard to turn graphical options on and off.

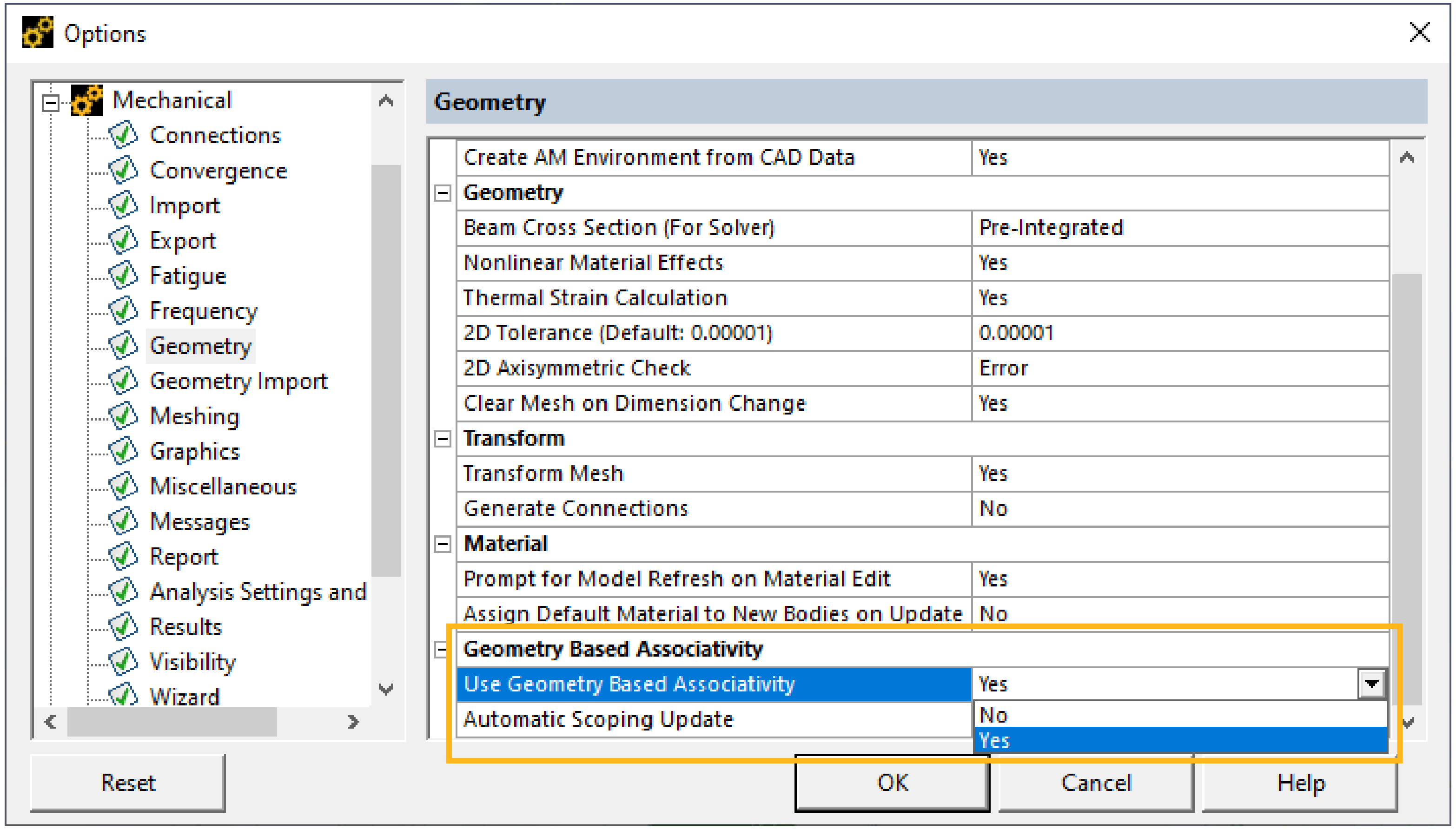




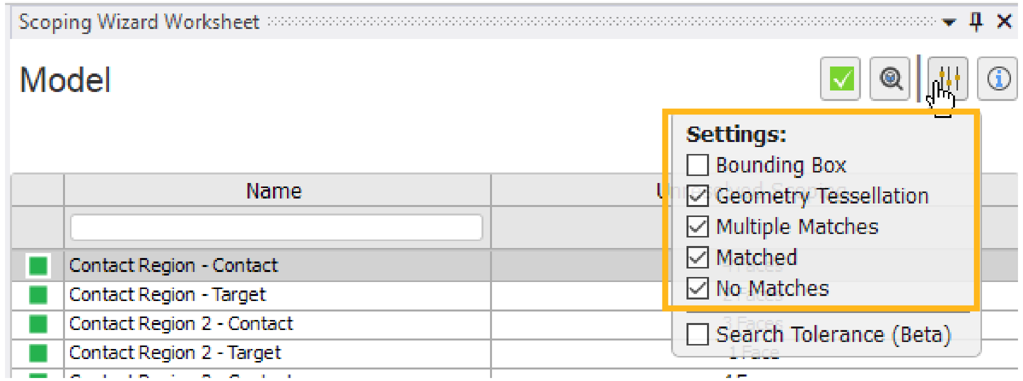
Ansys

GBA Graphics

- Chose “Yes” to Geometry preference option Use Geometry Based Associativity when updating the geometry in Mechanical.



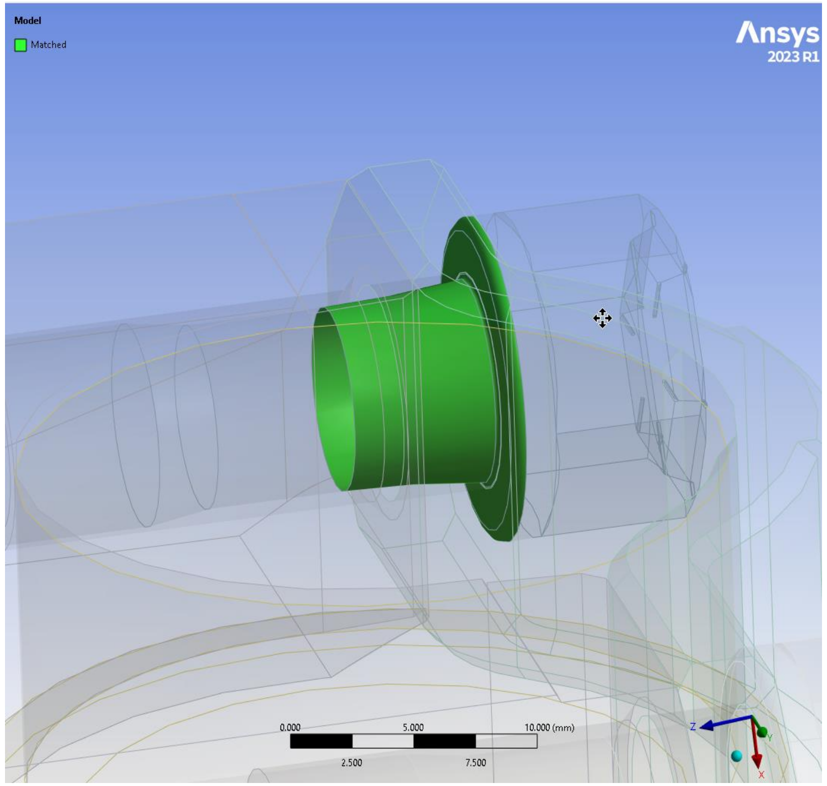
GBA Graphics

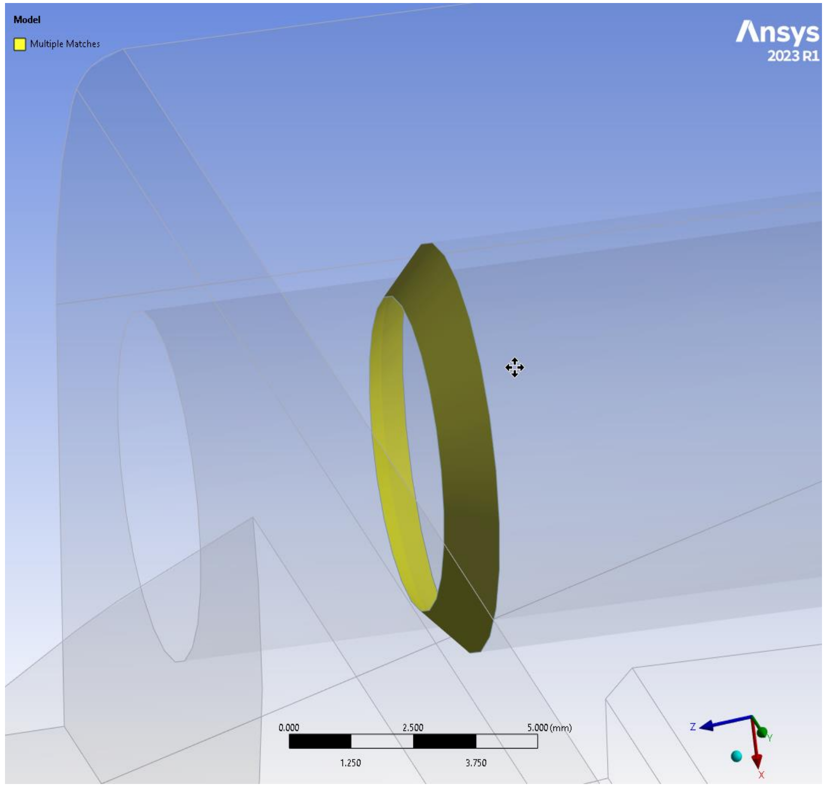


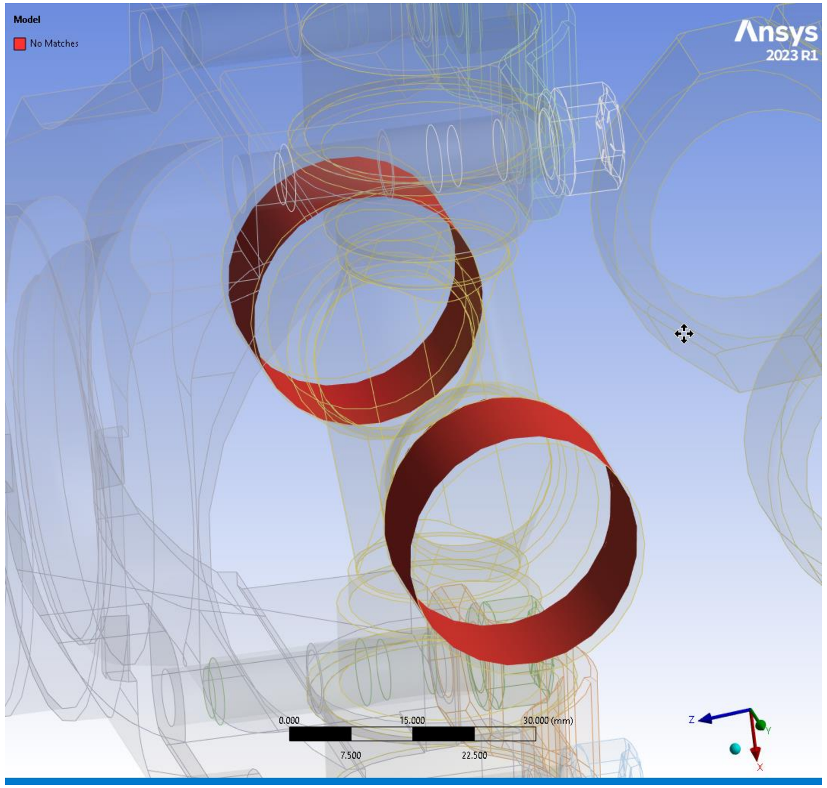
- Bounding Box - Displays bounding box of the scoped reference before geometry update. It displays for selected item(s) in the worksheet.
- Geometry Tessellation - Displays full tessellation of the scoped reference before geometry update. It displays for selected item(s) in the worksheet.
- Multiple Matches - Displays only references that are of type Multiple Matches.
- Matched - Displays only references that are of type Full Match.
- No Matches - Displays only references that are of type No Matches.

GBA Graphics

- At model level old geometry tessellations are displayed in green, yellow or red based on the matched type Full Match, Multiple Matches or No Matches, respectively.





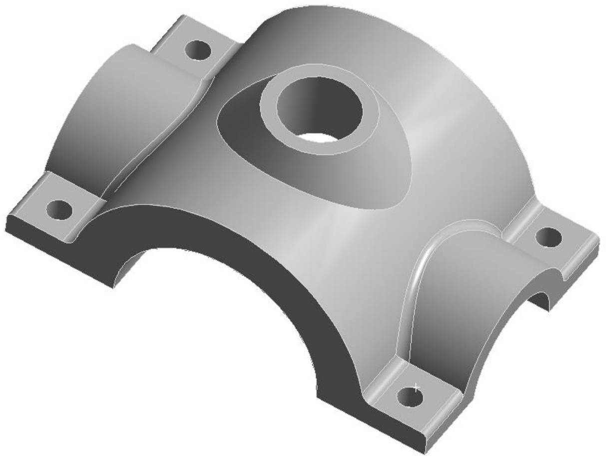


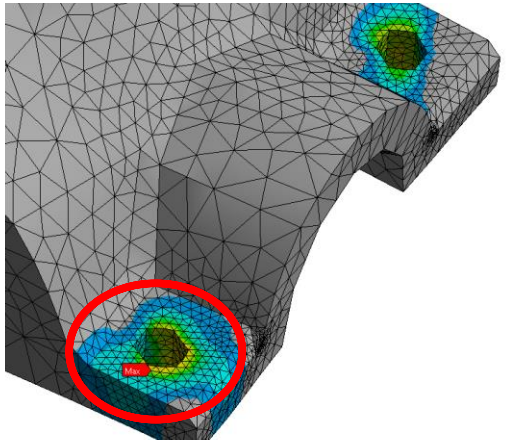
Geometry Preserving Adaptivity(GPAD)

Ansys

Geometry Integrity in Simulations

- Complicated geometries are difficult to mesh and may be impossible for user to determine proper mesh sizes a priori due to complicated geometries.
- Resulting meshes may not capture proper stress and deformation fields during analysis.
- GPAD capability leverages the Ansys meshing capability and the robust nonlinear adaptivity feature in Ansys Mechanical.
- Automatically refines the mesh to the initial geometry (instead of initial mesh boundaries) adaptively to improve solution accuracy.

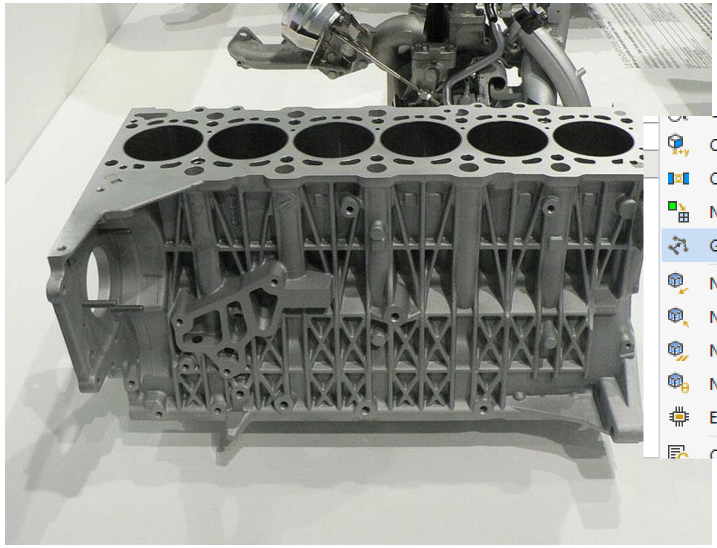


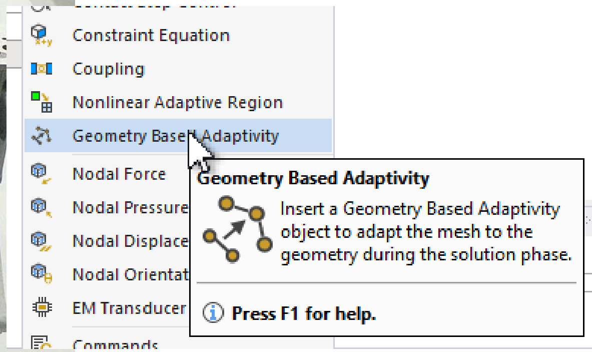


Nonlinear Adaptivity: Circular hole approximated by initial coarse mesh. Remeshing occurs within the same initial mesh boundaries.

Geometry Preserving Adaptivity

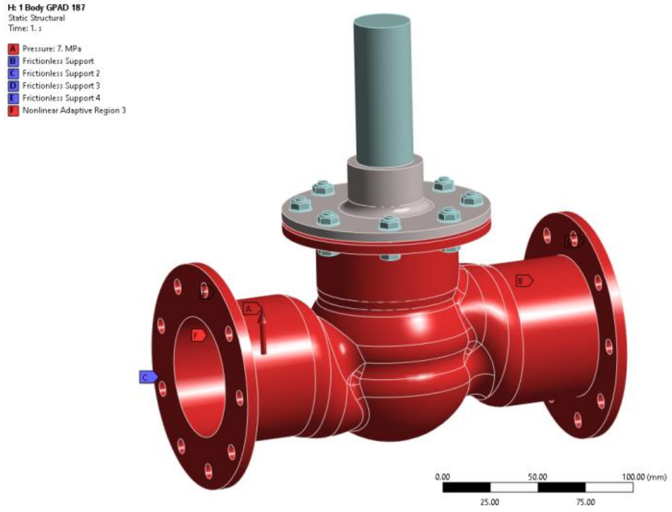
- Supports:
  - Linear analysis (NLGEOM, OFF)
  - Lower and higher-order tetrahedral elements
  - Elastic and elastoplastic materials
  - Contact capabilities
  - Incorporates defeaturing effects during remeshing
  - In-built sizing controls for remeshing:
    - Element size control
    - Equivalent stress or equivalent strain criteria
  - Mesh exploration feature for complicated models
  - Improved solver messages for a better understanding of the remeshing process
- Typical Use Case:
  - A complex model / part such as an engine block or manifold with several geometric features, where a user does not know where to refine the initial mesh

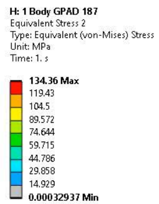


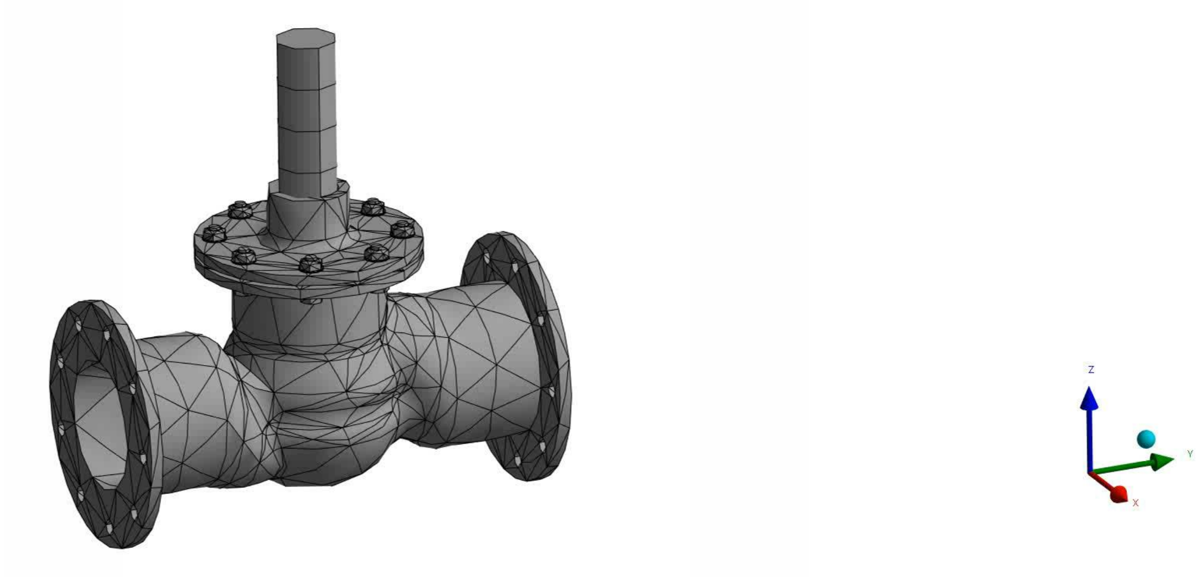


https://commons.wikimedia.org/wiki/File:CarterBMW1.JPG

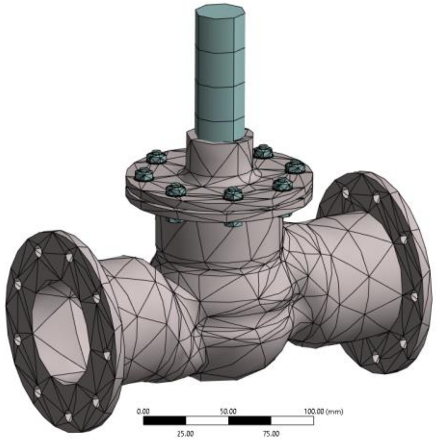
GPAD – Mesh Exploration or Aggressive Remeshing





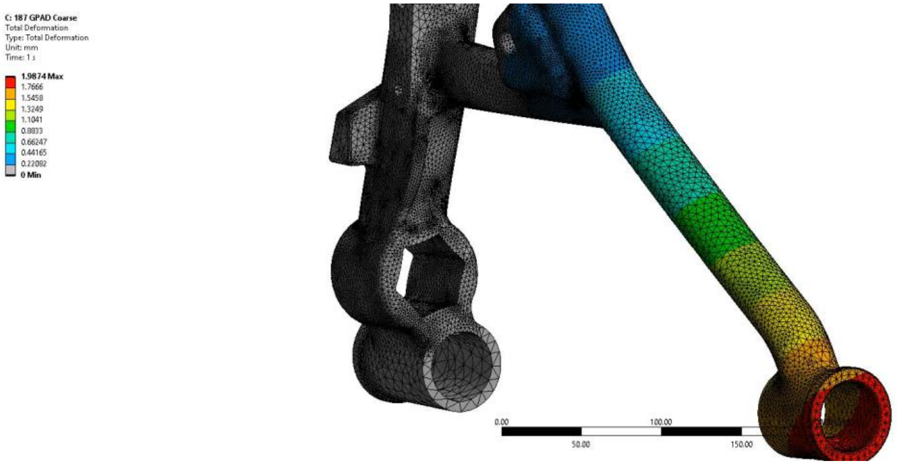


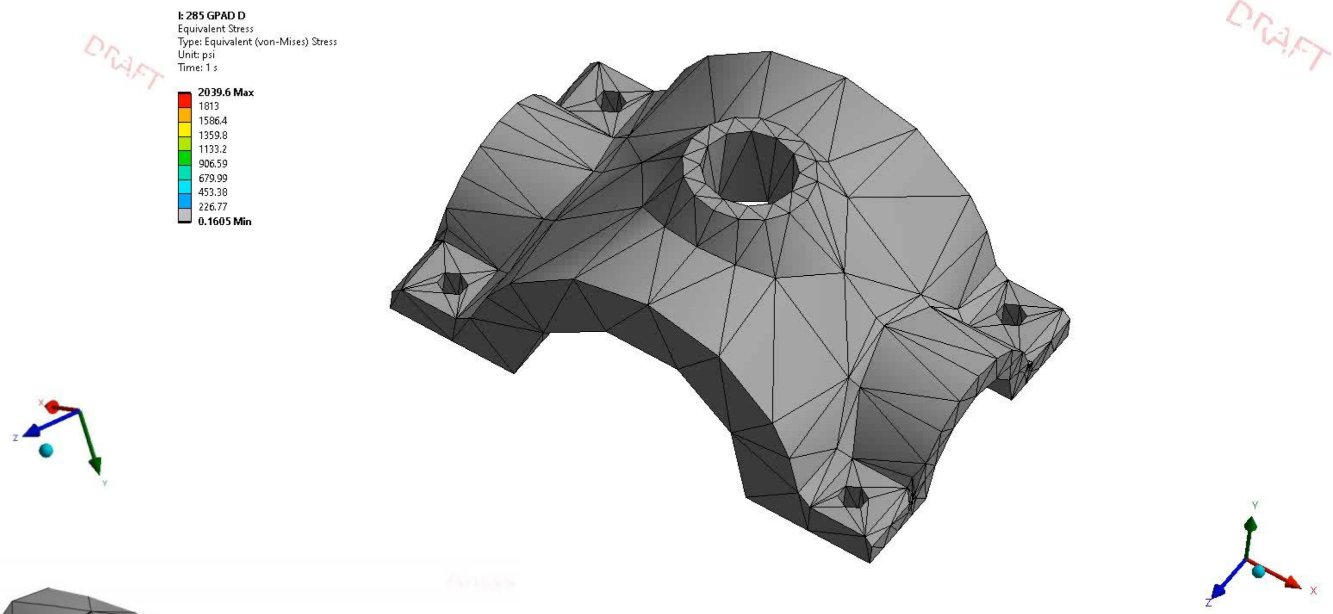
Initial Geometry & Mesh



Final Mesh with Aggressive Remeshing

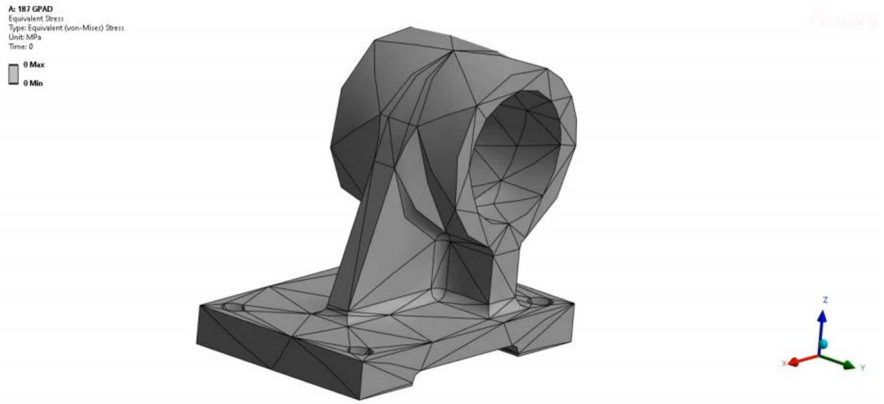
GPAD – Accurate Geometry Recovery Through Mesh adaptivity





91F26DEF.mechdat

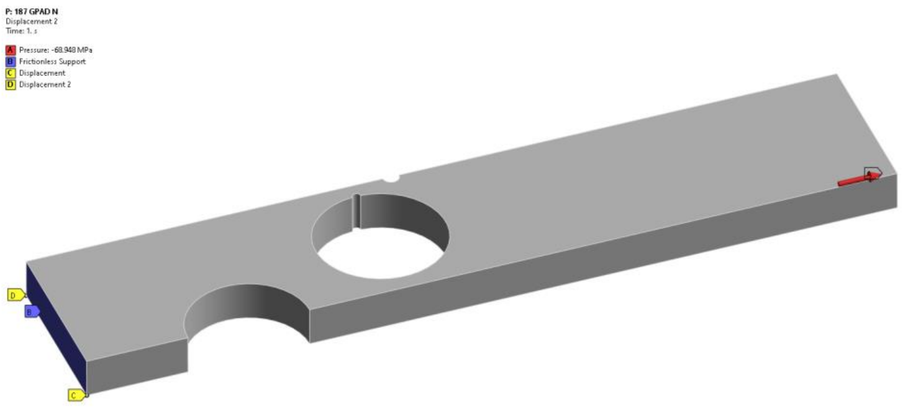
5E8E3601.mechdat

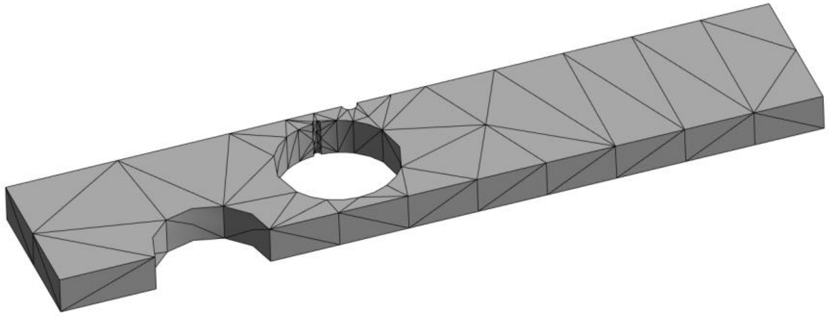


9084532D.mechdat

Ansys

GPAD - Element Size based remeshing control

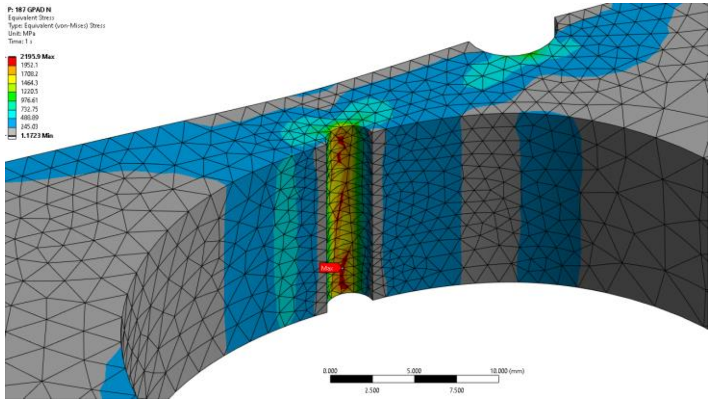




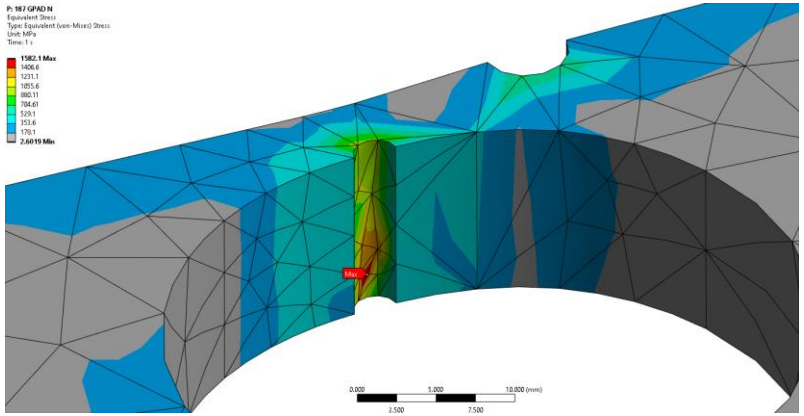
COMMAND:
NLMESH,ELSZ,10

5FE0B0A8.mechdat

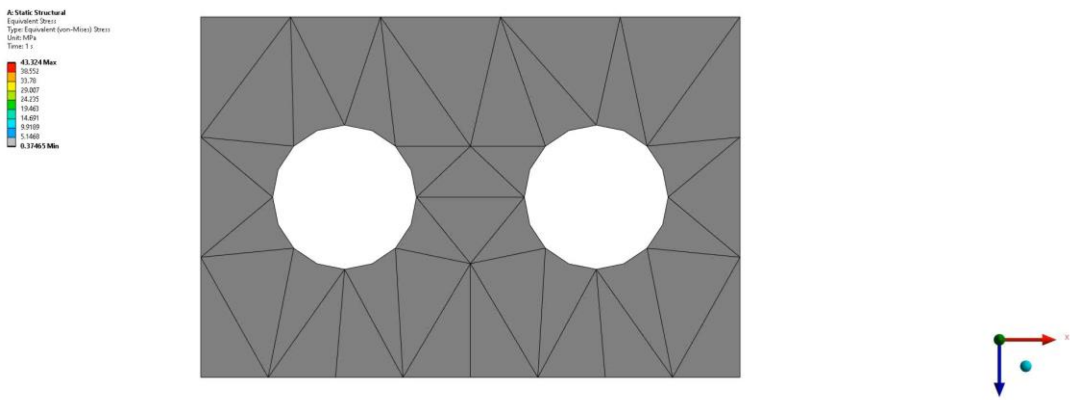
WITHOUT ELEMENT SIZE LIMIT (4 remeshings)

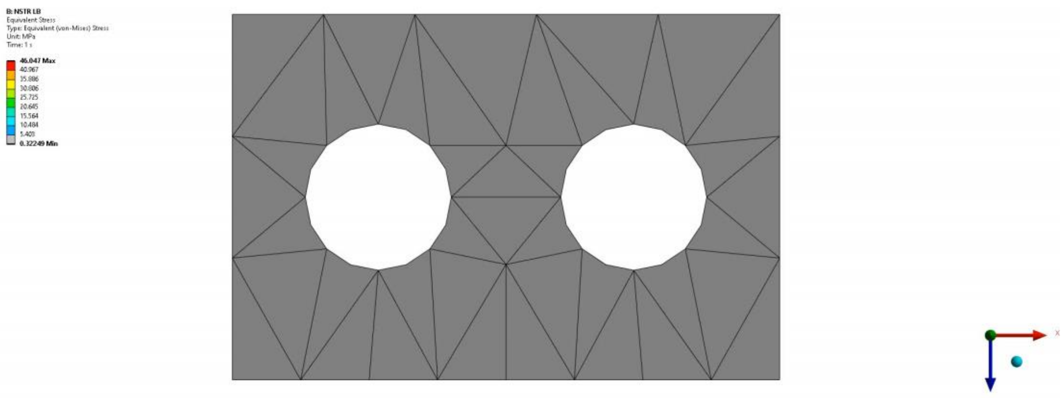


WITH ELEMENT SIZE LIMIT (Only 2 remeshings)

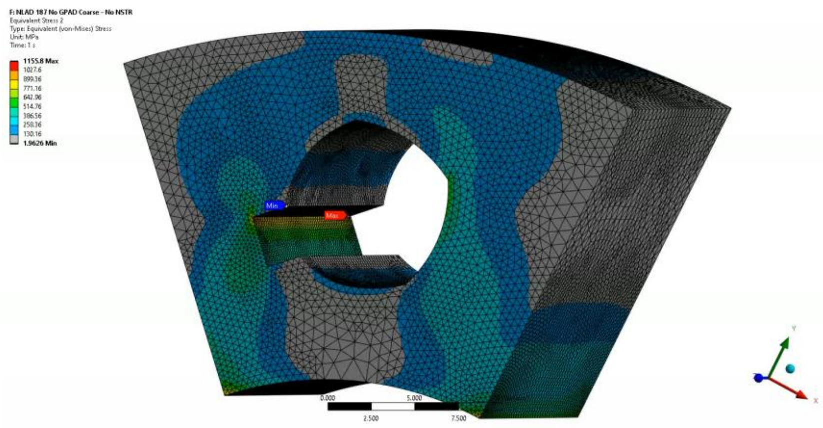


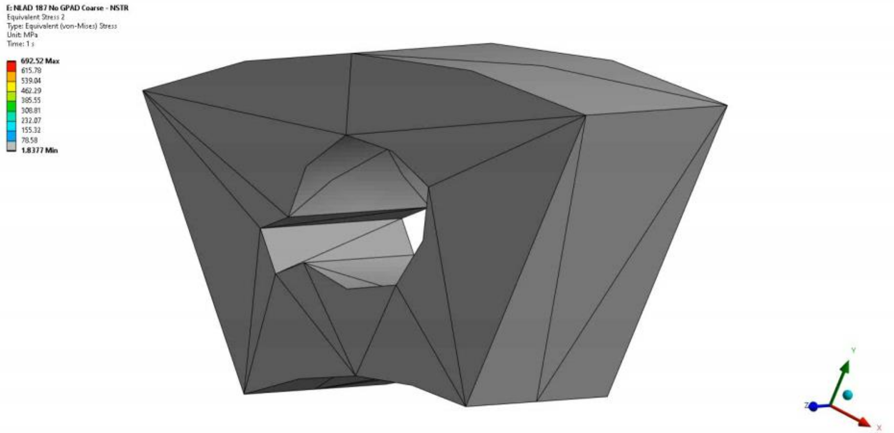
GPAD – Equivalent Stress and Strain based remeshing control





Remeshing Limited by Equivalent Stress or Strain levels (NLMESH,NSTR/NSTN)





GPAD and NLAD - Improved Solver Messages for Mesh Rejection

```
**** REGENERATE MESH AT SUBSTEP     8 OF LOAD STEP      1 BECAUSE OF
     NONLINEAR ADAPTIVE CRITERIA

     PREPARING DATA TO REMESH......

     REMESHING REGIONS ARE CREATED; GENERATING NEW MESH......

     REFINEMENT REMESHING PROCEDURE......


     REMESHING EXECUTABLE FAILURE:  PREPARING DATA TO SOLVE WITH THE OLD MESH......

**** NEW MESH HAS NOT BEEN CREATED. CONTINUE TO SOLVE WITH OLD MESH.
```

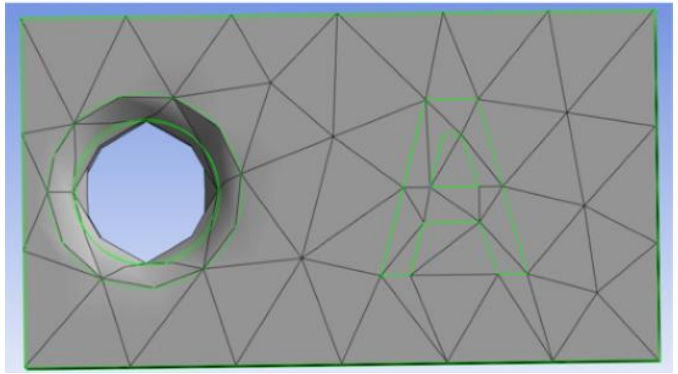
```
Element Average     : ------Source/342-------+------Target/1570-----
..Skewness(Vol)     :    0.6371                   0.5554
..JRatio(Node )     :    0.8597                   0.9289
..JRatio(Gauss)     :    0.9304                   0.9659
..Aspect Ratio      :    3.9488                   3.1531
Domain Volume       :   65.4272                  65.4280

Worst Element       :  --------Source-------+--------Target--------
..Skewness(Vol)     :    0.9932  (e1109    )      0.9977  (e57609   )
..JRatio(Node )     :    0.1152  (e352     )      0.2960  (e56865   )
..JRatio(Gauss)     :    0.4303  (e354     )      0.5772  (e56636   )
..Aspect Ratio      :   10.9505  (e1109    )     49.9090  (e56257   )
==================================================================
== Remeshing result statistics
==================================================================
Domain(s)           :   1
Region(s)           :   1
Patch(es)           :   3
nNod[New]           :   2716
nElm[New/Ef/Sd/Sr]  :   1570 / 342 / 66 / 1152

Peak memory         :   242 MB AmsMesher

- AmsMesher run completed in 2.242 seconds

 ======================== End Run ================================
 =================================================================


   NEW MESH QUALITY IS INADEQUATE:  PREPARING DATA TO SOLVE WITH THE OLD MESH......

MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS HAS BEEN MODIFIED
 TO BE, NEQIT = 25, BY SOLUTION CONTROL LOGIC.

**** NEW MESH HAS NOT BEEN CREATED. CONTINUE TO SOLVE WITH OLD MESH.
```

Inadequacy of mesh quality is determined by:

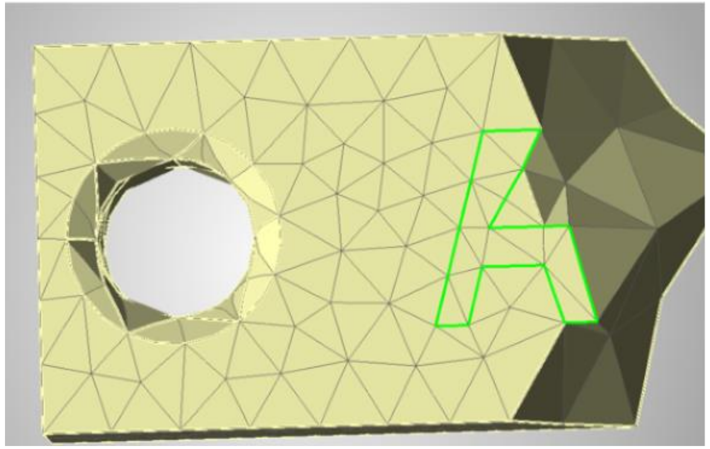
- Skewness
- Jacobian ratio
- Maximum element angle
- Or combination thereof

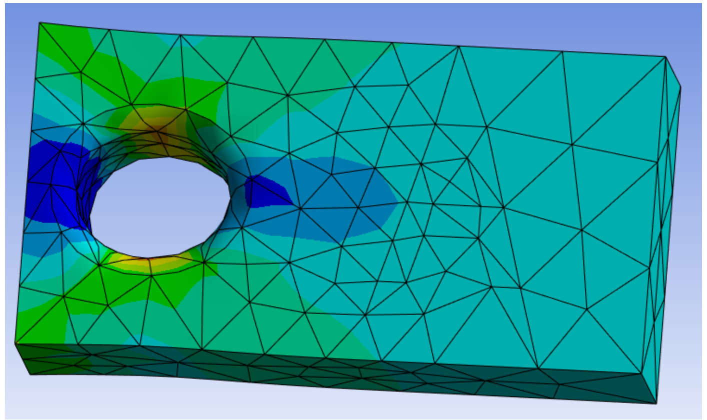
Ansys

GPAD Meshing – Incorporating Defeaturing during Remeshing



Part geometry and initial meshing
The embossment "𝔸" is defeatured during meshing



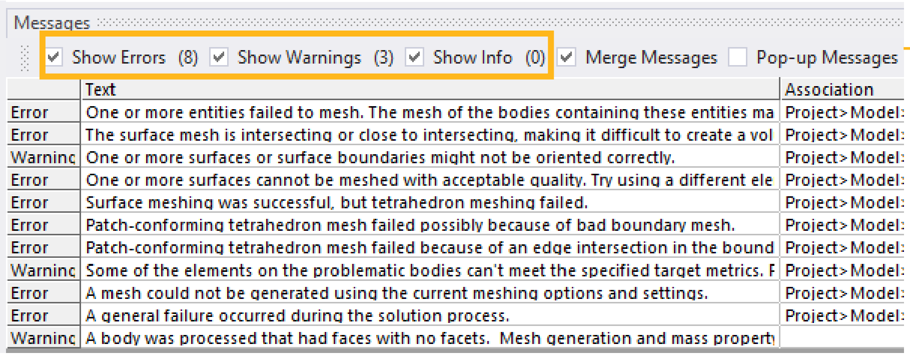


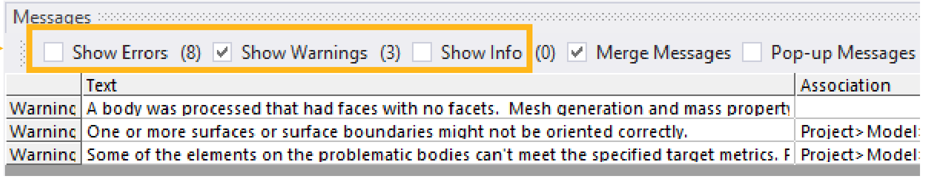
Embossment "𝔸" ignored

Messages Window Enhancements

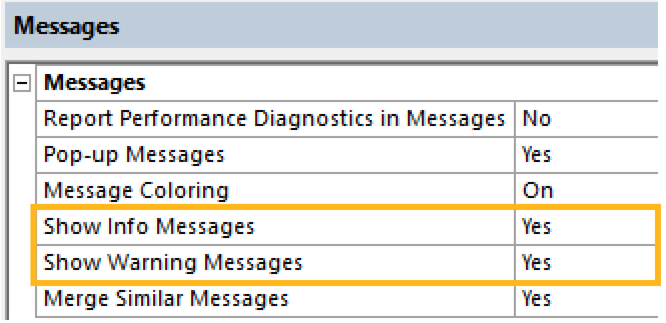
Message Filtering

- UI options to filter messages based on their severity.



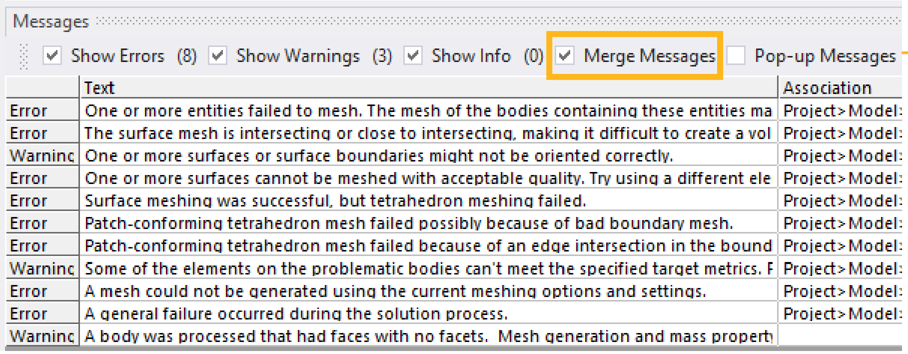


- This can also be controlled from **File->Options->Mechanical->Messages**



Message Filtering

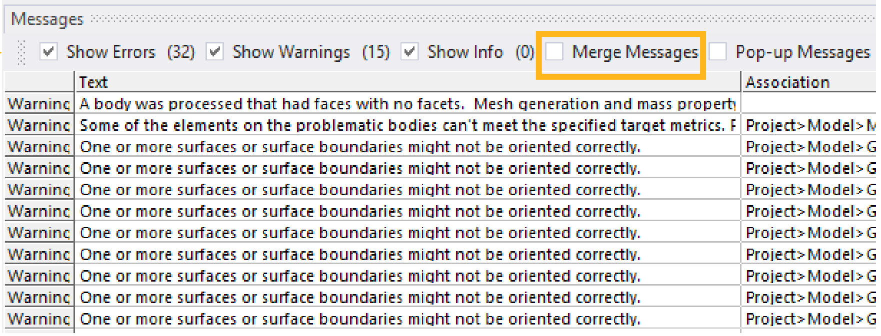
- ‘Merge Messages’ option will combine similar messages with the same severity into a single message. This is **On** by default.



Messages

☑ Show Errors (8) ☑ Show Warnings (3) ☑ Show Info (0) ☑ Merge Messages ☐ Pop-up Messages

	Text	Association
Error	One or more entities failed to mesh. The mesh of the bodies containing these entities ma	Project>Model:
Error	The surface mesh is intersecting or close to intersecting, making it difficult to create a vol	Project>Model:
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model:
Error	One or more surfaces cannot be meshed with acceptable quality. Try using a different ele	Project>Model:
Error	Surface meshing was successful, but tetrahedron meshing failed.	Project>Model:
Error	Patch-conforming tetrahedron mesh failed possibly because of bad boundary mesh.	Project>Model:
Error	Patch-conforming tetrahedron mesh failed because of an edge intersection in the bound	Project>Model:
Warninc	Some of the elements on the problematic bodies can't meet the specified target metrics. F	Project>Model:
Error	A mesh could not be generated using the current meshing options and settings.	Project>Model:
Error	A general failure occurred during the solution process.	Project>Model:
Warninc	A body was processed that had faces with no facets. Mesh generation and mass property	

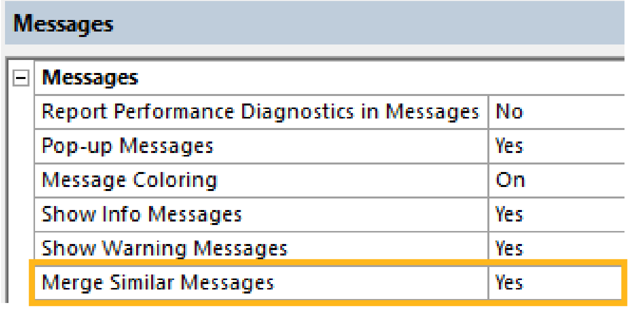


Messages

☑ Show Errors (32) ☑ Show Warnings (15) ☑ Show Info (0) ☐ Merge Messages ☐ Pop-up Messages

	Text	Association
Warninc	A body was processed that had faces with no facets. Mesh generation and mass property	
Warninc	Some of the elements on the problematic bodies can't meet the specified target metrics. F	Project>Model>M
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G
Warninc	One or more surfaces or surface boundaries might not be oriented correctly.	Project>Model>G

- This can also be controlled from **File->Options->Mechanical->Messages**

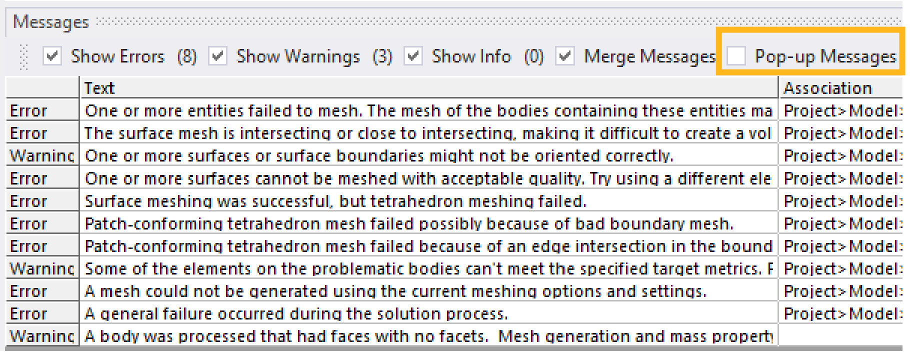


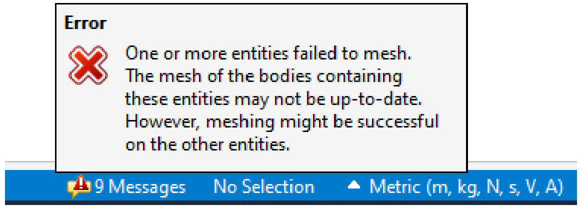
Messages

Messages	
Report Performance Diagnostics in Messages	No
Pop-up Messages	Yes
Message Coloring	On
Show Info Messages	Yes
Show Warning Messages	Yes
Merge Similar Messages	Yes

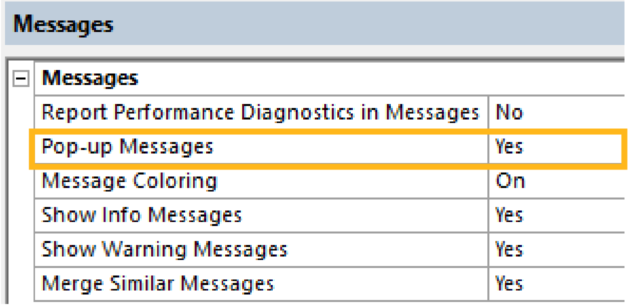
Message Pop-up

- UI option to control if messages should ‘pop-up’ when a message is issued. When **checked**, messages will pop-up when issued by Mechanical. When **unchecked**, these will be hidden.





- This can be also controlled from **File->Options->Mechanical->Messages**

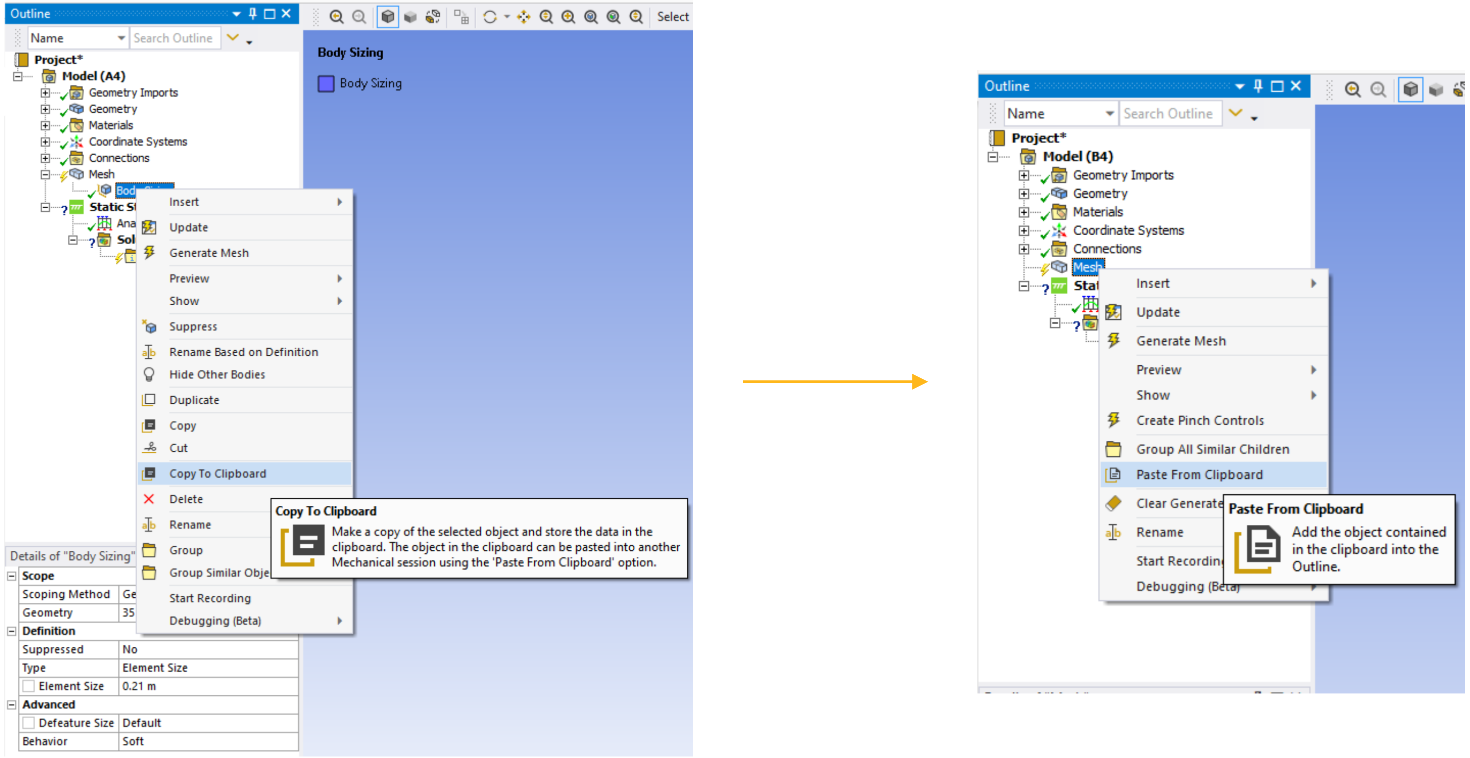


Copy and Paste of Objects Using the Clipboard

Ansys

Copy/Paste Between Sessions Using Clipboard

- In 2022R2, Copy-to-Clipboard and Paste-from-Clipboard were only supported on Joints, Contact Regions, Coordinate Systems, and Named Selections.
- For 2023R1, Copy-to-Clipboard and Paste-from-Clipboard has been made available for all copiable tree objects in Mechanical.



Command Snippet Output Search Prefix Supports Variables Inside Percentage (%) Symbols

Command Snippet Output Search Prefix – Variables Inside Percentage (%) Symbols

- Command Snippet's **Output Search Prefix** recognizes parameters with variables inside percentage (%) symbols.
- When creating the output parameters under **Results** category, the (%) symbols and the variable inside them are replaced by the value the variable has been set to using equal (=) symbol or *SET command, or if the variable is the iterator of a do loop, a range of parameters are created according to the limits of the do loop.

Details of "Commands (APDL)"

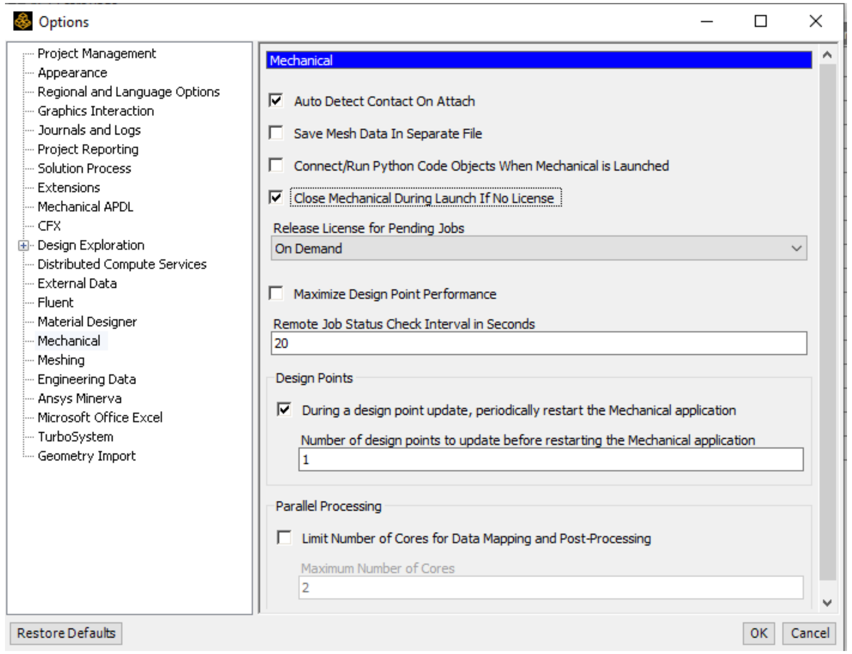
File	
File Name	
File Status	File not found
Definition	
Suppressed	No
Output Search Prefix	my_
Invalidate Solution	No
Target	Mechanical APDL
Input Arguments	
Results	
my_seqv_max_1	2.3498e+006
my_seqv_max_2	1.8128e+006
my_seqv_max_3	6.8018e+005
my_seqv_max_4	1.2609e+005

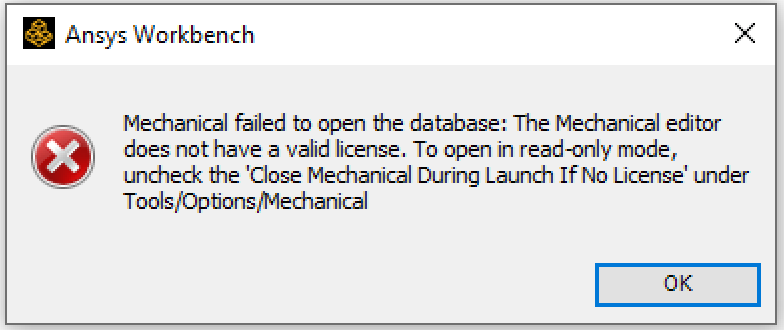
```
set,last

*do,i,1,4
    my_seqv_max_%i% = 0
    esel,s,type,,i
    nsle,s,all
    *get,curr,node,0,num,min
    *get,num_node,node,0,count
    *do,j,1,num_node
        *get,eq_s,node,curr,s,eqv
        *if,eq_s,gt,my_seqv_max_%i%,then
            my_seqv_max_%i% = eq_s
        *endif
        *get,curr,node,curr,nxth
    *enddo
    esel,all
    nsel,all
*enddo
```

Close Mechanical During Launch If No License

Close Mechanical During Launch if No License

- A new preference is available in WorkBench, Tools->Options->Mechanical, where user can stop from launching Mechanical when the checkout of licensing has failed
- Enabling this preference will display an error message whenever Mechanical is unable to checkout a License

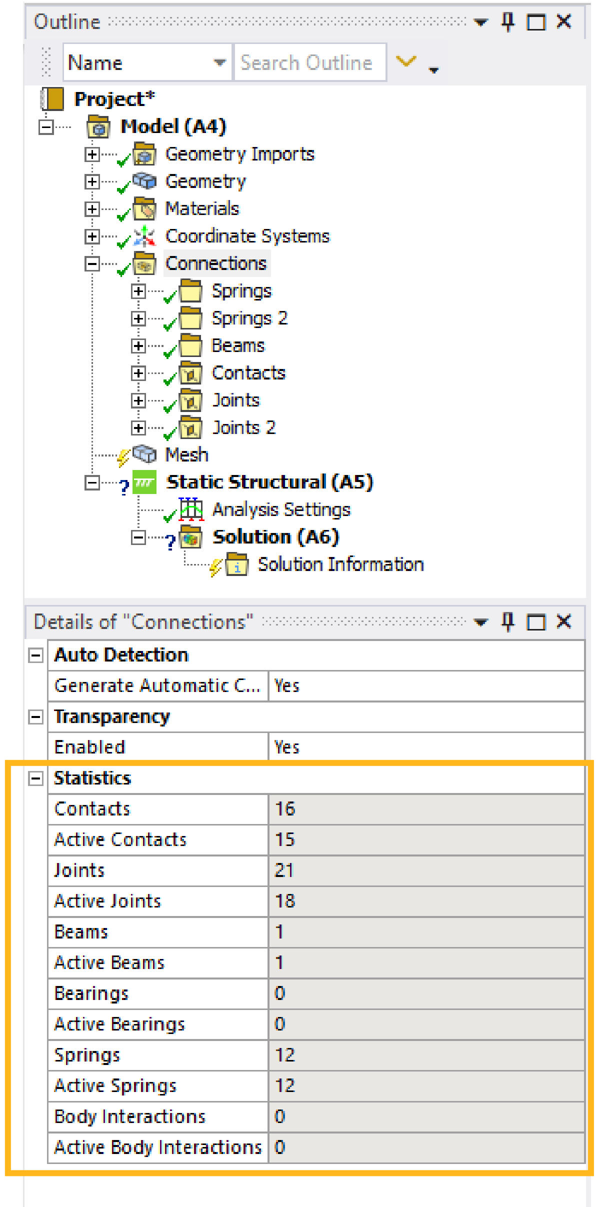




Connection Statistics

Connection Statistics

- Convenient access to overall connection counts without navigating to the Model Summary



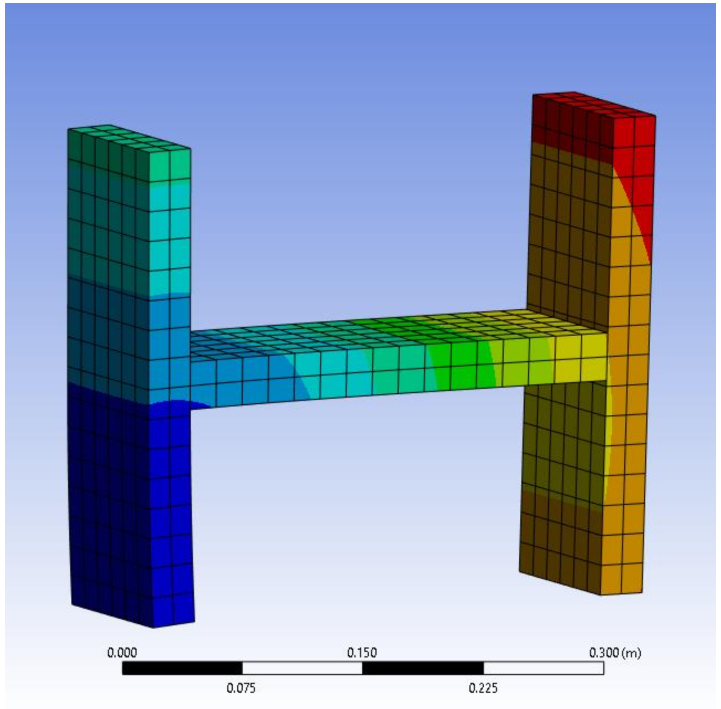
Post Processing

Separated scoping by entities in tabular data and graph

Utility Enhancements

Ansys

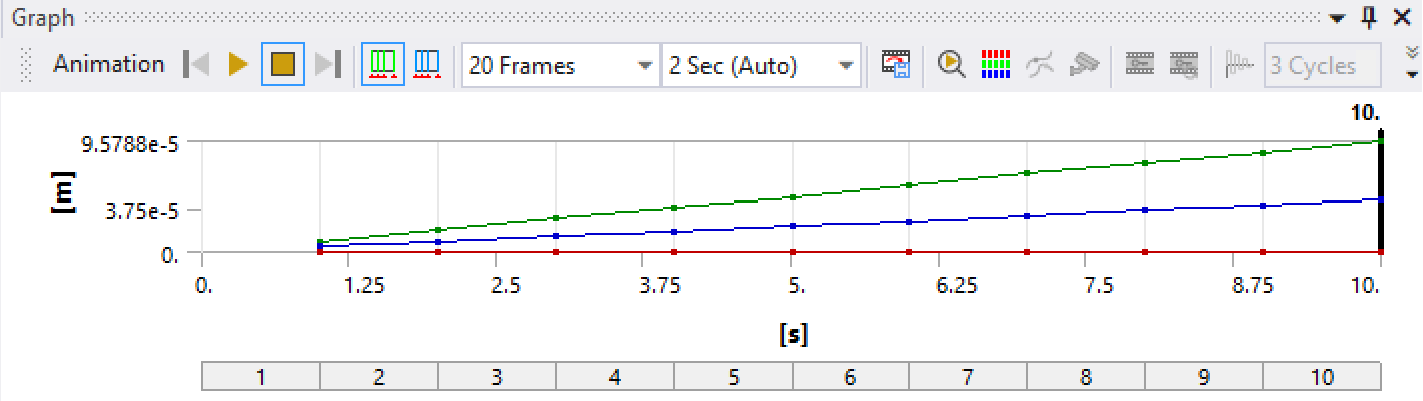
Before the enhancement



All bodies

Tabular Data

	Time [s]	Minimum [m]	Maximum [m]	Average [m]
1	1.	0.	9.5788e-006	4.4756e-006
2	2.	0.	1.9158e-005	8.9513e-006
3	3.	0.	2.8736e-005	1.3427e-005
4	4.	0.	3.8315e-005	1.7903e-005
5	5.	0.	4.7894e-005	2.2378e-005
6	6.	0.	5.7473e-005	2.6854e-005
7	7.	0.	6.7052e-005	3.1329e-005
8	8.	0.	7.6631e-005	3.5805e-005
9	9.	0.	8.6209e-005	4.0281e-005
10	10.	0.	9.5788e-005	4.4756e-005



Minimum, maximum and average values within ALL the bodies are reported in the Tabular Data and Graph

With the enhancement

Details of "Total Deformation-AllBodies"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
By	Time
Display Time	Last
Separate Data by Entity	Yes
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Minimum	
Maximum	
Average	
Minimum Occurs On	
Maximum Occurs On	
Minimum Value Over Ti	

- When **Separate Data By Entity** is set to Yes
- Both Tabular Data and Graph now display the values for each entity in the scope with different color

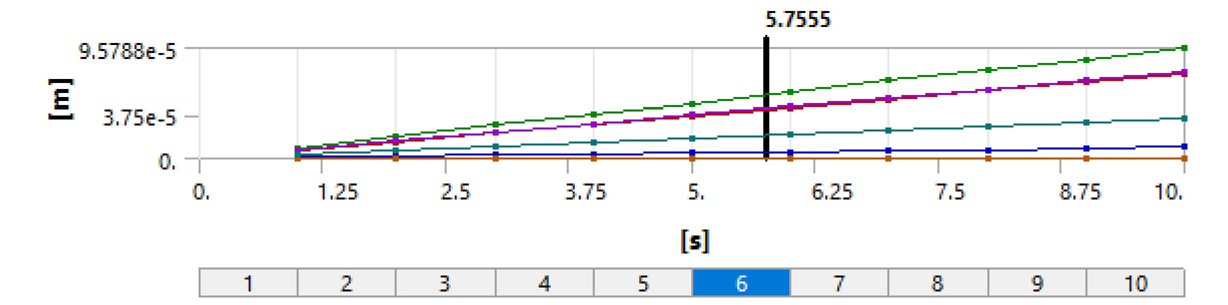
Body ID

Tabular Data

	Time [s]	Min 1 - Reference 6 [m]	Max 1 - Reference 6 [m]	Min 2 - Reference 33 [m]	Max 2 - Reference 33 [m]	Min 3 - Reference 60 [m]	Max 3 - Reference 60 [m]
1	1.	7.3201e-006	9.5788e-006	9.7814e-007	7.4912e-006	0.	3.5401e-006
2	2.	1.464e-005	1.9158e-005	1.9563e-006	1.4982e-005	0.	7.0803e-006
3	3.	2.196e-005	2.8736e-005	2.9344e-006	2.2474e-005	0.	1.062e-005
4	4.	2.928e-005	3.8315e-005	3.9126e-006	2.9965e-005	0.	1.4161e-005
5	5.					0.	1.7701e-005
6	6.					0.	2.1241e-005
7	7.					0.	2.4781e-005
8	8.					0.	2.8321e-005
9	9.					0.	3.1861e-005
10	10.					0.	3.5401e-005

Graph

Animation 20 Frames 2 Sec (Auto)

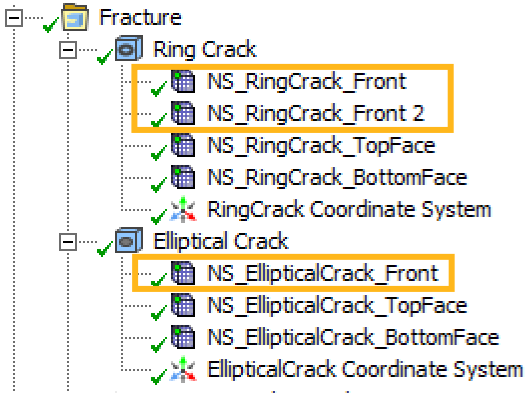


Ansys

Multiple Crack Fronts

Fracture Tool Results for Multiple Crack Fronts

- Fracture tool result (e.g., JINT) can be evaluated for all cracks, all or one of the crack fronts of a specific crack selection

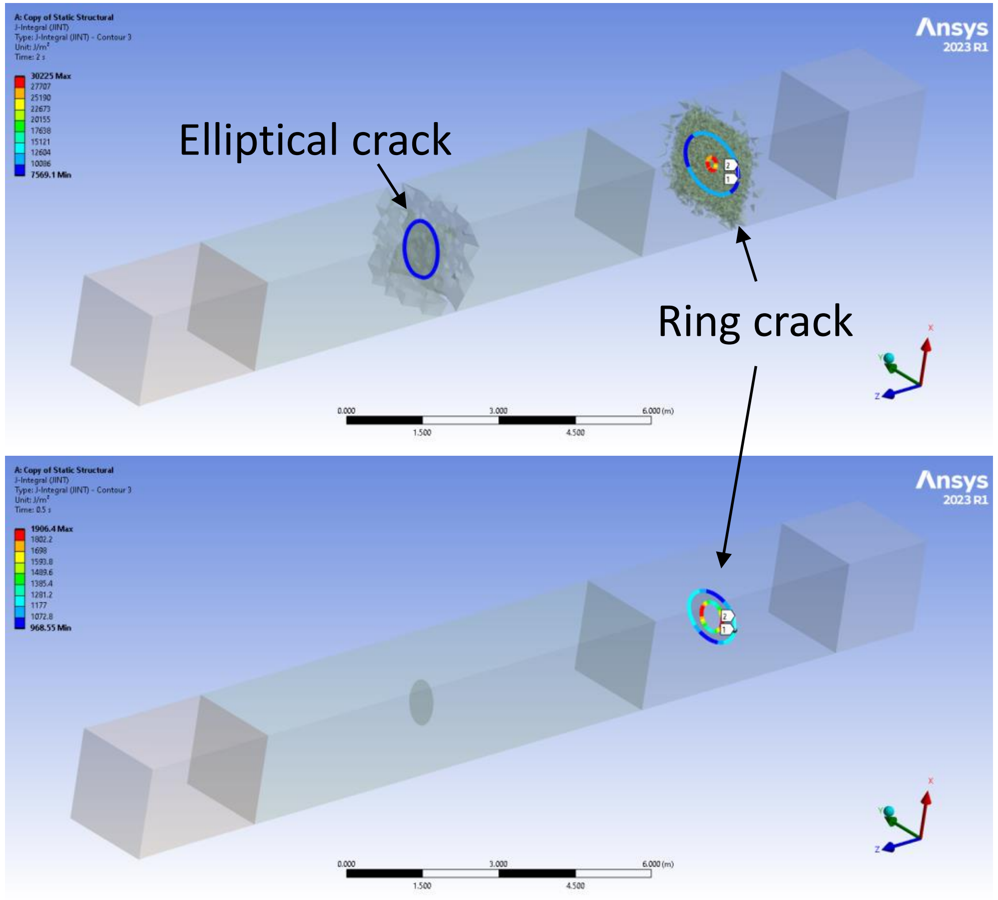


Display fronts of all cracks

Scope	
Scoping Method	Crack Selection
Crack Selection Mode	All Cracks
Definition	
Suppressed	No

Display fronts of the selected crack

Scope	
Scoping Method	Crack Selection
Crack Selection Mode	Single Crack
Crack Selection	Ring Crack
Crack Front Number	All Crack Fronts
Definition	
Suppressed	No



Tabular Data and Graph

- For All Cracks selection in the fracture tool, tabular data and graph show results as specified in the Details
- For multiple crack fronts, the crack Front Number is indicated in the tabular data

Scope	
Scoping Method	Crack Selection
Crack Selection Mode	All Cracks
Definition	
Suppressed	

Details of "J-Integral (JINT)"

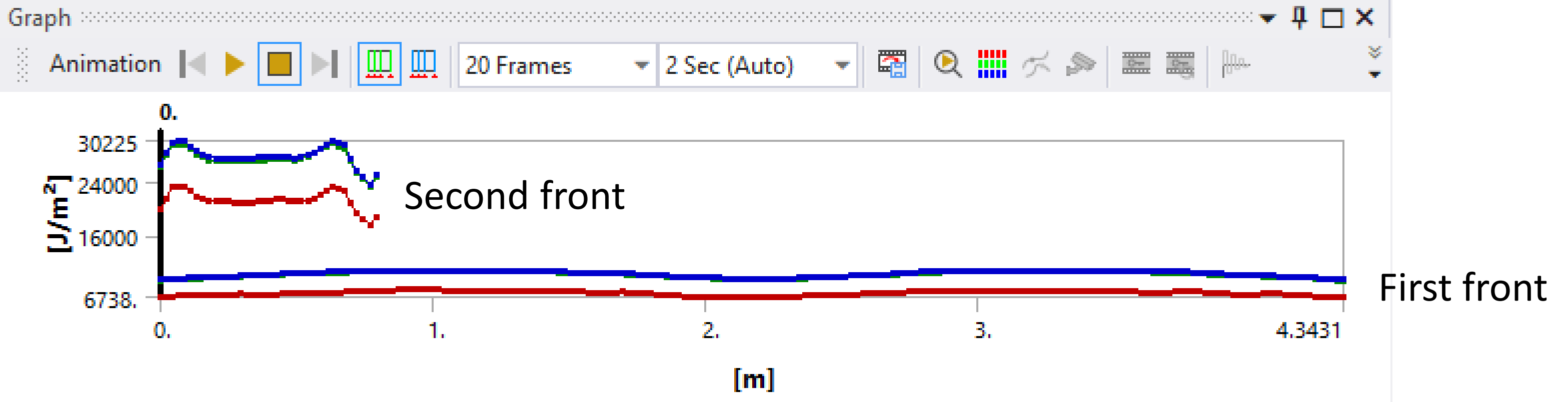
Definition	
Type	J-Integral (JINT)
Active Contour	Last
By	Result Set
Set Number	Last
Separate Data by Entity	No
Calculate Time History	Yes
Suppressed	No
Results	
Minimum	7569.1 J/m²
Maximum	30225 J/m²
Minimum Occurs On	Part2
Maximum Occurs On	Part2
Tabular and Graph Display	
Crack Selection	Ring Crack
Crack Front Number	All Crack Fronts

Tabular Data

	Length [m]	J-Integral (JINT) Contour 1 [J/m²]	J-Integral (JINT) Contour 2 [J/m²]	J-Integral (JINT) Contour 3 [J/m²]	Crack Front Number
201	4.2368	6859.6	9548.1	9617.7	1
202	4.2581	6820.7	9475.1	9537.5	1
203	4.2793	6781.9	9402.	9457.4	1
204	4.3006	6759.9	9343.3	9395.3	1
205	4.3218	6738.	9284.6	9333.2	1
206	4.3431	6751.2	9283.8	9341.5	1
207	0.	19940	26351	26677	2
208	2.255e-002	21572	28000	28363	2
209	4.4954e-002	23204	29649	30049	2
210	6.7449e-002	23259	29640	30076	2
211	8.9908e-002	23314	29630	30103	2

First front

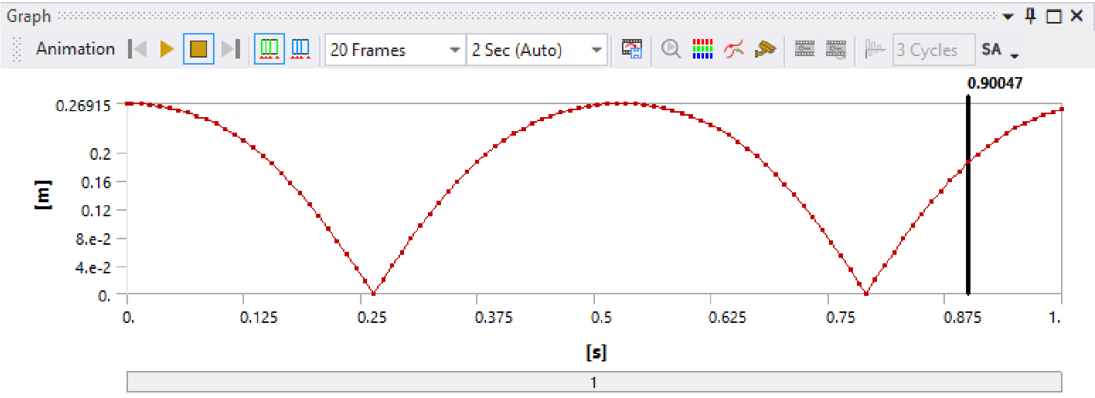
Second front



Distance Probe

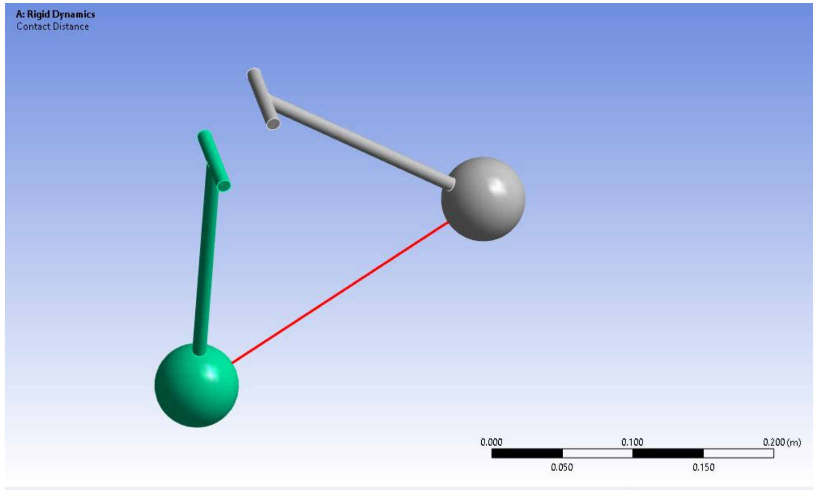
Contact Distance Probe graphics

- Draw red line connecting the two bodies participating in a contact distance probe, during the animation.
- This will appear as a rubber band stretching between the two bodies during animation.
- In chart and tabular data, the distance is used, but for the Graphics effect we need also the end positions of contacts at any time



Tabular Data

	Time [s]	Contact Distance [m]
1	0.	0.26915
2	5.e-003	0.26907
3	1.5e-002	0.26847
4	2.5e-002	0.26725
5	3.5e-002	0.26567
6	4.5e-002	0.26318
7	5.5e-002	0.26003
8	6.5e-002	0.2562
9	7.5e-002	0.25165
10	8.5e-002	0.24616
11	9.5e-002	0.2401
12	0.105	0.23306
13	0.115	0.22508
14	0.125	0.21619



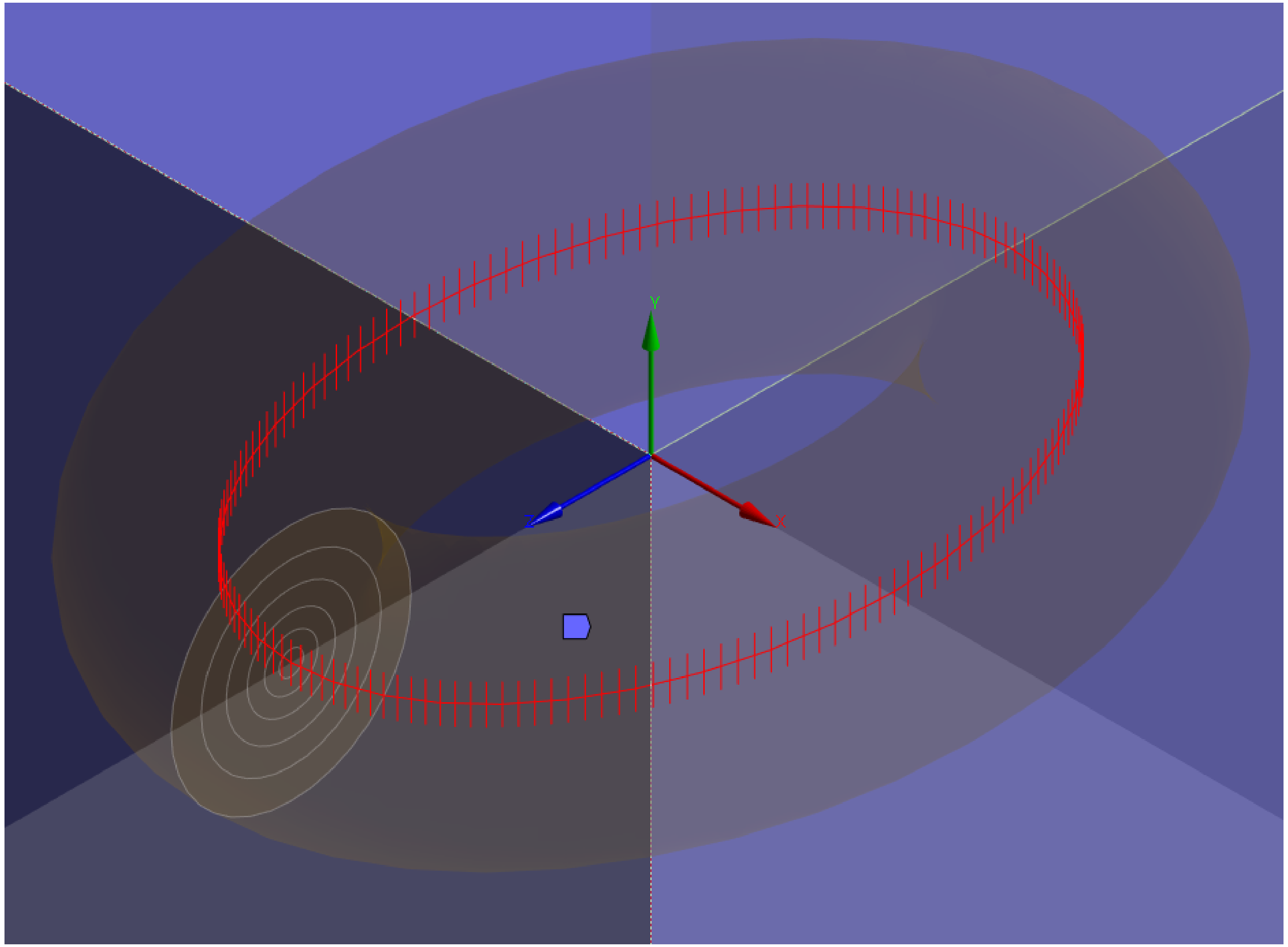
Graphics Annotations for new Crack objects

Introduction

- Previously, semi-elliptical and surface arbitrary cracks pre-existed. Now, with the work Somasekhar Kota did, we have new types of crack objects – namely elliptical, ring and embedded arbitrary cracks.
- We used to draw a half-torus in case of semi-elliptical cracks with two spider nets drawn at the ends of the half-torus tube. Now, in case of elliptical cracks, we draw the complete torus with just a single spider net, which indicates and corresponds to the element size set in the details and matches with the mouse cursor net.
- For surface arbitrary cracks, the origin of the coordinate system would be on the surface of the body and the buffer zone box would be only drawn in the positive X-axis (towards inside of body). But now, with embedded arbitrary cracks, the coordinate system lies inside the attached body and as a result, we extend the buffer zone box in the negative X-axis with the same amount as in the positive X-axis.

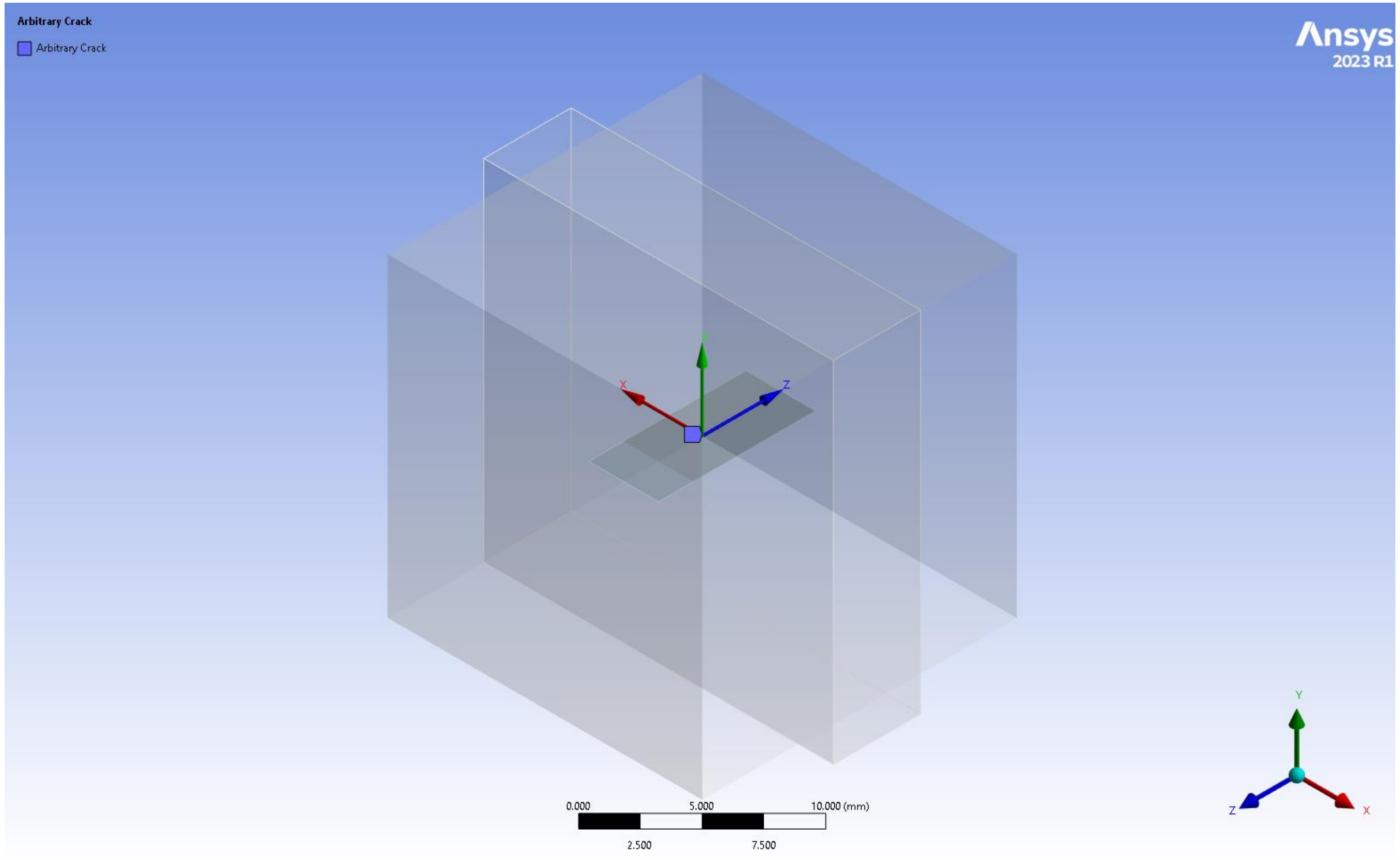
Elliptical Cracks (Tetrahedrons)

- Complete torus is now drawn as opposed to a half-torus in semi-elliptical cracks (pre-existing)
- Two Mesh Methods:
  - Tetrahedrons
  - Hex Dominant
- Ellipse dimensions controlled by Major and Minor Radius in details
- Number of vertical red ticks controlled by Growth Rate in details
- Number of circles controlled by Mesh Contours in details
- Radius of torus tube controlled by Largest Contour Radius in details



Arbitrary Through Crack (Embedded)

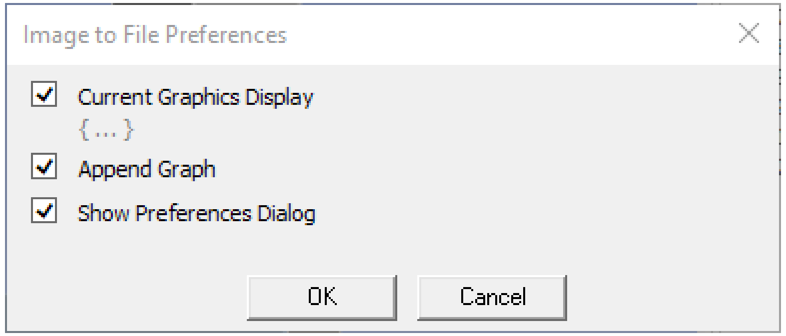
- The buffer zone box is extended in the negative X-axis by the same amount as in the positive direction
- If the crack edge on the negative X-axis direction intersects the attached body, it's considered an embedded arbitrary crack. Otherwise, it's a surface arbitrary crack

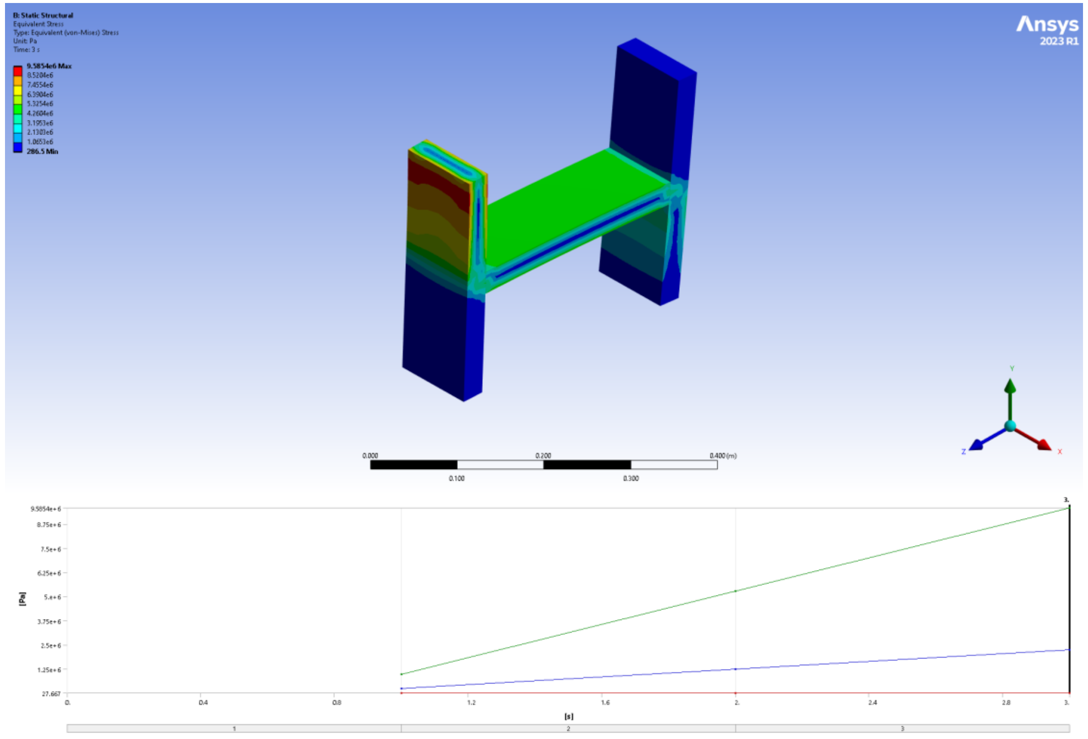


Graphics Export Enhancements

Appending the Graph to Image Exports

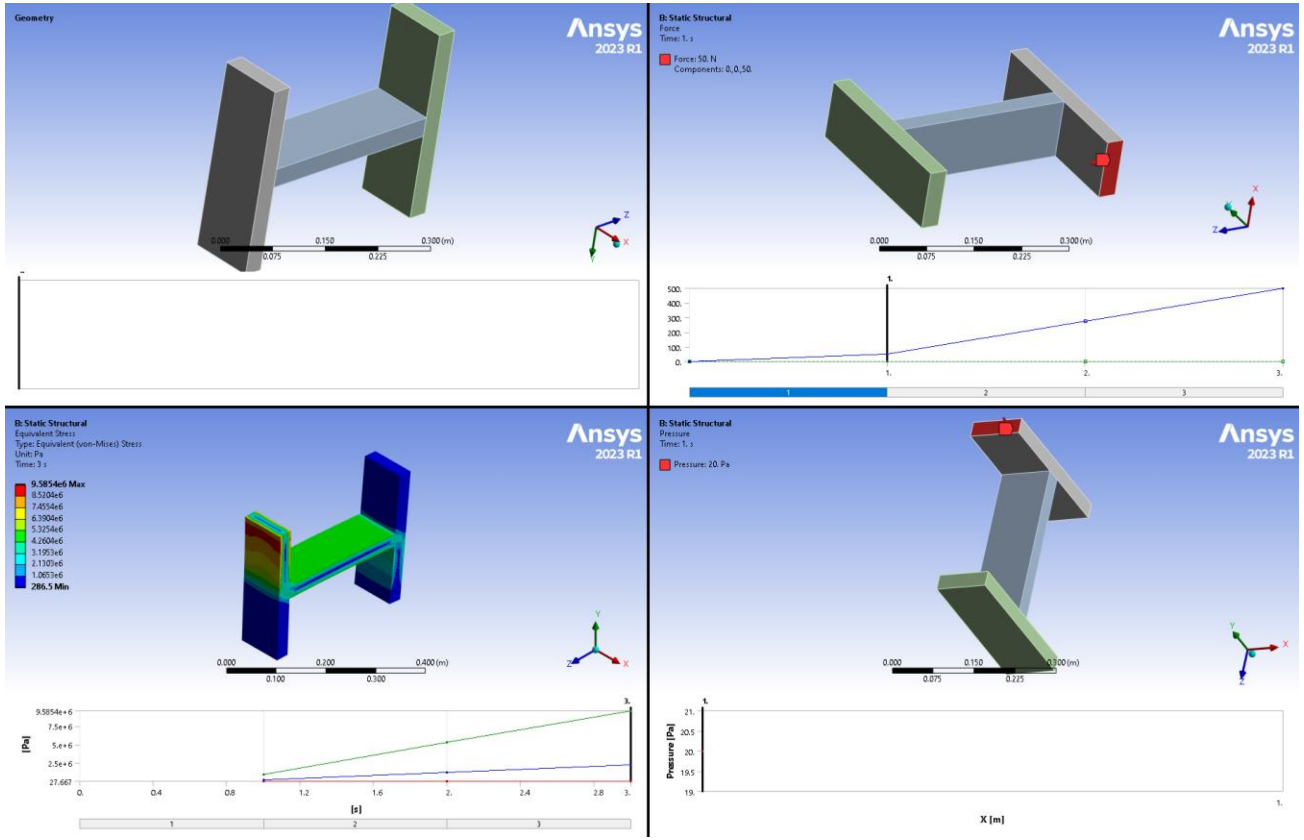
- The Image to File feature and the related Graphics.ExportImage scripting API now offer a new option which allows an image of the exported viewport's graph to be appended to the exported image of the viewport





Composite Image Export (Windows Only)

- It is now possible to export a composite image of all viewports
  - A thin black border can optionally be placed between or around viewport images to help visually separate each viewport image
  - Note: Linux does not support multiple viewports, so this feature is only available on Windows
- This feature is accessible in the GUI via Insert > Image > Composite Image to File… and in the scripting APIs via the Graphics object
- This feature can also make use of the new "Append Graph to Image Exports" feature



No-GUI Animation Export

- The Mechanical scripting API ExportAnimation (available on most result objects) can now be used in windowless batch scripting scenarios on Windows and Linux

- This API thus should be able to be used in all contexts where the Graphics.ExportImage API is available

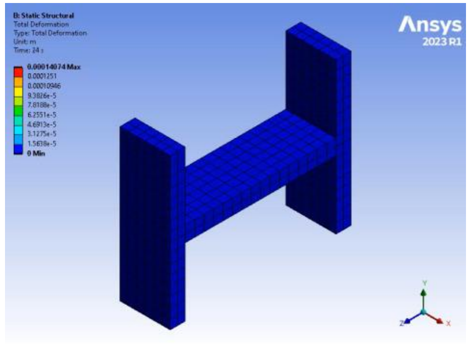
Animation Export Example

- One example of how to use this feature would be via the Workbench scripting console
- Open the console and run:
  - Simulation.RunScript(FilePath=“<path_to_mechanical_script.py>”, IsMeshing=False, ModelName=“<model_name>”)
    - The FilePath parameter should be a string representing the path to a Mechanical scripting file (Python file utilizing Mechanical APIs, such as <result_object>.ExportAnimation(…))
    - ModelName comes from the model cell in the Workbench schematic that you want to run this script for
      - Right-click the cell and click properties to open the cell’s properties. The “General > Component ID” property should contain the name to use

```
>>> Simulation.RunScript(FilePath=r"C:\demo-anim-export\mech.py", IsMeshing=False, ModelName="Model")
```

mech.py

```
res = DataModel.GetObjectById(151)
res.ExportAnimation("C:/demo-anim-export/videmo.mp4")
```

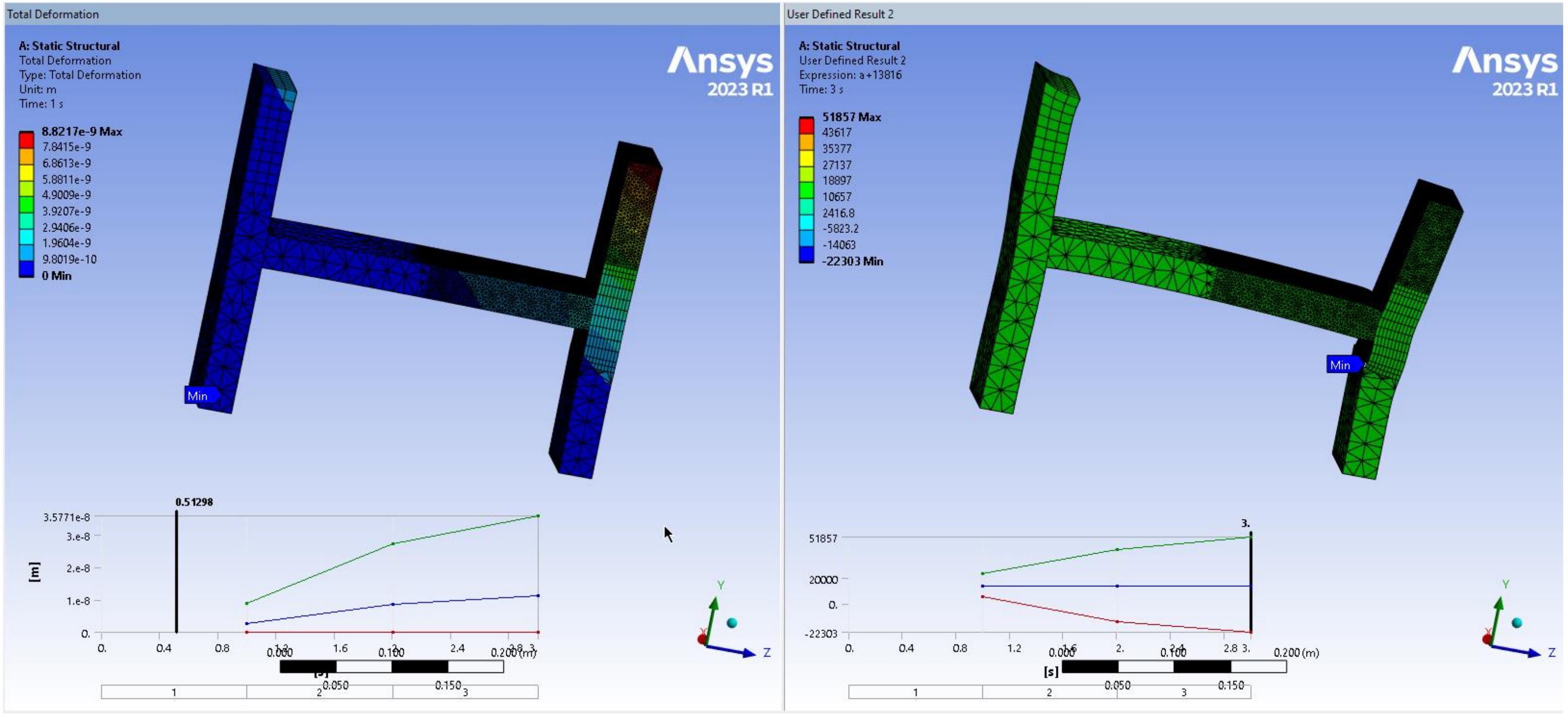


Display Graph in the Graphics window

Ansys

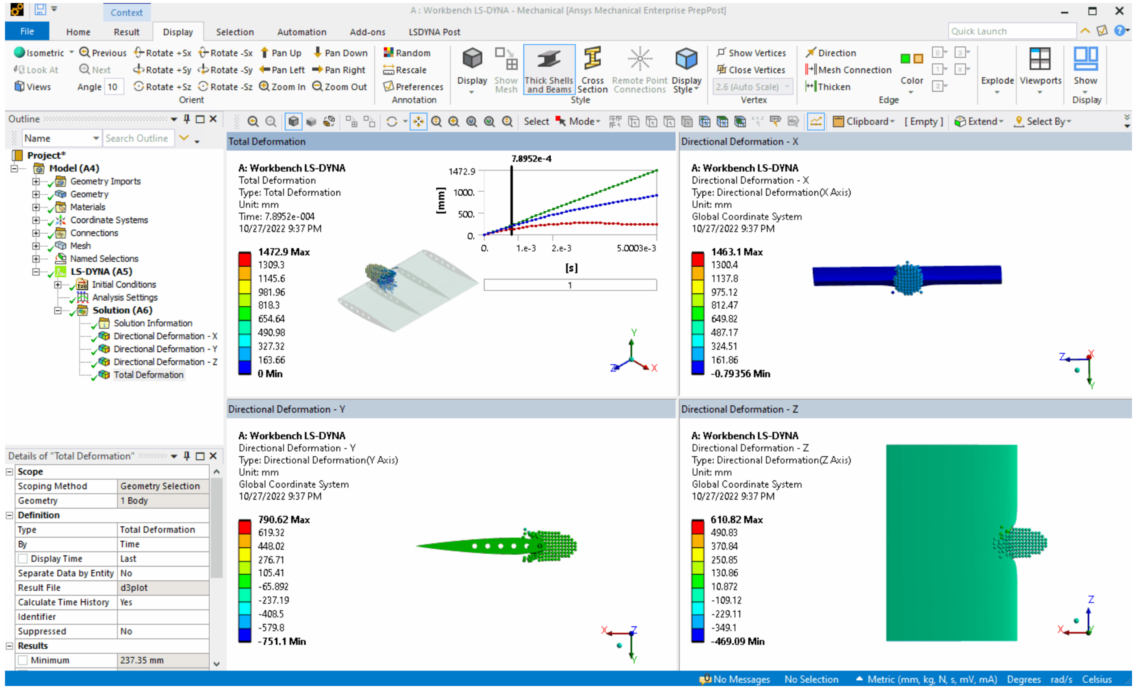
Display Graph in the current viewport window

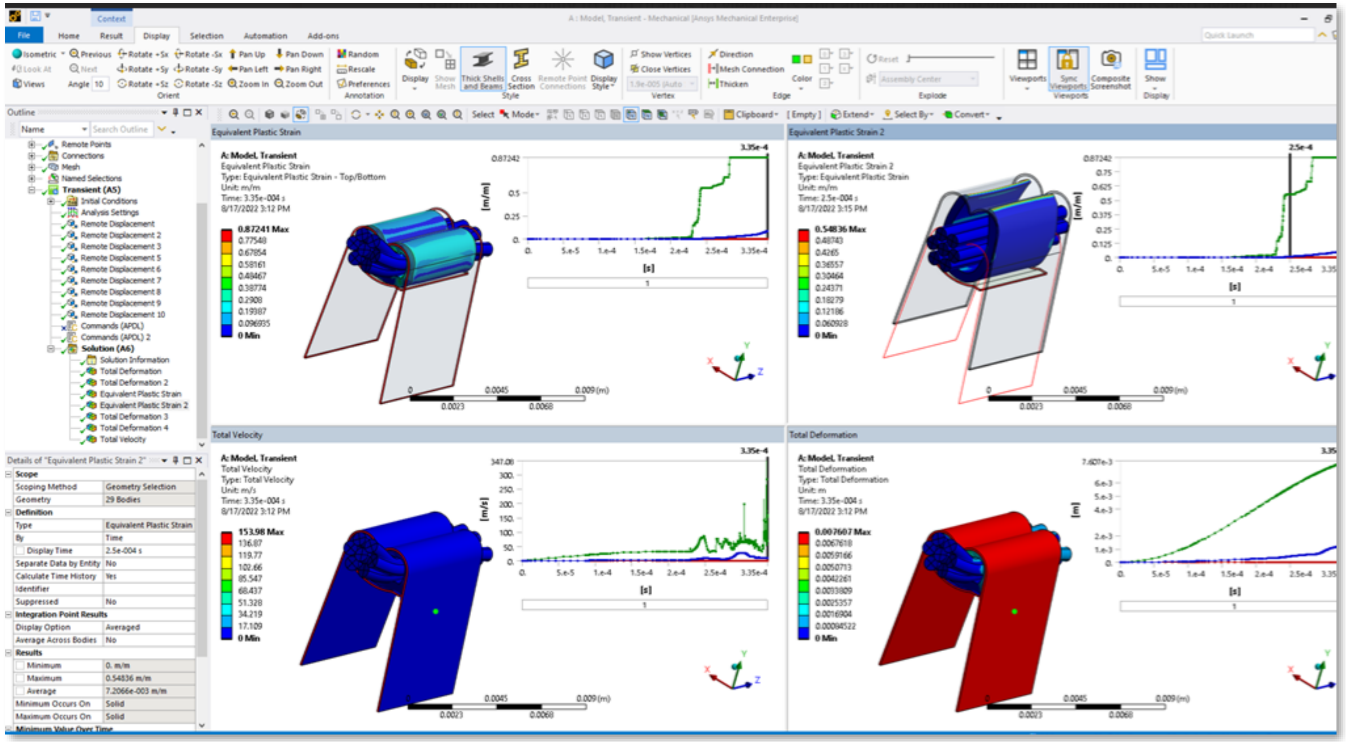
- This feature allows displaying of graph in the graphics window. You can select the **Display Graph** option on the Graphics Toolbar or use the keyboard key **G** or the context (right-click) menu option to turn on the display. This display supports multiple viewports and remains active until you select one of the options to remove the graph.



Embedded Charts in Graphics Window

- Enhanced results viewing with charts
- Multi-viewport animations – beta at 2023 R1





Expanded Support in Accelerated Animation and Multi-viewport Animation

Motivation for Accelerated Animation

- Accelerated Animation, despite the name, is not just about accelerating animation display. It also makes use of newer OpenGL technology to improve the fidelity of rendering, move away from (long) deprecated OpenGL calls, increase control over the rendering stack, and more. It also makes use of other techniques including asynchronous processing to improve the user experience.
- In many cases this does directly improve performance (both runtime and memory), sometimes drastically. In other cases this simply improves rendering and prepares for other enhancements and on the fly changes (such as on the fly legend updates without any graphics regeneration).
- Ultimately this feature is the first fruits of a larger initiative to revamp Mechanical result display for a variety of goals, of which performance is only one.

Additional Support in Accelerated Animation

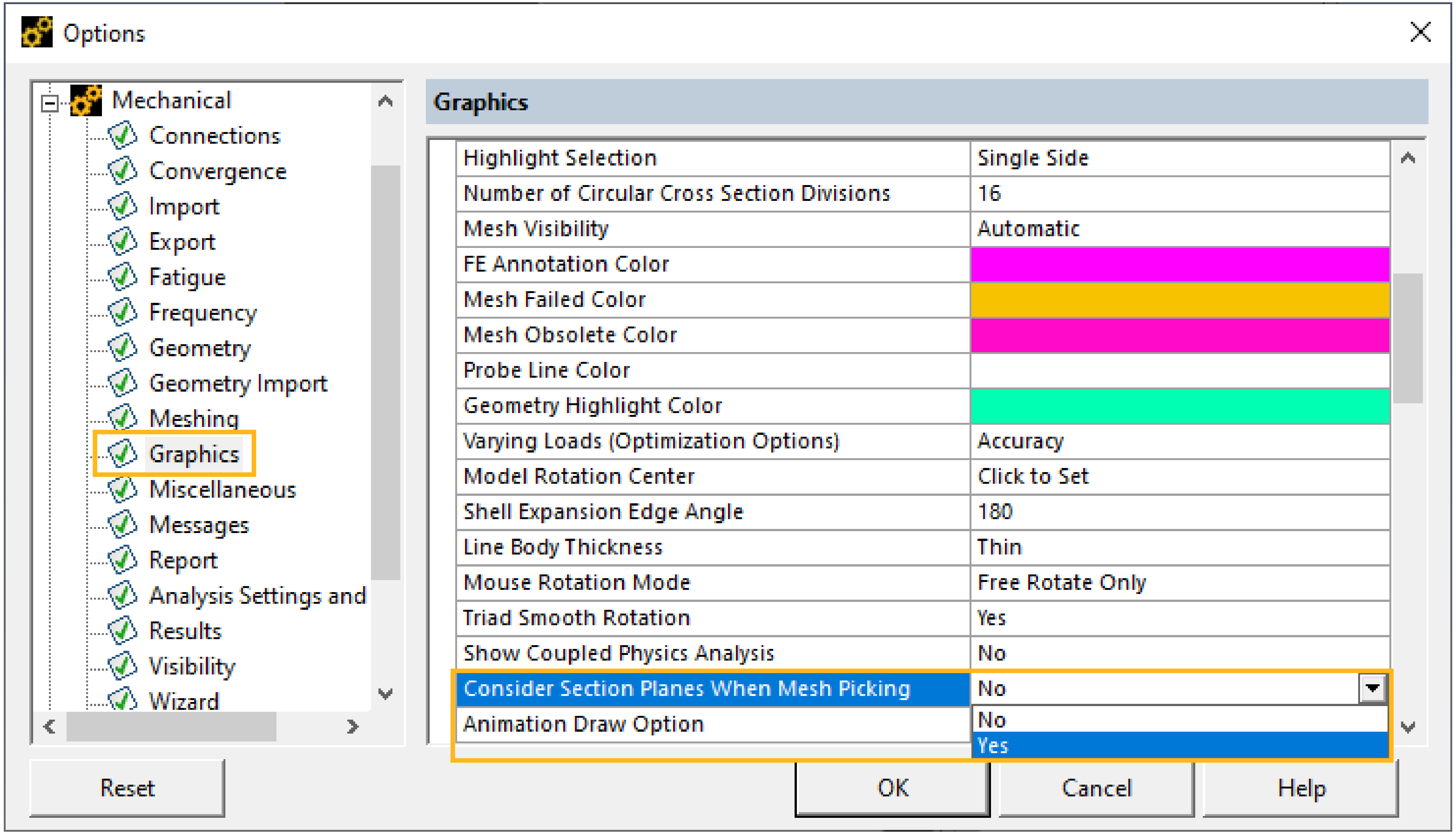
- Vector Results
- Tensor Results
- Results scoped to Faces, Edges, Vertex, Nodes, Elements or Element Faces
- Changing Meshes (NLAD, SMART Fracture, etc.)
- Harmonic Results
- Isoline display

FE Selections used with Section Plane Enhancements

Appending the Graph to Image Exports

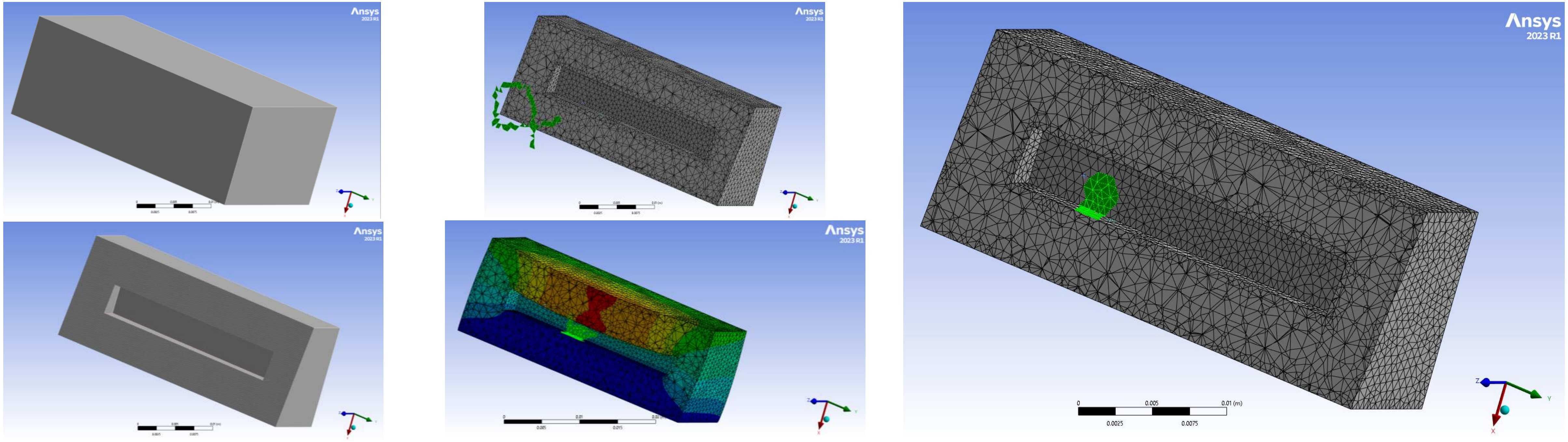
- Selecting mesh (FE) entities located on geometry that are hard to access directly, without using section planes
- Before, if section planes were used to visualize some entities located for ex, inside of the model, picking was not taking the section planes into account, creating confusion since apparently "invisible" entities were selected and not allowing to pick what was seen!
- Now, with this feature, the behavior it is more like "what you see is what you pick" WYSIWYP

How to activate...



Example (an enclosure that is not obvious without using section planes)

- Previously if section planes are present, faces on most exterior of the model are picked (slightly rotated here from the shoot direction to help understanding) - this is non intuitive for the user and not helpful
- Now, entities inside can be picked because the section plane information is considered



Notes

- ALL ACTIVE section planes are considered allowing customization of small areas hard to access otherwise (cut both to the left and right for ex.), but here the user must keep in mind that the true behavior is close with geometry (subtractive) section planes than with the mesh/results section planes behavior.
- Picking is not working on the capping - this is not the object of this work - capping is not normally part of the mesh.
- Hidden bodies will behave appropriately as before and while they could have been used before also to customize areas to pick, for single intricate bodies this was not an option as in the example above.
- Section planes have exterior and volume modes, this feature should work on both selection types and with any of single, box or lasso modes.
- Picking can be used also directly on (meshes of) results

Ansys

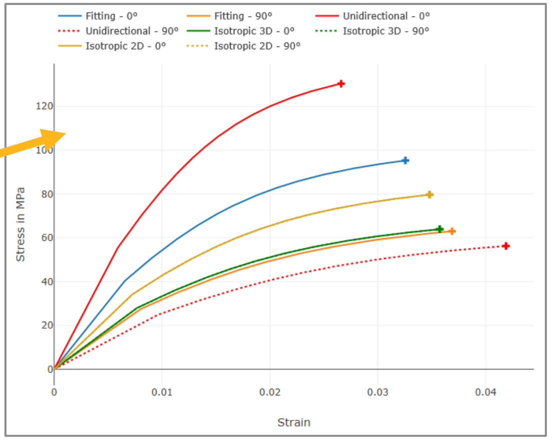
Composites

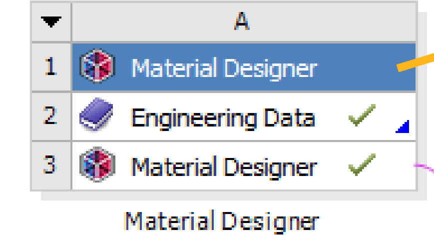
Short Fiber Composites

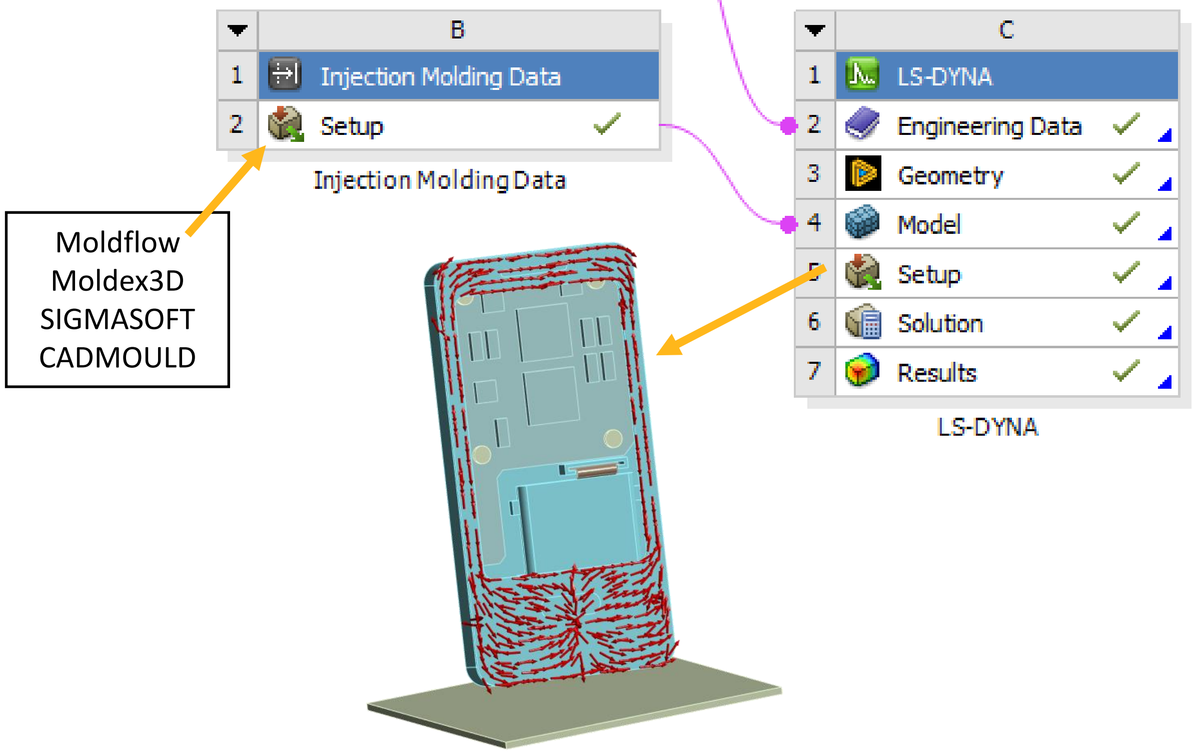
Short Fiber Composites in Workbench LS-DYNA

Same workflow for implicit (MAPDL) and explicit analyses (LS-DYNA):

- Import results from the most popular injection molding simulation tools using the **Injection Molding Data** system.
- Calibrate the anisotropic, orientation-dependent elasto-plastic material in **Material Designer.**
- Set up the model and post-process the results in **Mechanical.**







ACP and Composite Workflows

User Interface in ACP

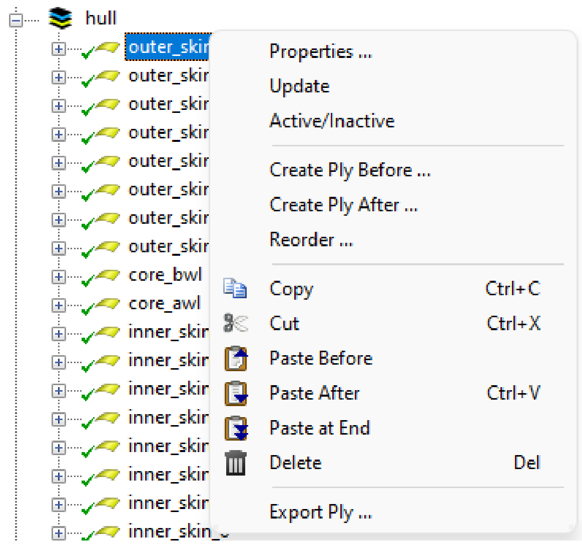
- The user interface has been improved in these areas:

  - Reordering the composite lay-up is made easier by introducing keyboard shortcuts and mouse actions.
    Modeling Plies, Modeling Groups, Scripts and Extrusion Guides can now be moved with Drag-and-Drop or Cut-and-Paste.

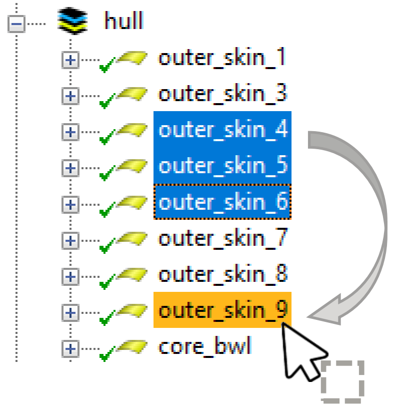
  - Support for the Context Menu key in the Tree View has been added.

  - Folders in the Tree View can be unfolded and folded with a double-click.

Context menu of a modeling ply with new actions and shortcuts



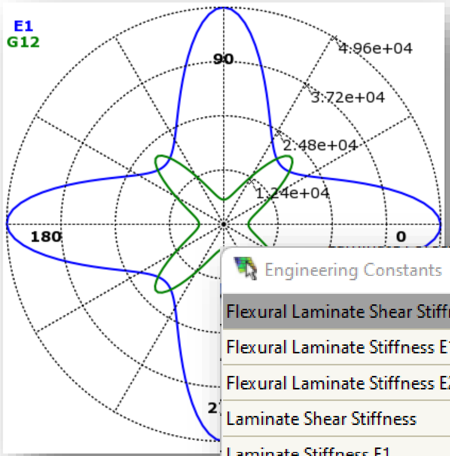
Drag & Drop: selected plies (in blue) are placed after the target ply (in orange)



Sampling Point - Laminate Properties of Variable Materials

- Laminate properties such as the stacking sequence and equivalent stiffnesses can be analyzed by the Sampling Point in ACP and Mechanical. The Sampling Point now supports variable materials as shown below for a ply with draping and shear-dependent properties.

Laminate properties without shear

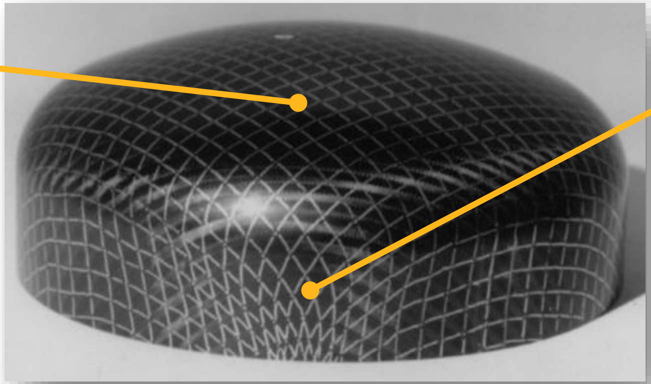


Engineering Constants

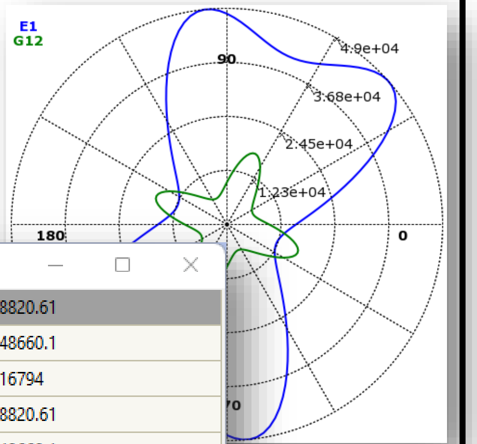
Flexural Laminate Shear Stiffness	5620.22
Flexural Laminate Stiffness E1	49574.6
Flexural Laminate Stiffness E2	49574.4
Laminate Shear Stiffness	5620.22
Laminate Stiffness E1	49574.6
Laminate Stiffness E2	49574.4
Out of Plane Shear G23	3741.32
Out of Plane Shear G31	3741.35
Shear Correction Factor k44 (G23)	0.833328
Shear Correction Factor k55 (G31)	0.833333

Sampling Points
- SP at 30 Degree Shear
  - Shear Dep Woven
    - P1__Shear Dep Woven
      - P1L1__Shear Dep Woven

Draping of a dome



Laminate properties at 30° shear



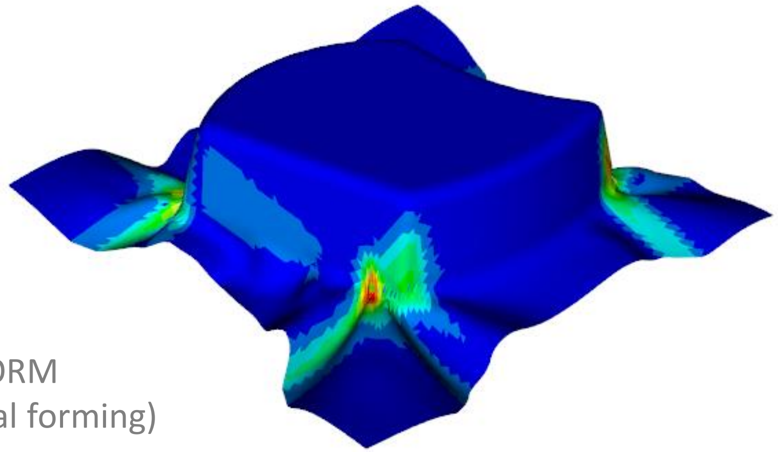
Engineering Constants

Flexural Laminate Shear Stiffness	8820.61
Flexural Laminate Stiffness E1	48660.1
Flexural Laminate Stiffness E2	16794
Laminate Shear Stiffness	8820.61
Laminate Stiffness E1	48660.1
Laminate Stiffness E2	16794
Out of Plane Shear G23	1426.48
Out of Plane Shear G31	4037.82
Shear Correction Factor k44 (G23)	0.354322
Shear Correction Factor k55 (G31)	0.806774

Ansys

HDF5 Composite CAE Interface

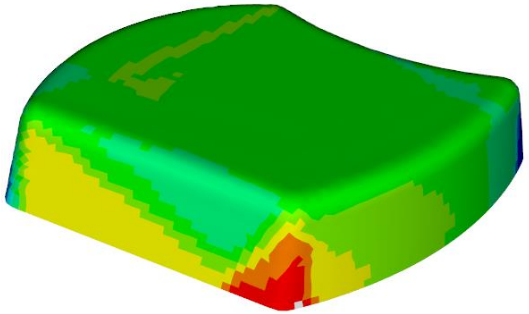
- The export interface of the HDF5 Composite CAE format has been improved in the area of layer representation. The user has now the choice between no offset, bottom, middle or top. This allows new usages, for example collision detection or solid modeling.
- The HDF5 Composite CAE interface now supports user defined scalar fields. The scalar fields can be imported into ACP where they are available as Lookup Tables and Field Definitions. This allows the user to define variable material properties based on the imported scalar fields.



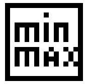
ANIFORM
(virtual forming)

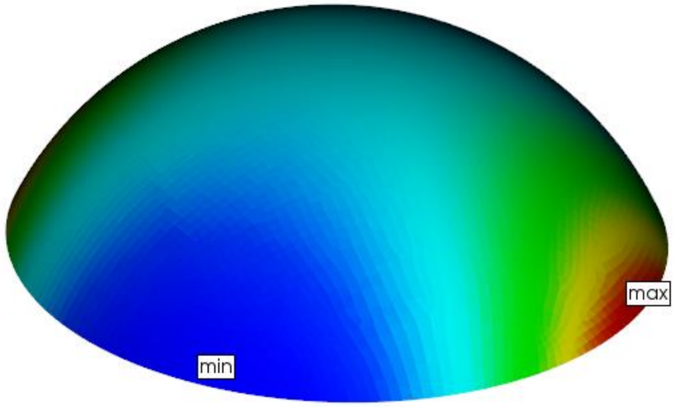
HDF5
Composite CAE

Ansys
ACP



Other Enhancements

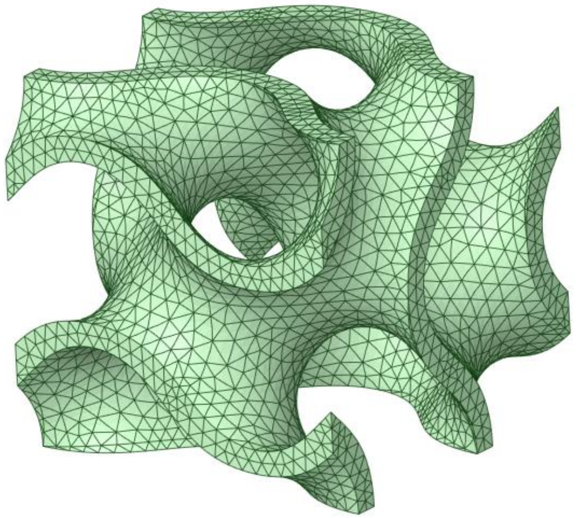
- The performance of the composite tools in Mechanical has been improved again, especially if multiple failure plots and sampling points are defined.
- Labels highlighting the minimum and maximum value of a plot can now be shown. They are toggled from this button  in the toolbar.
- Some issues in the visualization have been resolved.
- Support of file paths with Unicode characters.
- The file size of the ACP DB (.acph5) has been optimized.



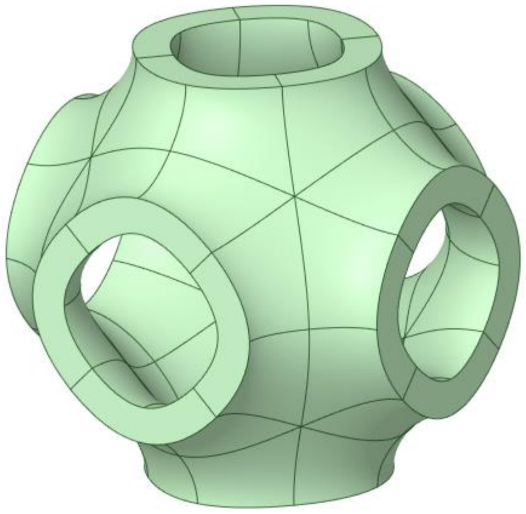
Material Designer

Triply Periodic Minimal Surface RVEs

- New pre-defined RVEs in Material Designer:



Gyroid

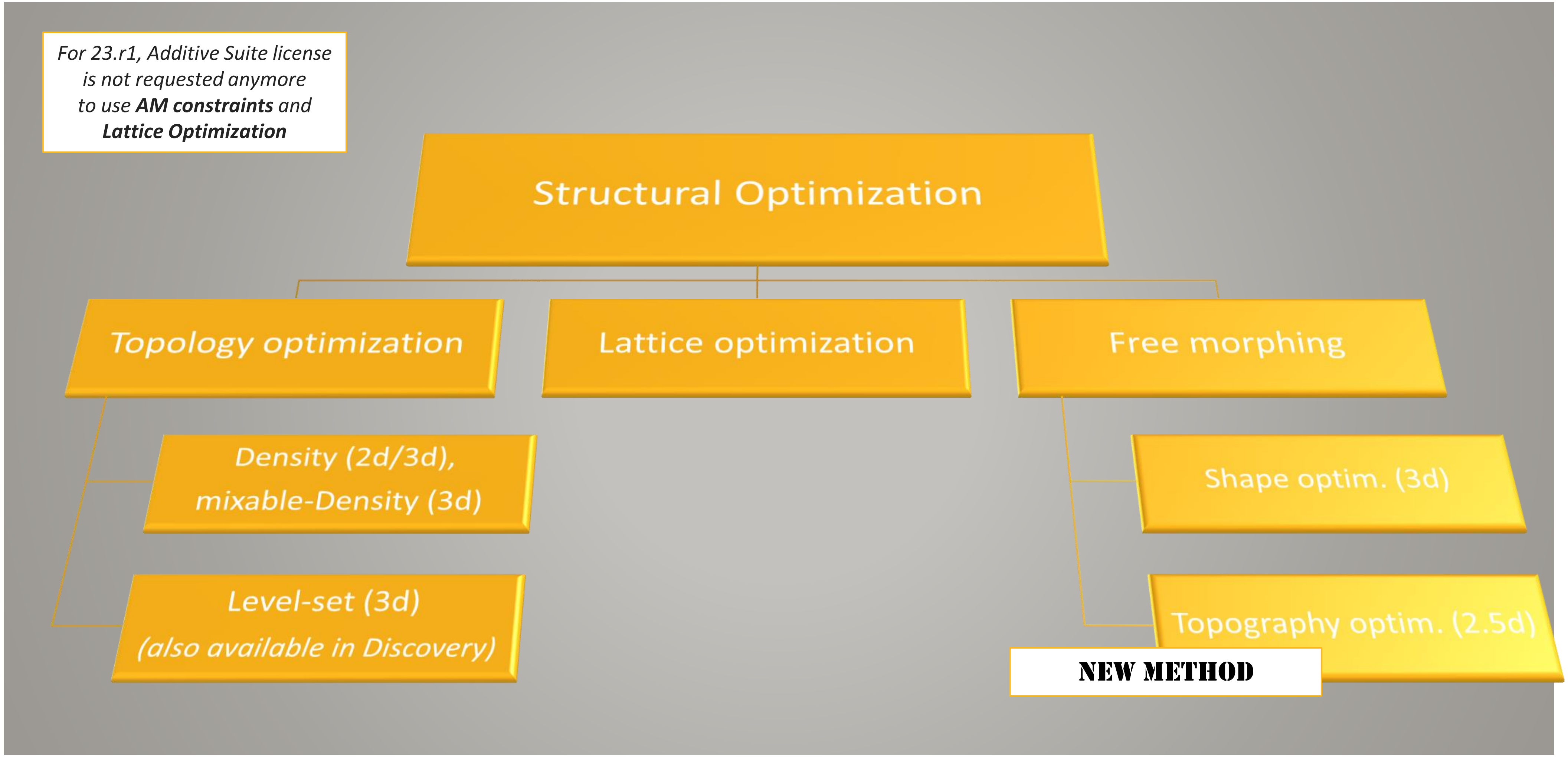


Schwarz P

- Easily simulate these commonly used types of infill

Structural Optimization

Overview of the different methods



New Optimization Methods

Topography & Mix-of-Methods

Structural Optimization

- New Analysis Settings options for X17:
  - Multi Optim Type Strategy: PG, On, Off
  - Algorithm: PG, MFD, SCPIP
- Exclusion Extension option for Levet-Set & Mixable Density
- Mixable Density: Advanced Options:
  - Initial Volume Fraction
  - Penalty Factor
  - Hyperbolic Projection

Details of "Analysis Settings"

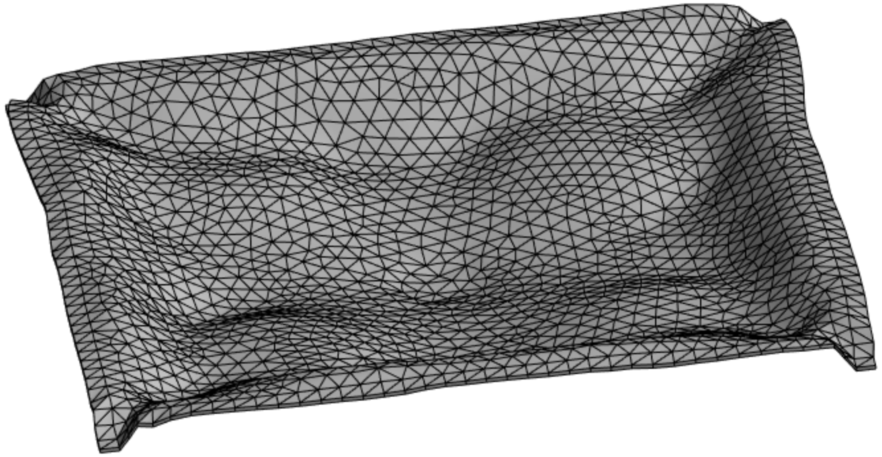
Definition	
Maximum Number Of Iterations	500.
Convergence Accuracy	0.1 %
Output Controls	
Solver Controls	
Solver Type	Program Controlled
Multi Optim Type Strategy	Program Controlled
Algorithm	Program Controlled

Details of "Optimization Region"

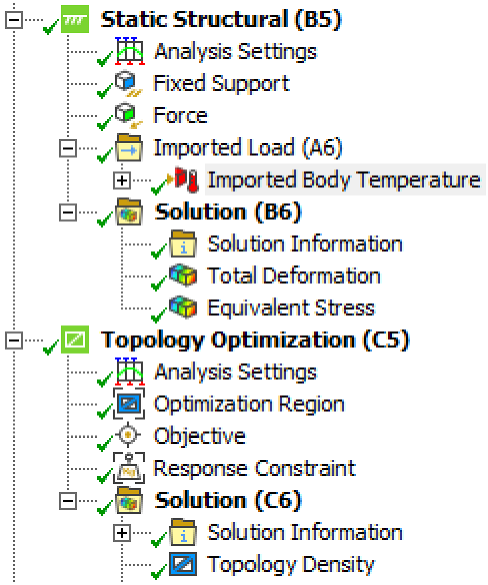
Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Boundary Conditions
Exclusion Thickness	Program Controlled
Exclusion Extension	Isotropic
Definition	
Suppressed	No
Optimization Option	
Optimization Type	Topology Optimization - Mixable Density
Initial Volume Fraction	Program Controlled
Penalty Factor (Stiffness)	Program Controlled
Hyperbolic Projection	Program Controlled

Structural Optimization

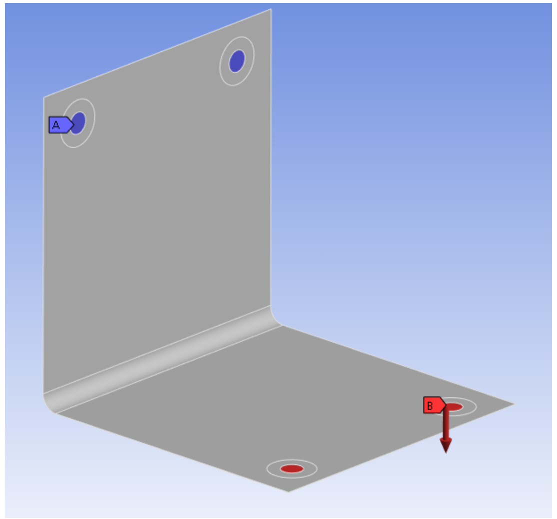
- New Topography Optimization Type



- Thermal Condition support for Mixable Density
- Multi Optim Region support for Mixable Density



Topography Optimization



Topography Optimization is similar to Shape Optimization in the sense that:

- it is a free-morphing optimization,
- the setup is user-friendly: select the body to optimize and define the non-optimizable region,
- the degrees of freedom for optimization are the nodes location,

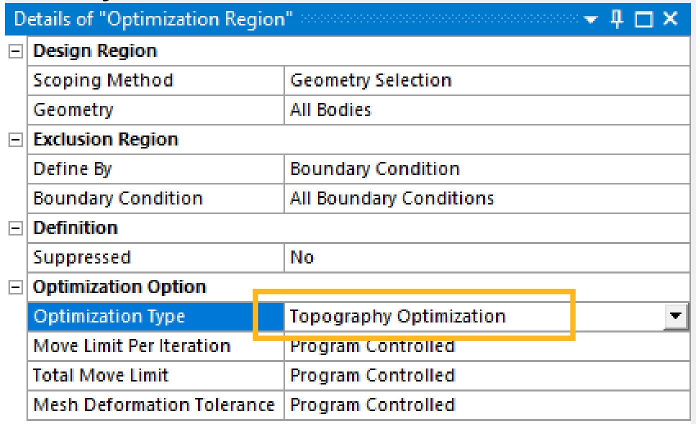
By contrast to shape optimization, Topography Optimization is dedicated for shell model.

The set-up is similar to Shape Optimization:

- **Move Limit Per Iteration** that enables you to define how far each node can move at each iteration (“dx” - element size- by default)
- **Total Move Limit** that enables you to define how far each node can move in total (5.dx by default).
- **Mesh Deformation Control**: That enables you to define how much the mesh can be stretched. (0.5 by default)

Current capabilities

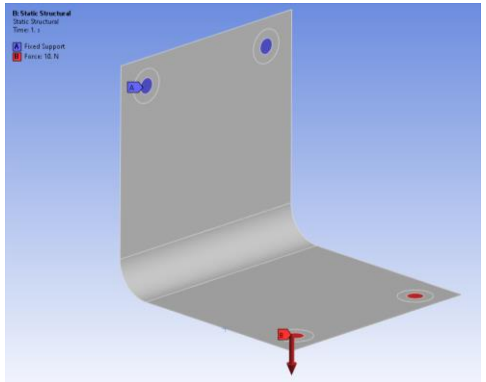
- about the Geometric Analysis: mass and volume
- Static Analisis: compliance and any static-UDC (User Defined Criterion)
- Modal Analysis: any modal-UDC (single frequency, robust frequency, etc)
- Element type and order: triangles/quadrangles, linear/quadratic
- compatible with the capabality « mix-of-methods »



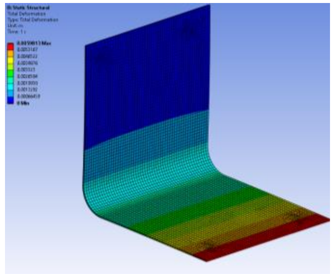
Details of "Optimization Region"

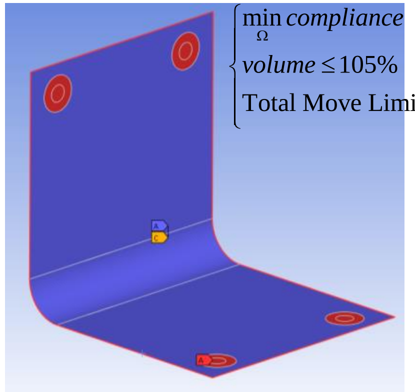
Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Boundary Conditions
Definition	
Suppressed	No
Optimization Option	
Optimization Type	Topography Optimization
Move Limit Per Iteration	Program Controlled
Total Move Limit	Program Controlled
Mesh Deformation Tolerance	Program Controlled

learn by examples (1/2)

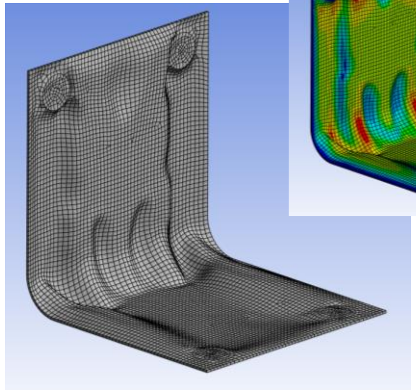


Static analysis:
Displacement-field

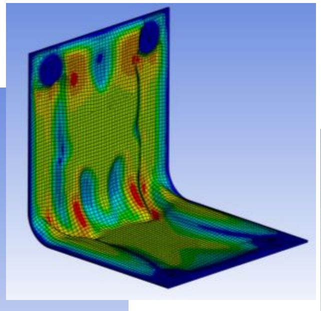




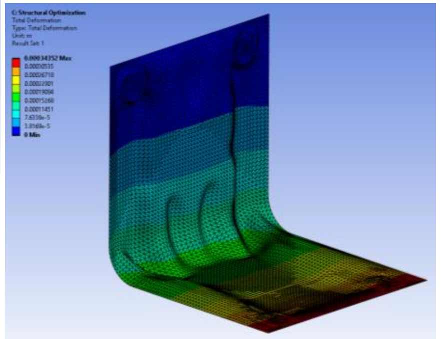
$$\begin{cases} \min_{\Omega} \textit{compliance} \\ \textit{volume} \le 105\% \\ \text{Total Move Limit} \le 2.5\text{mm} \end{cases}$$



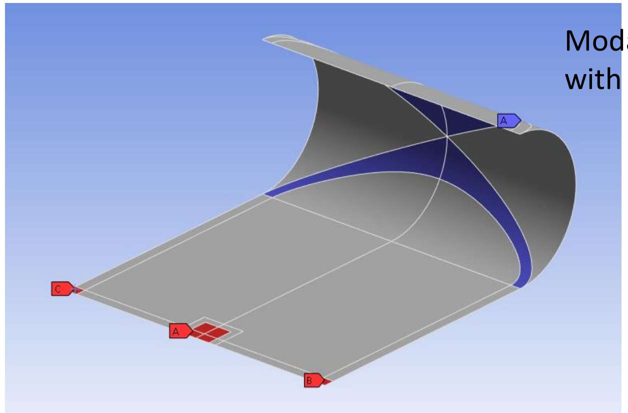
Nodes motion



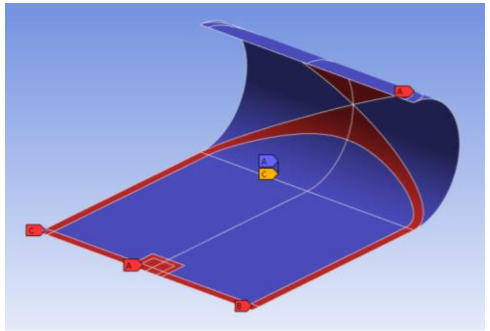
Disp-field of static analysis
20x stiffer



Mechanical setup & Mesh → Optimization setup → Solution

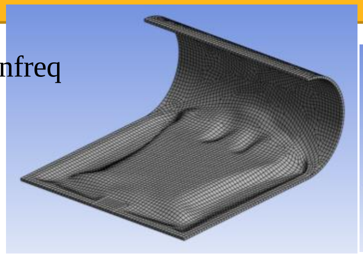


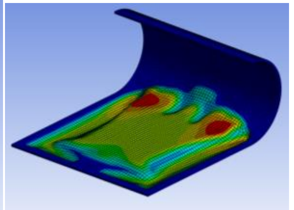
Modal Analysis,
with remote mass



Maximize 1st eigenfreq

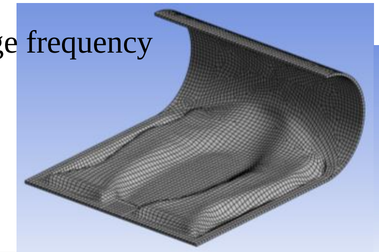
$$\begin{cases} \max_{\Omega} f_1 \\ \textit{mass} \le 105\% \end{cases}$$

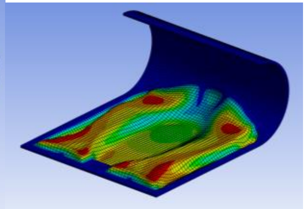




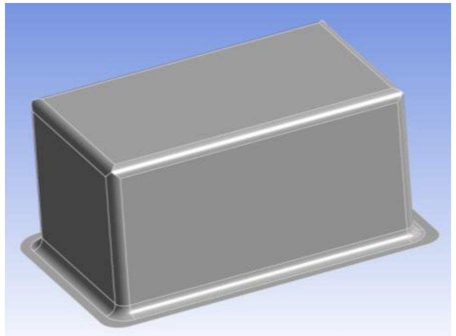
Maximize average frequency

$$\left\{ \max_{\Omega} \frac{1}{n} \sum_{i=1,..n} f_i \right.$$

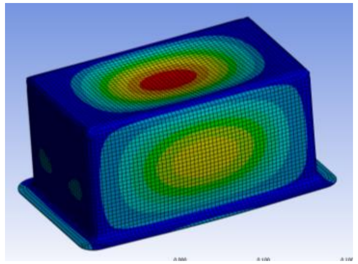




learn by examples (2/2)

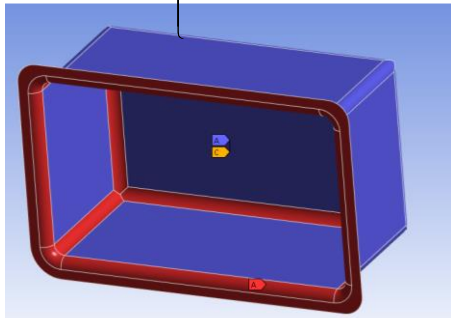


Modal analysis:
7-th eigenmode

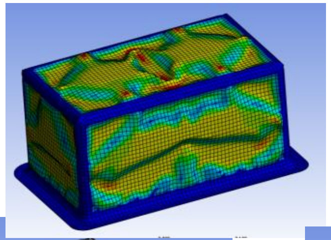


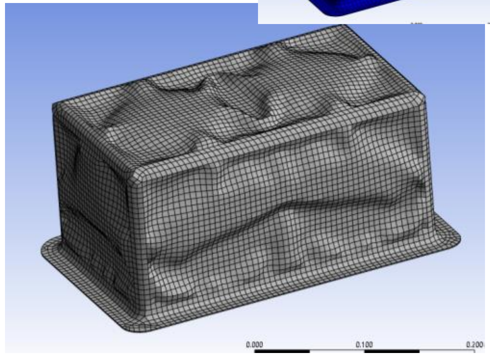
Maximize average frequency

$$\begin{cases} \max_{\Omega}\left(\frac{1}{n}\sum_{i=1,\dots n} f_i\right) \\ mass \leq 105\% \\ \text{Total Move Limit} \leq 5\text{mm} \end{cases}$$

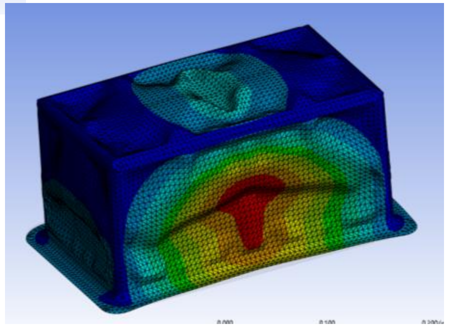


Nodes motion

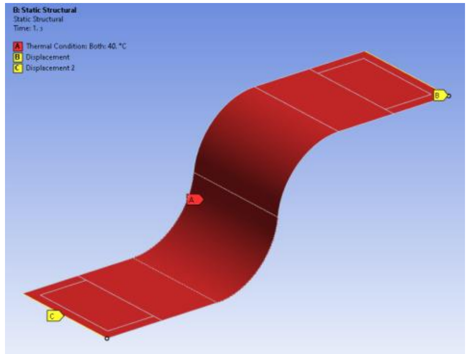




7[th] eigenmode

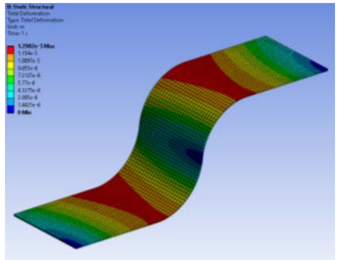


Mechanical setup & Mesh → Optimization setup → Solution

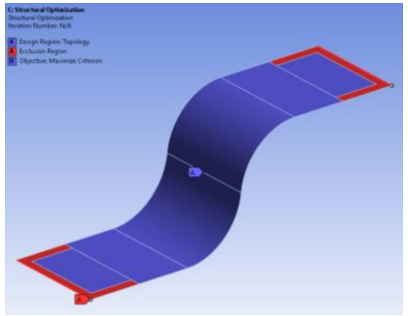


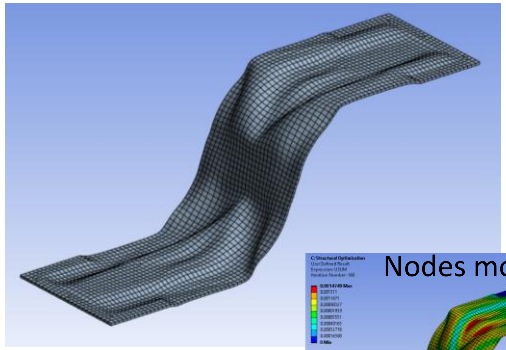
Static Analysis with thermal load only

Displacement-field

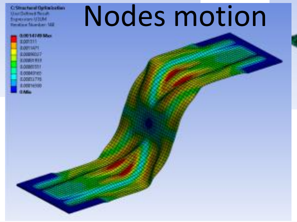


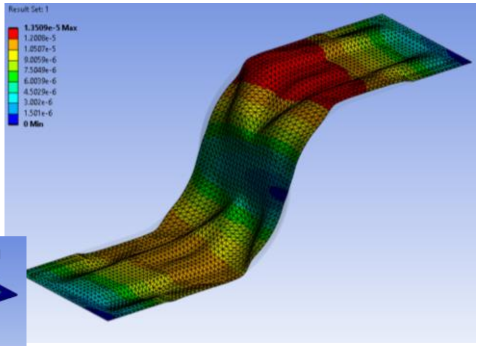
$$\begin{cases} \text{Maximize reaction-force} \\ \text{Total Move Limit} \leq 0.5\text{mm} \end{cases}$$





Nodes motion





static analysis displacement field

Ansys

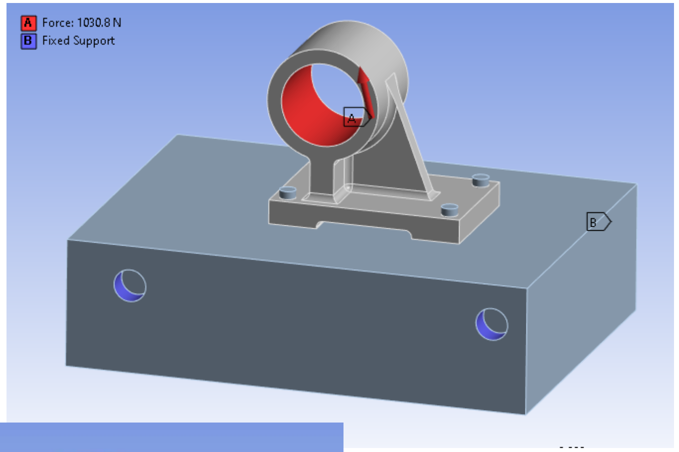
Multiple Optimization Types

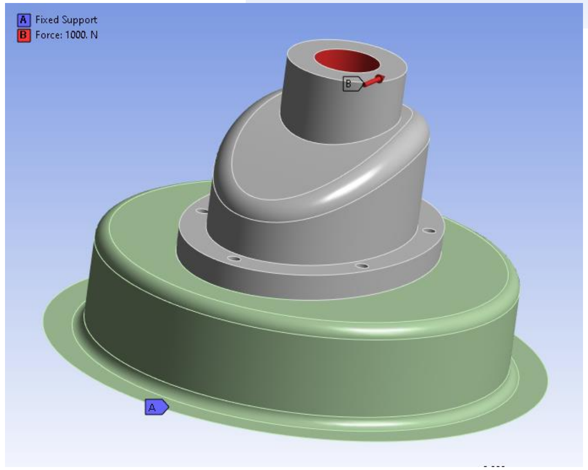
(i.e. Mix of optimization Methods)

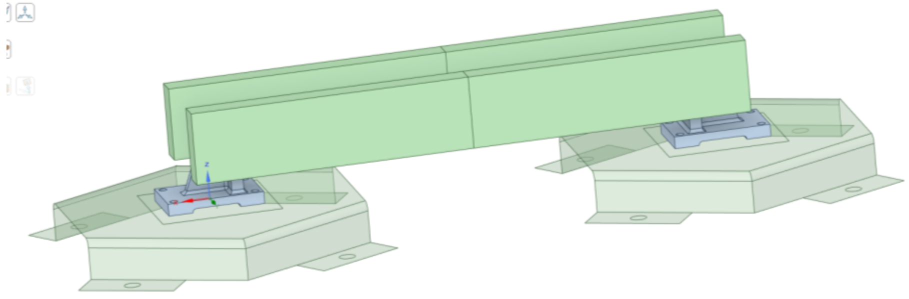
Multiple Optimization type Capability

Context
- there are multiple optim methods (topology, lattice, shape opt) but none fits for all needs
- each is dedicated for a specific context, specific needs and specific expectations
- Topology optimization:
  - aims to sketch a design from scratch;
  - rather fits in early stage of conception;
  - delivers the best design within a working domain;
  - is an IBM approach, not accurate but highly permissive.
- Shape optimization:
  - improves an existing design by bringing local modifications;
  - rather fits in late stage of conception;
  - delivers the best modification to improve a given design;
  - is a body-fitted approach, accurate but less permissive.
- For an orphan component, just chose the appropriate approach. But for a system with multiple components, one method is not always adequate. It is sometimes necessary to use specific optimization method on each component.

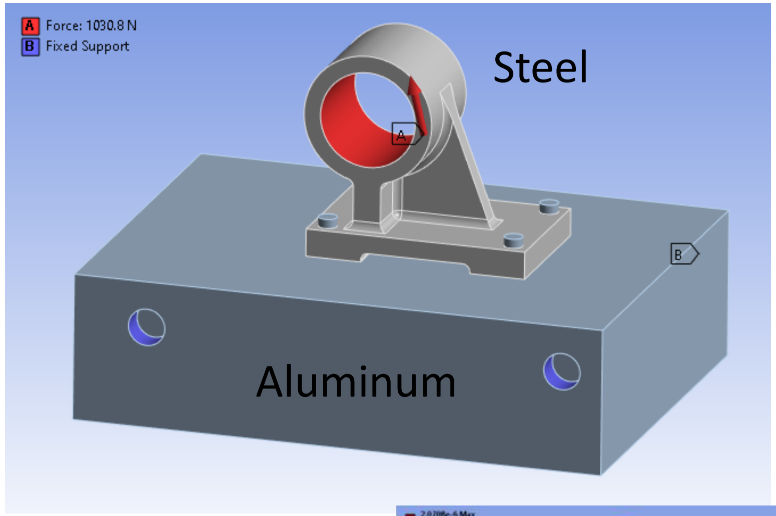
Since 2023 R1, it is possible to
- solve an optimization problem
- ... with multiple optimization regions
- ... having their own optim method (Topology, Shape opt. , Topography)



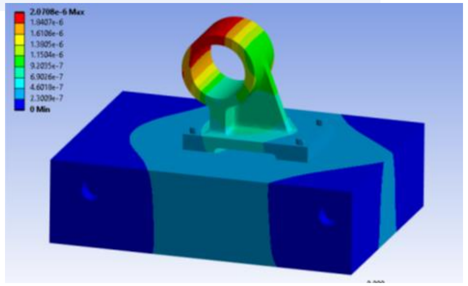




learn by examples (1/2)



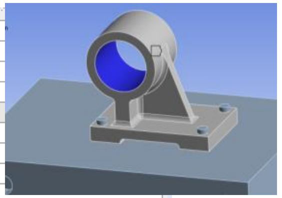
Static analysis:
Displacement-field



Optim problem $\begin{cases} \min \textit{mass} \\ DISP_A < 1.35e^{-6} m \end{cases}$

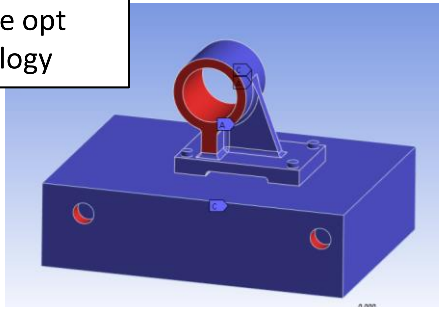
(this problem has no solution if the hook is not optimizable)

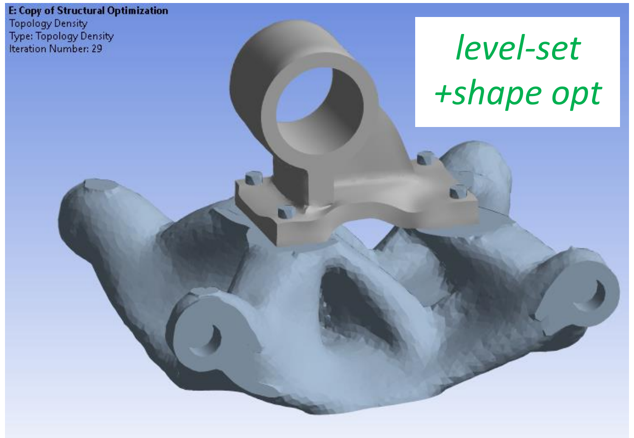
Definition	
Base Result	Displacement
Suppressed	No
Results	
Value	1.3884e-006 m
Scoping	
Scoping Method	Geometry
Geometry	1 Face
Load Step Selection	
Step	1
Vector Reduction	
Coordinate System	Global Coordinate System
Vector Reduction	Magnitude
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete

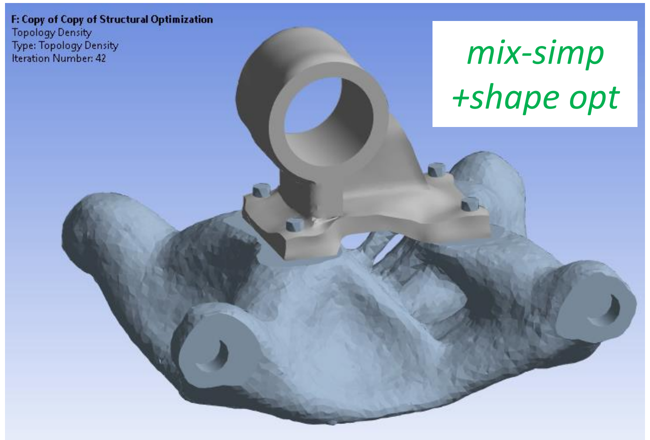


Criterion of interest: average disp of the loaded surface

Two optim region
-one by shape opt
-one by topology

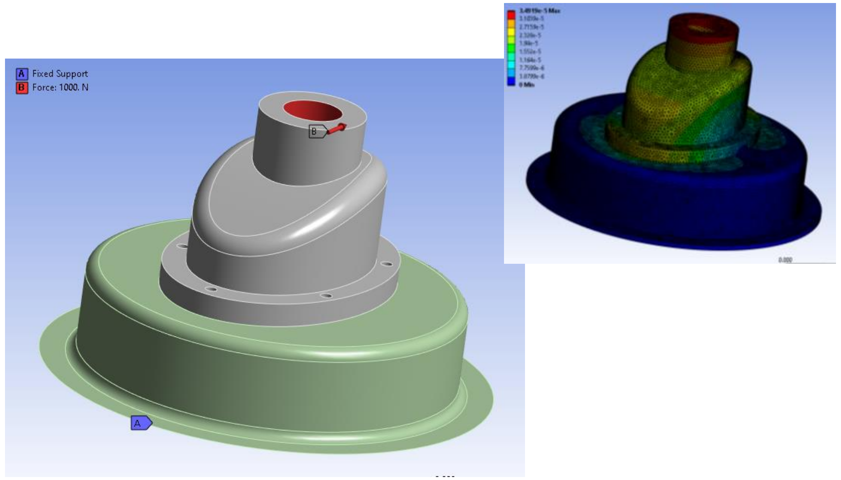




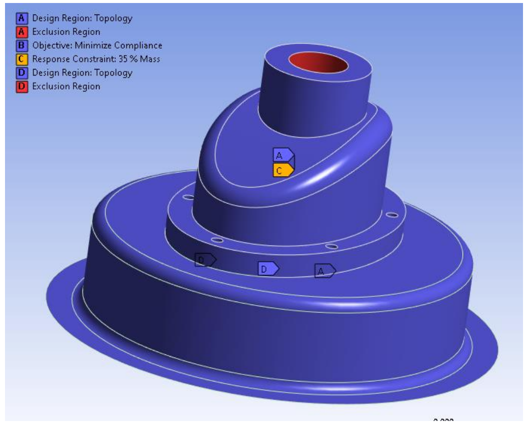


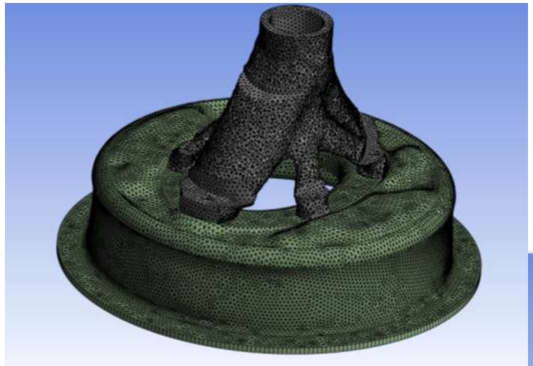
Mechanical setup & Mesh → Optimization setup → Solution

learn by examples (2/2)

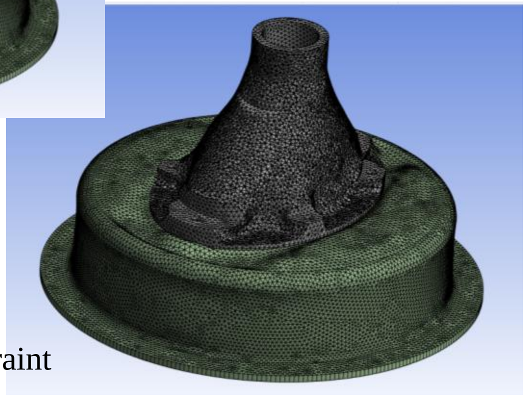


2 optim regions:
Topography & Topology



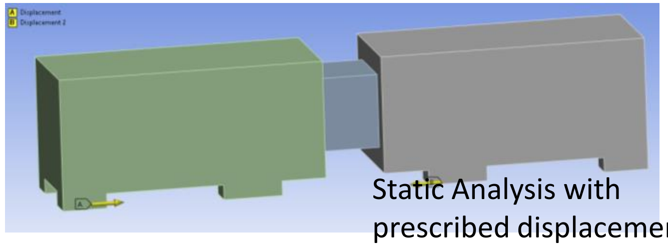


$$\begin{cases} \min_{\Omega} compliance \\ vol < 35\% \end{cases}$$

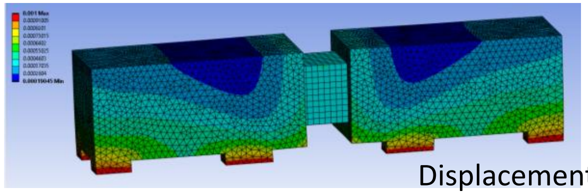


$$\begin{cases} \min_{\Omega} compliance \\ vol < 30\% \\ + \text{housing constraint} \end{cases}$$

Mechanical setup & Mesh → **Optimization setup** → **Solution**



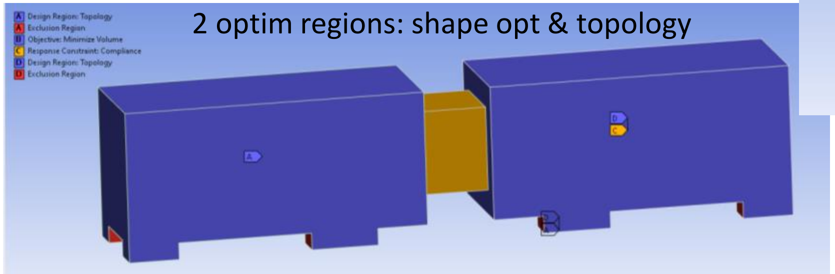
Static Analysis with prescribed displacement

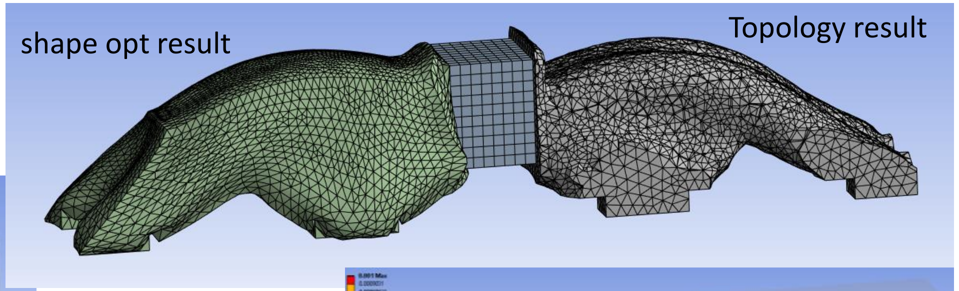


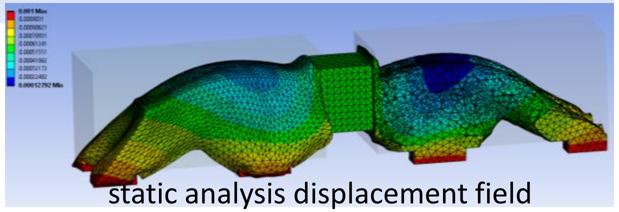
Displacement field

$$\begin{cases} \min_{\Omega} volume \\ compliance < -700 \end{cases}$$

2 optim regions: shape opt & topology







static analysis displacement field

New Features & Capabilities

new manufacturing constraint,
design constraint and more

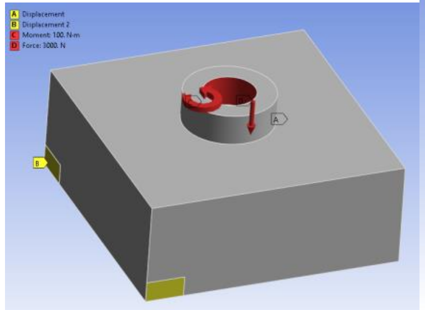
Ansys

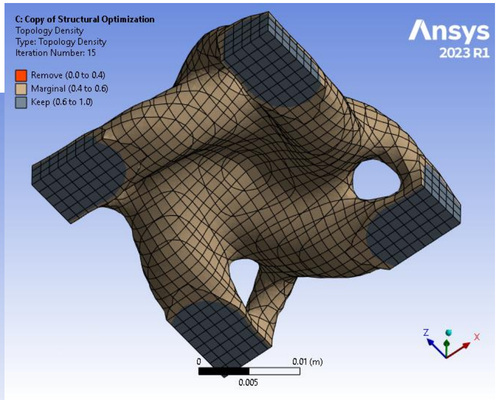
Topology Optimization – Housing constraint

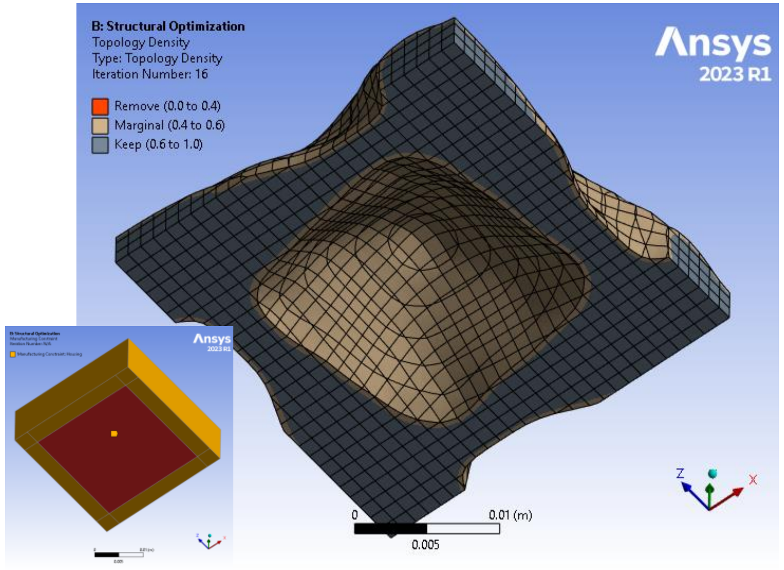
The Housing is a manufacturing constraint that has been introduced in 2022 R2.

It enables you to create a watertight design that encloses a given set of faces. Topology optimization often generates designs that include holes and perforations. Using this manufacturing constraint, you can create a container to house a given liquid.

Housing constraint is available both with Level-set and Mixable-Density Topology Optimization.





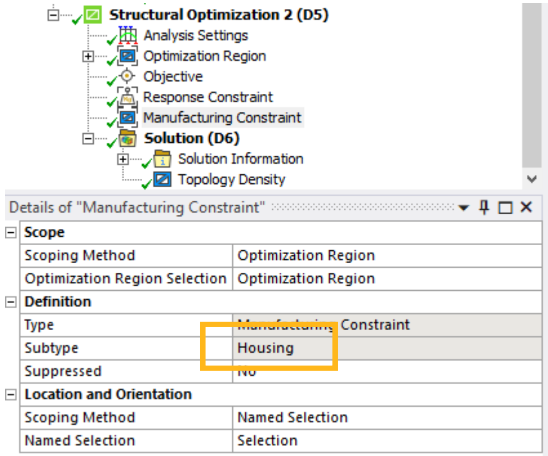


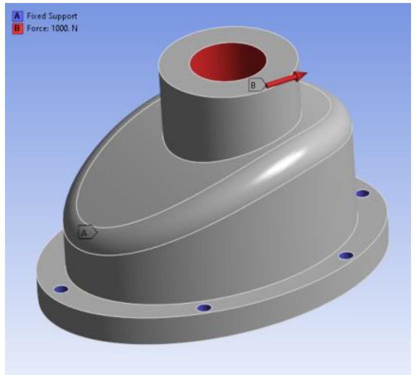
Static Analysis

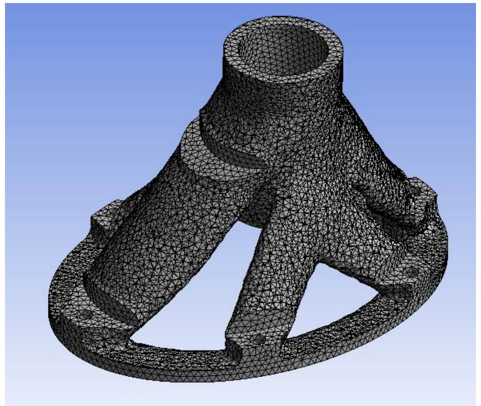
Solution **without** Housing Constraint

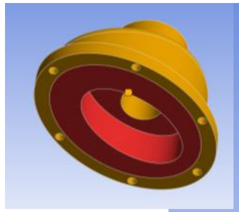
surfaces to enclose

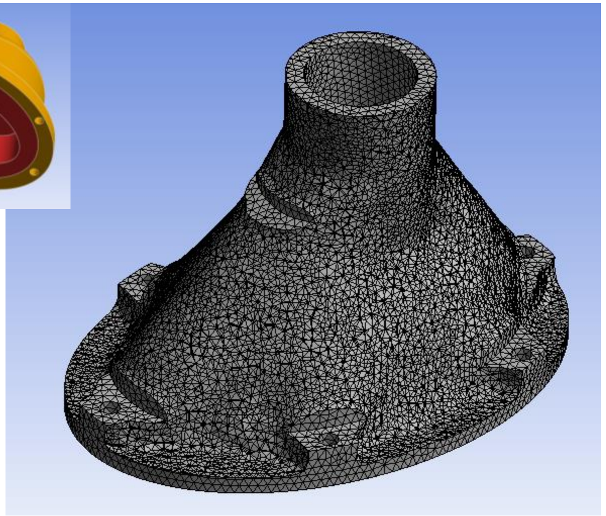
Solution **with** Housing Constraint











Shape Optimization

Design constraint

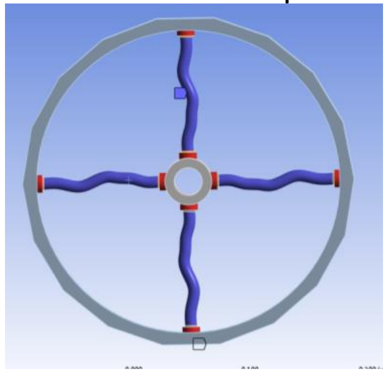
Design constraints are now available in Shape Optimization:

- cyclic symetry,
- plane symetry
- and pattern repetition (beta)

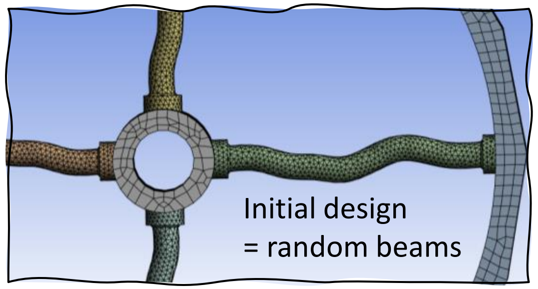
About the example:

- Wheel under static analysis
- the initial beams have been randomly designed
- how to improve to stiffness while keeping cyclic symmetry

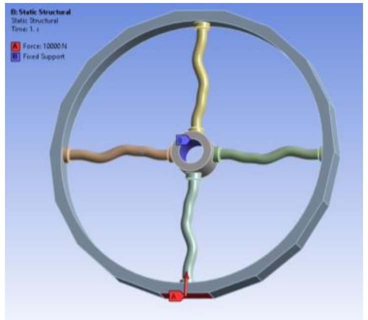
Optimizable faces



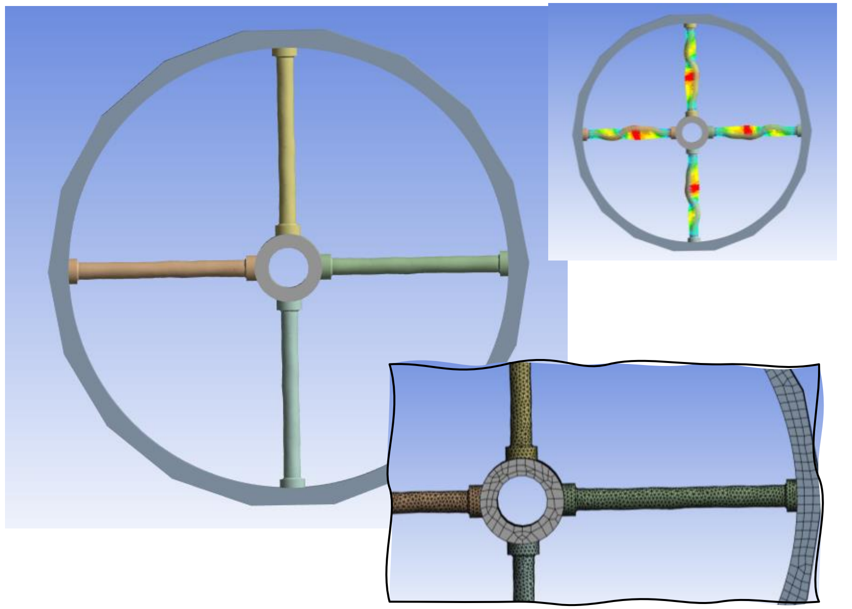
Details of "Design Constraint"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Design Constraint
Subtype	Cyclic Repetition
Suppressed	No
Location and Orientation	
Number of Sectors	4
Coordinate System	Global Coordinate System
Axis	X Axis



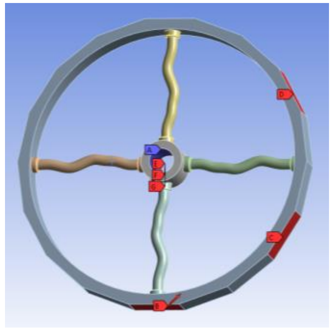
Case #1: Static Analysis, 1 loadcase



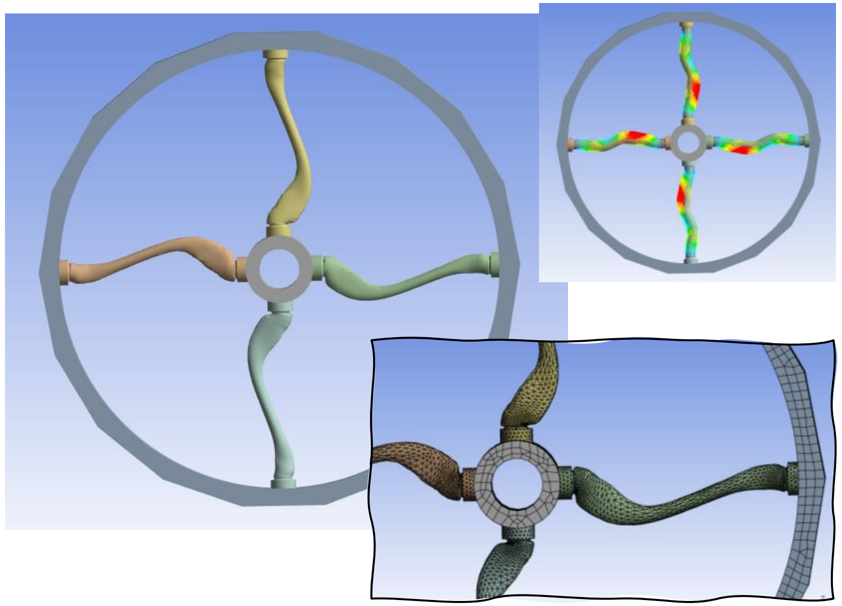
$$\begin{cases} \min\limits_{\Omega} compliance \\ mass \leq 100\% \\ +x\text{-cyclic symetry} \end{cases}$$



Case #2: Static Analysis, 6 loadsteps

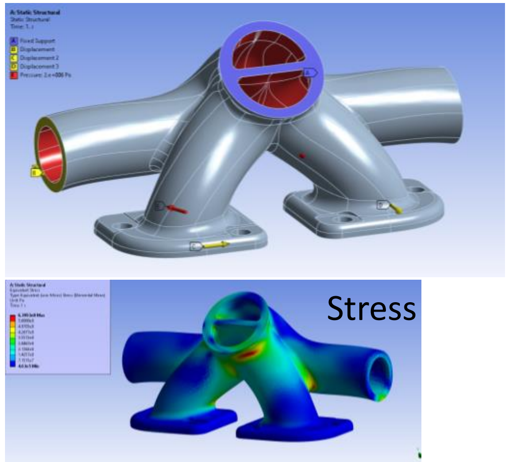


$$\begin{cases} \min\limits_{\Omega} \sum\limits_{k=1,3} Compliance_k \\ mass \leq 100\% \\ +x\text{-cyclic symetry} \end{cases}$$

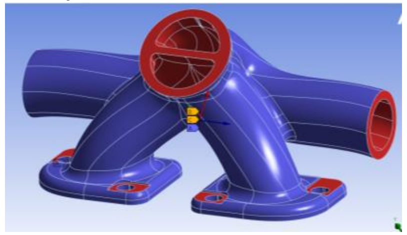


Learning by examples

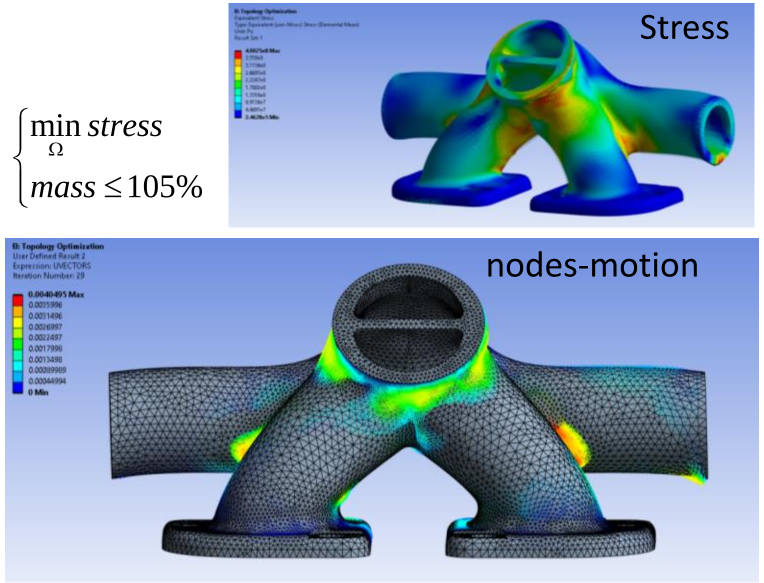
Stress-reduction for a manifold



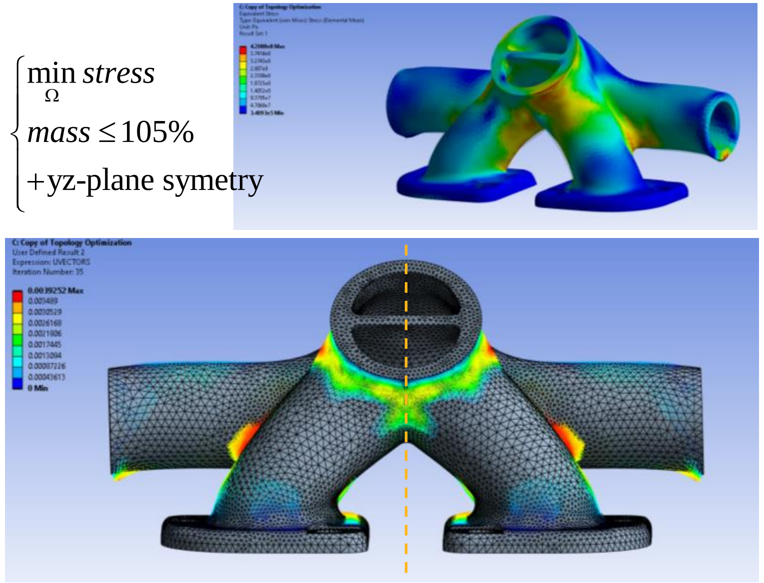
Optimizable surface



$$\begin{cases} \min_{\Omega} stress \\ mass \leq 105\% \end{cases}$$



$$\begin{cases} \min_{\Omega} stress \\ mass \leq 105\% \\ +\text{yz-plane symetry} \end{cases}$$



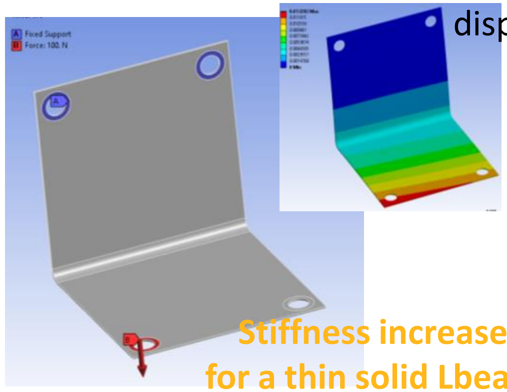
Mechanical setup & Mesh

Optimization setup

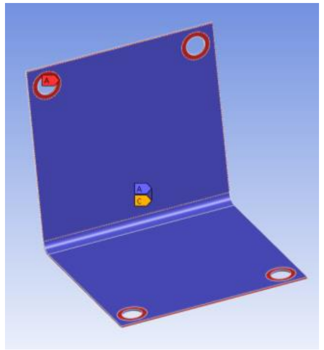
Solution

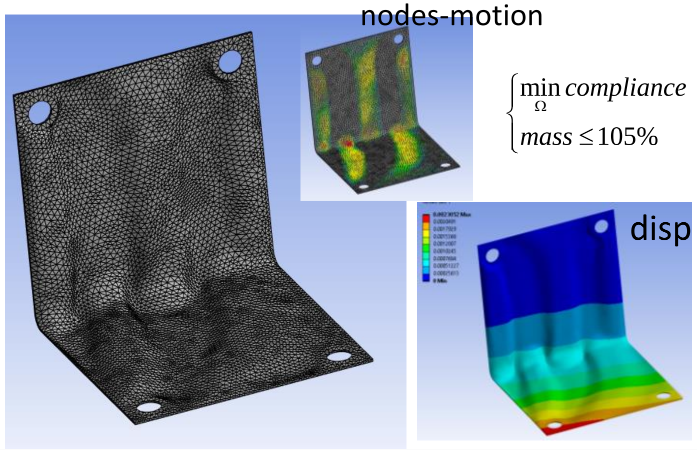
(***without*** *symetry-constraint*)

(***with*** *Symetry*)

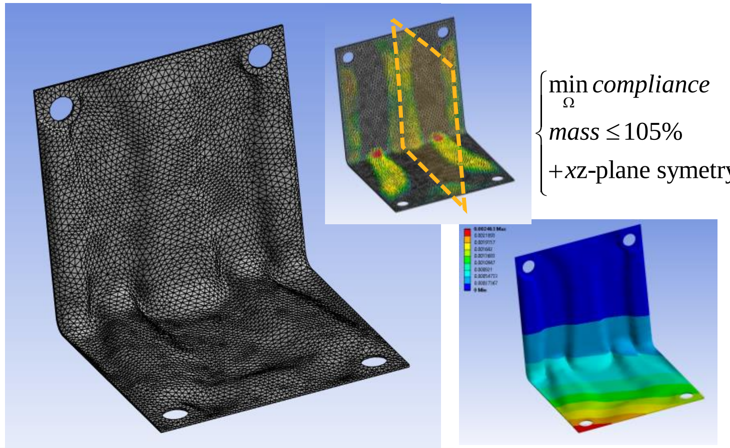


Stiffness increase for a thin solid Lbeam





$$\begin{cases} \min_{\Omega} compliance \\ mass \leq 105\% \end{cases}$$

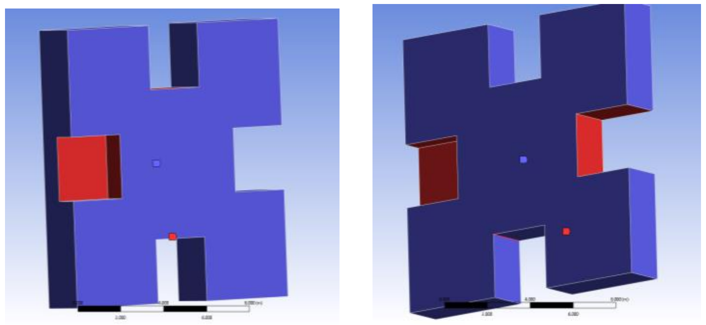


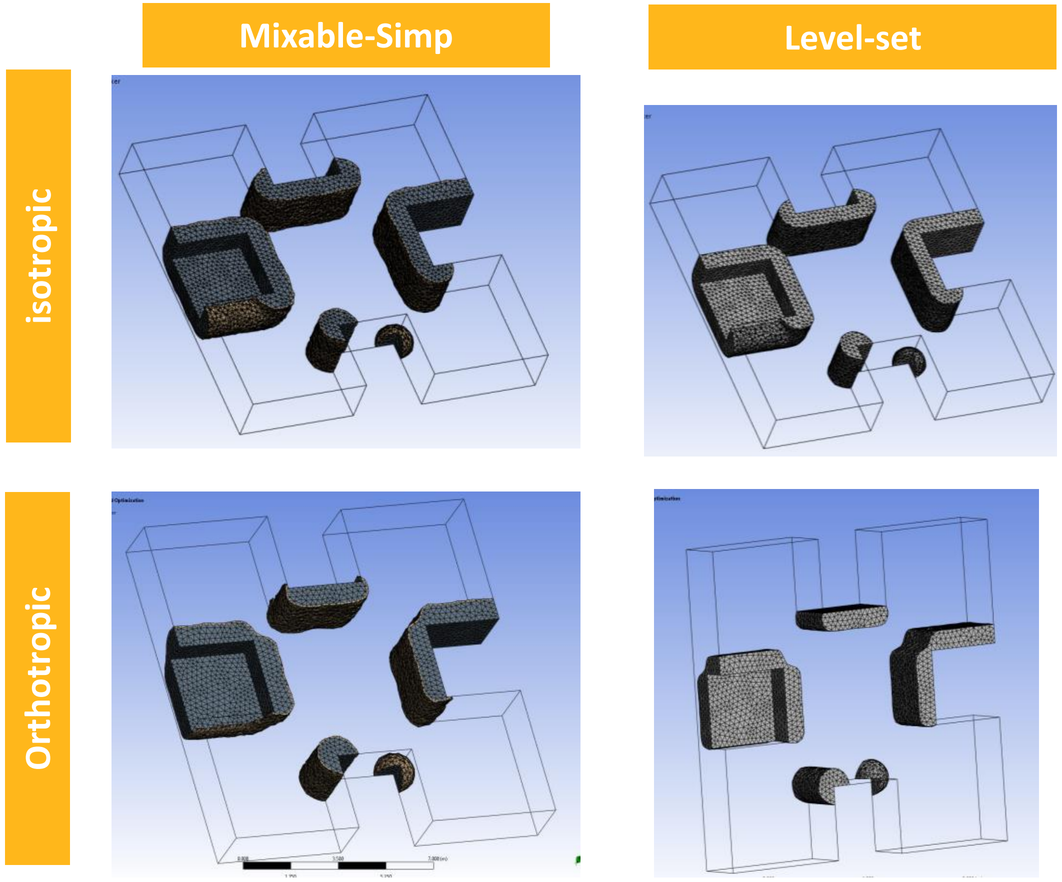
$$\begin{cases} \min_{\Omega} compliance \\ mass \leq 105\% \\ +\text{xz-plane symetry} \end{cases}$$

Exclusion Region: Special Extension

The **Exclusion Region** properties enable you to specify a region to be excluded from optimization. Since 22r2, the thickness is editable in Mixable-Simp and Level-set Topology optimization.
In 23r1, you can also choose the extension type:
-**Isotropic**: expansion in every direction from the selected surfaces
-**Orthotropic**: expansion along the surface normal. It is available when selecting a surface. For edge or node, it is Isotropic only.

Regions to exclude
(in red)





Details of "Optimization Region 2"

Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Boundary Conditions
Exclusion Thickness	Program Controlled
Exclusion Extension	Orthotropic
Definition	
Suppressed	No
Optimization Option	
Optimization Type	Topology Optimization - Level Set Based
Initialization Modification (Beta)	None

Improvement & Corrections

✓ **Mixable-Density Topology Optimization**

- Thermal-load in Static Analysis is supported in 23r1. The handling is fully consistent between Level-set, Mixable-Simp, Shape opt. and Topography.
- Correction in the computation of the derivative for static criteria in the presence of acceleration.
- Three advanced parameters are available :
  "initial volume fraction",
  "penalty factor" and "hyperbolic projection"

Details of "Optimization Region"

Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Supports
Exclusion Extension	Isotropic
Definition	
Suppressed	No
Optimization Option	
Optimization Type	Topology Optimization - Mixable Density
Initial Volume Fraction	Program Controlled
Penalty Factor (Stiffness)	Program Controlled
Hyperbolic Projection	Program Controlled

✓ **Level-set Topology Optimization**

- Many improvements have been introduced leading to a faster machinery. The computation per iteration is more than 2x faster.

✓ **Multi Optim Type Strategy**

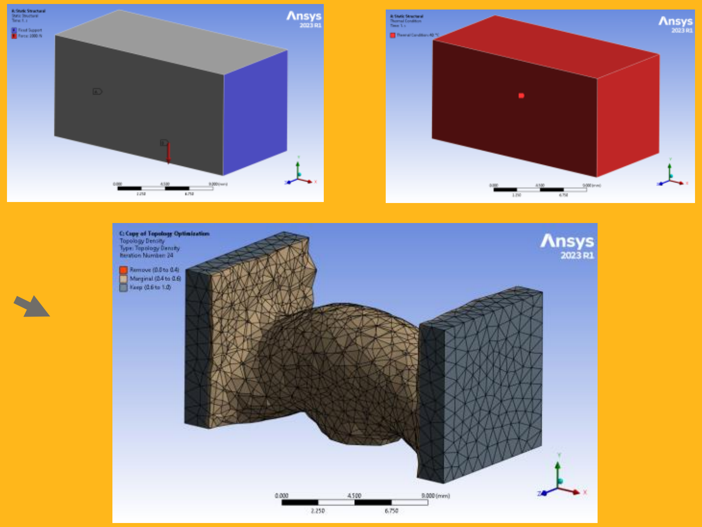
- A special tuning has been introduced to handle the mix of optimization methods. This new tuning is automatically activated for this context
- It can also be used for single optim method.
  It will affect the behavior of Level-Set and Shape Optimization.
- The strategy is more agressive and leads to better and faster results, but possibly less smooth.

Topology Optimization (B5) LVLS TOpt (B5)
- Analysis Settings
- Optimization Region
- Objective
- Response Constraint
- Commands (TOPO) (Beta)

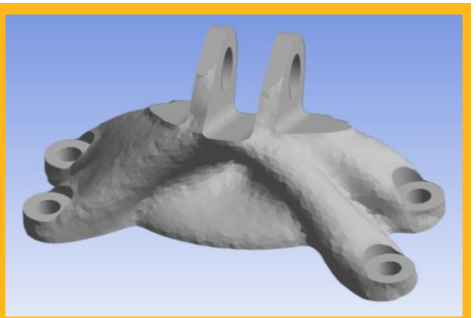
Details of "Analysis Settings"

Definition	
Maximum Number Of Iterations	500.
Convergence Accuracy	0.1 %
Output Controls	
Solver Controls	
Solver Type	Program Controlled
Multi Optim Type Strategy	On
Algorithm	Program Controlled
Analysis Data Management	

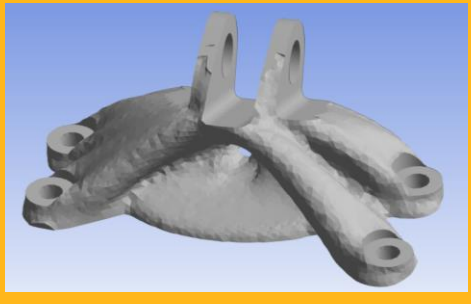
Thermal load with Mix-Simp



Multi Optim Strategy = off
nb iterations: 61
final compliance: 0.340J



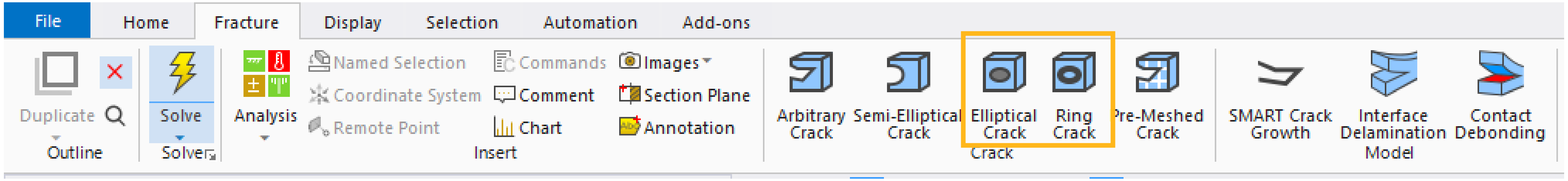
Multi Optim Strategy = ON
nb iterations: 36
final compliance: 0.338j



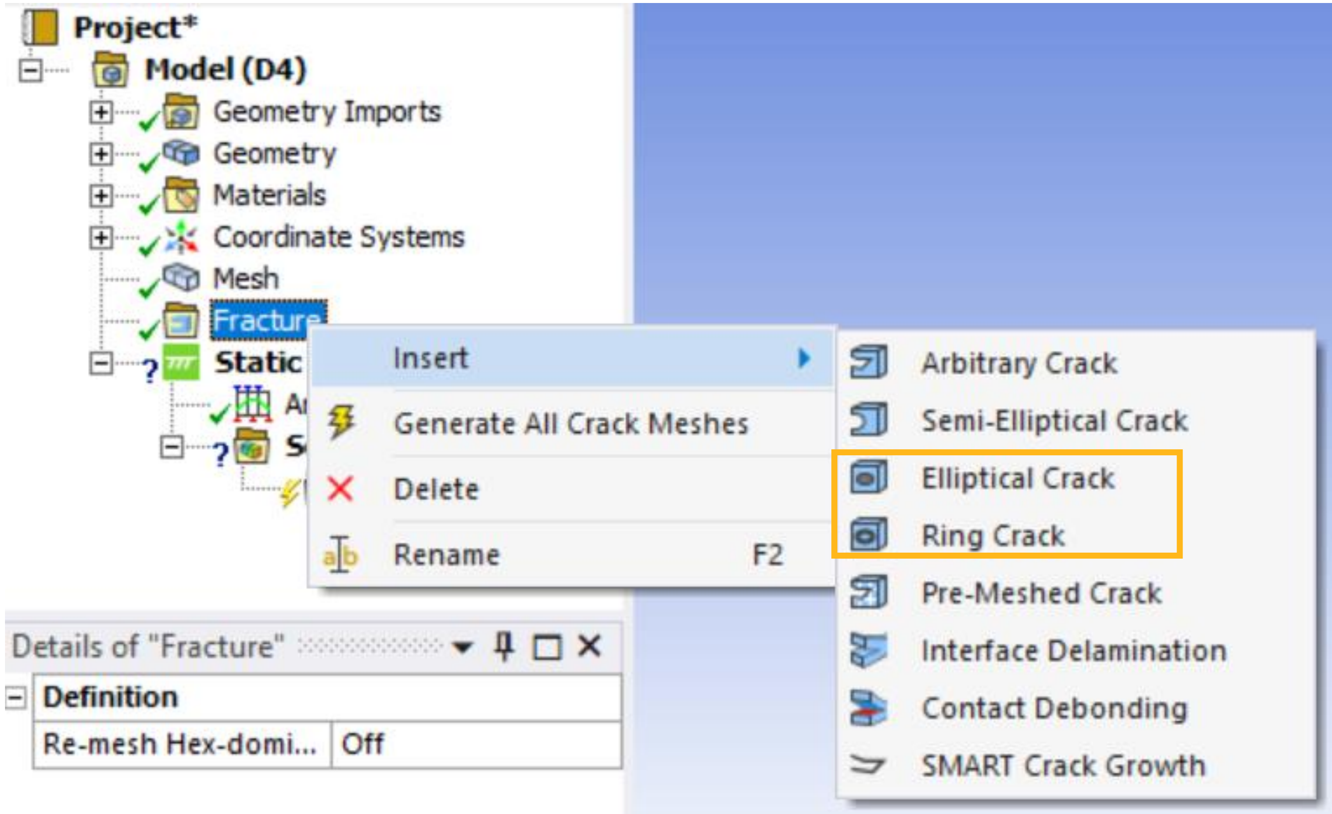
Ansys

Fracture Enhancements

Fracture Enhancements for 2023 R1

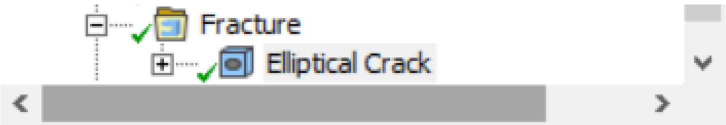


- Mechanical supports embedded analytical cracks of shape Elliptical and Ring inside a solid body
- Using Arbitrary Crack, Mechanical supports Through cracks, Embedded cracks and cracks intersecting at multiple corners
- Fracture results can be evaluated either for all crack fronts or a particular crack front associated to crack object



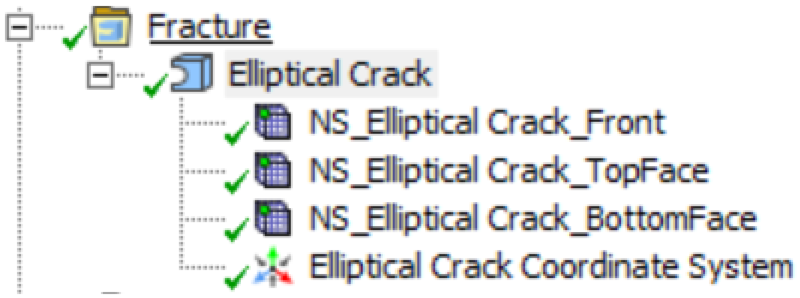
Analytical Crack : Elliptical Crack

- Mechanical can now define an embedded analytical crack of elliptical shape using the Coordinate system and the Major Radius and Minor Radius

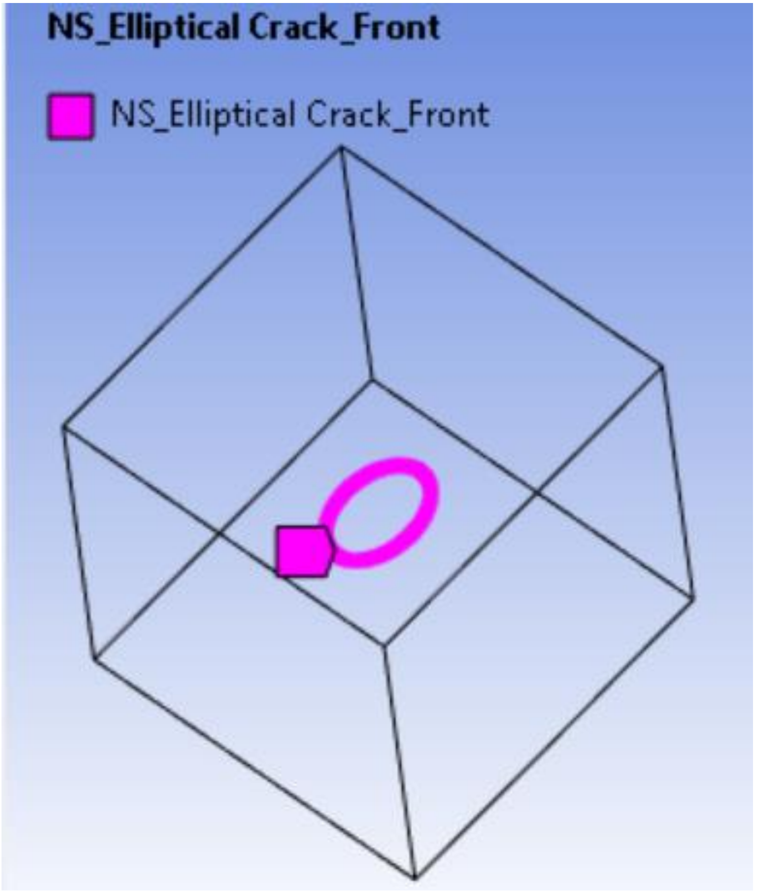


Details of "Elliptical Crack"

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	88
Coordinate System	Coordinate System
Crack Shape	Elliptical
--Major Radius	3. mm
--Minor Radius	2. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	1. mm
Growth Rate	Default (1.2)
Front Element Size	Default (0.10071 mm)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No

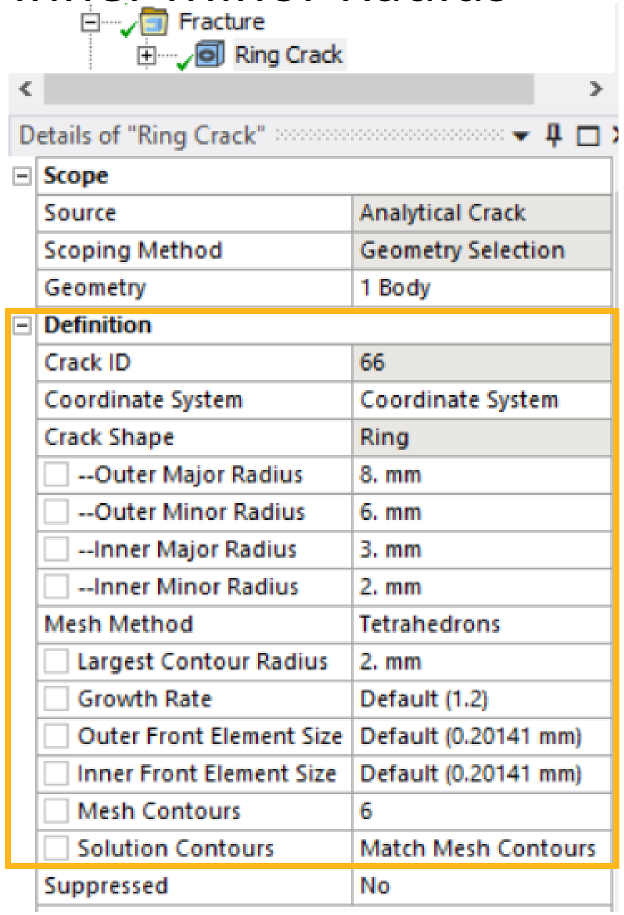


Mesh generates one elliptical crack front

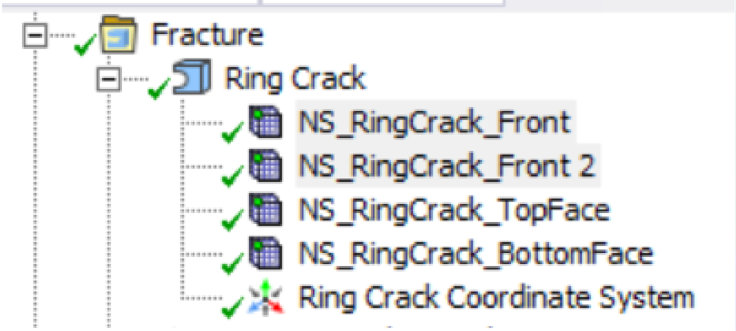


Analytical Crack : Ring Crack

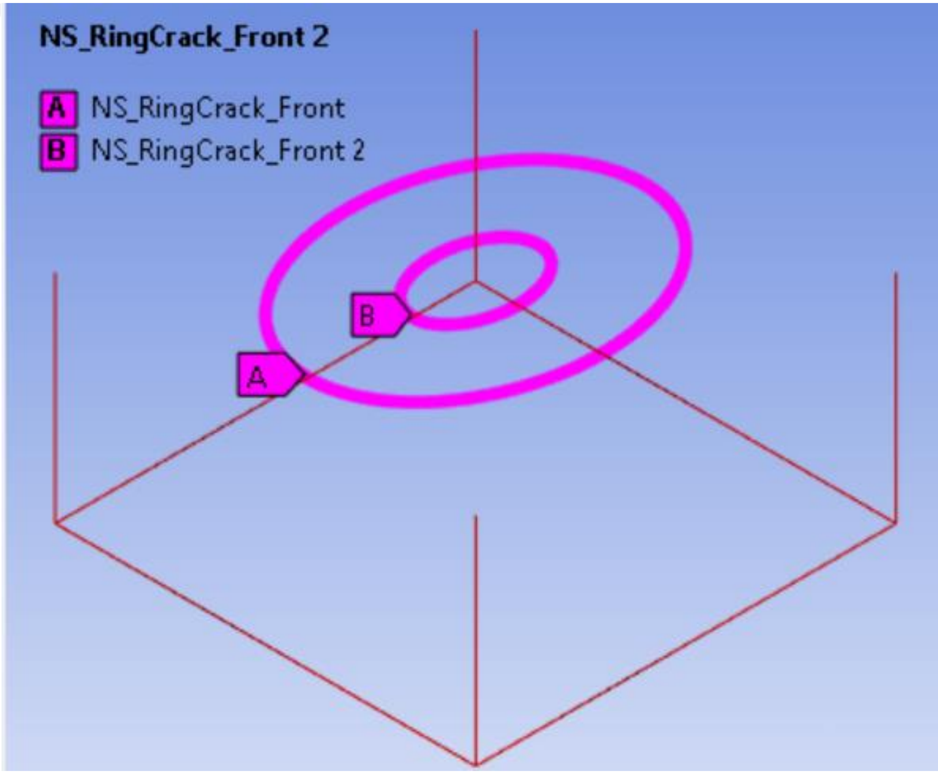
- Mechanical can now define an analytical crack of ring shape using the Coordinate system and the Outer Major Radius and Outer Minor Radius, Inner Major Radius and Inner Minor Radius



Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Crack ID	66
Coordinate System	Coordinate System
Crack Shape	Ring
--Outer Major Radius	8. mm
--Outer Minor Radius	6. mm
--Inner Major Radius	3. mm
--Inner Minor Radius	2. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	2. mm
Growth Rate	Default (1.2)
Outer Front Element Size	Default (0.20141 mm)
Inner Front Element Size	Default (0.20141 mm)
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No



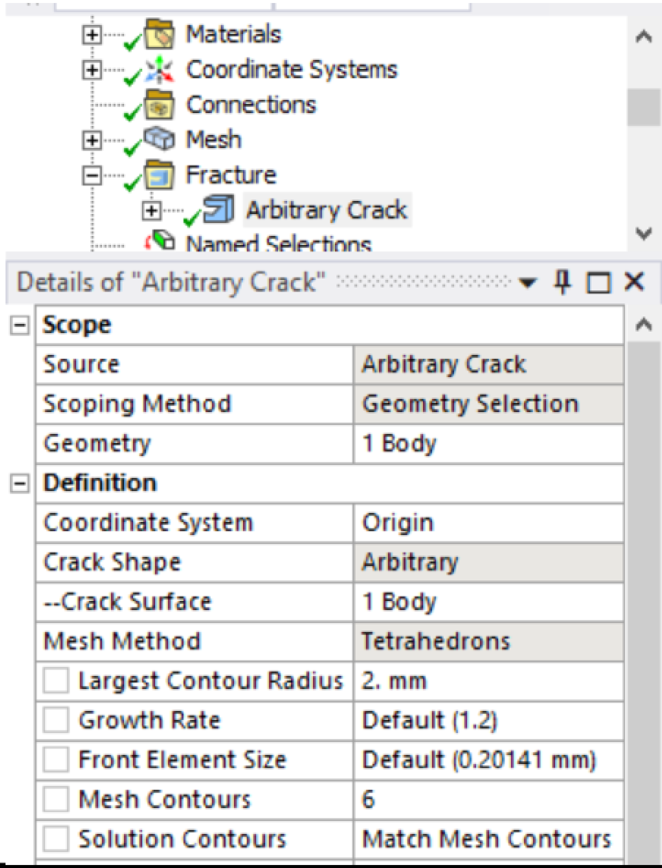
Mesh generates two crack fronts

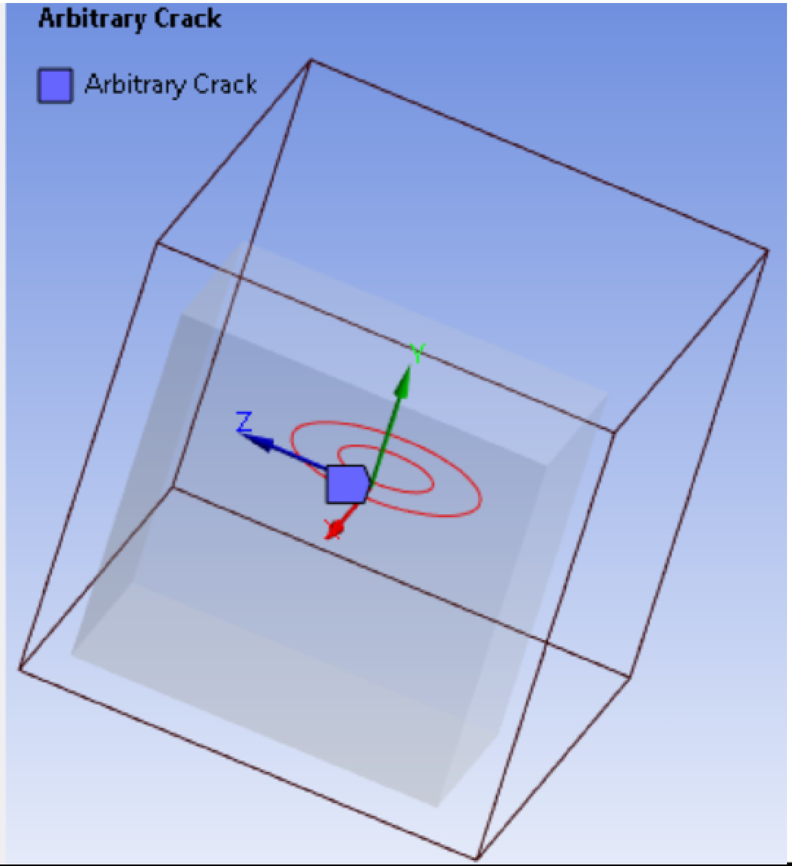


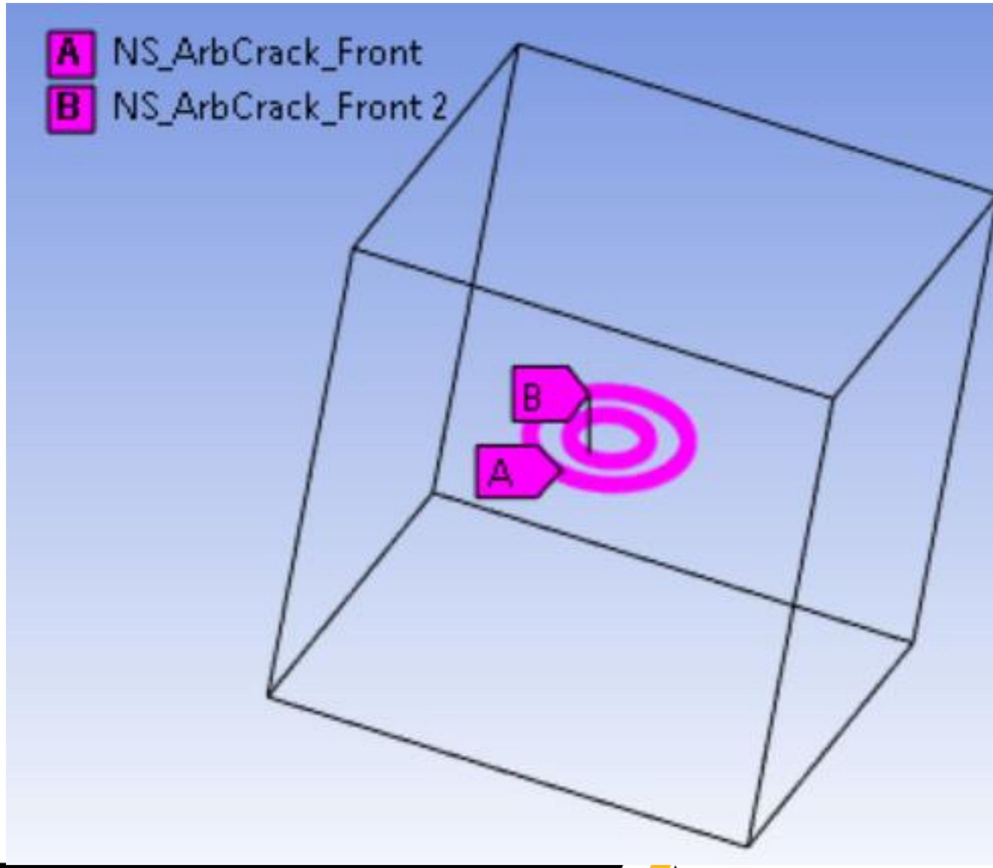
Ansys

Arbitrary Crack - Elliptical and Ring shaped

- Mechanical supports Arbitrary crack where crack mesh can be generated on an elliptical or ring shape crack surface body and embedded crack related fracture parameters can be computed

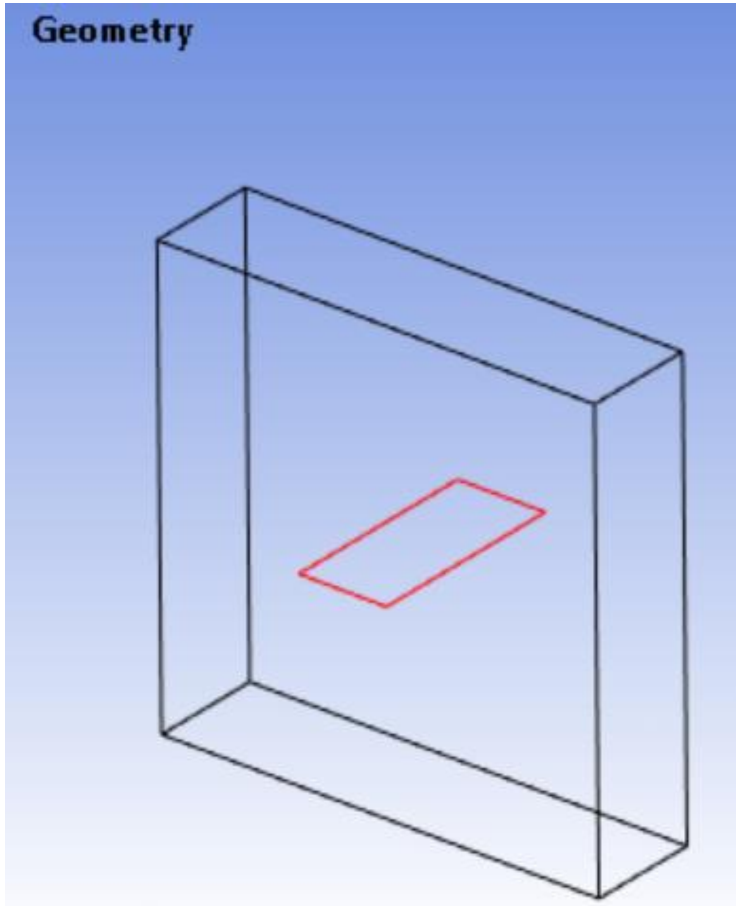




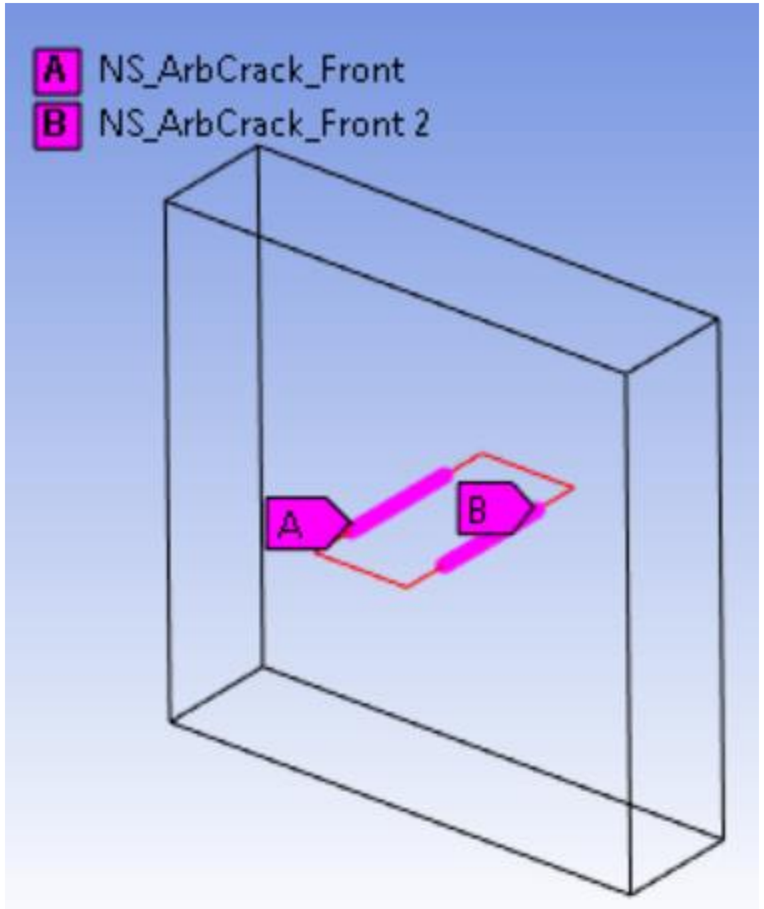


Arbitrary Crack - Through Crack

- A crack surface body which cuts through the solid body will generate two crack fronts

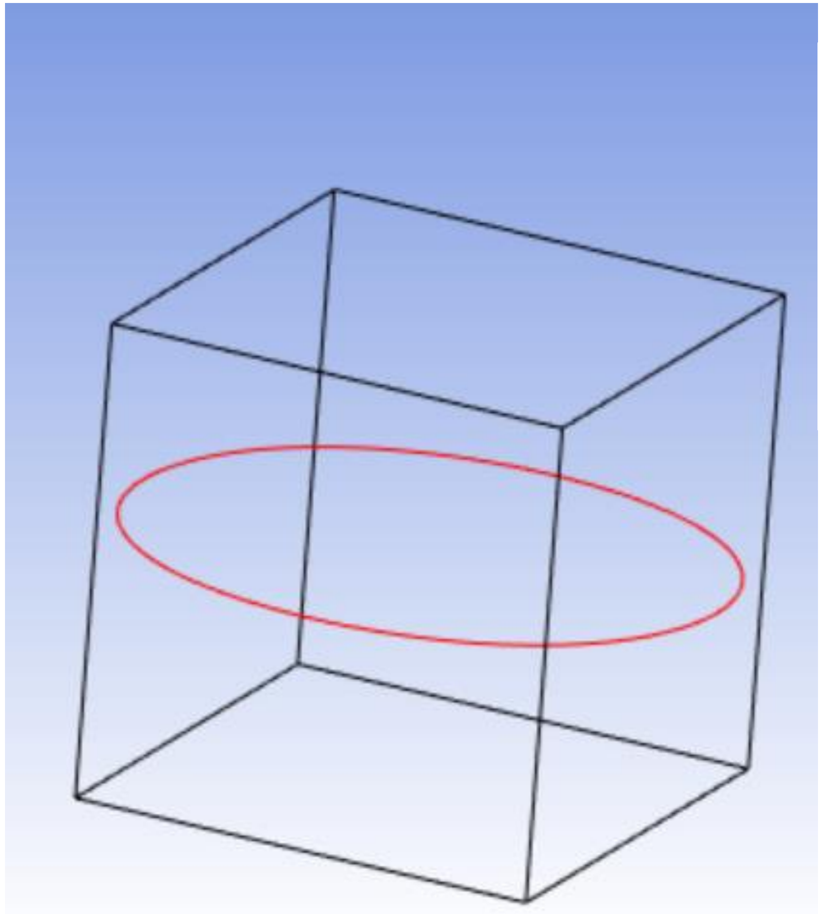


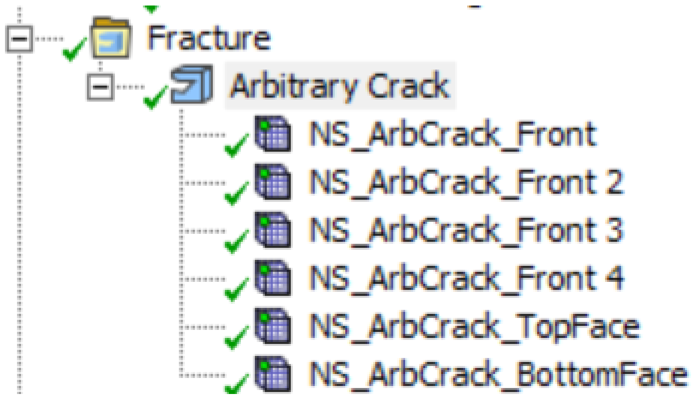
Mesh generates two crack fronts



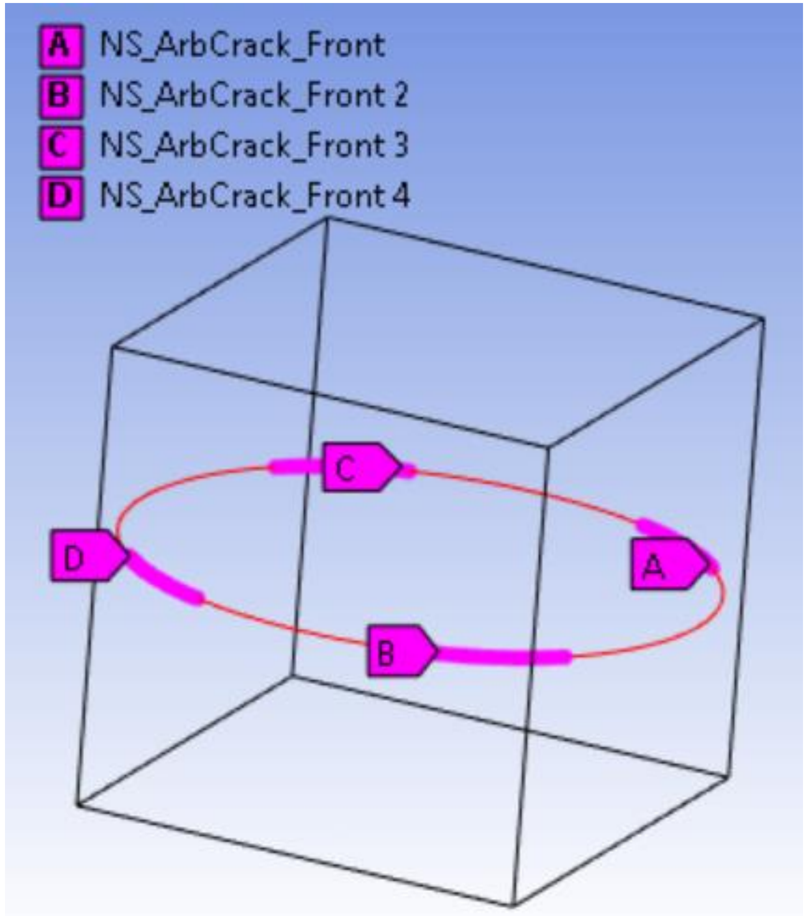
Arbitrary Crack – Intersecting at multiple corners

- A crack surface body which intersects the free surfaces of the solid body at corners creates multiple crack fronts





Mesh generates four crack fronts



Crack Front Number property on named selections

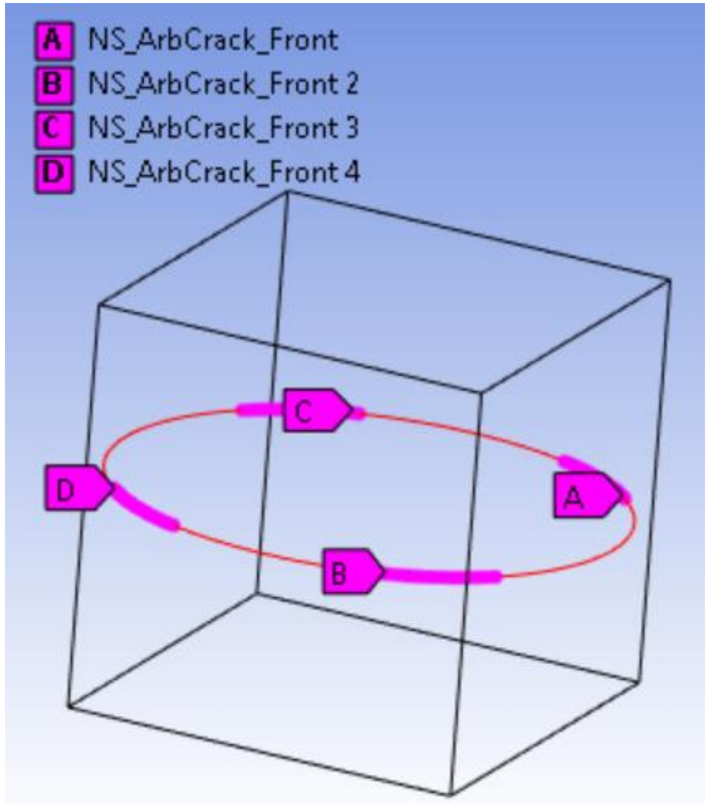
- Crack front named selections are numbered in the same order as they are created, and this number is used while post processing, as **Crack Front Number** property of Fracture Tool to extract fracture results and probes on a specific crack front

Details of "NS_Elliptical Crack_Front"

Scope	
Geometry	316 Nodes
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Generated
Total Selection	316 Nodes
Suppressed	0
Used by Mesh Worksheet	No
Created For Crack	Elliptical Crack
-- Crack Front Number	1

Details of "NS_Elliptical Crack_TopFace"

Scope	
Geometry	3255 Nodes
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Generated
Total Selection	3255 Nodes
Suppressed	0
Used by Mesh Worksheet	No
Created For Crack	Elliptical Crack



Details of "Fracture Tool"

Scope	
Scoping Method	Crack Selection
Crack Selection Mode	Single Crack
Crack Selection	Arbitrary Crack
Crack Front Number	1
Definition	

Details of "Fracture Tool 4"

Scope	
Scoping Method	Crack Selection
Crack Selection Mode	Single Crack
Crack Selection	Arbitrary Crack
Crack Front Number	4
Definition	

Unique solver ids for fracture parameters

- We create and send unique solver ids for each crack front, we create like:

_CRACK66_FRONT1_SIFS,1

_CRACK66_FRONT1_JINT,2

_CRACK66_FRONT2 _SIFS,3

_CRACK66_FRONT2 _JINT,4

Which can be used to post process fracture results through command snippets

Fracture
Ring Crack

Details of "Ring Crack"

Scope	
Definition	
Crack Id	66
Coordinate System	Coordinate System
Crack Shape	Ring
--Outer Major Radius	6. mm
--Outer Minor Radius	4. mm
--Inner Major Radius	3. mm
--Inner Minor Radius	2. mm
Mesh Method	Tetrahedrons
Largest Contour Radius	2. mm
Growth Rate	Default (1.2)
Front Element Size	Default (0.20141 mm)
Mesh Contours	6
Solution Contours	Match Mesh Conto...
Suppressed	No

Details of "NS_RingCrack_Front"

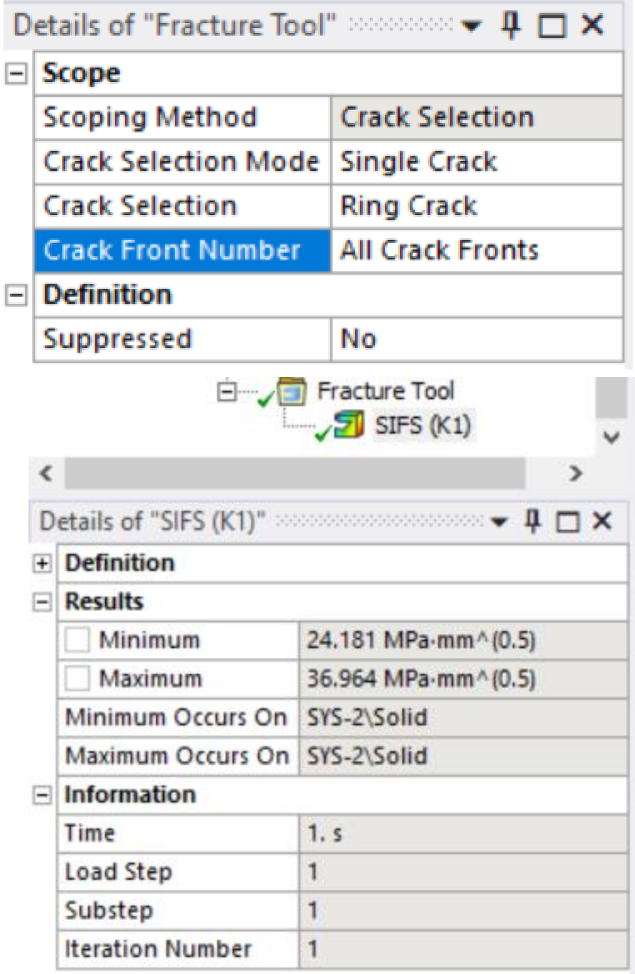
Scope	
Definition	
Statistics	
Type	Generated
Total Selection	158 Nodes
Suppressed	0
Used by Mesh Worksheet	No
Created For Crack	Ring Crack
-- Crack Front Number	1

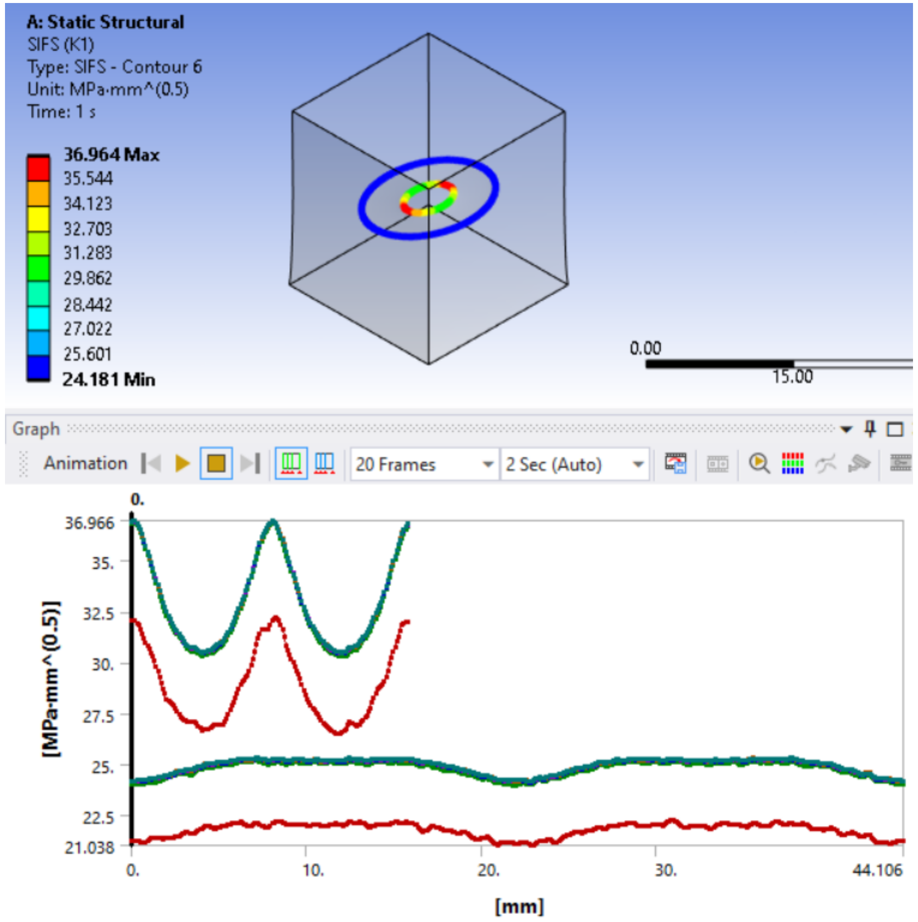
Details of "NS_RingCrack_Front 2'

Scope	
Definition	
Statistics	
Type	Generated
Total Selection	316 Nodes
Suppressed	0
Used by Mesh Worksheet	No
Created For Crack	Ring Crack
-- Crack Front Number	2

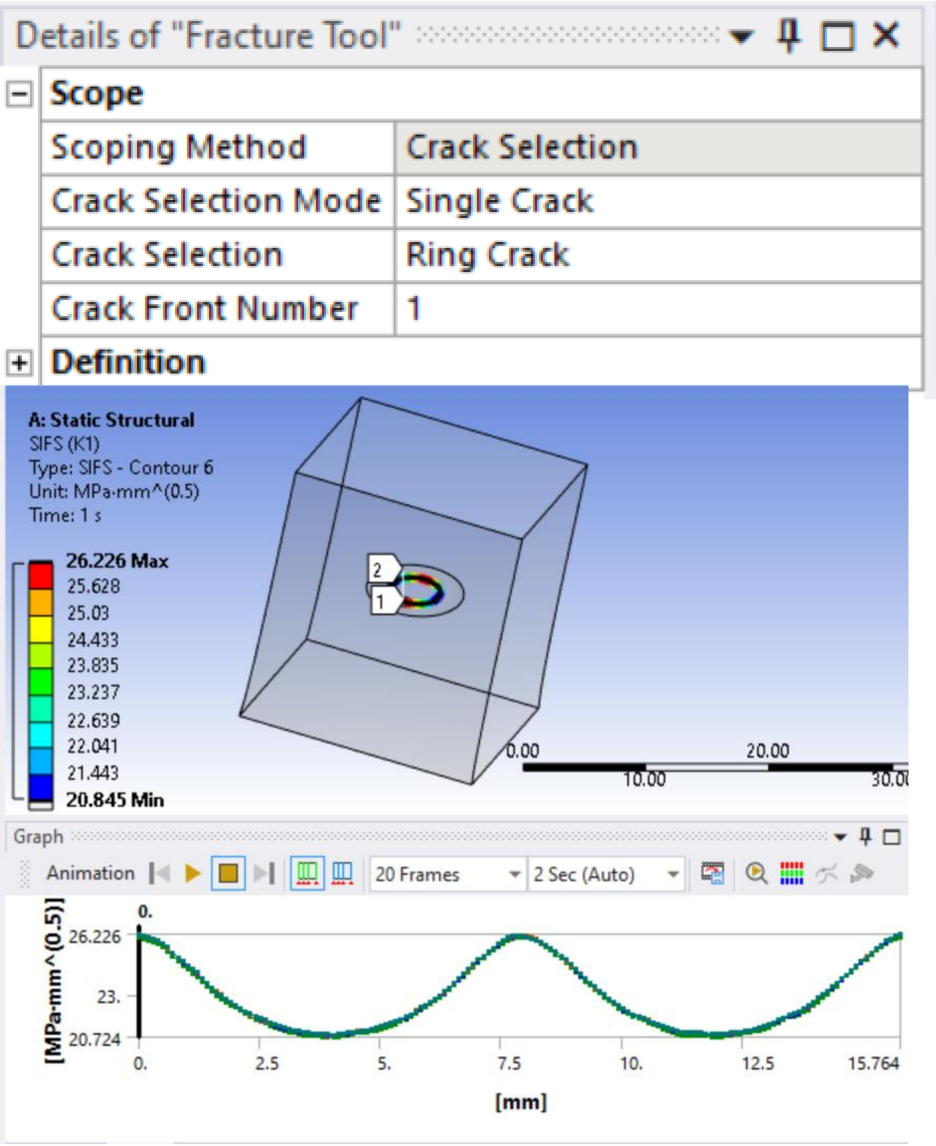
Fracture results for all crack fronts

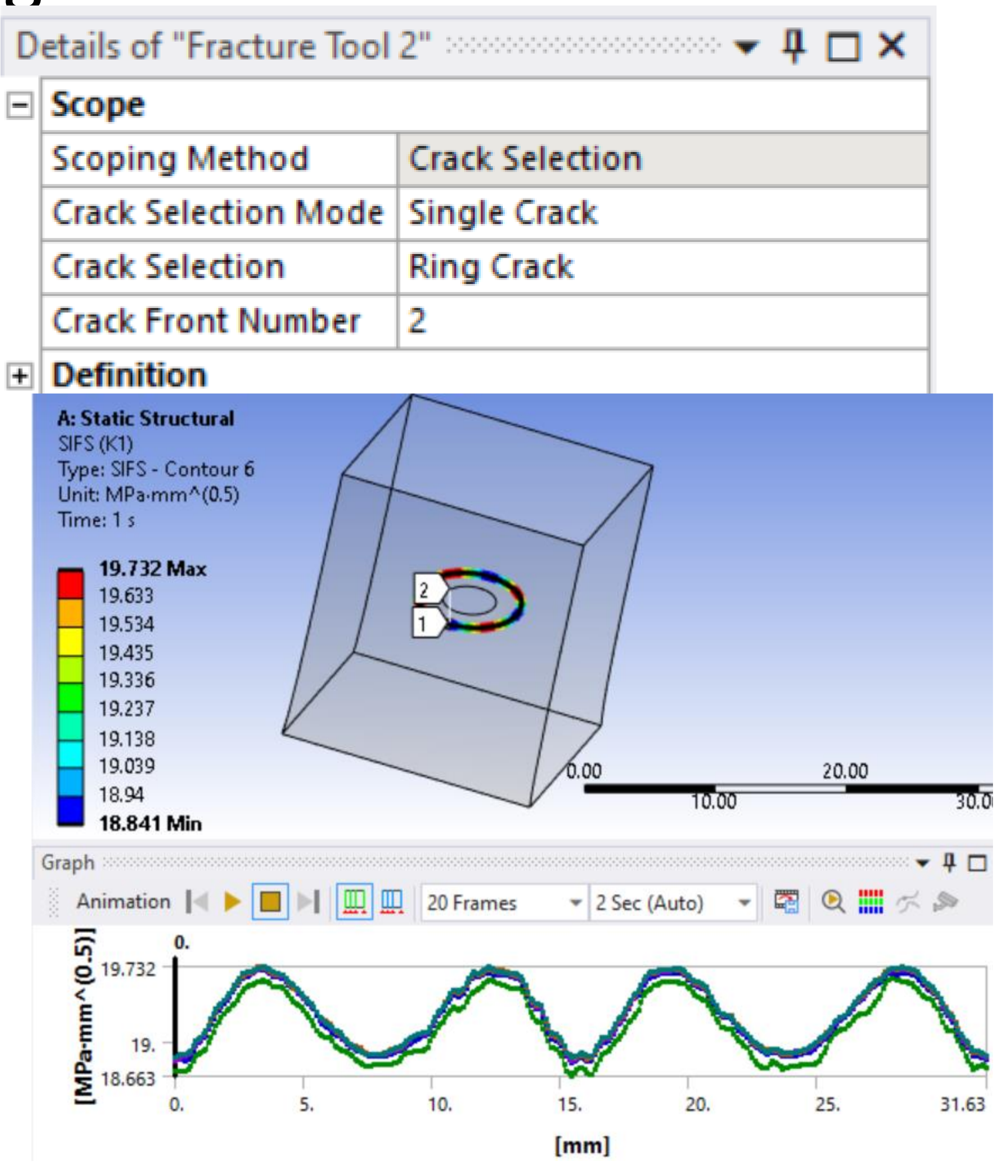
All Crack Fronts will be the default option for Crack Front Number. This option plots the results of all crack fronts and displays cumulative results





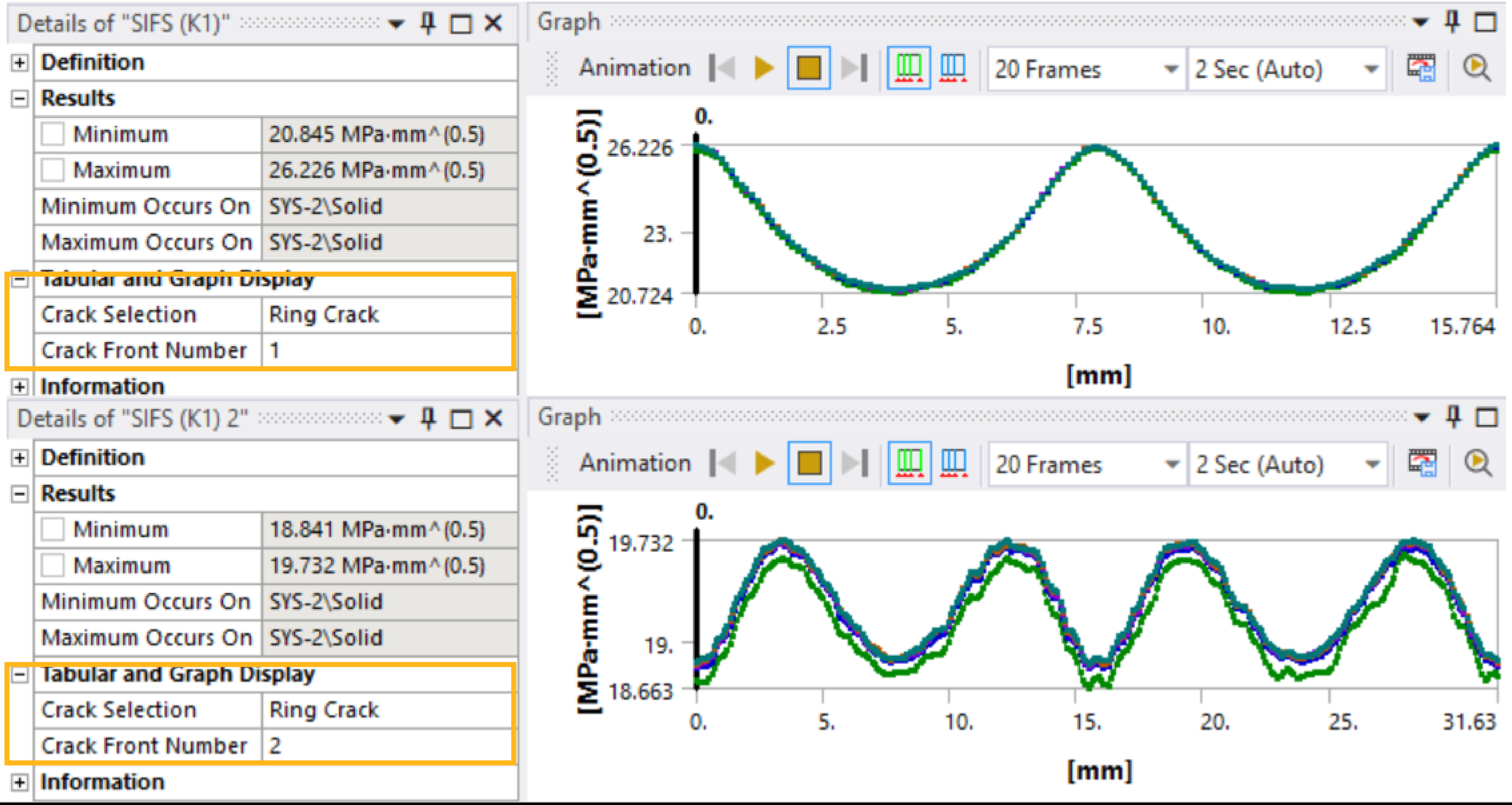
Results on specific crack front using Crack Front Number





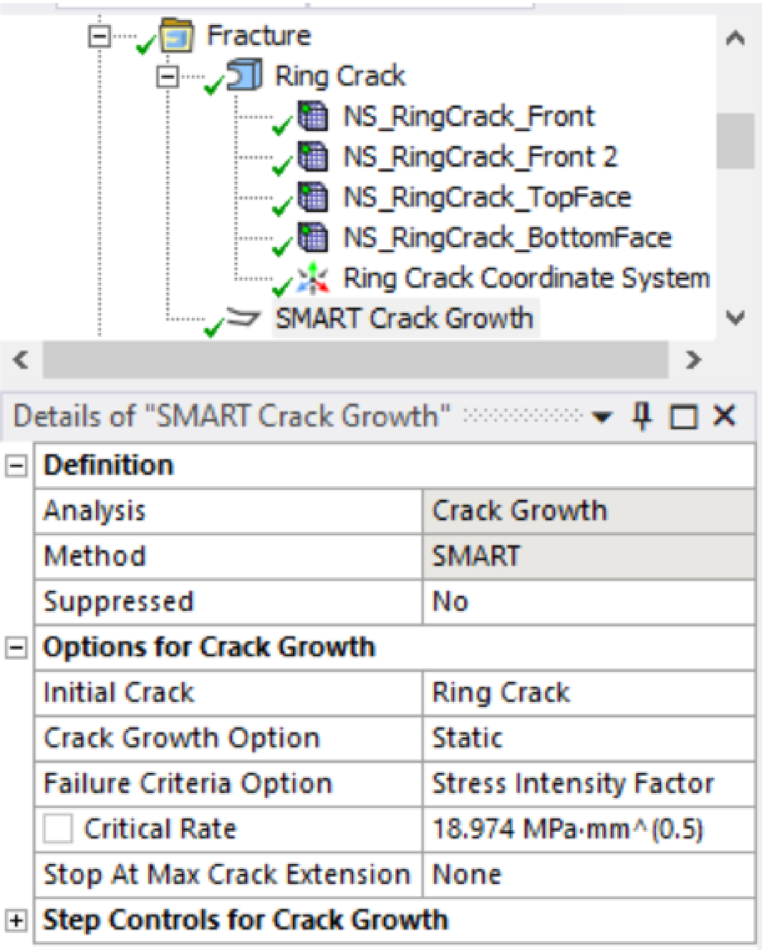
Results - All Cracks selection mode on Fracture Tool

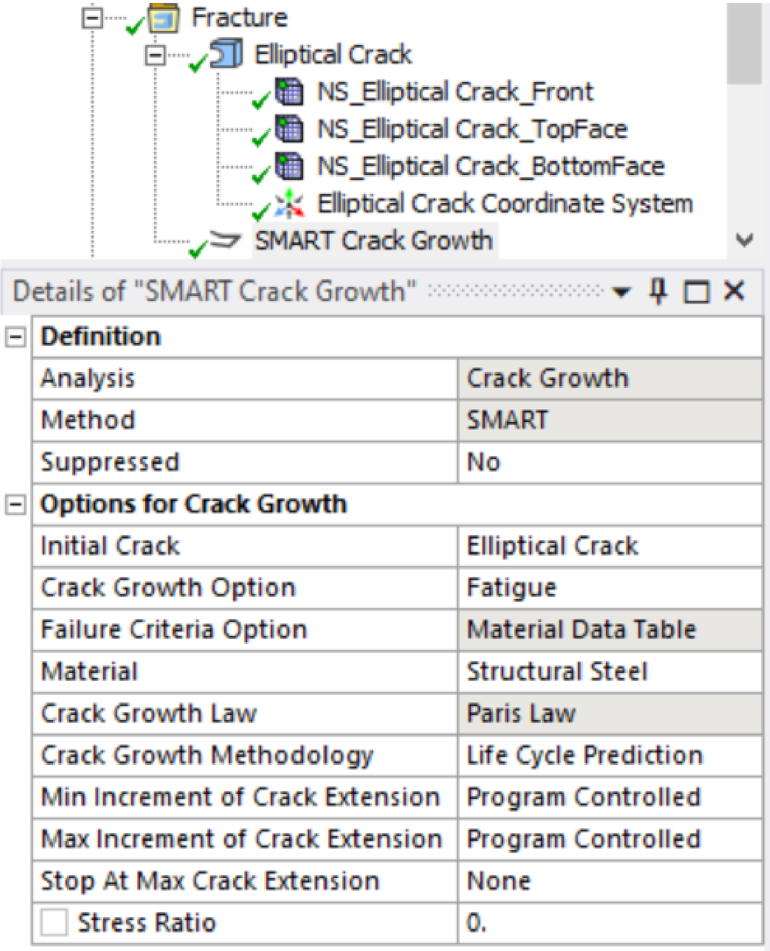
- When All Cracks selection mode is selected on the Fracture Tool, you can specify the Crack Front Number below Crack Selection to plot the graph and table of fracture results on a specific crack front of the selected crack



SMART Crack Growth support for new crack shapes

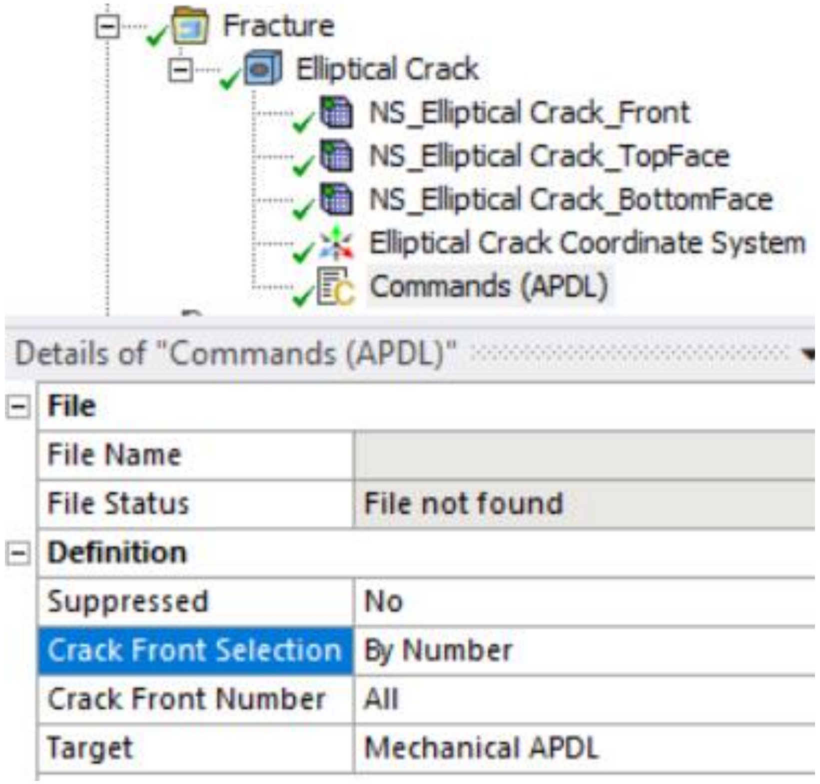
- SMART Crack Growth can be defined for these new crack types to study the crack growth propagation and related fracture results and probes





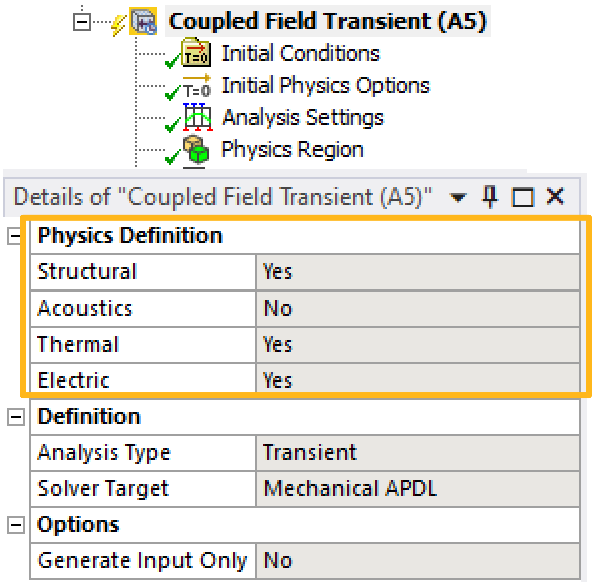
Commands (APDL) - For specific crack front

The Commands (APDL) object provides a **Crack Front Selection** property that enables you to execute commands specific to a crack front. Property options include **All** (default), **First**, **Last**, and **By Number**. When you select the **By Number** option, an additional property displays: **Crack Front Number**.

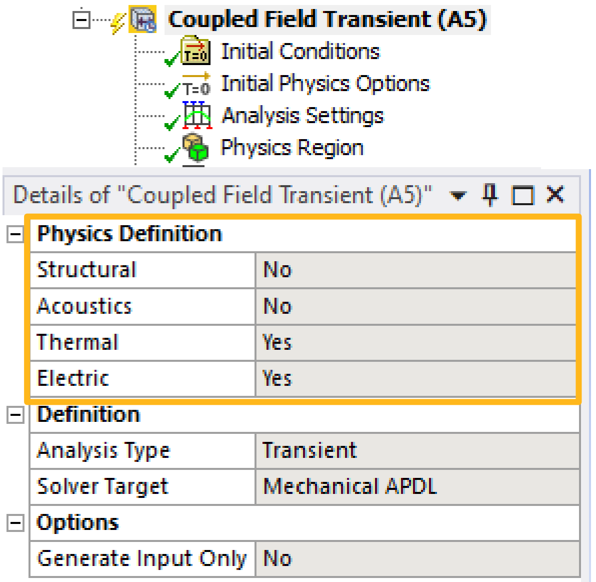


Coupled Field Features

Electric Conduction in Coupled Field Transient



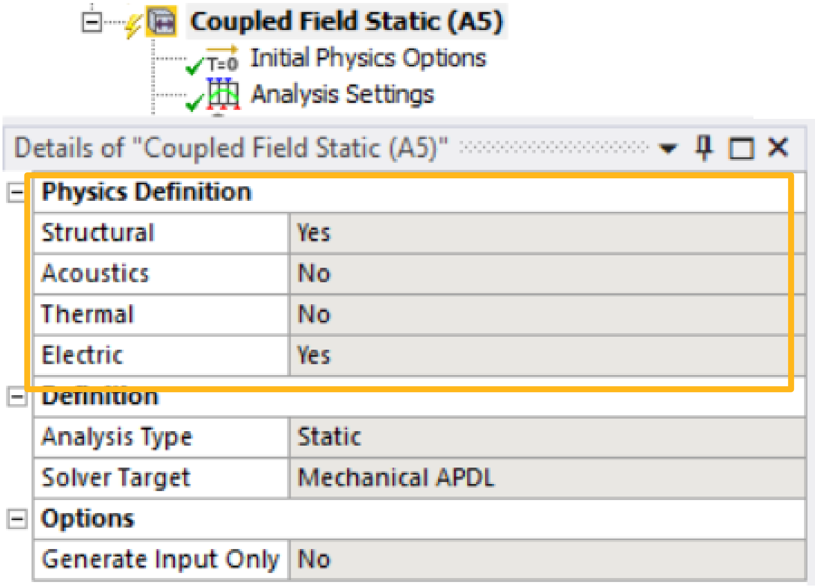
- Electric Conduction is now supported in Coupled Field Transient
- Coupling of Thermal and Electric Conduction can be performed
- Coupling of Structural and Thermoelectric Conduction can be performed
- Applications include
  - Joule heating in various application
  - Thermocouple design
  - Thermoelectric Micro-actuators
  - Many others



Ansys

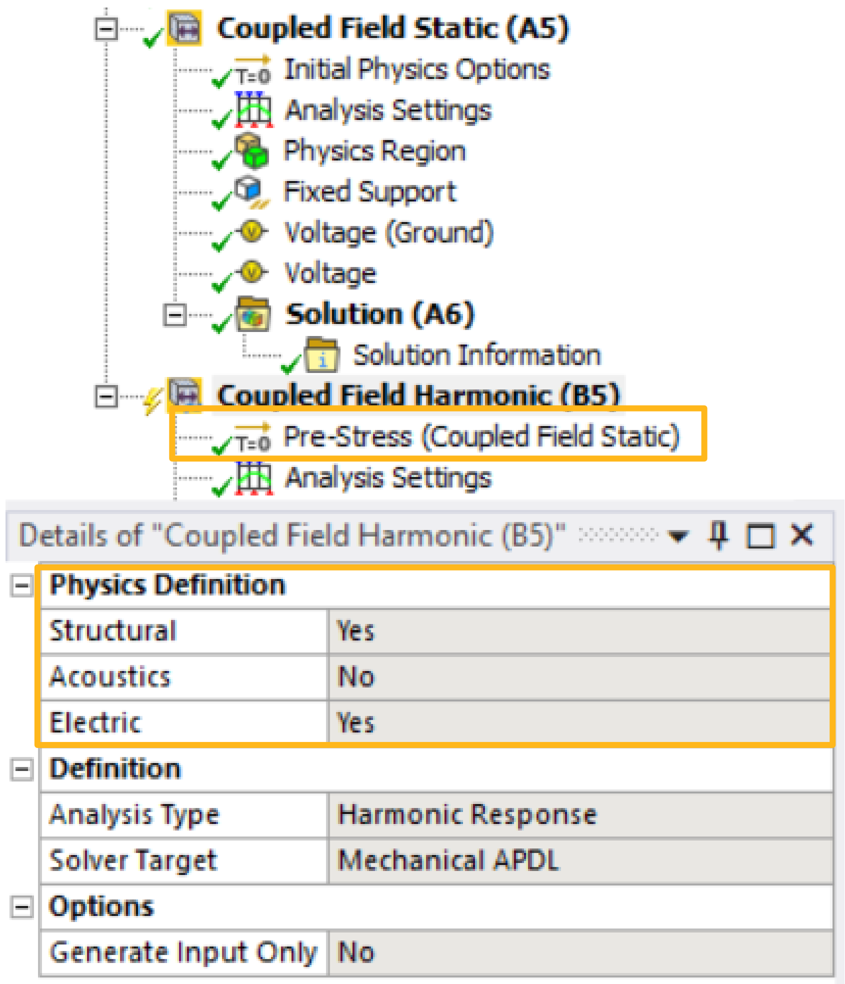
Electrostatic Force Coupling in Coupled Field Static

- Electrostatic Structural analysis with Electrostatic Force coupling is now supported in Coupled Field Static
- This coupling is also supported with
  - Electrostatic Structural coupling with Piezoelectric
  - Electrostatic Structural coupling with Acoustics
- Applications include
  - Electroactive polymer actuators
  - Electrostatic micro-electromechanical-mechanical devices (MEMS) such as electromechanical switches, sensors and actuators
  - Comb drives, accelerometers, torsional
  - Micromirrors and Gyroscopes



Electrostatic Force Coupling in Prestressed Coupled Field Harmonic

- Prestressed Coupled Field Harmonic analysis can be performed with these couplings:
  - Electrostatic Structural coupling
  - Electrostatic Structural coupling with Piezoelectric
  - Electrostatic Structural coupling with Acoustics
- These couplings need to be defined in the upstream Coupled Field Static



4. General Miscellaneous Enhancements

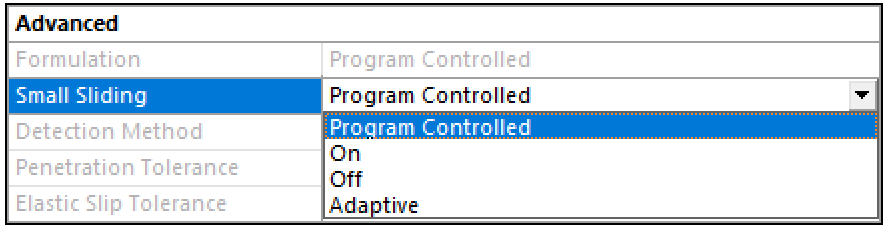
- Acoustics PML is supported in Coupled Field Transient
- Charge Residuals supported on Solution information object. Charge residuals can be plotted and viewed for nonlinear solutions that either do not converge or that where aborted during the solution.
- System Coupling is now supported with Coupled Field when either Structural or Thermal physics are present or when both physics are present.
- For Structural-Thermal physics, the lower order mesh now uses the SOLID225 element formulation instead of SOLID226 (a high-order element) with the mid-side nodes dropped.

Contact and Connection Enhancements

Contact Enhancements

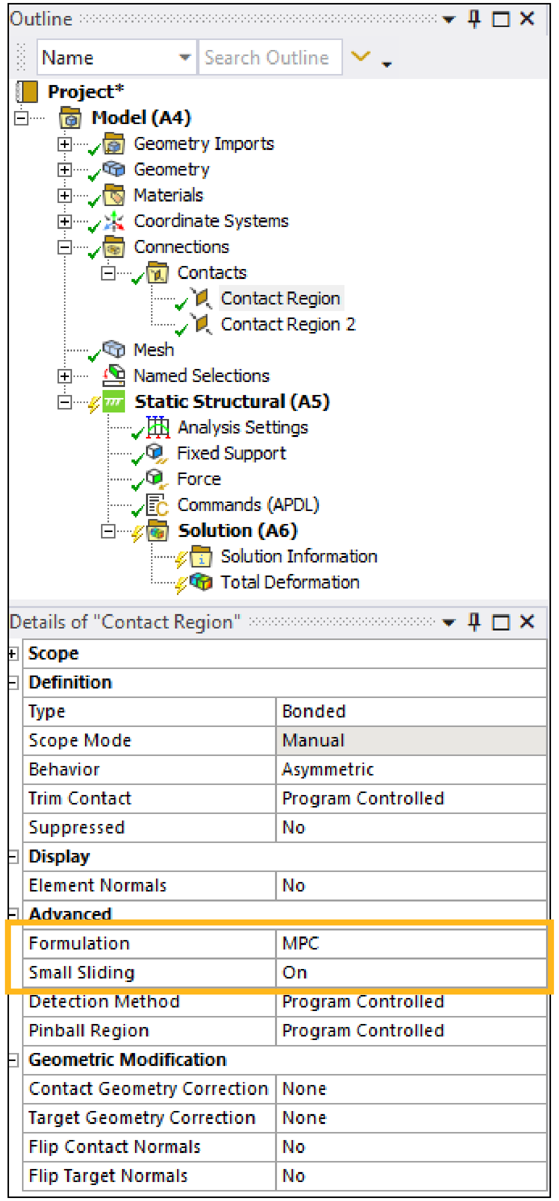
- Small Sliding property is now supported when the Formulation property is set to MPC.

- Property options include:



- The small-sliding logic improves solution robustness. It can easily solve complex contact models having a bad quality geometry or mesh and non-smooth contact interfaces.

- Small Sliding is turned On by the Program Controlled setting for a No-Separation contact with Large Deflection turned off and for a Bonded contact.

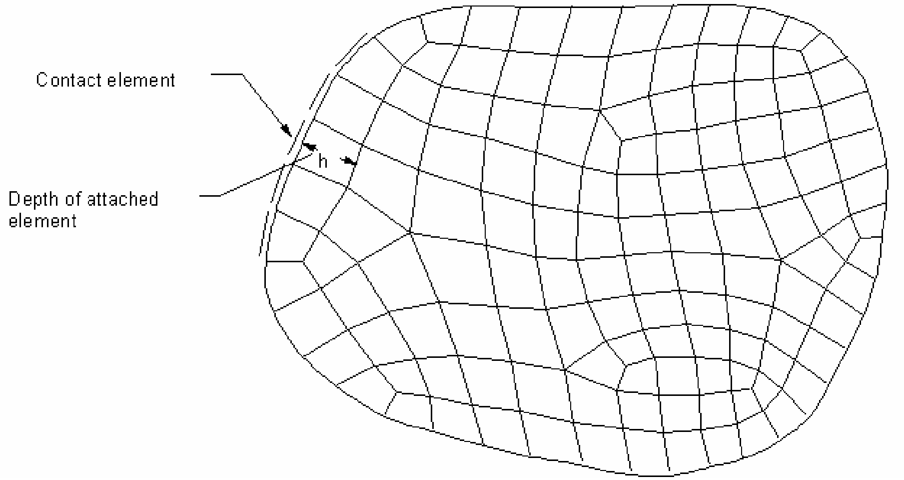


Ansys

Default Pinball Radius Factor for Shells

- When we define a contact, the APDL calculates a default pinball radius
- Circle for 2D or a sphere for 3D of radius 2*depth (rigid-flex contact) or 1*depth (flex-flex contact) of the underlying element.
- In Mechanical, the default pinball radius factor when using Program Controlled option is always set to
  - 0.0 for most contacts -> solver will decide the factor.
  - 1.0 for shells (spurious contact and slow contact searching). Too Large!!
  - Solver recommends 1.75 to 0.5 (rigid – flex), 0.5 to 0.25 (flex-flex) for bonded and no separation contacts
    - Solution: Issue 0.0 and let solver decide the pinball radius factor

Figure 3.12: Depth of the Underlying Element




```
rmod,tid,6,1.          ! PINB
rmod,tid,10,0.         ! CNOF
rmod,tid,12,0.         ! FKT
rmod,tid,36,1816       ! WB DSID
rmod,cid,3,10.         ! FKN
rmod,cid,5,0.          ! ICONT
rmod,cid,6,1.          ! PINB
```

Default Pinball Radius Factor for Shells

- In 2023 R1 issue a factor of 0.0 for **MPC Bonded** and **No Separation** Contacts on shells

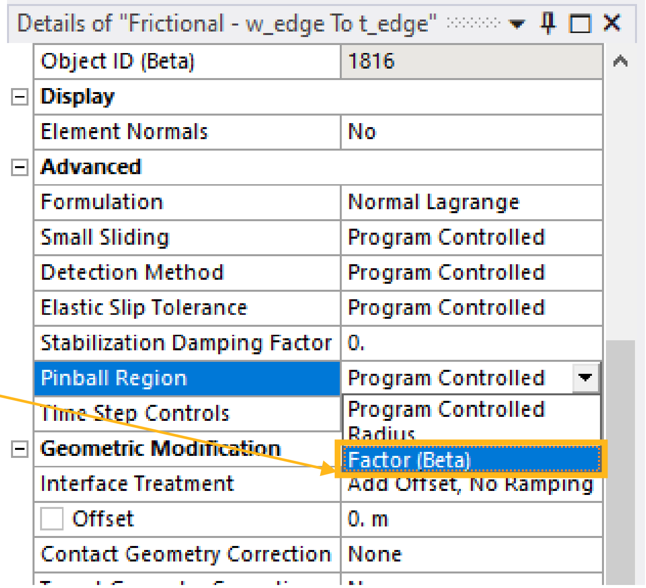
Input File

```
rmod,tid,6,0.          ! PINB
rmod,tid,10,0.         ! CNOF
rmod,tid,12,0.         ! FKT
rmod,tid,36,1816       ! WB DSID
rmod,cid,3,10.         ! FKN
rmod,cid,5,0.          ! ICONT
rmod,cid,6,0.          ! PINB
```

Output file (solve.out)

```
*** NOTE ***                            CP =       1.000   TIME= 16:33
Contact related postprocess items (ETABLE, pressure ...) are not
available.
Contact detection at: nodal point (Dual shape function based)
Nominal variation of contact stiffness is activated,
MPC will be built internally to handle bonded contact.
*WARNING*: Certain contact elements (for example 99&141) overlap each
other.  Overconstraint may occur.
Average contact surface length                 0.10612E-03
Average contact pair depth                     0.60493E-04
Average target surface length                  0.11289E-03
Default pinball region factor PINB             0.25000
The resulting pinball region                   0.15123E-04
```

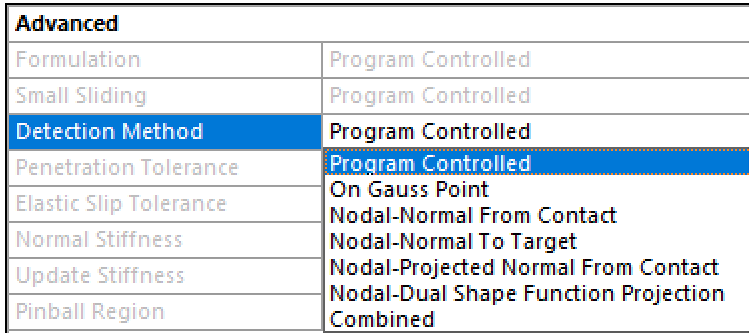
- Please use the beta feature to specify pinball radius factor manually for legacy databases if required



- Future work: Extend for other contact formulations

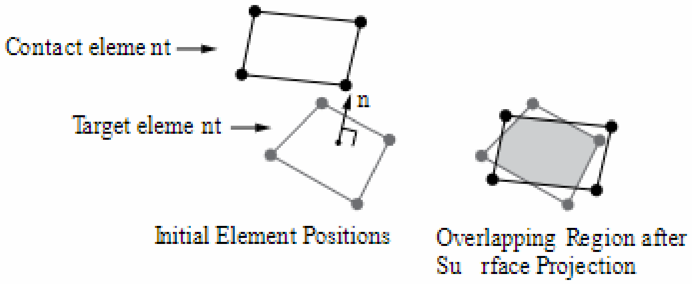
Projected Contact for Gaskets

- Send the KEYOPT(4) = 3 when
  - If Detection Method is set to Program Controlled AND
  - The contact is scoped to a gasket



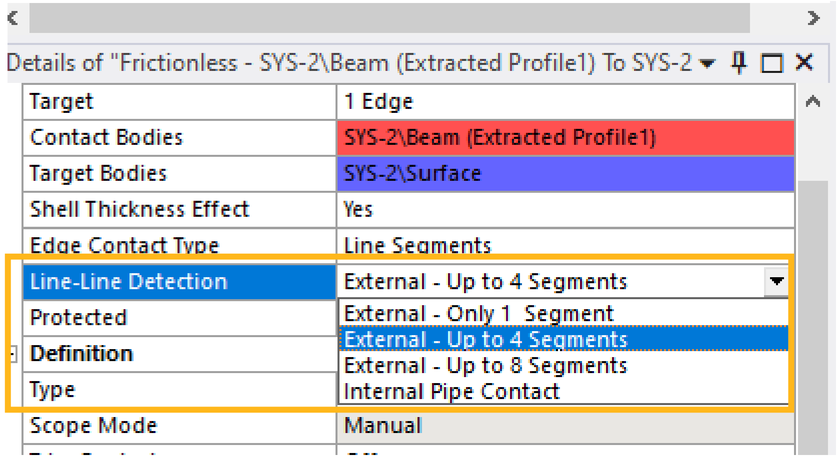
- Advantages:
  - This setting enforces a contact constraint on an overlapping region of the contact and target surfaces rather than on individual contact nodes
  - Prevents displacement overshoot for gasket elements
- Disadvantage:
  - Computationally more expensive

Figure 3.22: Surface Projection Based Contact



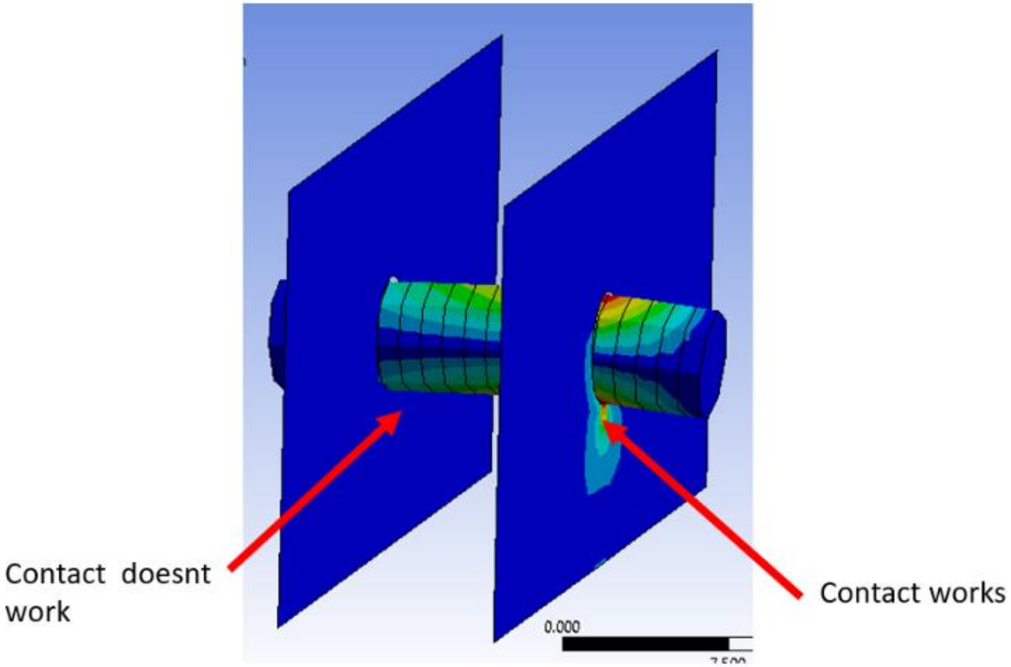
Edge-Edge Detection

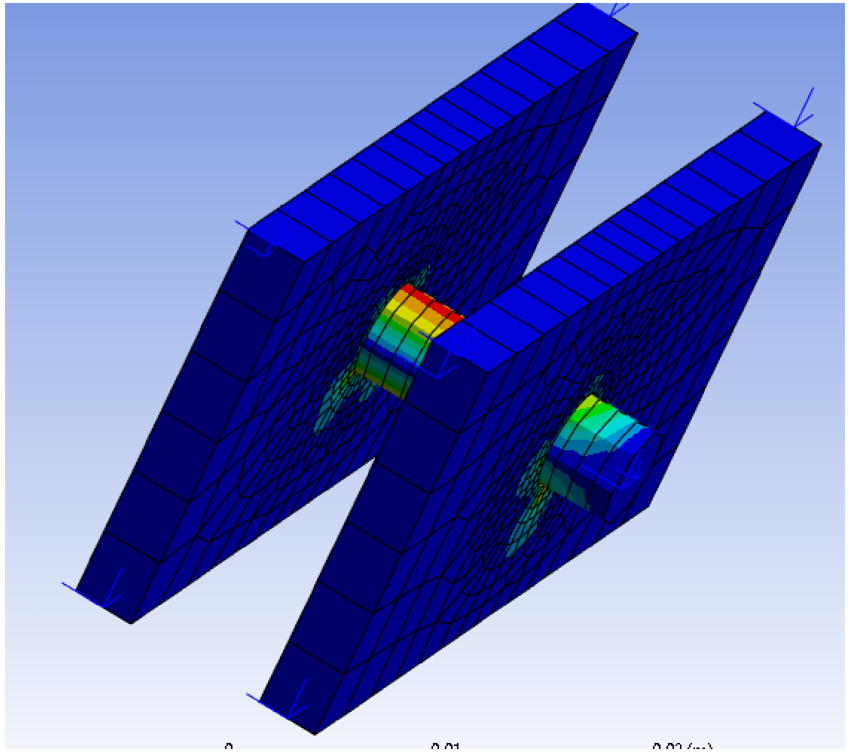
- For beams bodies, Beam-Beam Detection property was available. Rename it to Line-Line Detection.
- This property Line-Line Detection available when Edge Contact Type is set to Line Segments.
- The option defaults to External-Up to 4 segment which will set Keyop, cid, 14,1




```
enorm,_elow,,,,,NOWARN        ! enorm in case the edges are not aligned. don't issues warnings
keyo,cid,3,2                 ! beam-beam or line segments edge contact, include all possible contact
keyo,cid,14,1                ! beam-beam or line segments edge contact, allow up to four target segments
keyo,cid,10,0                ! adjust contact stiffness each NR iteration (from Program Controlled setting)
keyo,cid,11,1                ! include shell thickness effect
keyo,cid,12,0                ! standard contact
keyo,cid,18,1                ! small sliding turned on by application
```

An example shown below where the edge contact with Line-Line detection option gives better results

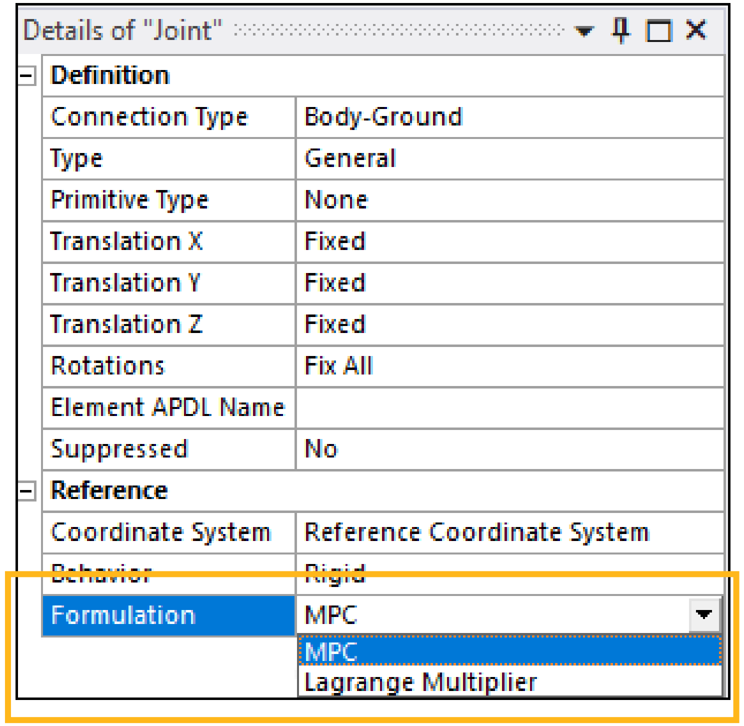




External –up to 4 segment option gives better results

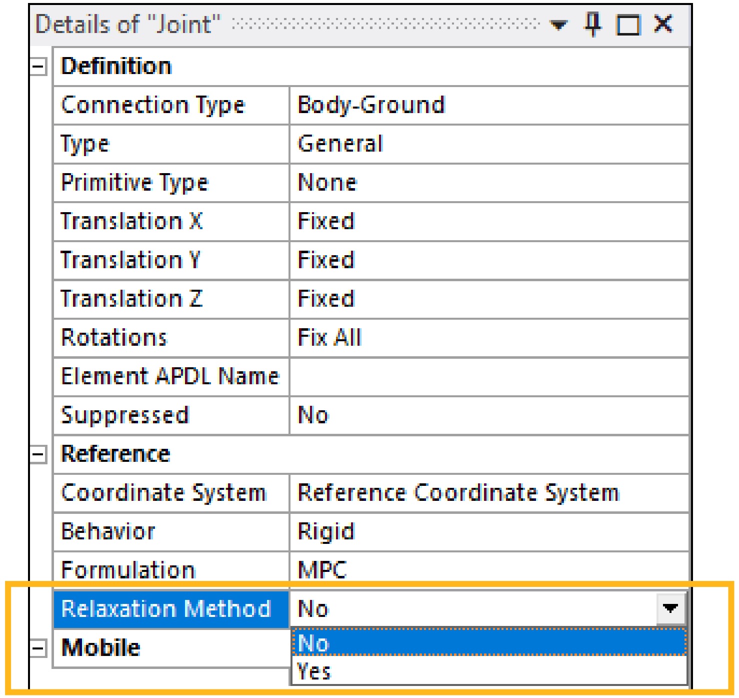
Connection Enhancements

- Lagrange Multiplier for Remote Points and Joints:
  - Remote Point and Joint objects have a new property:
  - Formulation: This property enables you to specify the contact algorithm the application uses for a particular remote point computation. Options include-
    - MPC (default): This option creates multipoint constraint equations internally during the Mechanical APDL solution to tie the bodies together.
    - Lagrange Multiplier: This option enforces zero penetration when the contact is closed, making use of a Lagrange multiplier on the normal direction and a penalty method in the tangential direction. This formulation helps to overcome over-constraint problems better than the MPC formulation.
  - In previous releases, these remote point-based boundary conditions used the MPC formulation internally. Now, with the option to specify Lagrange Multiplier, user can **better eliminate over-constraints.**



Connection Enhancements

- Relaxation Method for Remote Points and Joints:
  - When Formulation property is set to MPC, the Remote Point and Joint objects display a new property:
    - Relaxation Method- When MPC-based surfaced-based constraints or rigid bodies are subjected to over-constraints, this method (when set to Yes) relaxes the constraint between contact-generated internal constraint equations and other constraint equations.



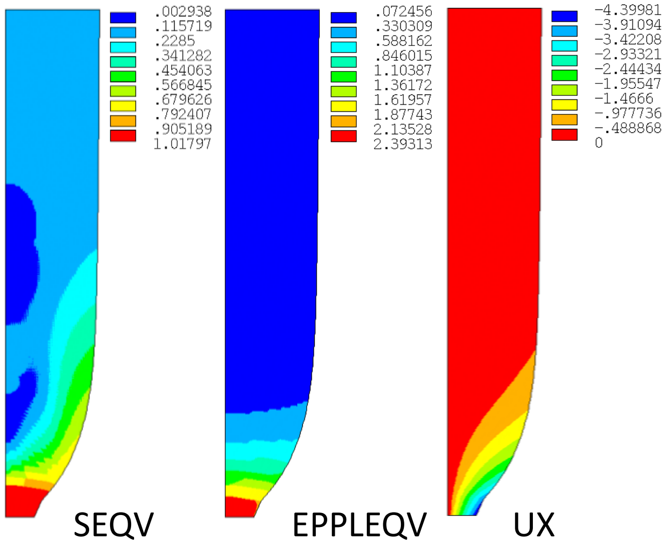
Materials

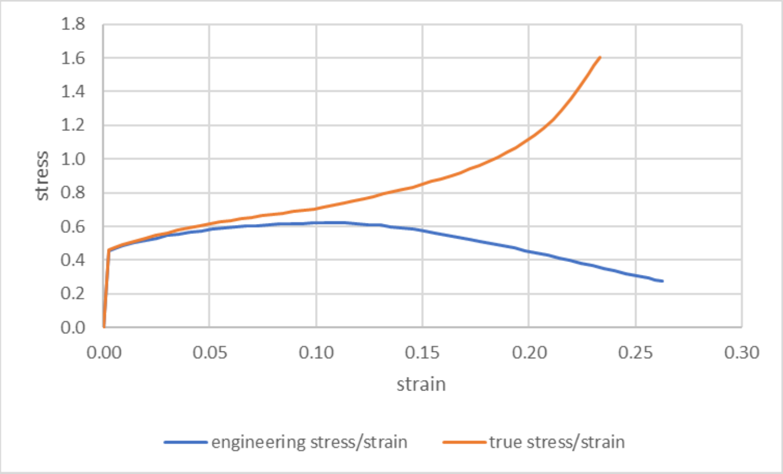
Ansys

Finite strain plasticity material model

- Combines hyperelasticity and von Mises plasticity with isotropic hardening
  TB,HYPER + TB,PLASTIC,,,,BISO|MISO or TB,NLISO
- Multiplicative split of mechanical deformation gradient into elastic and plastic parts: $\boldsymbol{F}^m = \boldsymbol{F}^e \boldsymbol{F}^p$
- Allows for large elastic strains and large plastic strains
- Supports displacement based and mixed u-P element formulations
- Supports three-dimensional, plane strain, generalized plane strain, axisymmetric, axisymmetric with torsion and general axisymmetric conditions

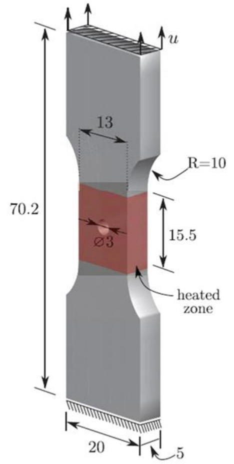
Necking of circular bar



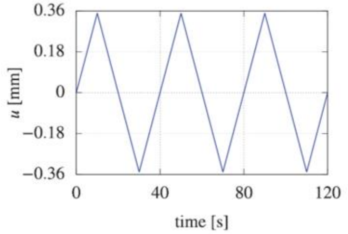


Material-agnostic Generalized TMF Damage

- Fatigue failure is widespread in many engineering applications
  - E.g. structures with simultaneous cyclic thermal and structural loading
- Need for material-agnostic regularized damage model
  - Damage materials lead to strain softening behavior: use regularization
  - Pervasiveness of fatigue failure in different industries: many different materials
  - Gradient regularized damage greatly improved solution robustness
- TMF damage model can be applied to most native Ansys nonlinear materials
  - Formulate based on incremental inelasticity
  - Damage driven by inelasticity
  - Applicable to bilinear and multilinear isotropic/kinematic hardening, rate-dependent Chaboche (viscoplasticity), Drucker-Prager materials
- TMF damage can be directly coupled with thermal and pore fluid diffusion
  - Pore fluid diffusion is based on Darcy Law



- TMF damage model
- 3 fields coupled (structure, thermal and damage)
- 20 cycles of load

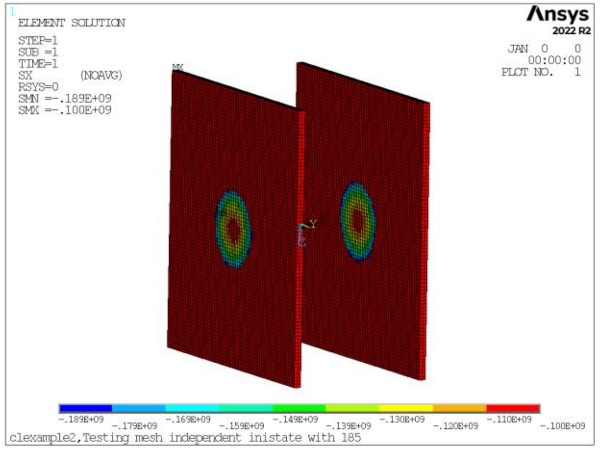


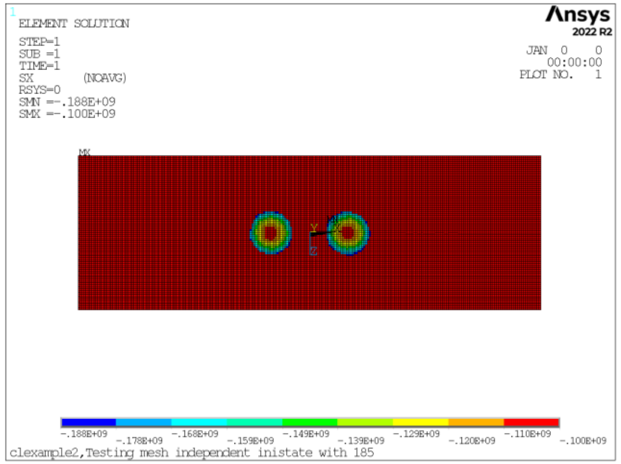
Damage evolution

us-522039e.dat

Mesh Independent INISTATE

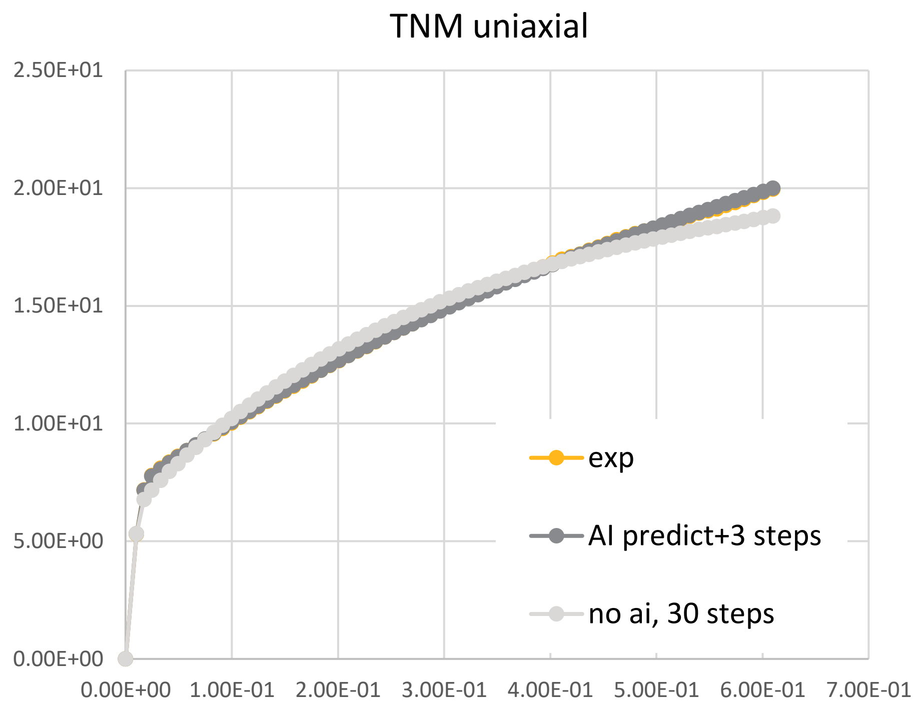
- Ability to apply INISTATE on a cloud of points to be used later by the generated finite element mesh.
- User Field Variables and Elastic Stress or Elastic Strain supported and applied as functions of Location, Time, Temperature and Frequency
- Supported for Solid Elements PLANE182/183, SOLID185/186/285
- Can be applied as Node-Based or Element-Based or Free-Form to be used for any other purpose (ex :fracture)
- Multiple convex zones of data can be applied to portions of the model to reduce data size and improve performance when and where applicable





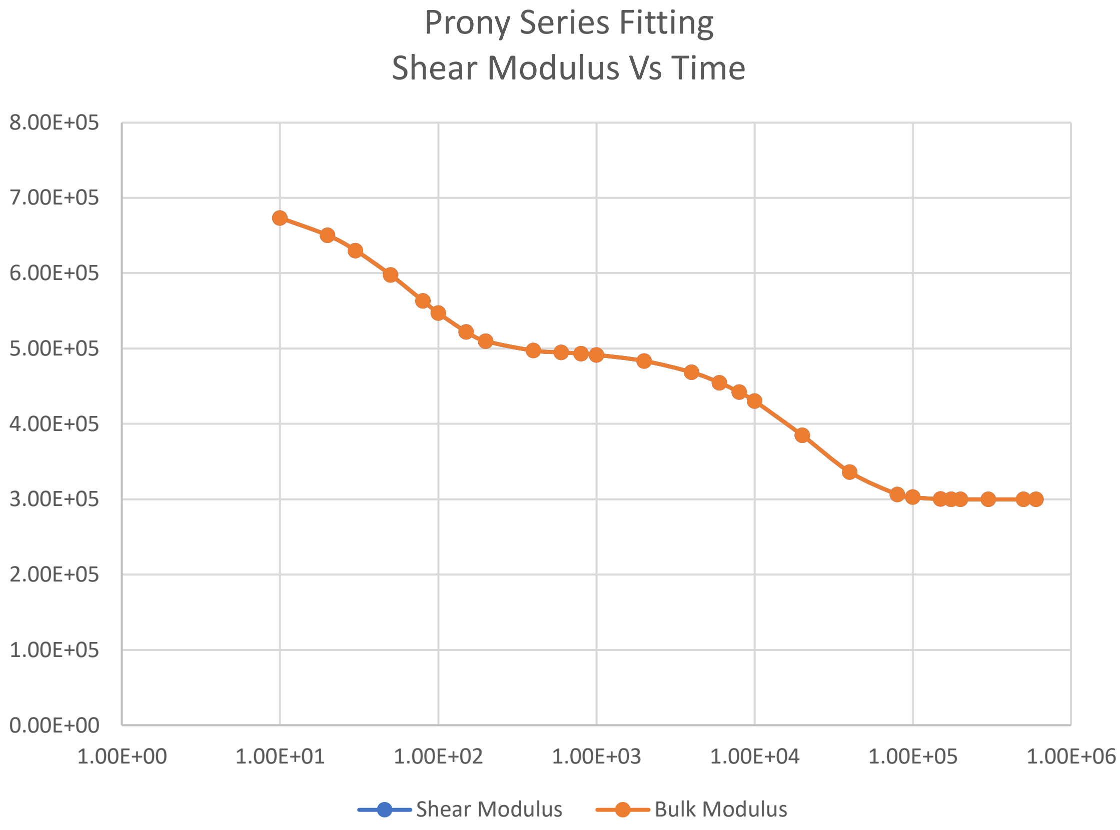
AML Development Update

- Models to be covered for calibration in the order of priority
  - Viscoelasticity with Prony series
  - Frequency viscoelasticity with Prony series
- ML/AI assisted calibration for robustness and performance
  - Parameter initiation for BB and TNM
- GRPC (by WB EDA Team)
  - GRPC Server and additional C Sharp Layer to support HTML close to completion.
  - Addition implementations in the GRPC Server for the future for other AML features
  - GRPC Server extensions to support Python interface(by WB EDA Team) scheduled for the future
- HTML Based UI by Materials BU to support hyperelasticity in progress



Viscoelastic Materials Fitting

- Parameter fitting is now supported for Prony Series with Shift Functions
  - TB,ELAS with TB,PRONY and TB,SHIFT(if needed)
- Experimental Data Supported
  - Shear Modulus Vs Time
  - Bulk Modulus Vs Time
  - Shear Modulus (Real + Imaginary Components) Vs Frequency
  - Bulk Modulus (Real + Imaginary Components) Vs Frequency
- Temperature dependency supported



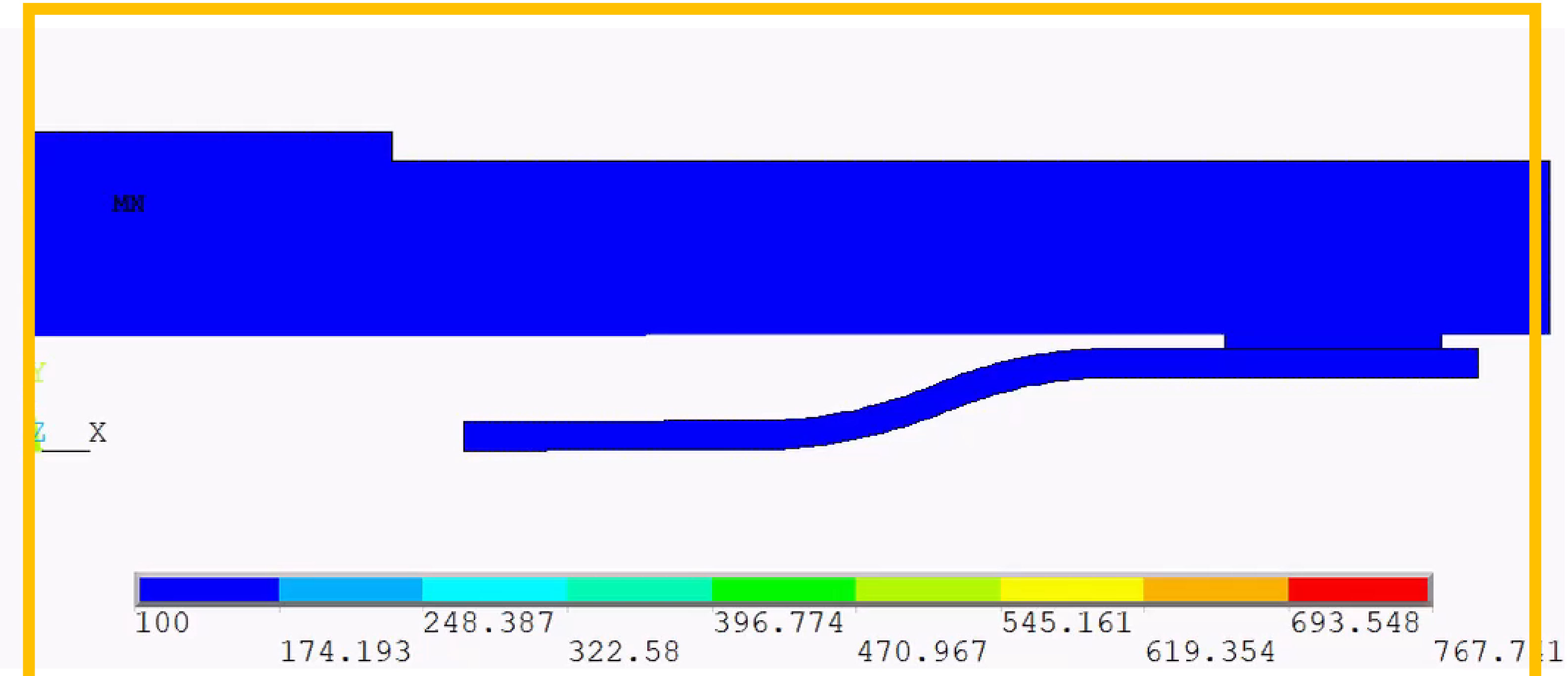
Contacts

2023R1 Contact & Nonlinear Solver Heuristics Enhancements

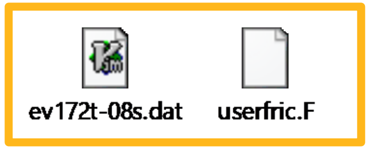
- Support contact for 2D axisymmetric elements with torsion ROTY
- DMP contact performance improvement:
  - Distribute underlying elements of target surface into different domains for node-surface contact CONYA175
  - Concurrent contact trimming and pair splitting to avoid external element numbering overflow
- Contact options for shells – BIW workflows
  - Double sided target surface
  - Limited CEs for rigid constraint under small deflection
- Contact result, convergence history tracking in PyMAPDL
- Enhance NLHIST stop criteria: tracking result variables to terminate solution besides contact results
- Improve robustness for transient dynamics: initialized acceleration, backward-Euler integration scheme as default for 1st order equations.
- Improve accuracy for New-Raphson iterations: refine force/displacement convergence tolerances.

2D axisymmetric elements with torsion ROTY

- Frictional contact was traditionally unsupported for 2D elements with torsion DOF
- Frictional forces in radial, axial and circumferential directions are now supported
- Example below shows the frictional heat dissipation in a coupled thermal-structural model

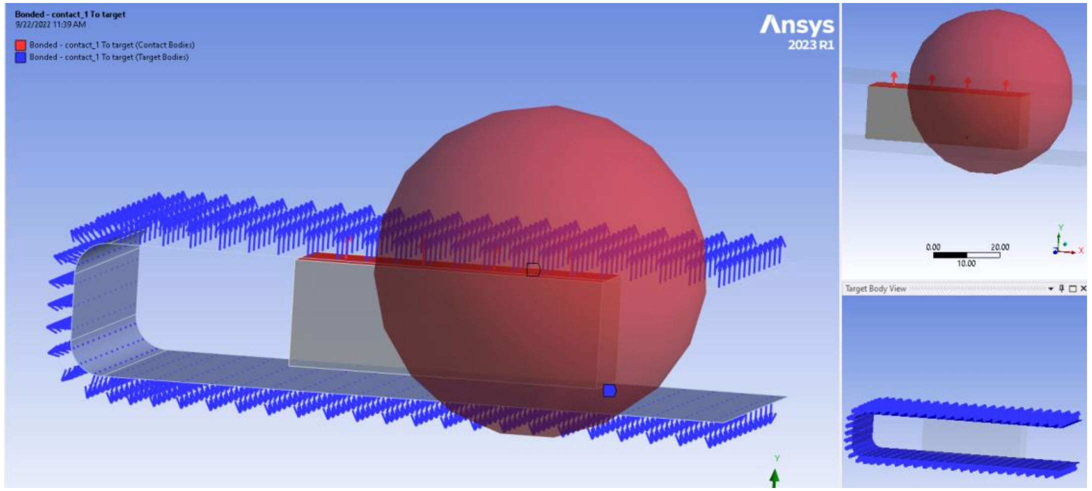


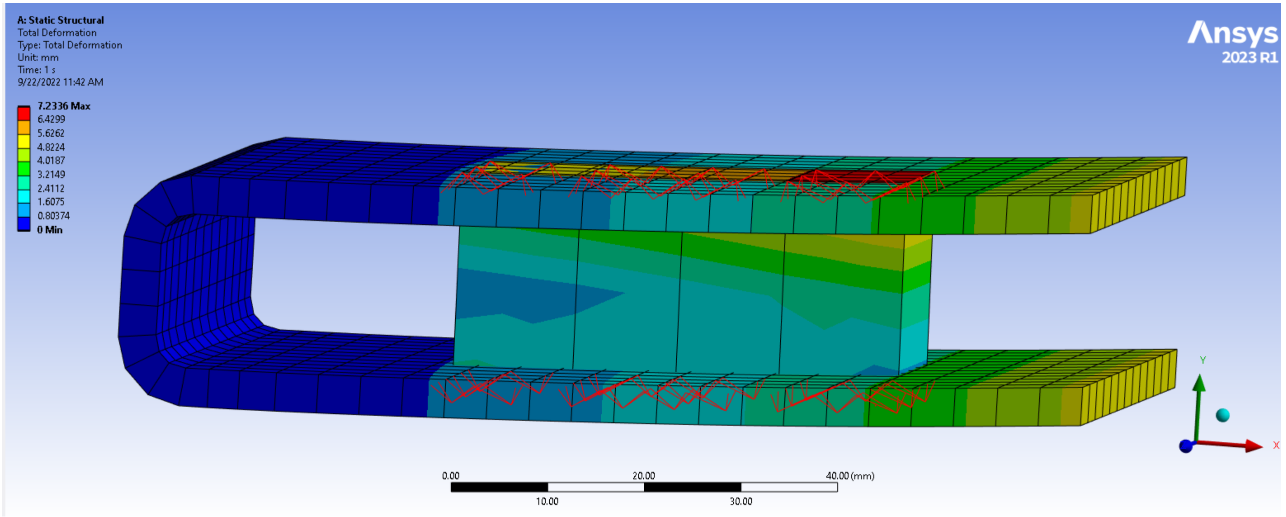
Coupled field brake transient analysis



Double Sided Target Surface

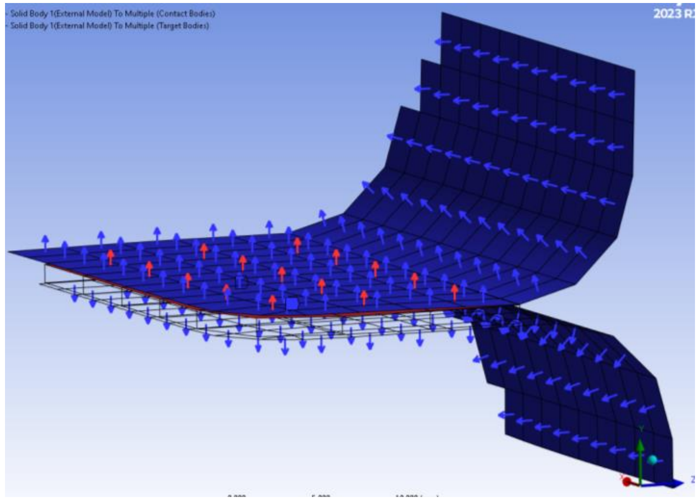
- Adhesives in BIW models may have to be bonded with both top/bottom shell surfaces – requires two separate definitions
- Double-sided target surface option reduces setup to only one set of the target surface.
  - Reduces the total number of target elements
  - Make setup easier for models in which orientations of underlying shell elements are arbitrary or not consistent.



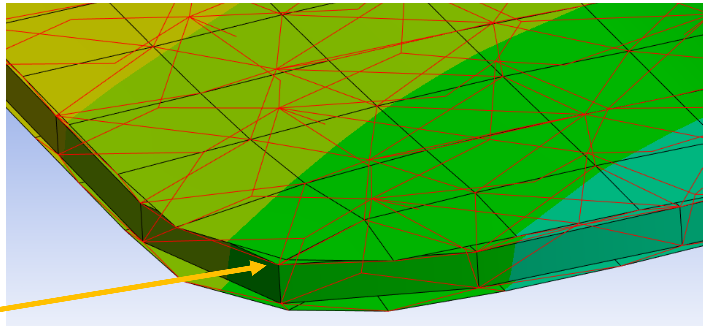


Double Sided Target Surface

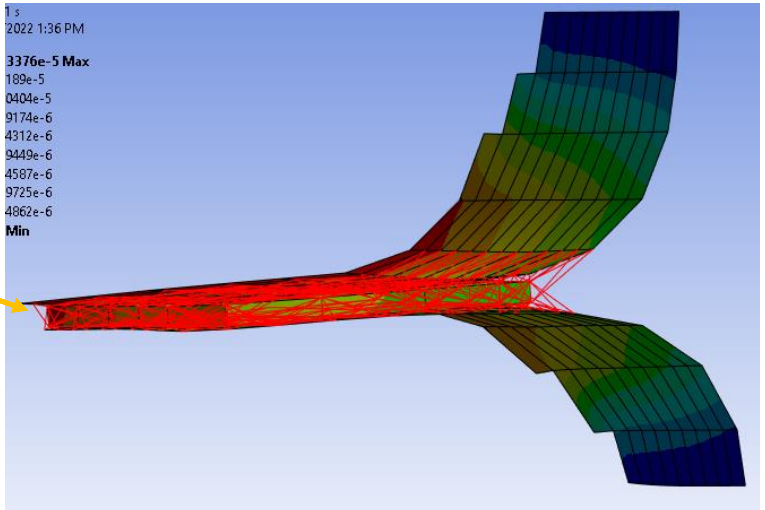
Arbitrary target surface normal over shell elements



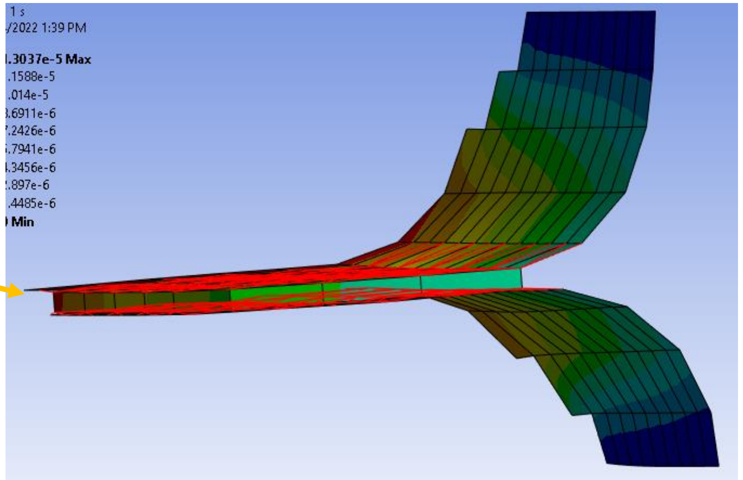
The double-sided target definition builds CEs on stick-out gasket nodes which improves robustness ✓



For arbitrary target surface normal, the single sided target definition either misses contact detections or builds CEs across over both sides of gasket elements ✗

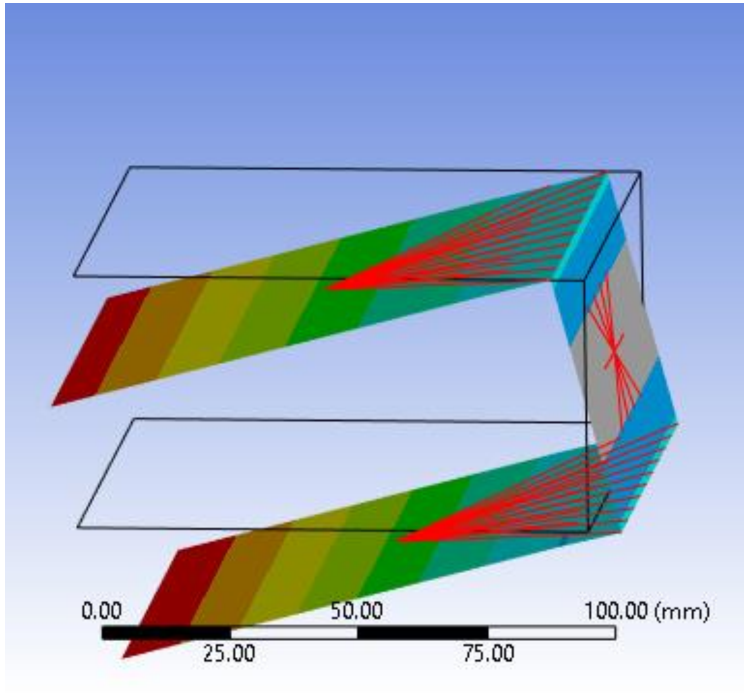


The double-sided target definition builds CEs on correct sides ✓

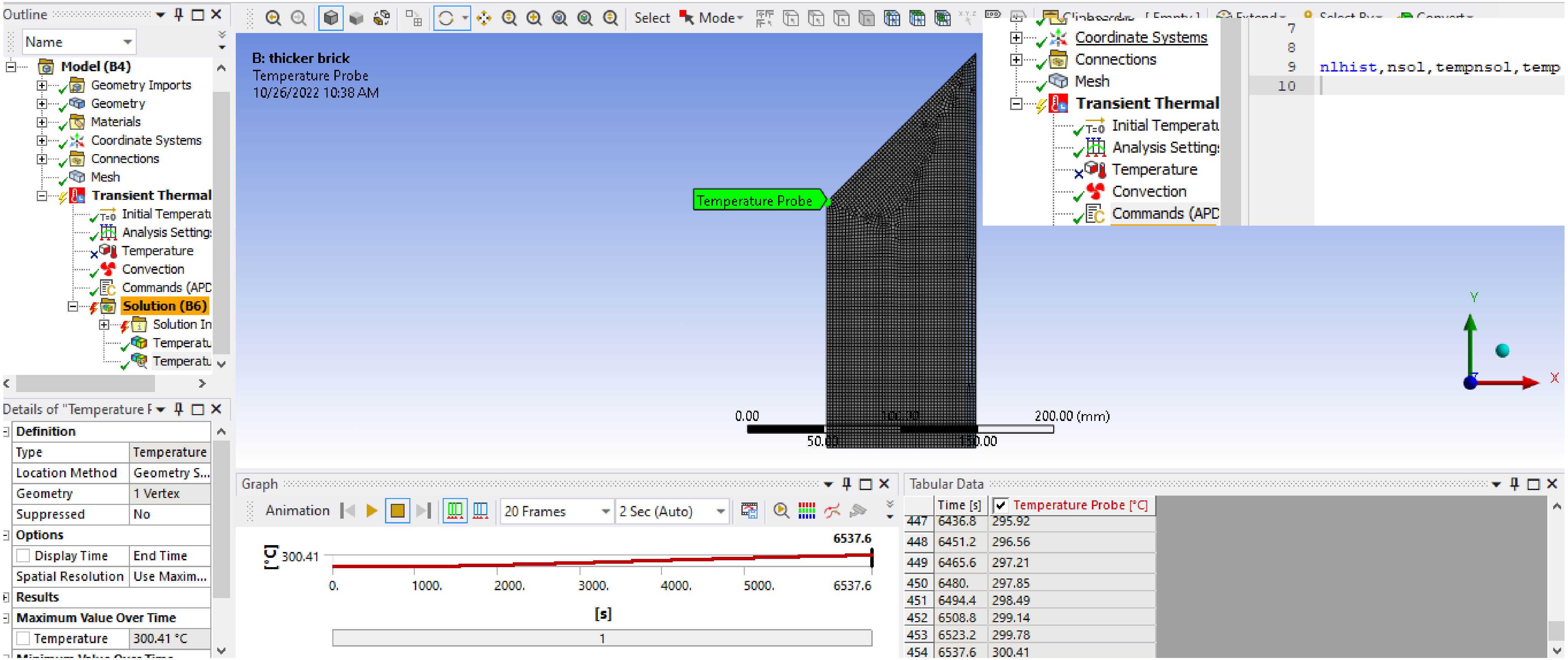


Limited CEs for Rigid Constraint under Small Deflection

- Improvement in performance and reduction in memory usage for rigid surface constraint under small defection
  - Only builds CEs on rigid nodes which connect to other elements or have applied boundary conditions
  - No internal CEs are built for free rigid nodes (no connection to any other elements and no B.C.s)
  - Geometry correction is made on free rigid body nodes during solution



Terminate analysis automatically based on tracked results



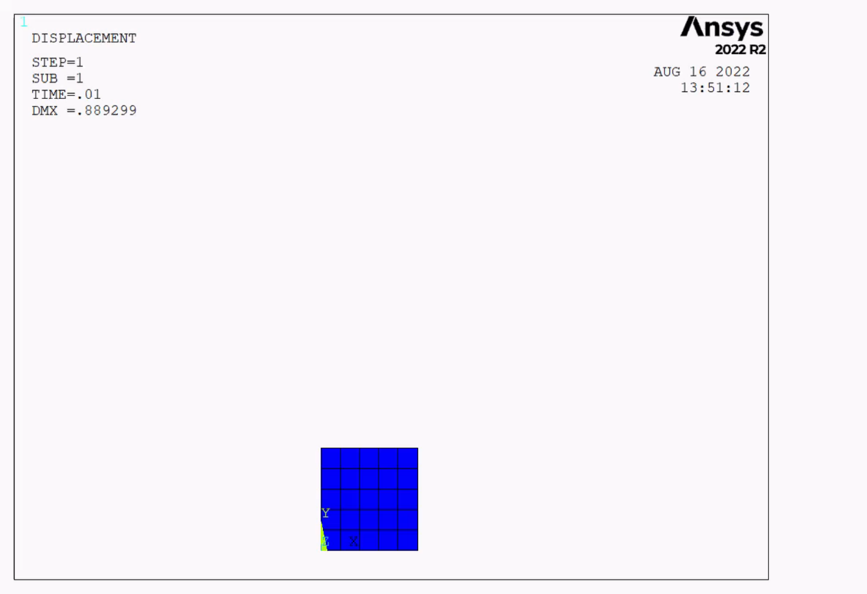
Analysis terminates automatically when the temperature of the probed node exceed 300 C in the transient run

Behavior of Deleted Displacements (DDELE) in Restart and NLAD

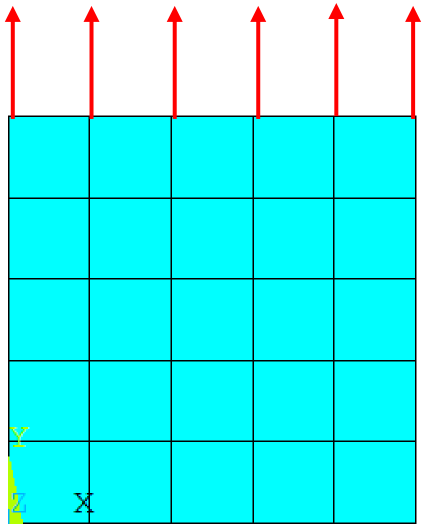
DDELE in Restart and NLAD

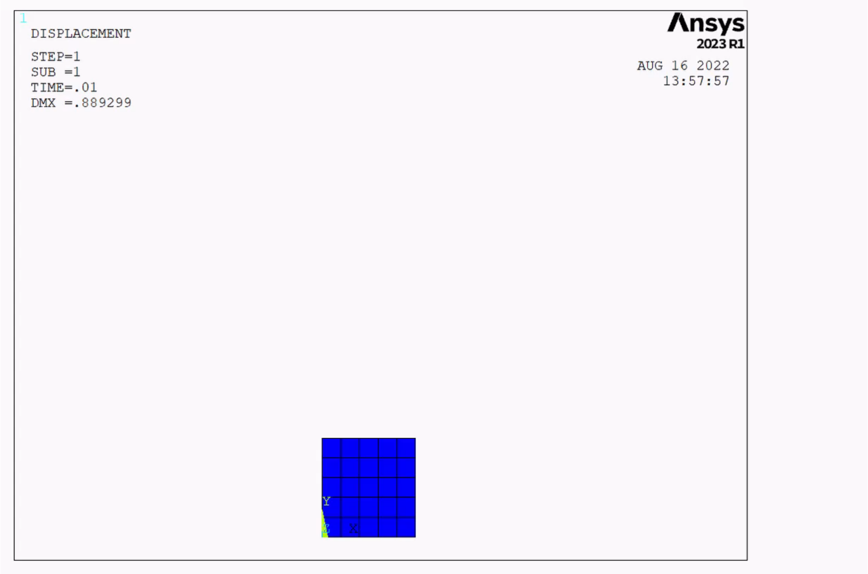
- In version 2022 R2 and before:
  - If a displacement boundary condition was deleted (using the DDELE command) during a regular analysis step, then this deletion was not being honored in a restart analysis of the same step.
  - Analyses using nonlinear adaptivity (NLAD) framework also disallowed deleting previously imposed displacements.
- Solution:
  - In version 2023 R1, the restart analysis will ensure that the previous deleted displacements are correctly observed.
  - In the nonlinear adaptivity framework, previously imposed displacements can be deleted if these regions are not remeshed.

Simple Example of DDELE behavior with Restart



Behavior in 2022R2



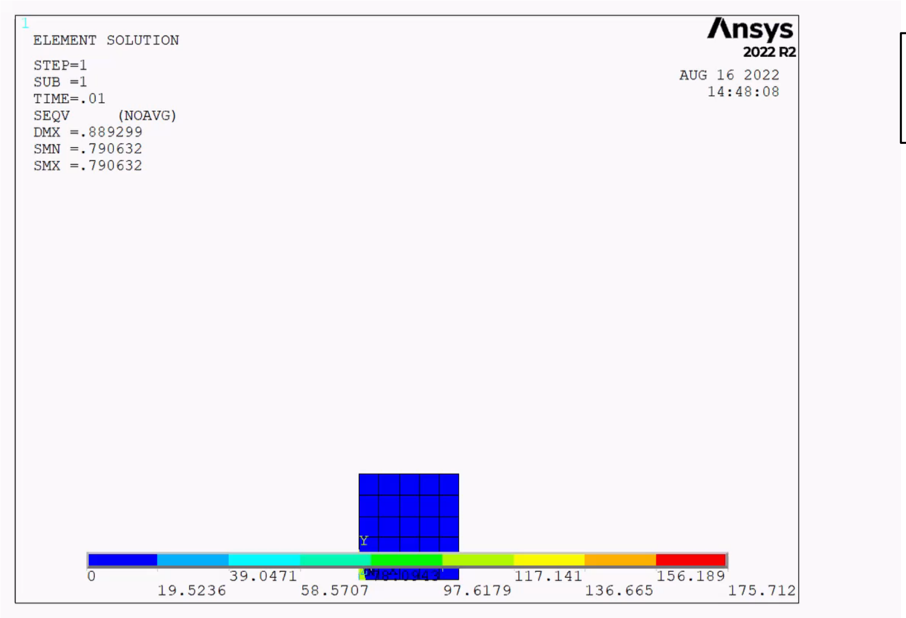


Behavior in 2023R1

The loading is as follows:

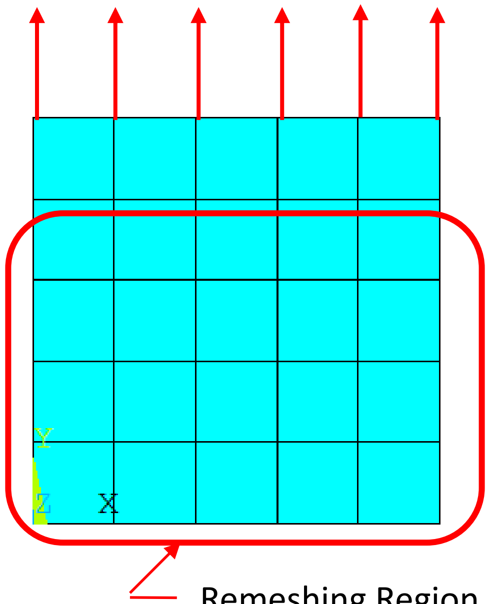
- Apply displacements on top edge in load step 1
- DDELE the applied displacement in load step 2
- Restart from middle of load step 2

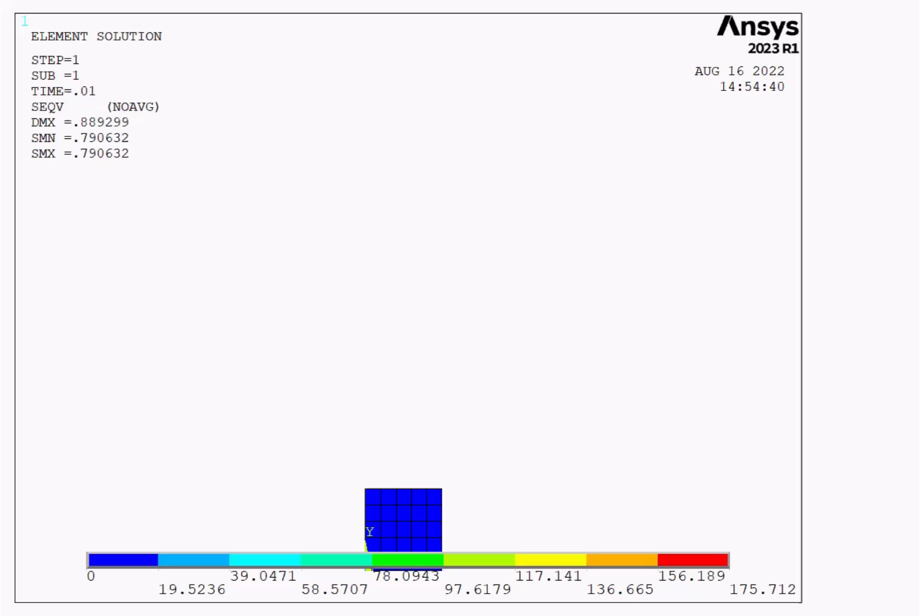
Simple Example of DDELE in NLAD



Behavior in 2022R2

For NLAD, the region where DDELE is applied is not remeshed



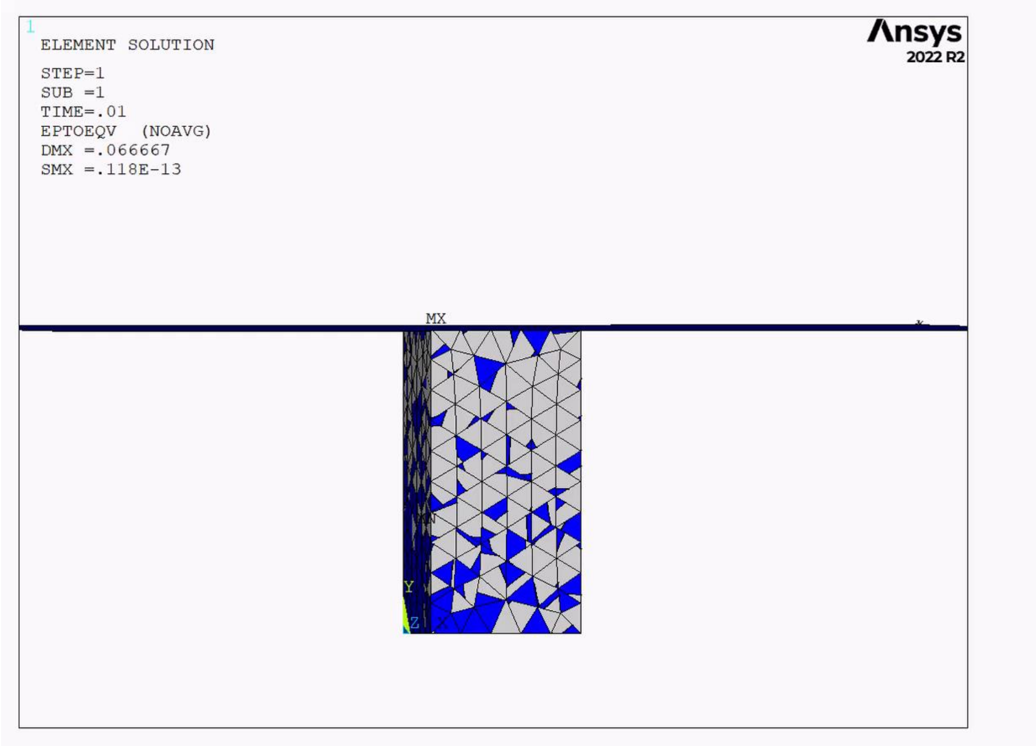


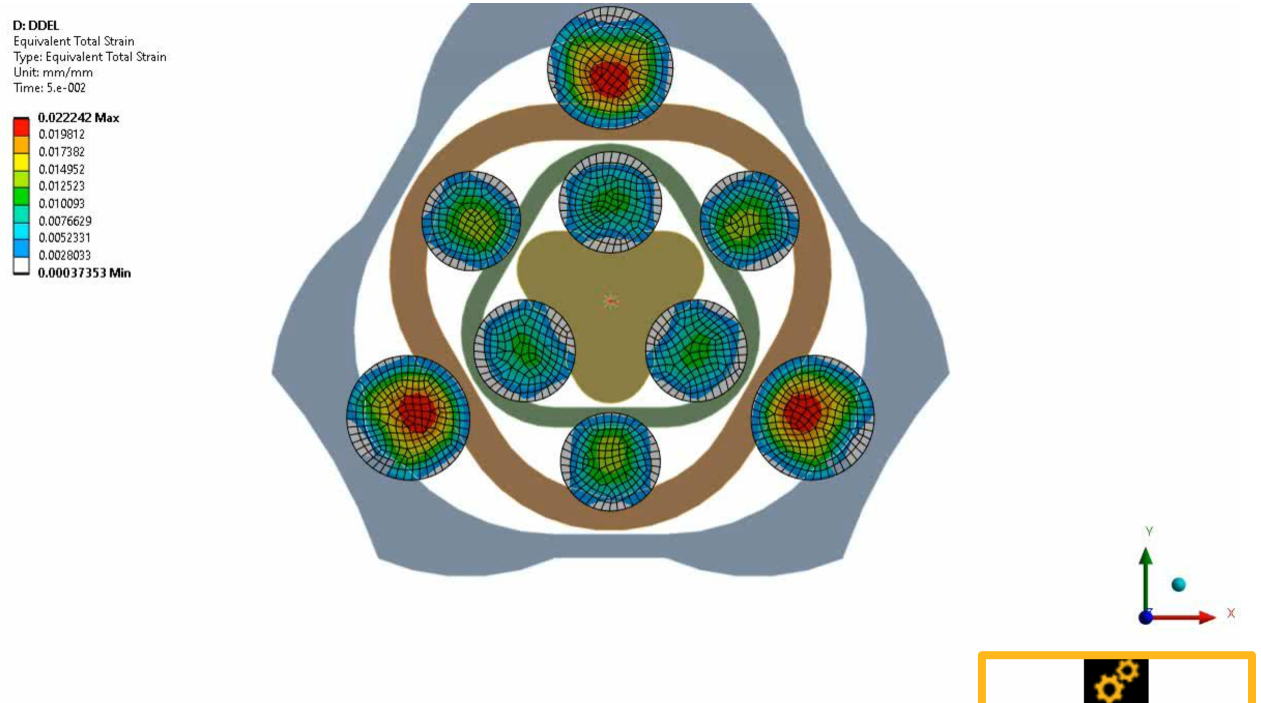
Behavior in 2023R1

The loading is as follows:

- Apply displacements on top edge in load step 1
- DDELE the applied displacement in load step 2
- Restart from middle of load step 2

Examples of DDELE in RESTART and NLAD





D0D395A1.mechdat

The loading is as follows:

- Apply displacement at the pilot node
- DDELE the applied displacement in load step 2
- Re-apply the displacement at the pilot node in load step 3
- Uses the restart framework

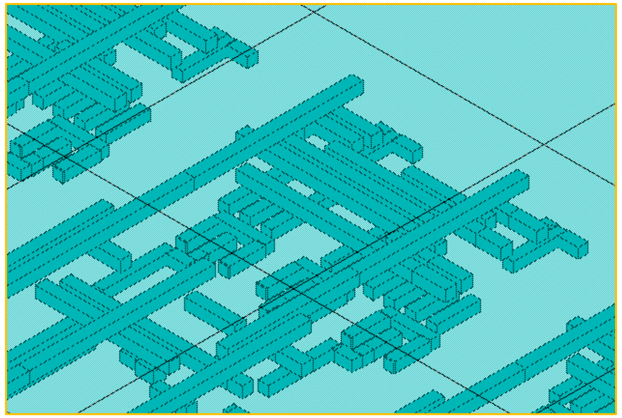
The loading is as follows:

- Interference resolution in load step 1
- Rotation in load step 2
- DDELE rotation in load step 3
- Uses the NLAD framework

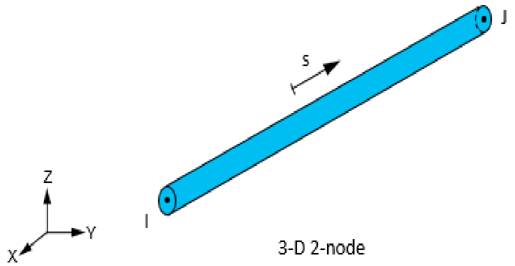
Elements

Coupled-Field Link Element (LINK228)

- Motivation
  - Element embedding method already adopted in electronic reliability study
  - Many PCB/Chip components can be simulated with line elements
  - Need for coupled-field line elements to properly capture the physics



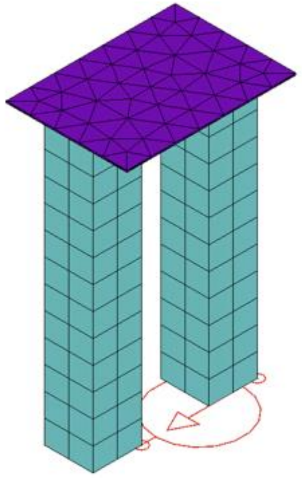
- Overview
  - LINK228 : 3D 2-Node Coupled-Field Link
  - Supported coupling types
    - Structural-Thermal , Thermal-Electric, Structural-Thermal-Electric coupling
  - Supported analyses
    - Static, Transient, Harmonic
  - Max. DOFs : UX, UY, UZ, TEMP, VOLT



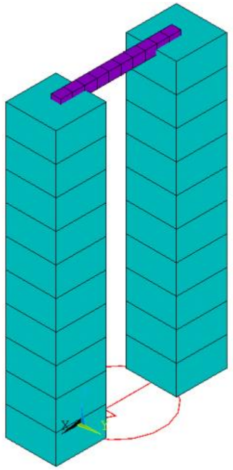
Case 1: Thermo-Electric coupling : Peltier Cooler

- Comparison between SOLID225 and LINK228
- Accurate results with smaller models

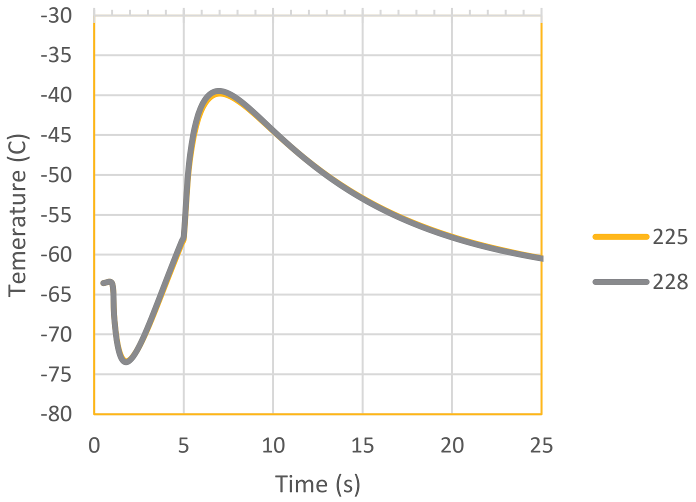
SOLID225

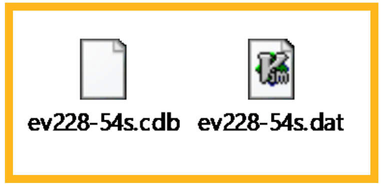


LINK228



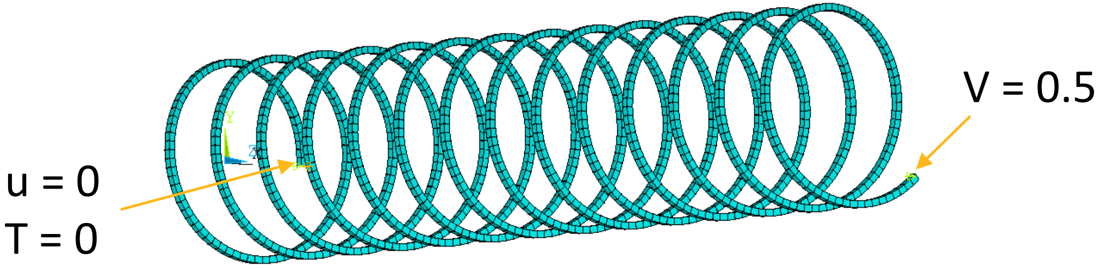
Cold Junction Temperature

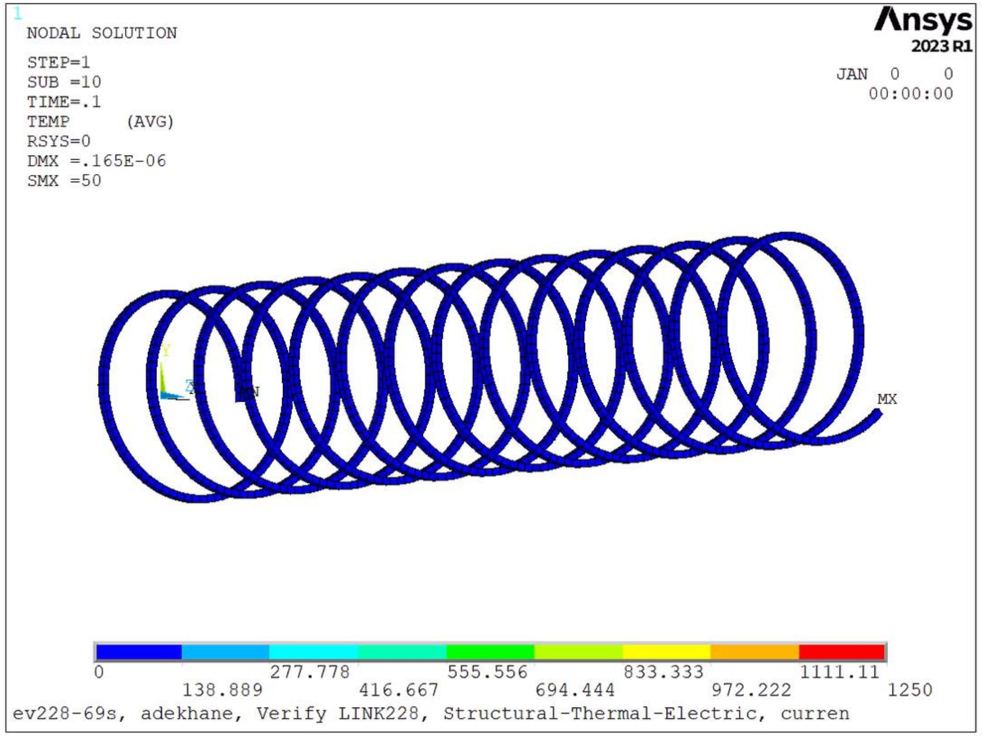




Case 2: Structural - Thermo-Electric Coupling

- Electric field induced deformation

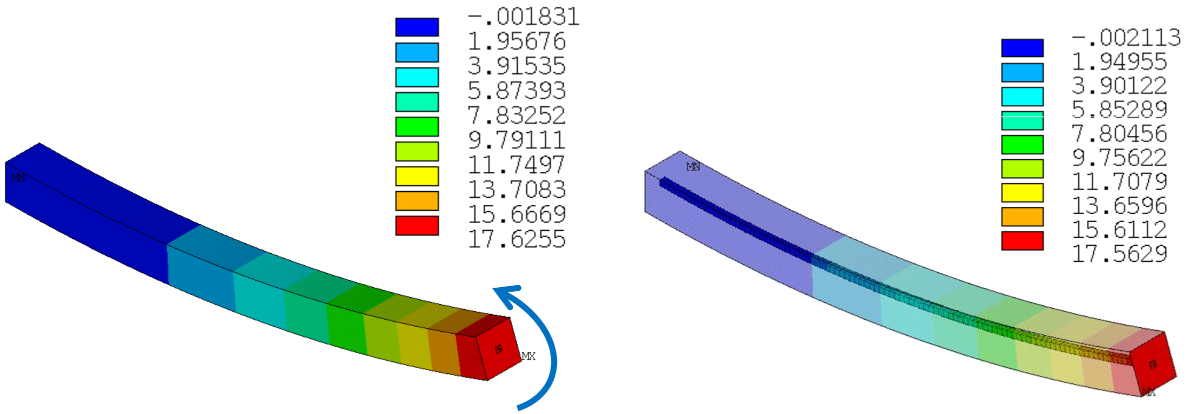




Bending & Torsional Stiffness for Discrete Reinforcing Element

- Discrete reinforcing element REINF264 capable of uniaxial stiffness only before R23.
- Bending and torsional stiffness required for embedded electronic (vias), civil (steel rebars), or biomedical (stents) components

Case 1: Reinforcing vs. full solid models

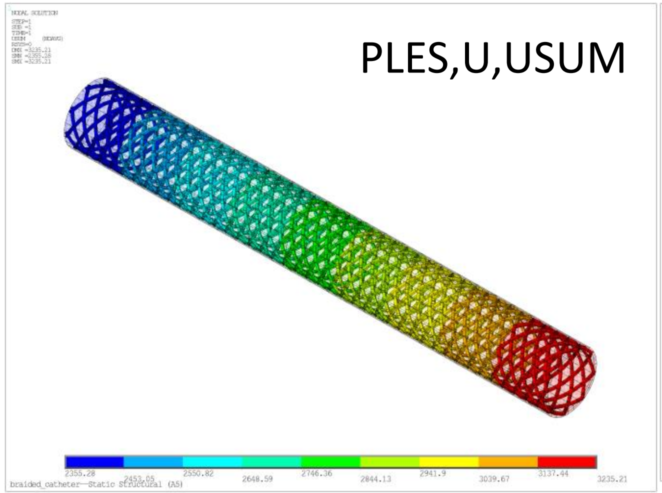


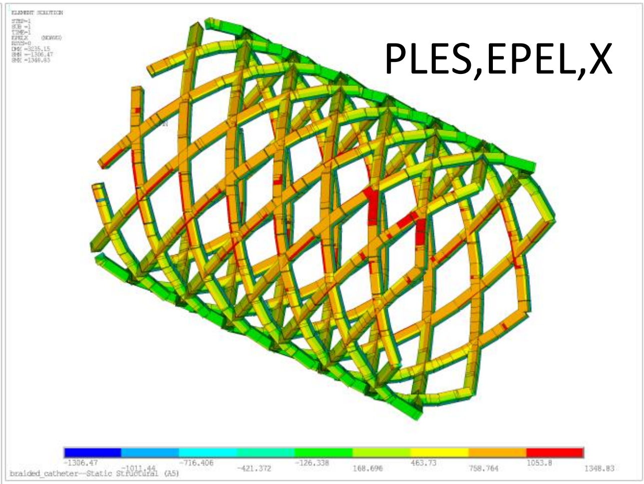
SOLID186 model

REINF264 model
w/ bending & torsional stiffness

Case 2: Bending of a braided catheter

ev264-mi210s.dat



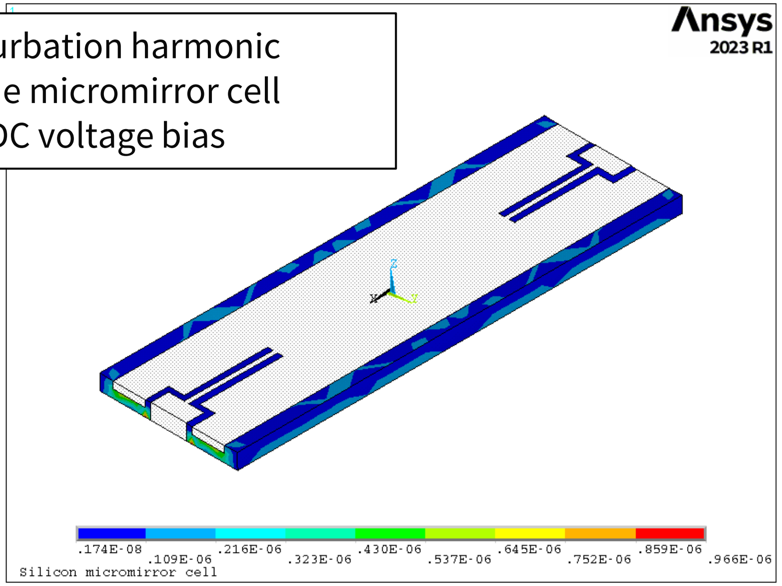


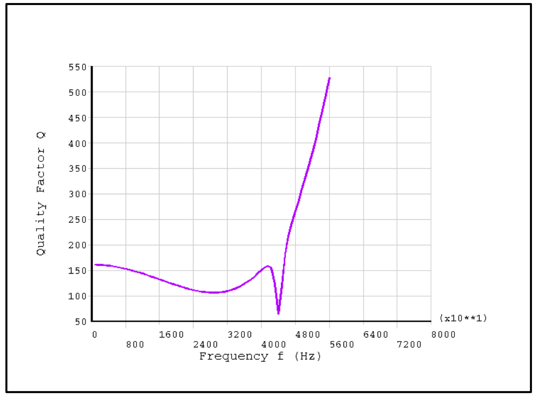
Electrostatic-Structural Analysis Enhancements

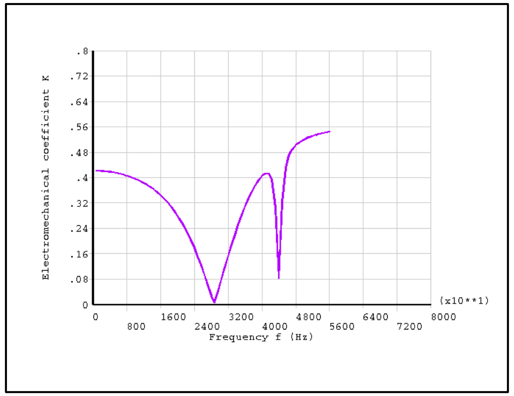
- The electrostatic-structural analysis (KEYOPT(1)=1001) of elements PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 has
  - A new keyoption (KEYOPT(4) = 4) to turn off the default electric force coupling
  - New element output quantities available with the electric force coupling to make result post-processing consistent with the piezoelectric coupling:
    - electric current density (JS),
    - energies (Ue, Ud, Um, SENE, KENE, DAMP)
    - Joule heat (JHEAT)

demo_mirror.dat

A linear perturbation harmonic analysis of the micromirror cell subject to a DC voltage bias



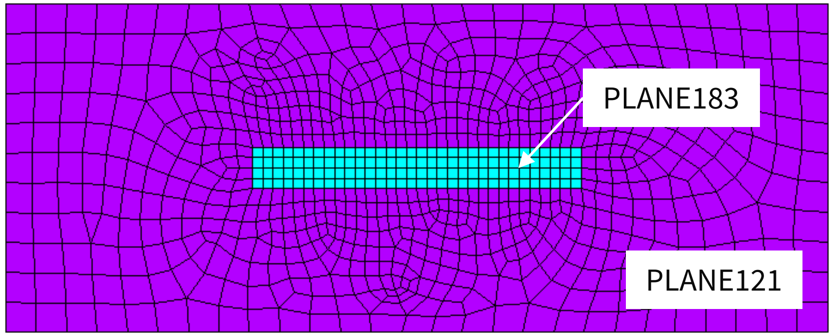




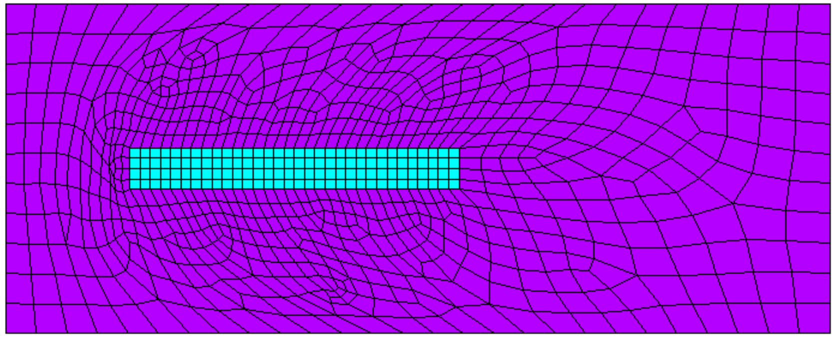
MORPH Command Enhancement

- The MORPH command with the option that allows structural elements in the model (StrOpt = YES) now supports the morphing of the following meshes:
  - Electrostatic
  - Electric
  - Thermal
  - Diffusion
  - Electromagnetic
  - Coupled-field with no structural degrees of freedom

Undeformed electrostatic mesh
Capacitance C0 = 62.4pF



Morphed electrostatic mesh following the displacement of a structural mesh to the left
Capacitance C1 = 64 pF



demo_morph.dat

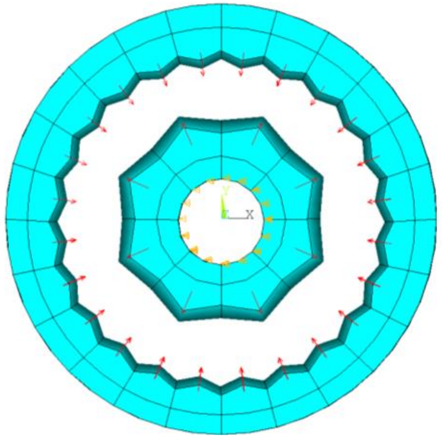
Radiosity Enhancements

RADIOSITY ENHANCEMENTS

- View Factor smoothing enhancements
  - Viewfactor smoothing (VFSM command) with options 2 & 4 produced different results if the surface facets are reordered
- New algorithm has been used to circumvent this issue.
- Energy balance for higher order elements
  - In rare cases with complex geometries the energy balance may not be rigorously satisfied
  - For higher order elements, the midside node TEMP is not transferred to the radiosity solver.
- RADOPT command will have a conservative option to address this issue which will reduce the energy imbalance to < 1%

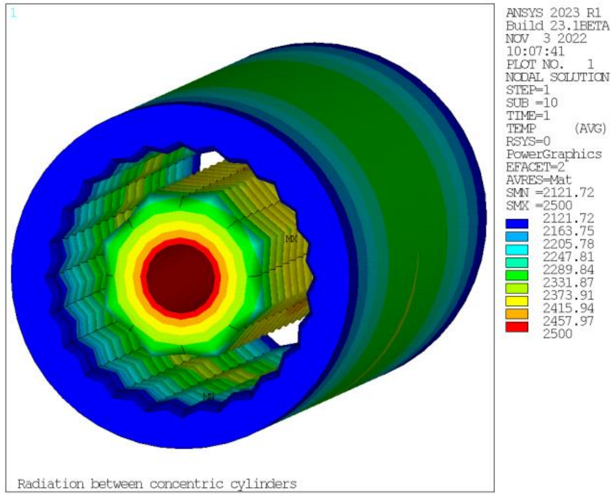
Radiosity Enhancements

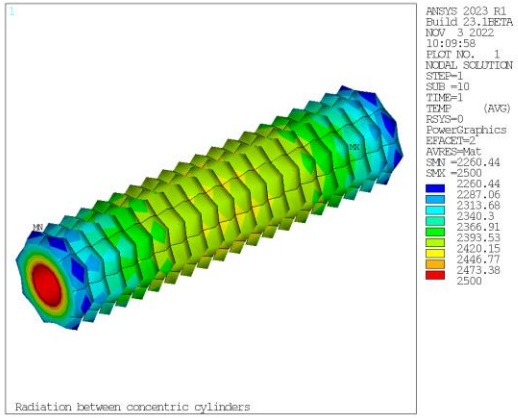
- New algorithm to improve View Factor Smoothing (VFSM command)
- New energy conservative option on the RADOPT command to effectively reduce energy imbalance.



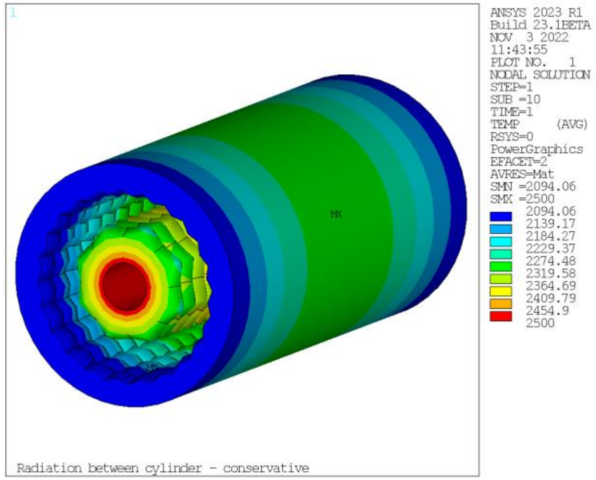
Radiation between cylinders (dimpled surfaces)

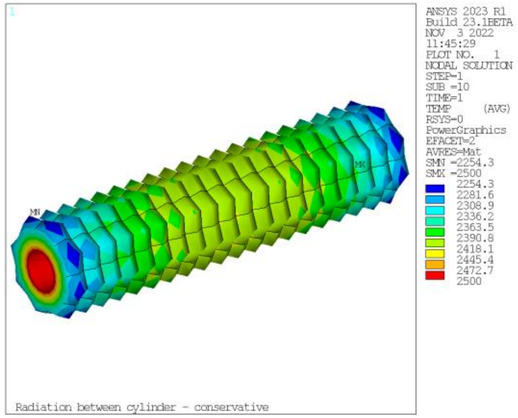
Nonconservative
Energy imbalance 7.6%





Conservative
Energy imbalance 0.0043%





ccyl.dat

Fracture

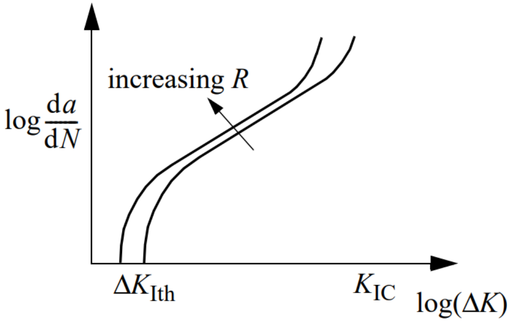
SMART support for non-proportional loading

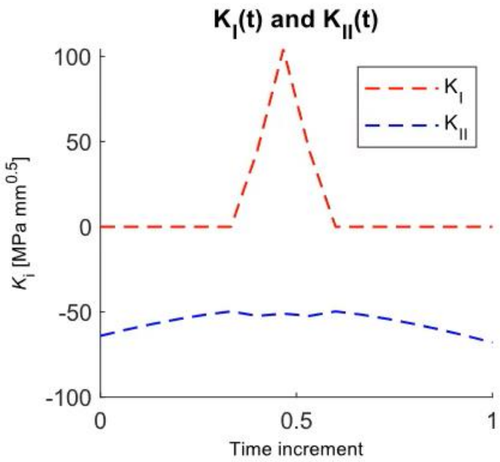
Non-proportional load support

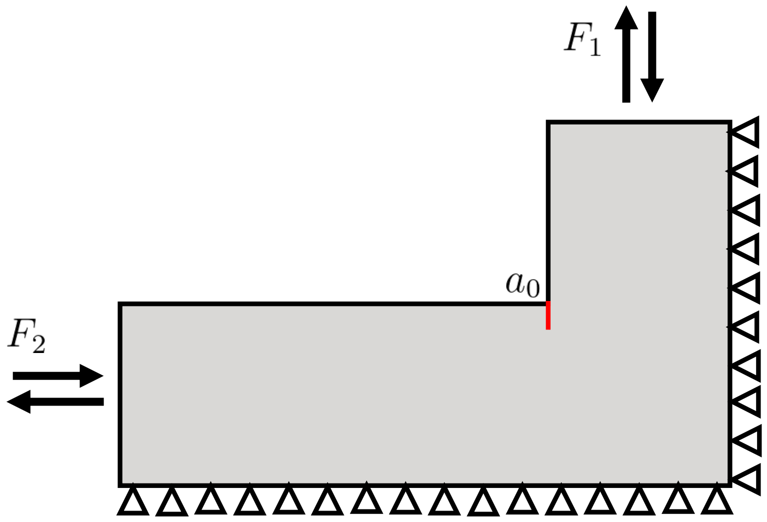
- Support varying stress ratio across crack front
- Direct calculation of stress intensity factor range for fatigue crack growth prediction

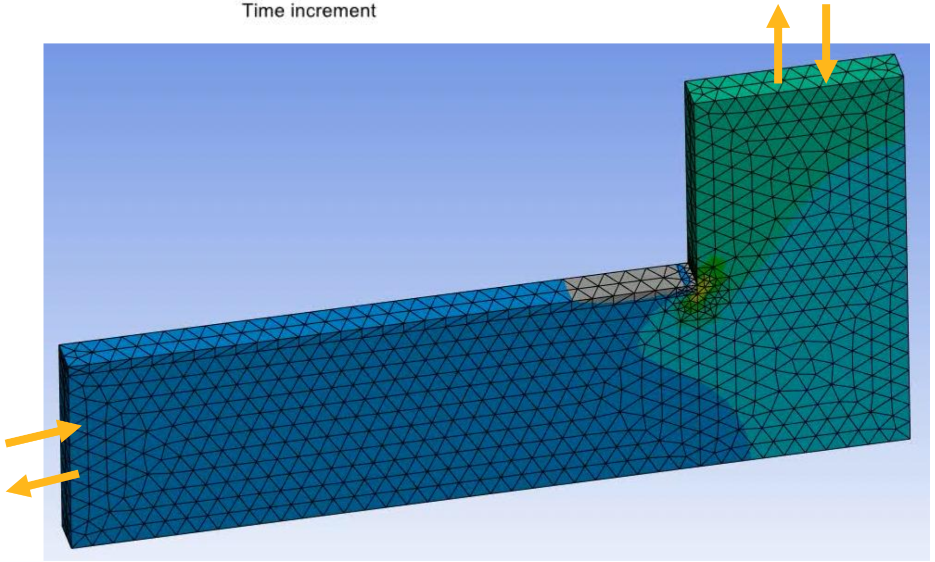
$$\Delta K_{eff} = \Delta K_{max} - \Delta K_{min}$$

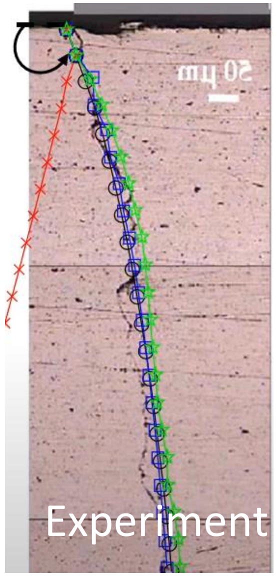
- Crack propagation direction is calculated based on
    - Maximum stress intensity factor
    - Minimum stress intensity factor
    - Local maximum circumferential stress









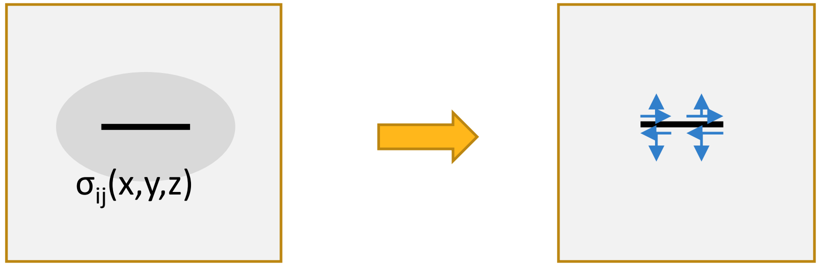


Infante-Garcia, Diego, et al., 7th International Conf. on Crack Paths, CP2021, https://www.youtube.com/watch?v=HfPMCfUNLRA

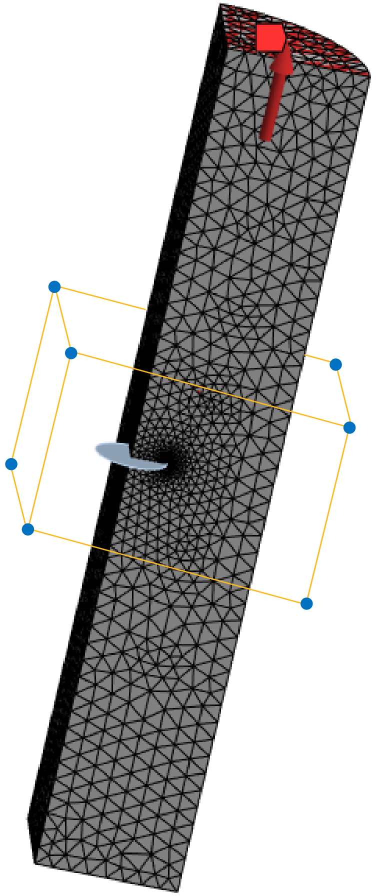
Crack-face-traction from initial stress

Approach:
- Initial stress is internally converted to crack surface traction for fracture calculation
- Support both static and fatigue crack growth
- Limited to linear application

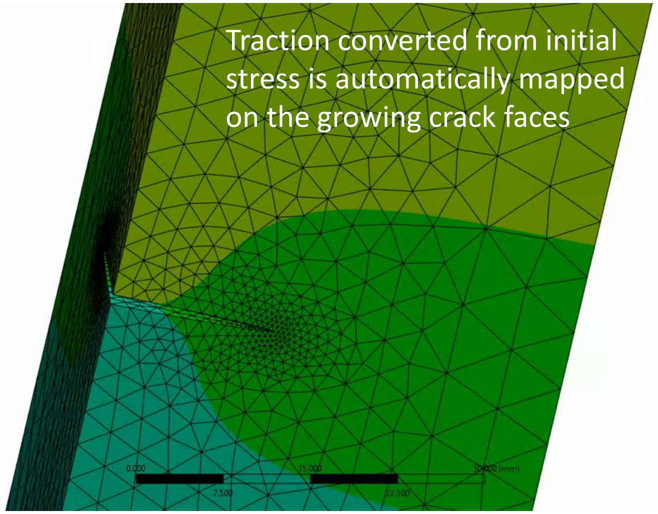
Benefit:
- Computationally faster (No Newton Raphson)
- Requires initial stress data only in the immediate vicinity of crack faces

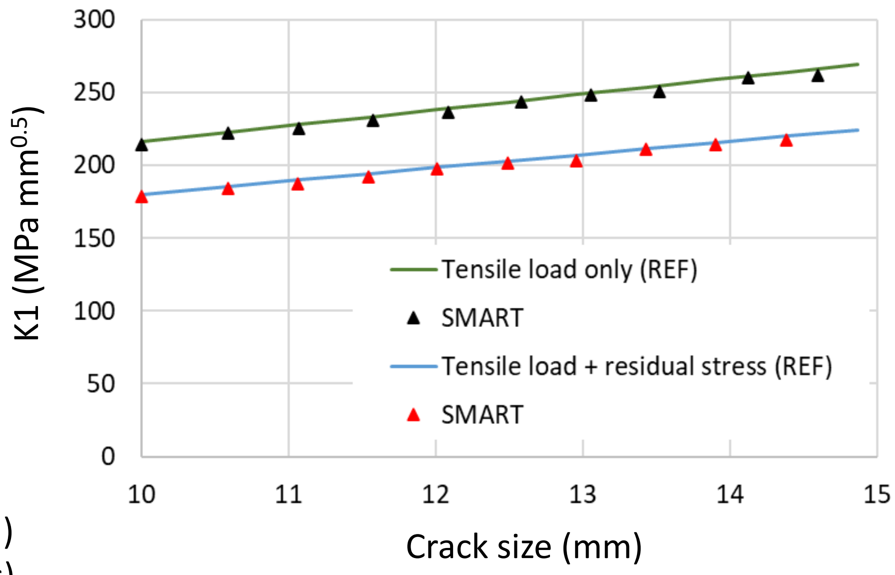


Residual stress field   Equivalent traction load at crack faces



Remote tensile load (60 MPa)
+
Initial (residual) stress (-10 MPa)
(Mesh-independent data points)





Reference: Tada, H., Paris, P.C. and Irwin, G.R., 1973. The Stress Analysis of Cracks Handbook, 3rd ed, ASME Press, 2000, pg 396.

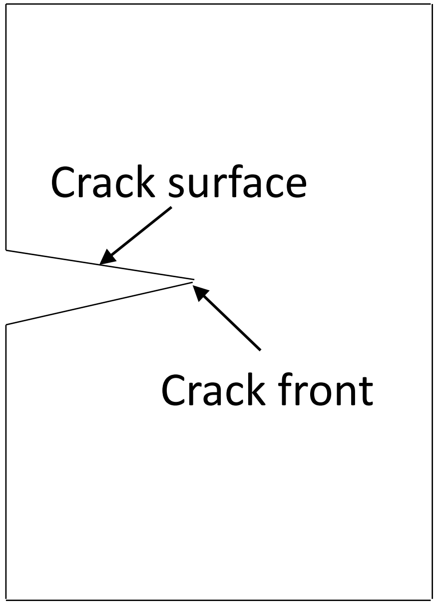
SMART with Cohesive Zone Modeling

SMART crack growth with automatically inserting of interface element (INTER204)

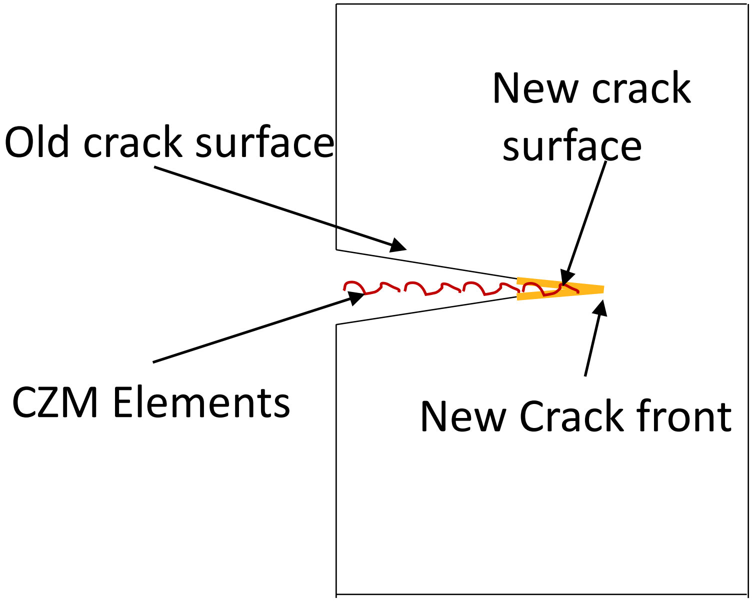
- Interface elements for initial crack surfaces
- Interface elements for new crack surfaces

Cohesive modeling with interface elements (CZM elements) can be

- Modeling the crack closure (penetration) only
- Modeling crack closure for compression and decohesion for tension

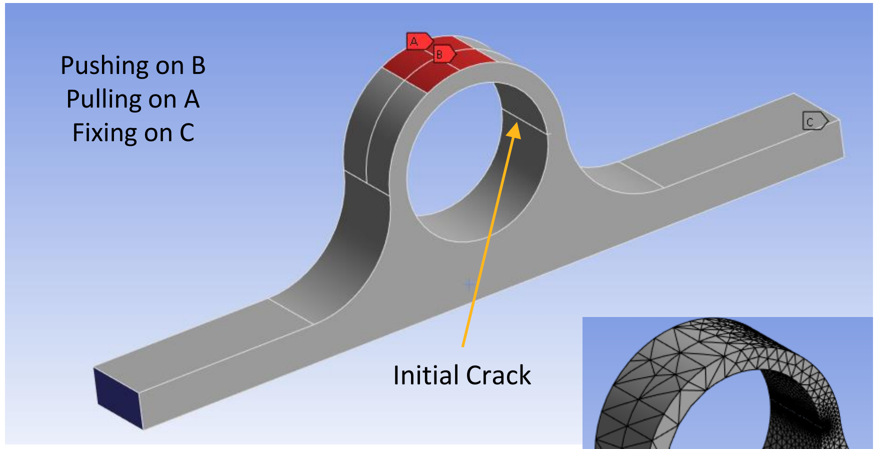


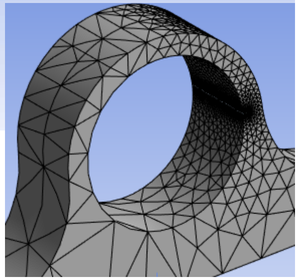
Before insertion

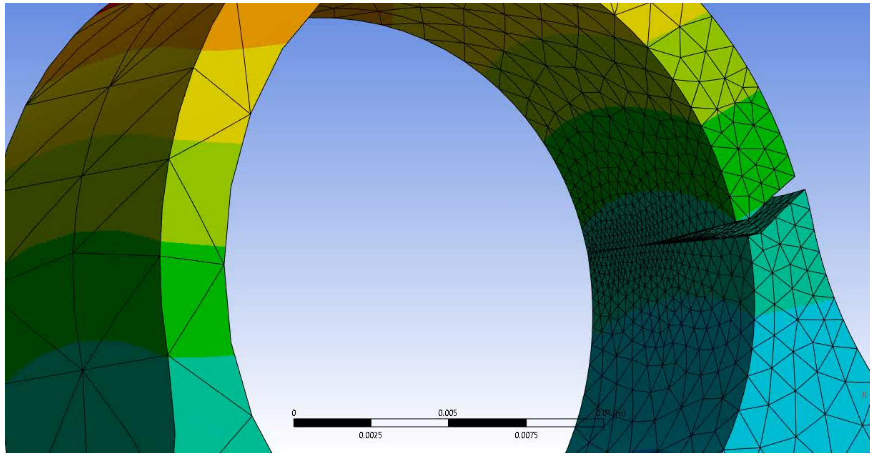


After insertion

Crack Growth under asymmetric loadings

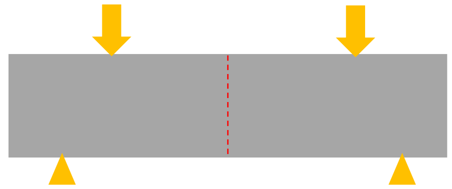




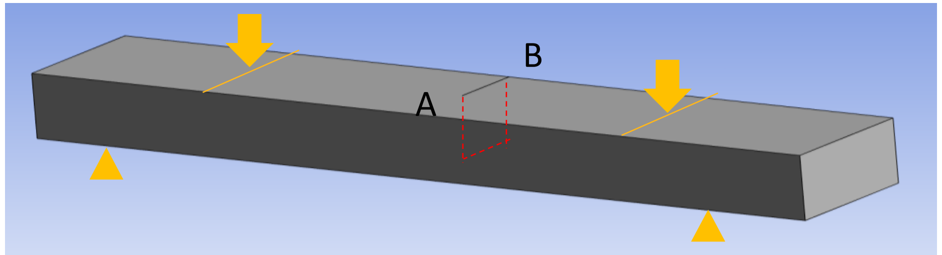


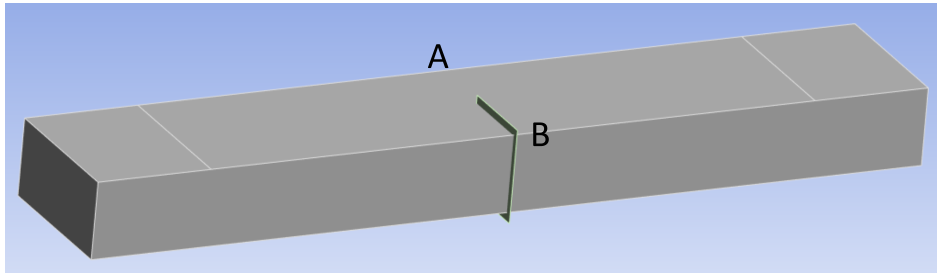
Crack Arrest Modeling

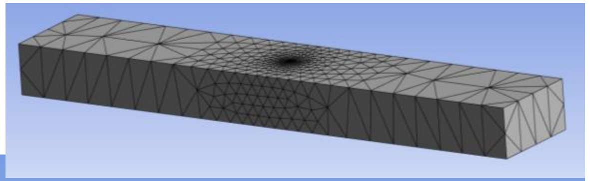
- Four point bending beam with a side crack
  - Line pressure loads are applied
    - The model forms an asymmetric load pattern to crack where top part is subjected to compression and bottom part id under tension
  - FCG is conducted
  - Paris law with threshold is used
  - CZM elements are automatically inserted at first substep and subsequent crack propagation substeps
  - CZM elements are only to use for preventing crack surface penetration
  - A small threshold value is used to prevent negative KI value used in the effective stress intensity factor range calculation

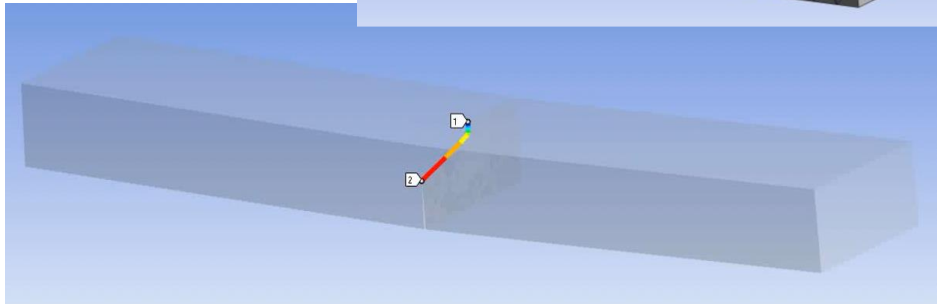


Four point bending specimen







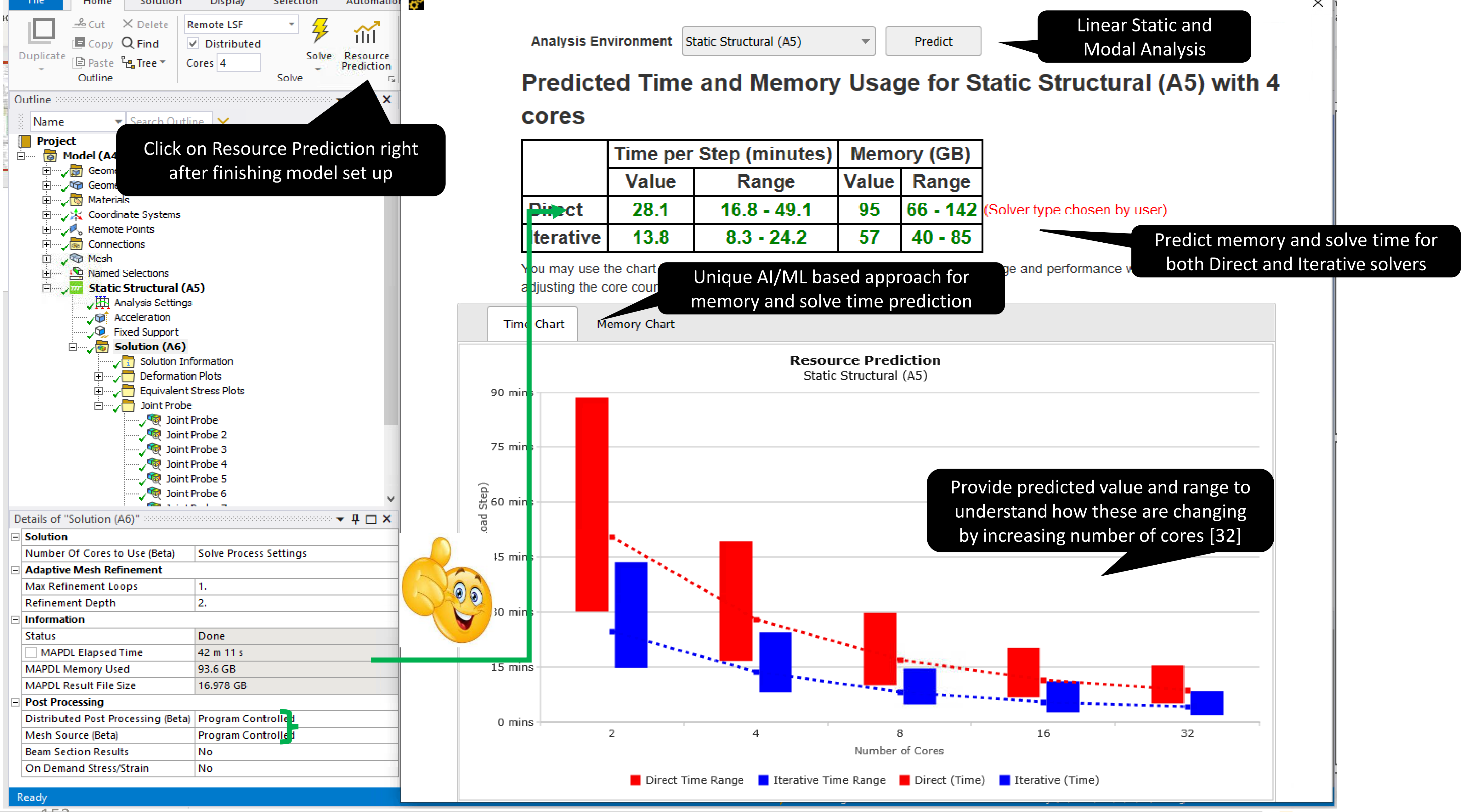


Crack front evolution with SIF K_I

MAPDL Solver

Overview

- Resource Prediction
- Krylov Method for single physics harmonic – Primary target: NVH
- Advanced Iterative Solver for single physics full harmonic – Primary target: NVH
- Support for new libraries and AMD GPU


Click on Resource Prediction right after finishing model set up
Linear Static and Modal Analysis
Analysis Environment
Static Structural (A5)
Predict
Predicted Time and Memory Usage for Static Structural (A5) with 4 cores
Time per Step (minutes)
Memory (GB)
Value
Range
Direct
28.1
16.8 - 49.1
95
66 - 142
(Solver type chosen by user)
Iterative
13.8
8.3 - 24.2
57
40 - 85
Predict memory and solve time for both Direct and Iterative solvers
Unique AI/ML based approach for memory and solve time prediction
Time Chart
Memory Chart
Resource Prediction
Static Structural (A5)
Provide predicted value and range to understand how these are changing by increasing number of cores [32]
Number of Cores
Direct Time Range
Iterative Time Range
Direct (Time)
Iterative (Time)
Details of "Solution (A6)"
Solution
Number Of Cores to Use (Beta)
Solve Process Settings
Adaptive Mesh Refinement
Max Refinement Loops
1.
Refinement Depth
2.
Information
Status
Done
MAPDL Elapsed Time
42 m 11 s
MAPDL Memory Used
93.6 GB
MAPDL Result File Size
16.978 GB
Post Processing
Distributed Post Processing (Beta)
Program Controlled
Mesh Source (Beta)
Program Controlled
Beam Section Results
No
On Demand Stress/Strain
No
Ready

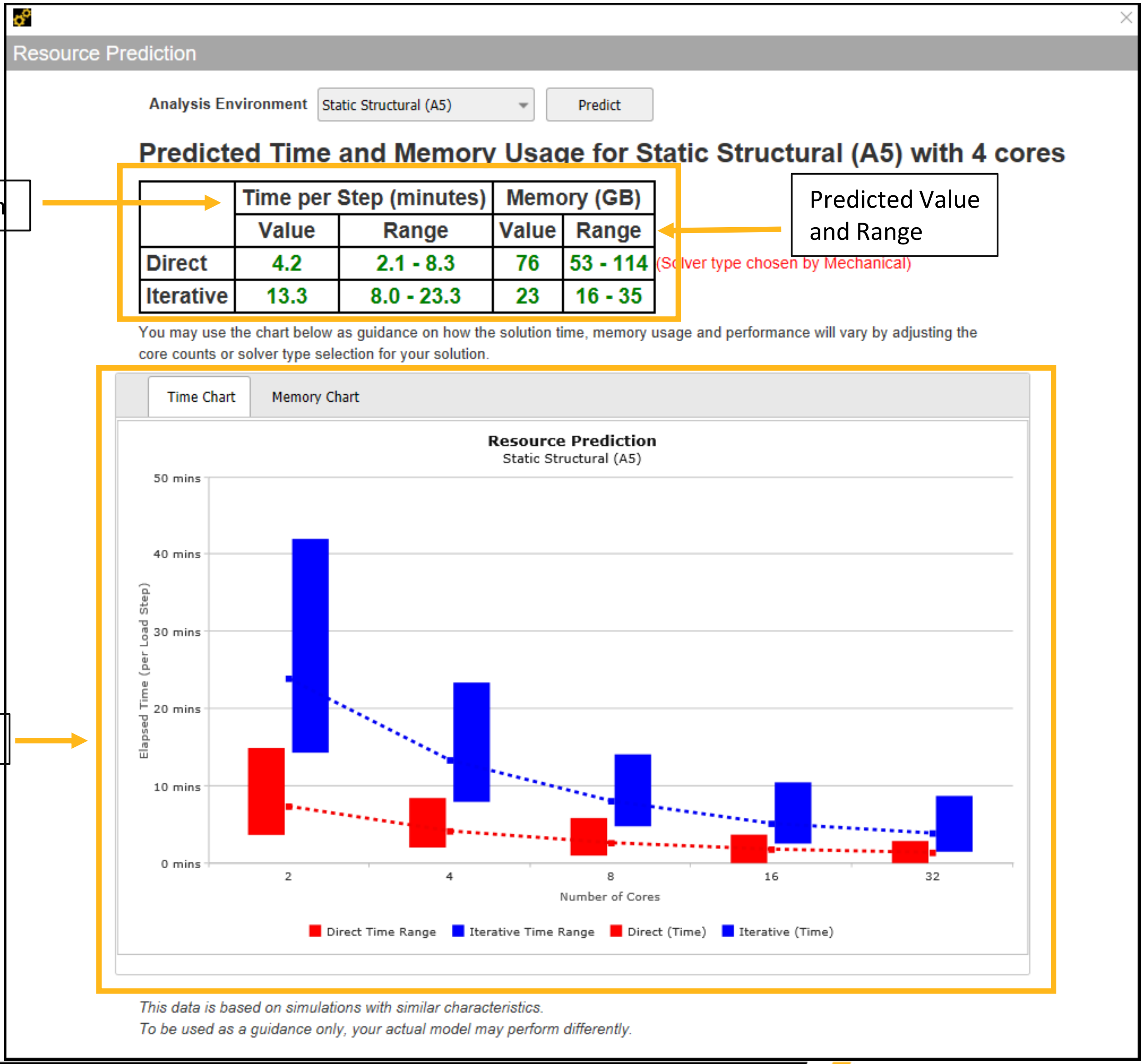
What’s New?

- Shows Time Prediction in addition to Memory Prediction.
- Displays table data that includes a predicted value as well as a range of lower and upper values for time and memory.
- Provides Time and Memory charts with both predicted value and range for better understanding of computational resources.
- Supports model imported using External Model systems

Time Prediction

Predicted Value and Range

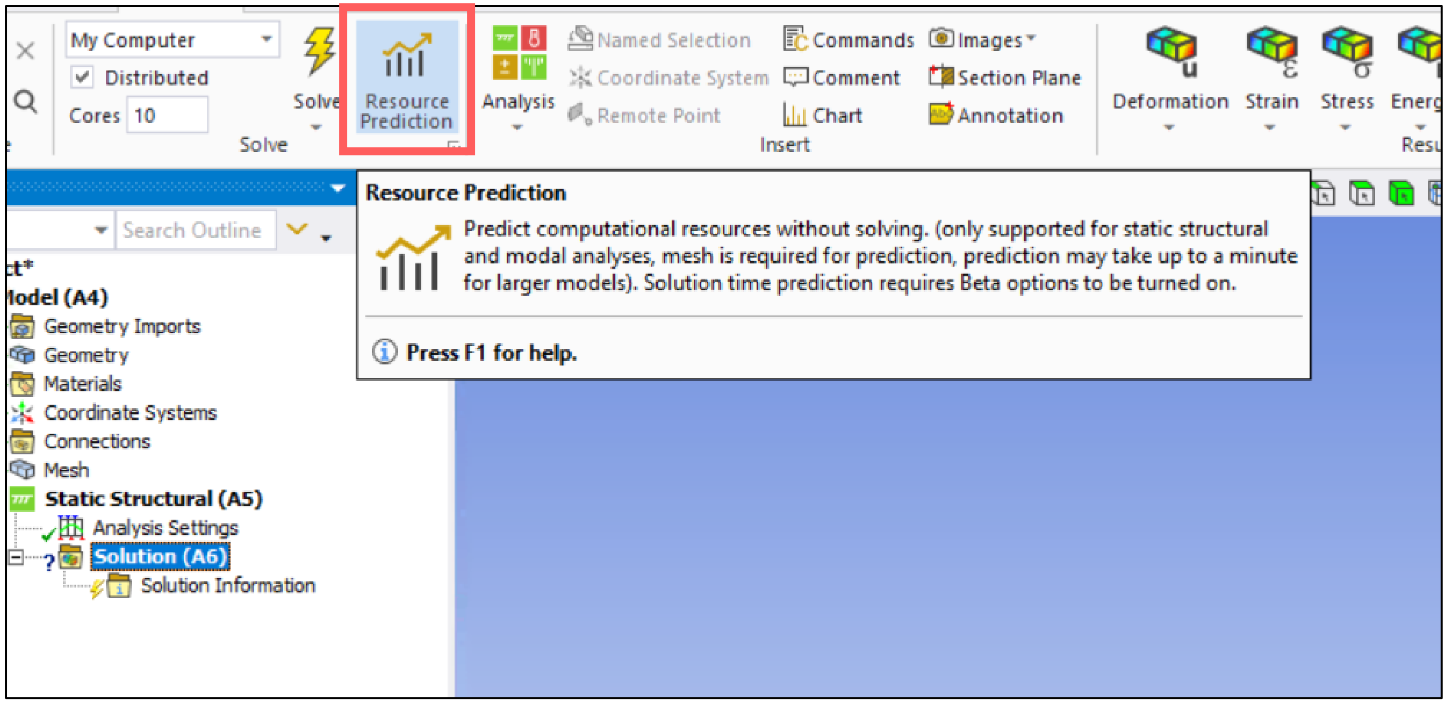
Time and Memory chart



Compute Resource Prediction

Solve time predictions are now available in addition to expected memory needed

- Exposure of resource prediction is only via Mechanical GUI
- Improved accuracy for memory requirement predictions
- No longer using fixed classification bins but instead dynamic error ranges



Resource Prediction Enhancements

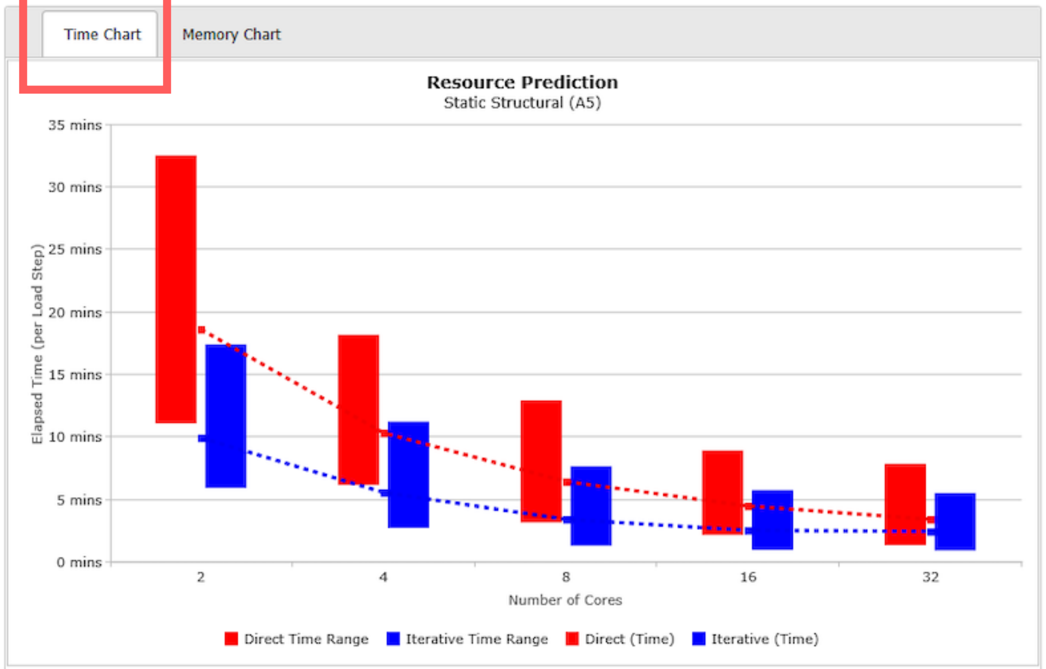
Expected solve time predictions can help

- Get a sense of how long a simulation will take
- See which equation solver will likely run faster
- Gain insights about using more CPU cores (HPC)

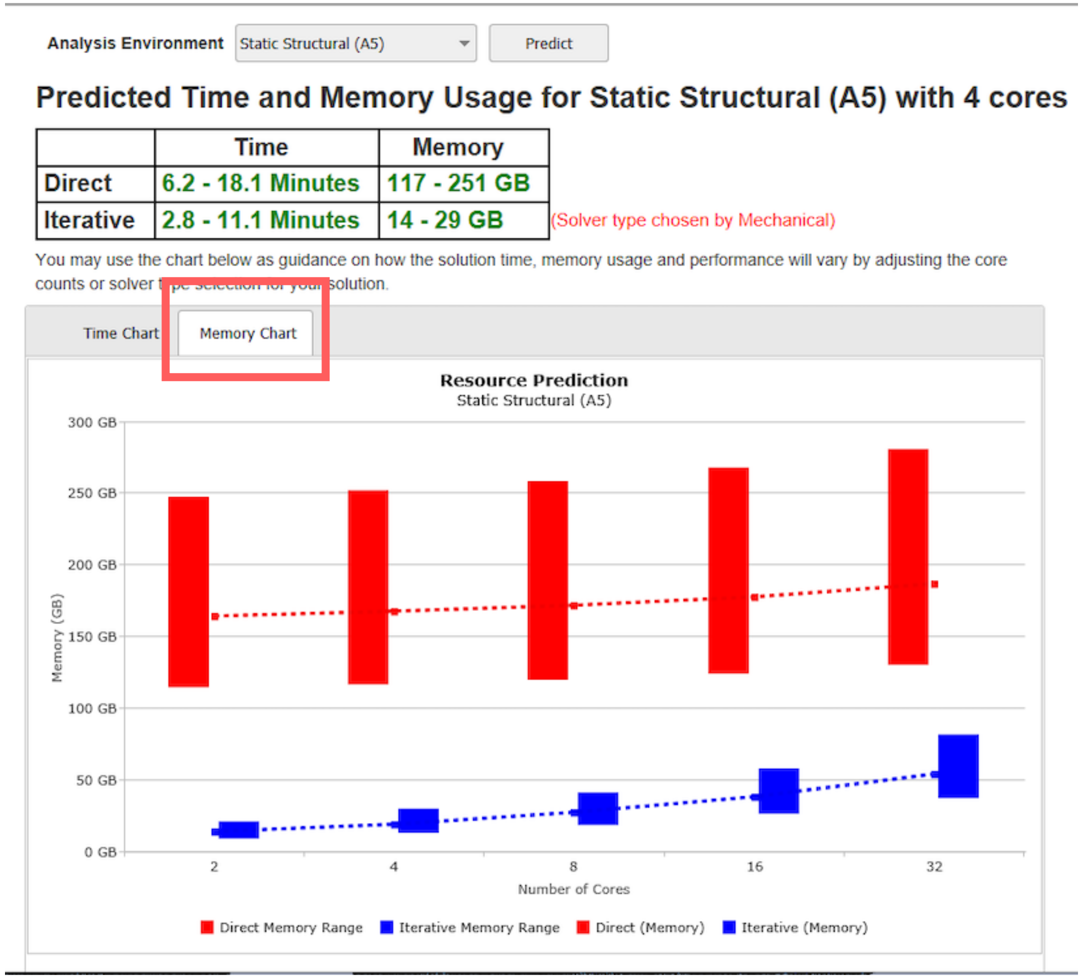
Predicted Time and Memory Usage for Static Structural (A5) with 4 cores

	Time	Memory	
Direct	6.2 - 18.1 Minutes	117 - 251 GB	
Iterative	2.8 - 11.1 Minutes	14 - 29 GB	(Solver type chosen by Mechanical)

You may use the chart below as guidance on how the solution time, memory usage and performance will vary by adjusting the core counts or solver type selection for your solution.



Compute Resource Prediction: Solver Memory Requirements



Memory predictions are improved

- Get a sense of how much memory is required
- See which equation solver will use less memory
- Gain insights about using more CPU cores (HPC)

Ansys

Harmonic Analysis Enhancements

Objective → speedup harmonic analyses

– FULL method → very computationally expensive (matrix factorization at every freq)
– VT method → reduced method, limitations (must have smooth variation of load)
– MSUP method → reduced method, limitations (frequency dependent materials)

New “reduced” method for solving harmonic analyses → KRYLOV

– Based on Krylov subspace → aims to improve performance over FULL method
– Available via new harmonic analysis option (HROPT,KRYLOV)
– Targeting single-field acoustic or structural harmonic analyses

KRYLOV Method

- **Build subspace (Q) to reduce FEA model**
  - Step 1 → Form **K**, **M**, **C**, and **f** at middle of the specified frequency range (ω_0) → symbolic assembly → .full file
  - Step 2 → Form $\check{\mathbf{K}}$ and $\check{\mathbf{C}}$ where $\check{\mathbf{K}}(\omega_0) = \mathbf{K} - \omega_0^2\mathbf{M} + i\omega_0\mathbf{C}$ and $\check{\mathbf{C}}(\omega_0) = \mathbf{C} + 2i\omega_0\mathbf{M}$
  - Step 3 → Factor $\check{\mathbf{K}}^{-1}$ using direct solver
  - Step 4 → Compute $\mathbf{v}_1 = \mathbf{g}_1/|\mathbf{g}_1|$ where $\mathbf{g}_1 = \check{\mathbf{K}}^{-1}\mathbf{f}$ and $\mathbf{f}(\omega_0) = \mathbf{f}_r + i\mathbf{f}_i$
  - Step 5 → Compute $\mathbf{v}_2 = \breve{\mathbf{g}}_2/|\breve{\mathbf{g}}_2|$ where $\breve{\mathbf{g}}_2=\breve{\mathbf{g}}_2-(\mathbf{v}_1\bullet\breve{\mathbf{g}}_2)\mathbf{v}_1$ $\breve{\mathbf{g}}_2 = \mathbf{g}_2/|\mathbf{g}_2|$ and $\mathbf{g}_2 = -\check{\mathbf{K}}^{-1}\check{\mathbf{C}}\mathbf{g}_1$
  - Step 6 → Compute $\mathbf{v}_n = \breve{\mathbf{g}}_n/|\breve{\mathbf{g}}_n|$ where $\breve{\mathbf{g}}_n=\breve{\mathbf{g}}_n-(\mathbf{v}_{n-1}\bullet\breve{\mathbf{g}}_n)\mathbf{v}_1$ $\breve{\mathbf{g}}_n = \mathbf{g}_n/|\mathbf{g}_n|$ and $\mathbf{g}_n = -\check{\mathbf{K}}^{-1}(\check{\mathbf{C}}\mathbf{g}_{n-1} + \mathbf{M}\mathbf{g}_{n-2})$ *for n=3,dimQ*

- Steps 5 & 6 use Modified Gram-Schmidt Orthonormalization
  - Generates a set of complex-value basis vectors $\{\mathbf{v}_1,\mathbf{v}_2,\ldots,\mathbf{v}_n\}$ that are almost orthonormal
  - *Structure preserving dimension reduction* → subspace **Q** = span($\mathbf{v}_1, \mathbf{v}_2,\ldots,\mathbf{v}_n$) is used to reduce the harmonic FEA

KRYLOV Method

- **Reduce system of equations and solve**
  - Step 1 → Form **K**, **M**, **C**, and **f** at the given frequency value (ω_i) → symbolic assembly → .full file
  - Step 2 → Build **A** where $\mathbf{A}(\omega_i) = \mathbf{K} - \omega_i^2\mathbf{M} + i\omega_i\mathbf{C}$
  - Step 3 → Use subspace **Q** to reduce **A** and **f** at each frequency value → $[\mathbf{Q}^T\mathbf{A}(\omega_i)\mathbf{Q}]\mathbf{y} = \{\mathbf{Q}^T\mathbf{f}(\omega_i)\}$
  - Step 4 → Solve (dense matrix) reduced system via LAPACK to compute **y** at each frequency value

Project **K, C, M, f** on $\mathbf{Q}=\{\mathbf{v}_1, \ldots, \mathbf{v}_n\}$ to obtain reduced system at frequency ω

System of equations in Step 4 will be [dimQ] in rank → extremely fast

KRYLOV Method

- **Expand system of equations back to FEA model**
  - Step 1 → Expand $\mathbf{y}_i$ → $\mathbf{x}_i = \mathbf{Q}\mathbf{y}_i$ at each frequency value (ω_i)
  - Step 2 → Compute residual $\mathbf{R}_i$ where $\mathbf{R}_i = \mathbf{F}_i - \mathbf{A}_i \mathbf{x}_i$ where $\mathbf{A}_i = \mathbf{K} - \omega_i^2\mathbf{M} + i\omega_i\mathbf{C}$
  - Step 3 → Normalize residual $\mathbf{R}_i$ where $\mathbf{R}_i = |\mathbf{R}_i| / |\mathbf{F}_i|$

KRYLOV Method

New commands

HROPT, *Krylov, FrqVal*

- Enable the Krylov method for the harmonic analysis

- [Optional] Specifies the frequency at which to generate the subspace (defaults to middle of supplied frequency range)

KRYOPTION, *MaxDim,,, ResTol, CheckOrtho*

- Optional parameters to control the following:

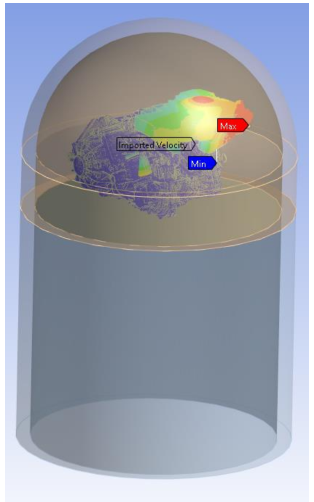
Maximum subspace dimension

Tolerance used for the residual check

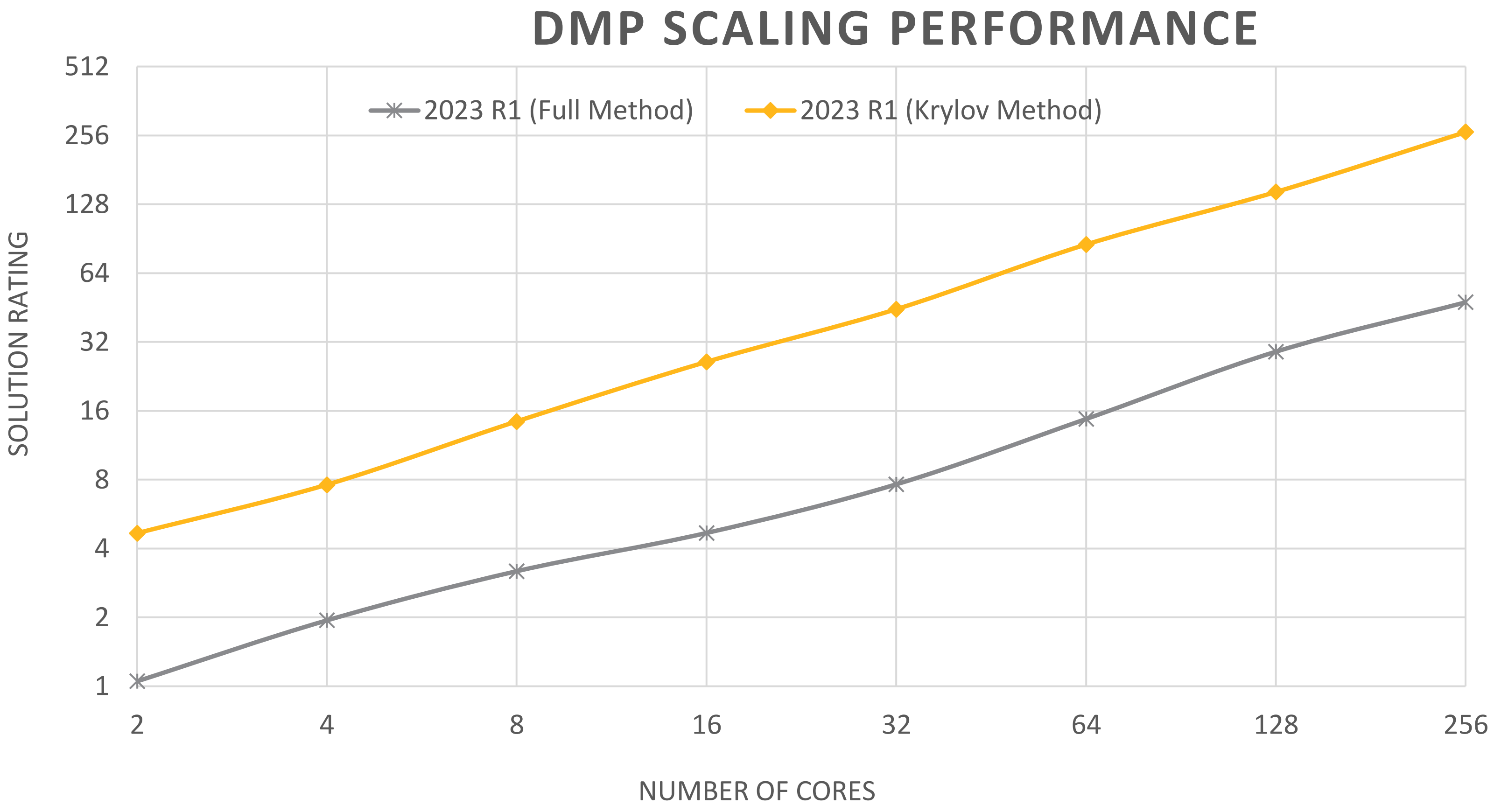
Activate the check on the orthogonality of the subspace vectors

KRYLOV Method

- Significantly faster than using the FULL method for harmonic analyses



- 1.8 MDOF; Sparse direct solver
- Harmonic analysis with 122 frequency points from 0 to 2000 Hz
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6142 processors (32 cores), 384GB RAM, SSD, CentOS 7.9, Mellanox Infiniband



Miscellaneous Enhancements

- Upgraded to v3.2.1 BLIS library for AMD processors
  - Initial support for AVX-512 instructions for Genoa and future architectures by AMD
- Upgraded to CUDA 11.7 libraries for NVIDIA GPUs

Beta Features

- PCG iterative solver now supports harmonic analyses (BETA)
  - Complex-value operations now supported in the PCG solver path
  - Works well for single-field structural and acoustic models
  - Fluid-structure interaction supported with some limitations
  - Can significantly decrease the memory requirements over using the direct solver but still fully supports DMP and thus is significantly faster than using the ICCG or QMR solvers

- PCG iterative solver now supported with AMD GPUs (BETA)
  - Sparse-matrix vector operation is now offloaded onto AMD GPUs
  - Up to 10x faster operations for this offloaded solver kernel with same accuracy as CPU-only path

Linear Dynamics

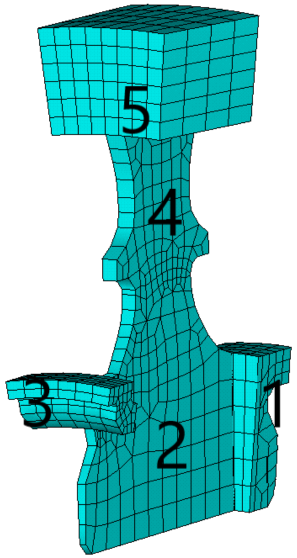
Ansys

Overview: Solver + UI/UX

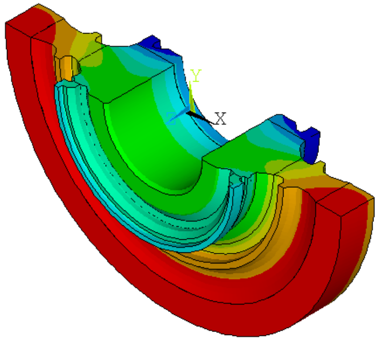
- Multistage Cyclic Symmetry Analysis
- Substructuring/CMS
- Harmonic Balance Method (Beta)
- Elcalc support in spectrum analysis (Beta)
- Residual Vector Support with QRDAMP (Beta)
- Miscellaneous

Multistage Cyclic Symmetry Analysis

- Stage Connections
  - Many stages can be connected to any given stage, overcoming the prior limit of two stage connections per stage. This allows models of greater complexity to be easily assembled. This also works with multiharmonic stages having multiple stage clones.

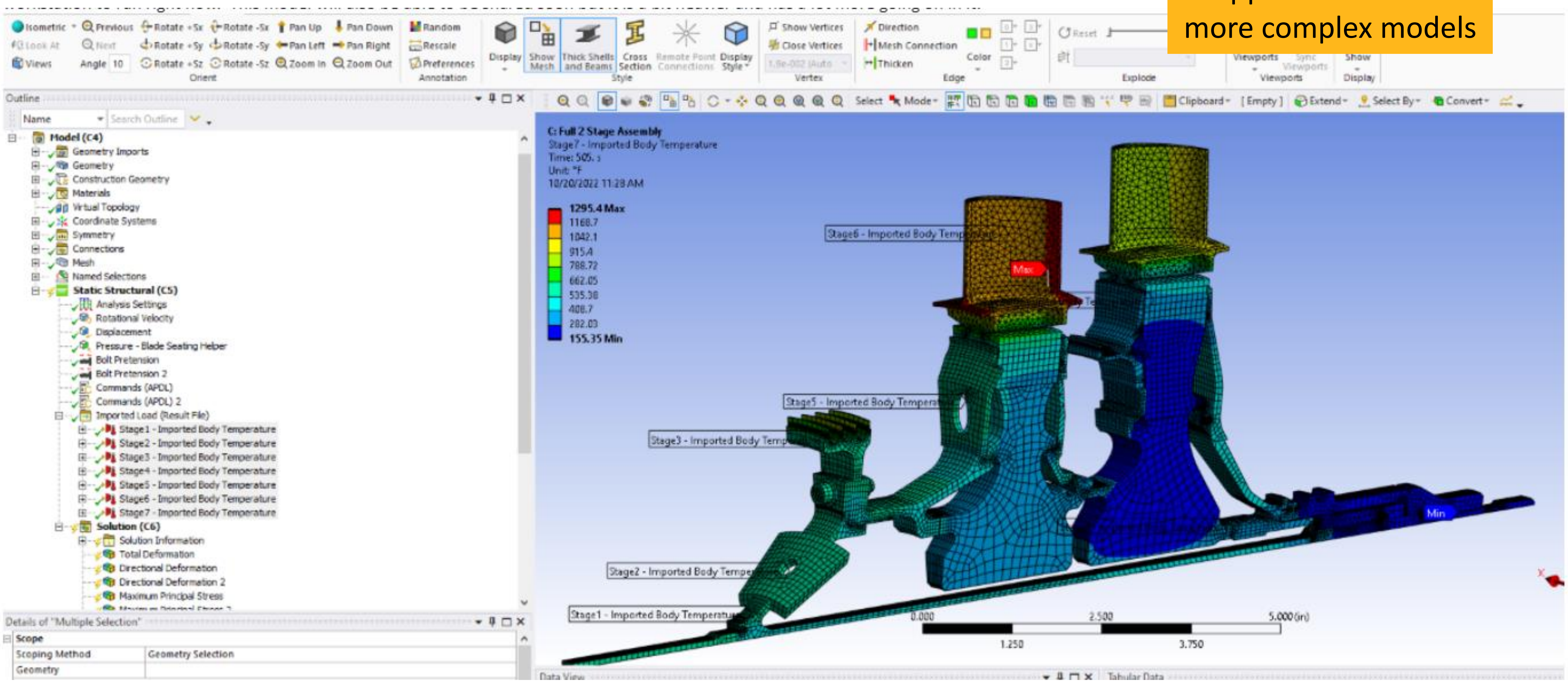


Multistage 5-Stage Model
Stage 2 With 3 Connections



Cross-Section of Expanded Displacements

Support Mechanical more complex models



Multistage Cyclic Symmetry Analysis

- Tabular Loads
  - Node and Element based tabular loads applied to harmonic index 0 base sectors in a static analysis are automatically copied to the harmonic index 0 duplicate sectors and stage clones.

Support Mechanical Imported Loads

WB Mechanical Imported Load Tabular Data

```
/com,*********** Create Load Variation Table for Imported Load "Imported Body Temperature" ***********
*DIM, _lv_254__0, TABLE, 2, 54395, 1, TIME, NODE, ,
*PREAD, _lv_254__0, 163188
  0.000000000000e+00  0.000000000000e+00  1.000000000000e+00  1.000000000000e+00
  2.200000000000e+01  9.978023529053e+01  2.000000000000e+00  2.200000000000e+01
  9.813773345947e+01  3.000000000000e+00  2.200000000000e+01  9.943675994873e+01
  4.000000000000e+00  2.200000000000e+01  9.919849395752e+01  5.000000000000e+00
…
```

Multistage Cyclic Symmetry Analysis

- Multistage Multiharmonic Harmonic Selection Tool (Beta)
  - A python based tool offers guidance as to which harmonics for each stage may contribute to the overall multistage solution.

Help customer pick the relevant harmonics

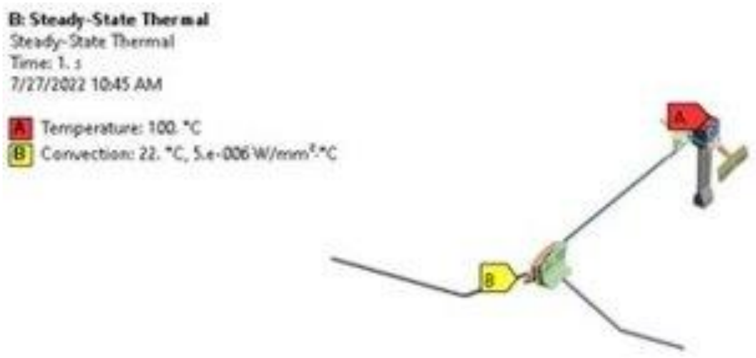
```
# %% USER INPUT

# >>>>>>>>>>>>>>>>>>>><<<<<<<<<<<<<<<<<<<<<<< #
# >>>>>>>>>>>>>>> USER INPUT <<<<<<<<<<<<<<<< #
#                                             #
N_sectors = [8,16,4]     # Number of sectors for each stage
mainHI     = 1          # Main harmonic index
connection = [[2],      # Connections relating stages
              [1,3],
              [2]]
#                                             #
# >>>>>>>>>>>>>>> USER INPUT <<<<<<<<<<<<<<<< #
# >>>>>>>>>>>>>>>>>>>><<<<<<<<<<<<<<<<<<<<<<< #
```

User Input for HI Selection

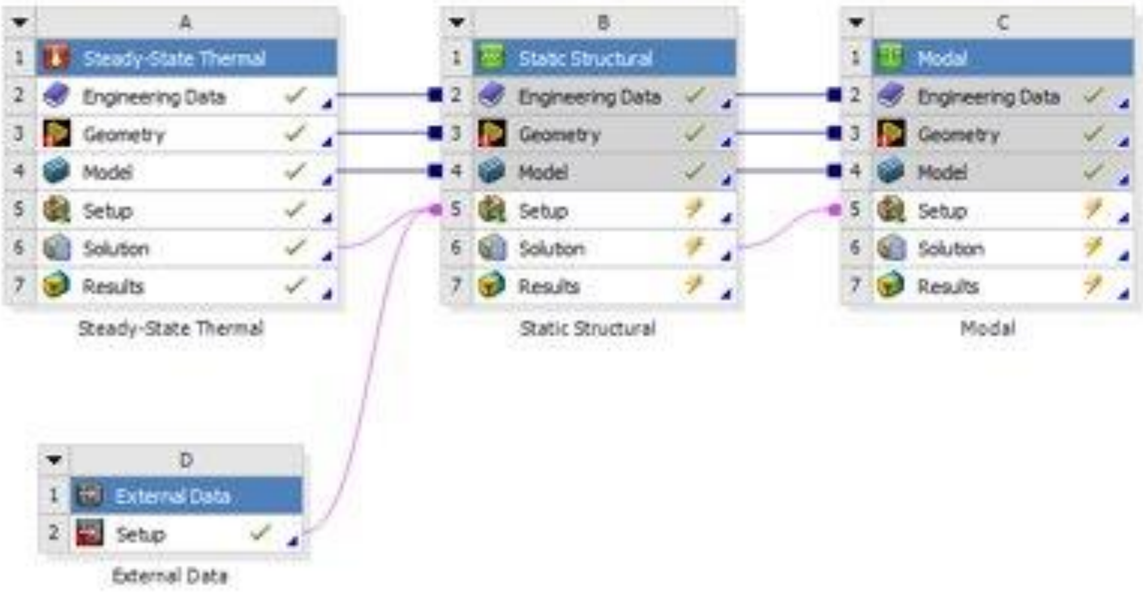
```
*****************************************************************
**** Main HIs + extra HIs based on nodal diameters matching *****
*****************************************************************

HIs stage 1: [1, 3]
HIs stage 2: [1, 3, 5, 7]
HIs stage 3: [1]
```

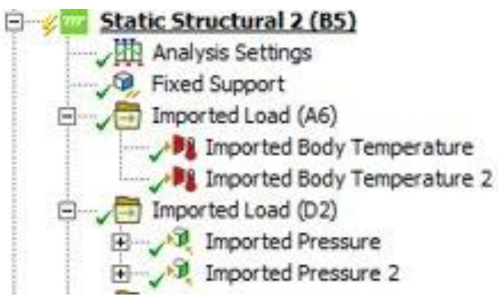
Output for HI Selection

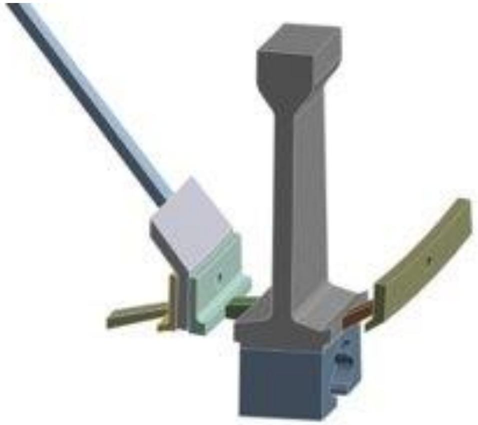
Multistage Exposure in Mechanical



- Allow multistage for thermal analysis (HI=0)
- Stage can be connected to more than one upstream & one downstream stage
- Allow multiple harmonic indices for Static Structural analysis
- Allow Imported loads in Thermal & Static Structural (beta)







All Connect

Improved performance for Substructuring Analyses

Improve default settings for better performances

In R18.0 the default for the maximum number of load vectors that can be generated and stored in Sename.sub file, NUMSUBLV, has been increased from 31 to 1000.

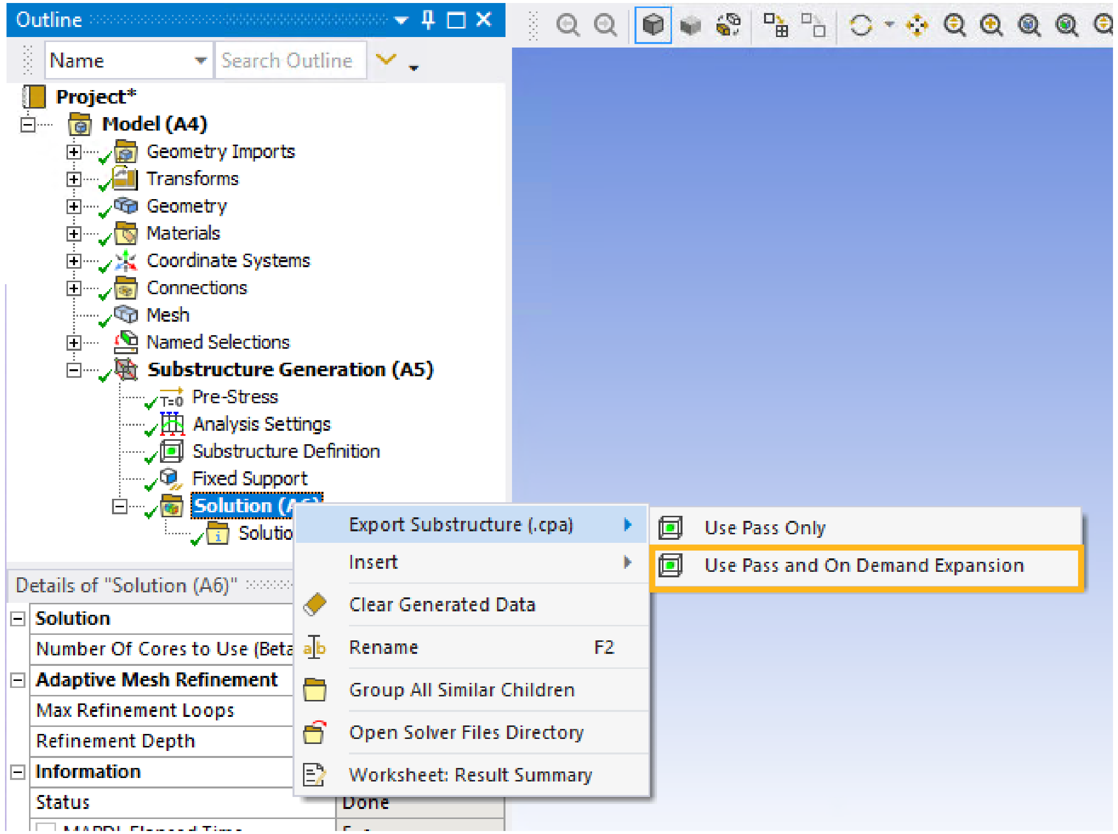
After developments done in R19.2, in the .esav file of a use pass, a record size associated to any superelement has been increased, which could slow down performance when a large number of time points are calculated during the use pass. A workaround was provided into the documentation to overcome this issue.

Caution: In the use pass, an **.ESAV** file record of size 2*NUMSUBLV*NUMSUBLV is stored for each load step. Consequently, if the use pass analysis has a high number of load steps (for instance, a transient analysis with thousands of time steps), the default or higher value for NUMSUBLV can significantly slow it down. To prevent this, set the NUMSUBLV argument of **/CONFIG** before the first generation pass to limit the maximum number of load steps in all generation passes. For example, issue **/CONFIG**,NUMSUBLV,2 if no more than 2 load steps are done in all generation passes.

To improve performance, in R23.1 the default NUMSUBLV has been changed from 1000 to 31 in MAPDL and in Mechanical, and the workaround is no more needed.

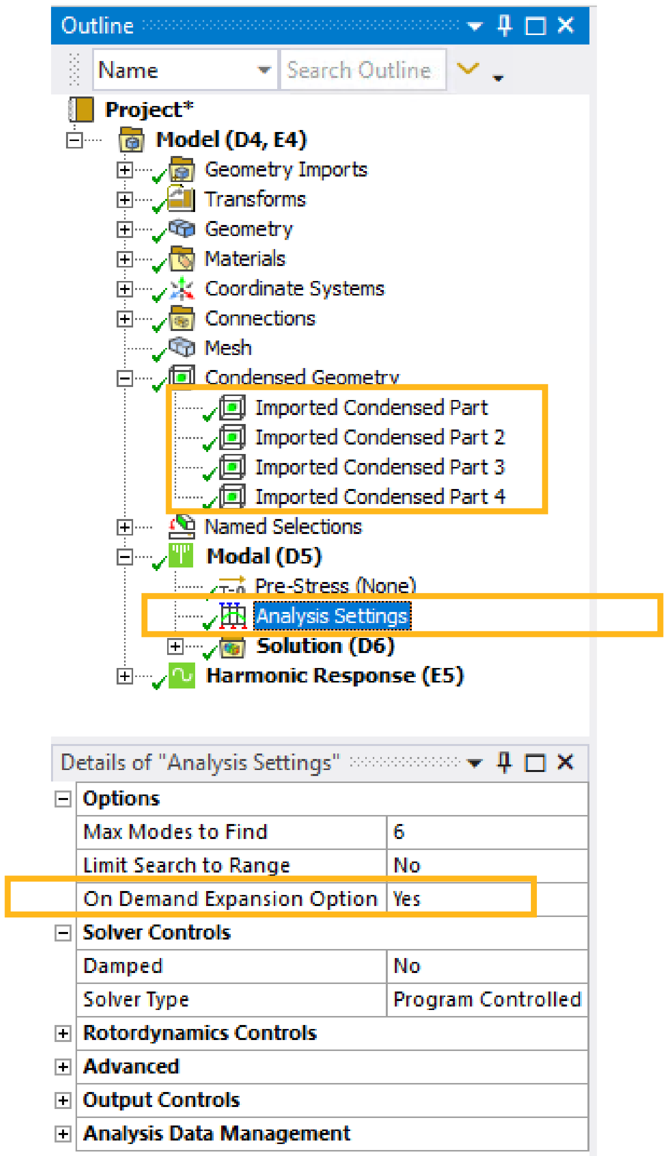
Bottom-up CMS: Export files for On Demand Expansion

- In Substructure Generation analysis, files required for Use Pass and On Demand Expansion can be exported as a .cpa substructure file.



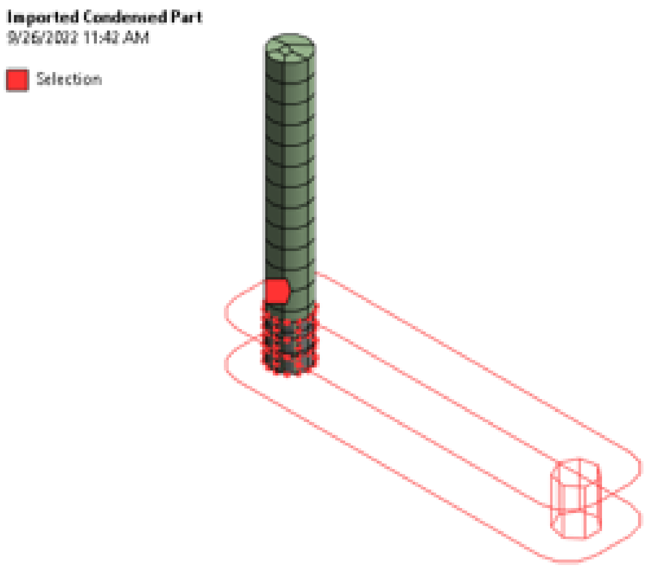
Bottom-up CMS: Import Substructure file

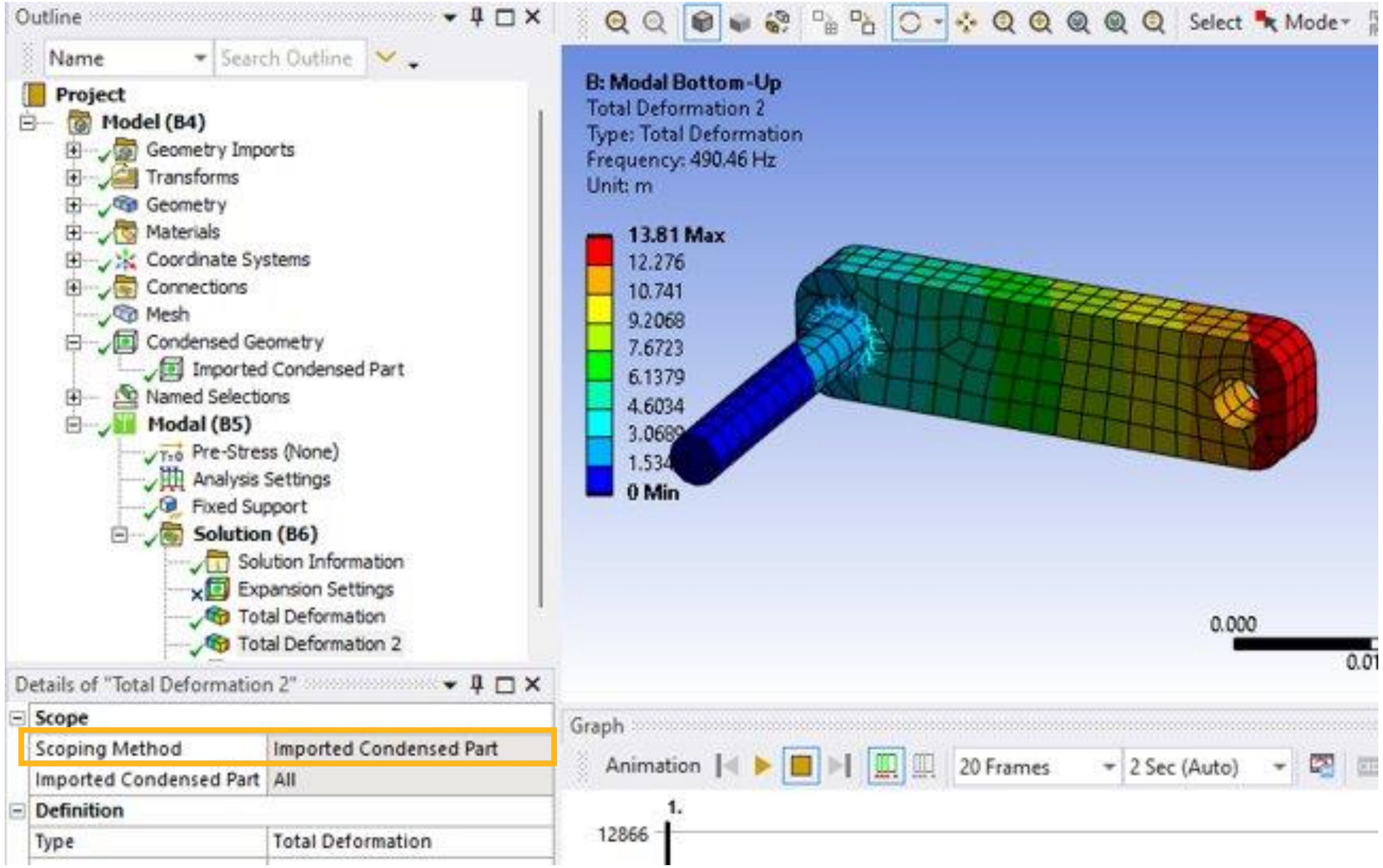
- The substructure file (*.cpa) can be imported using the Imported Condensed part object.
- The On Demand Expansion Option must be set to Yes for expansion to work with Imported Condensed Part.



Bottom-up CMS: Perform on demand expansion

- Results can be expanded on the Imported Condensed part by selecting the scoping to Imported Condensed part



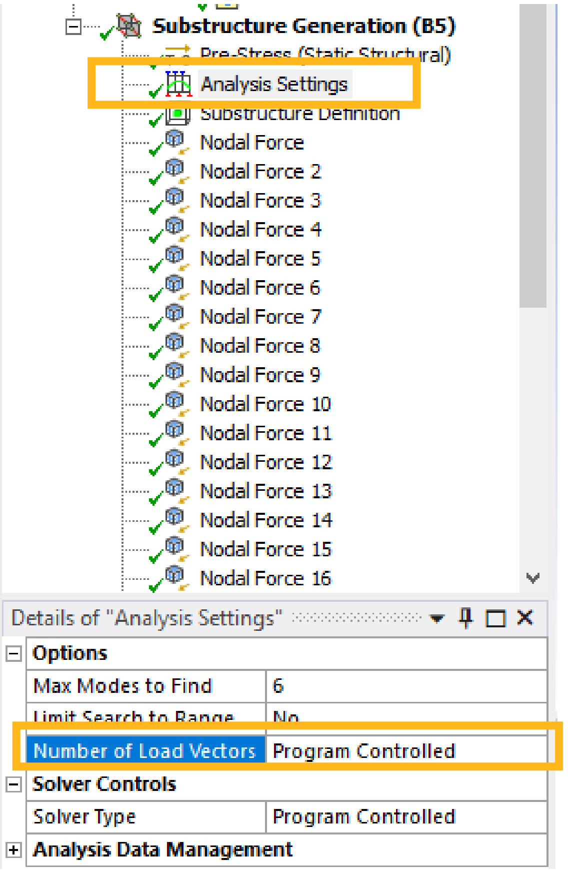


Ansys

Bottom-up CMS: Load vector limit for Substructure Generation

- **Number of Load Vectors** property is available for Substructure Generation analysis in Analysis Settings object
- When set to 0, which is the Program Controlled value and is the default, the application automatically detects the number of load vectors. The user can also specify the value using this property
- This setting enables efficient storage of load vector data and improves memory and performance of the application if this data is needed for many use pass steps.
- Based on the user specified input, /config,numsublv command is written to the input file

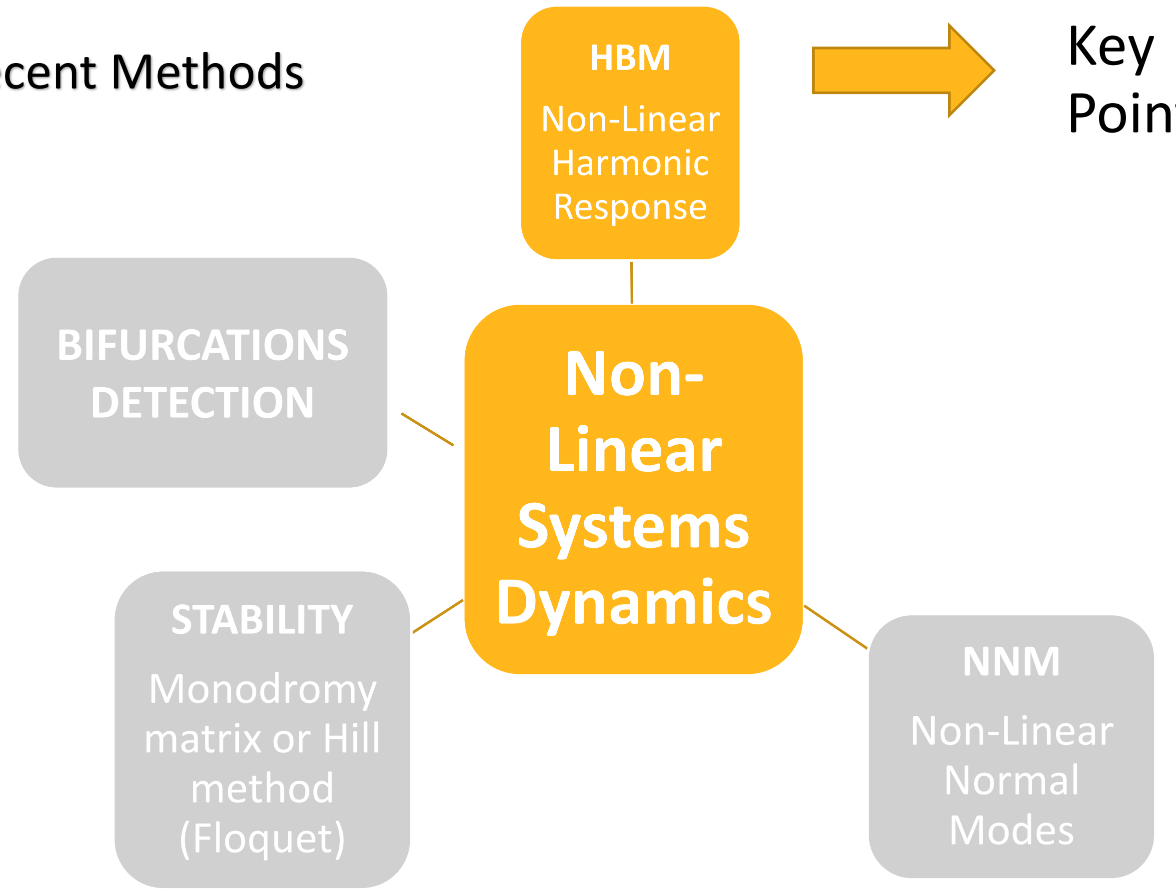
```
/batch
/config,noeldb,1        ! force off writing results to database
/config,numsublv,32
*get,_wallstrt,active,,time,wall
! ANSYS input file written by Workbench version 2023 R1
resume,file,rdb
```



Ansys

Scope of Harmonic Balance Method Initiative - beta

Recent Methods



Key Points	Model is mostly linear with some localized non-linearity such as friction dampers. Only periodic solutions are of interest.
	Transient analysis is the currently only solution, but it is costly: large number of runs, long runs to reach steady-state.
	HBM provides an efficient way to determine the multi-harmonic solution.

Harmonic Balance Method Workflow

Pre-processing

- Linear elements and materials: no specifics - CMS superelements are also supported
- Non-linear elements:
  - COMBIN39
  - COMBIN40
  - USERELEM
  - CONTA178

Solver

New option for harmonic analysis HROPT, HBM

HARMONIC

AFT

TRANSIENT

Results Files

rst HI 0

rst HI 1

rst HI 2

. . .

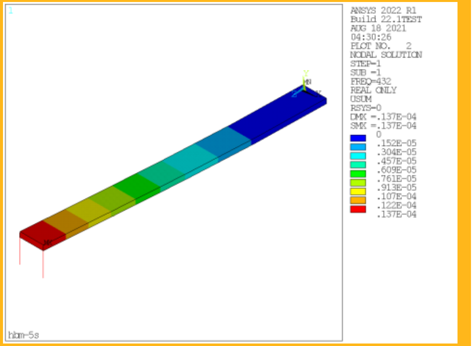
rst HI N

MAPDL script

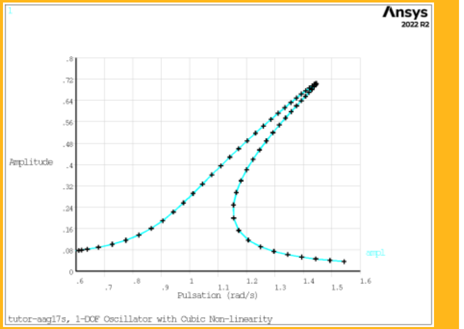
Mechanical DPF

Post-processing

Displacement solution at given frequency



Frequency Response Function



Each harmonic result file has the same format as for a regular harmonic analysis. No specifics for the post-processing of 1 harmonic.

2023R1 Example: Simplified model of blade-disk assembly with under platform damper [Poudou thesis 2007]

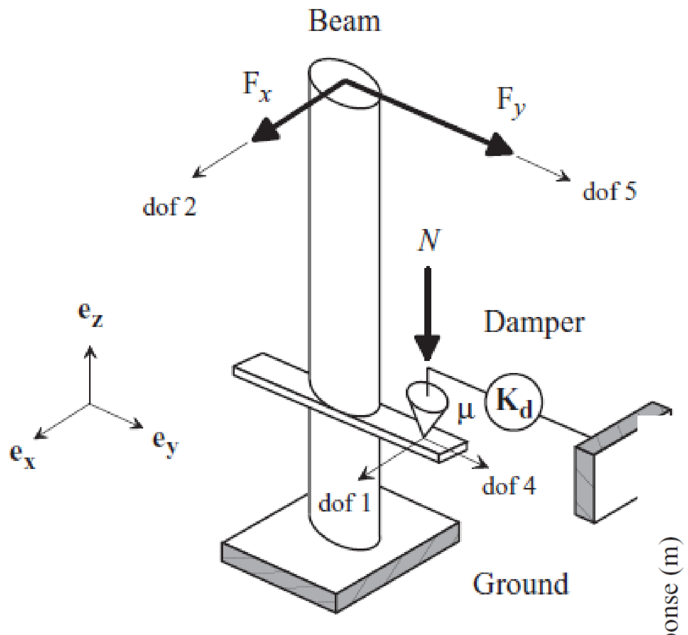
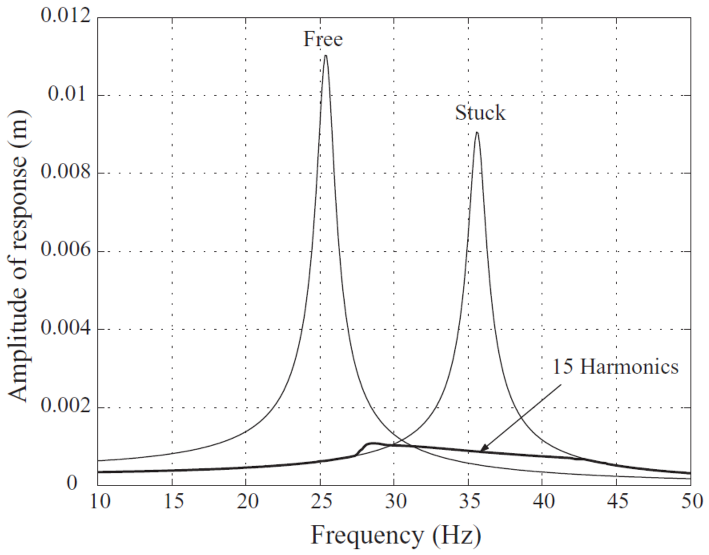
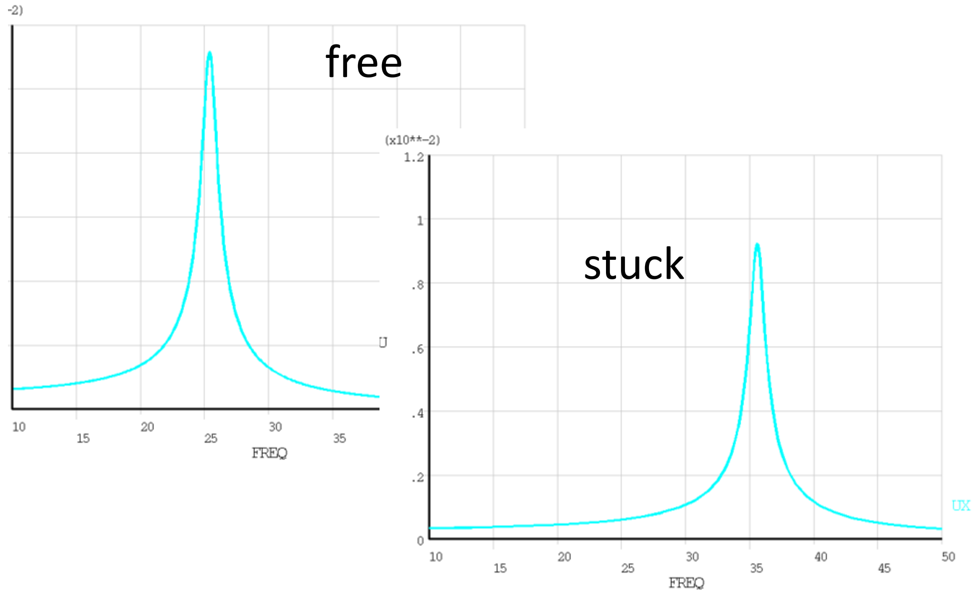


Figure 3.1: Simple beam model with a friction damper

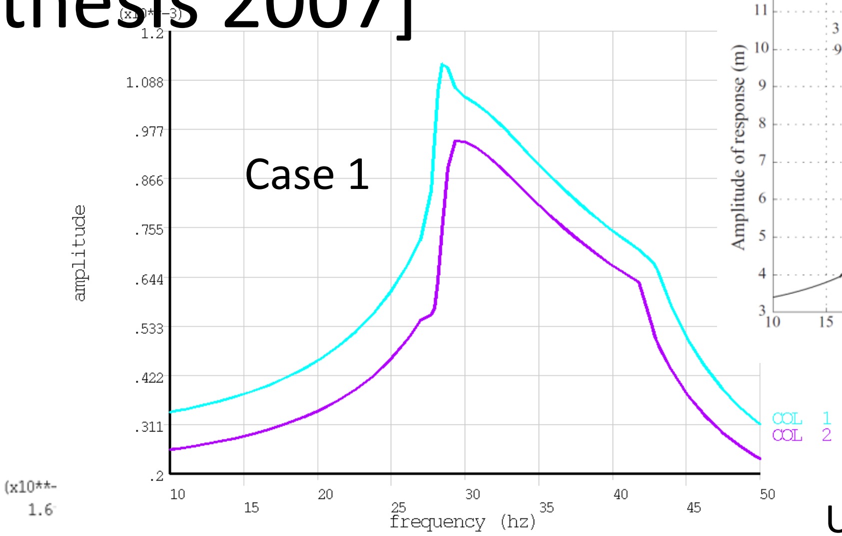
- Reduced order model of a beam:
  - 2 master nodes: tip of the beam and friction point
  - 2 normal modes
- Only transverse displacement allowed - UZ=0
- Friction is 2D - constant normal force
- Reduced matrices are provided

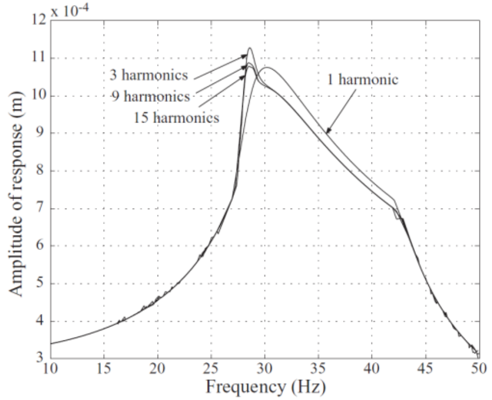




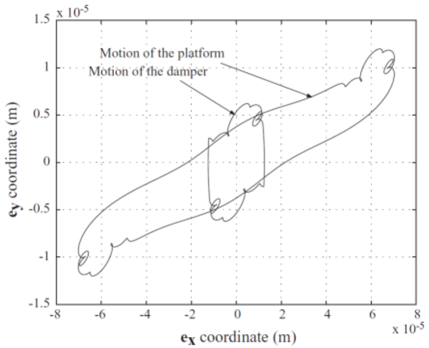
2023R1 Example: Simplified model of blade-disk assembly with under platform damper [Poudou thesis 2007]

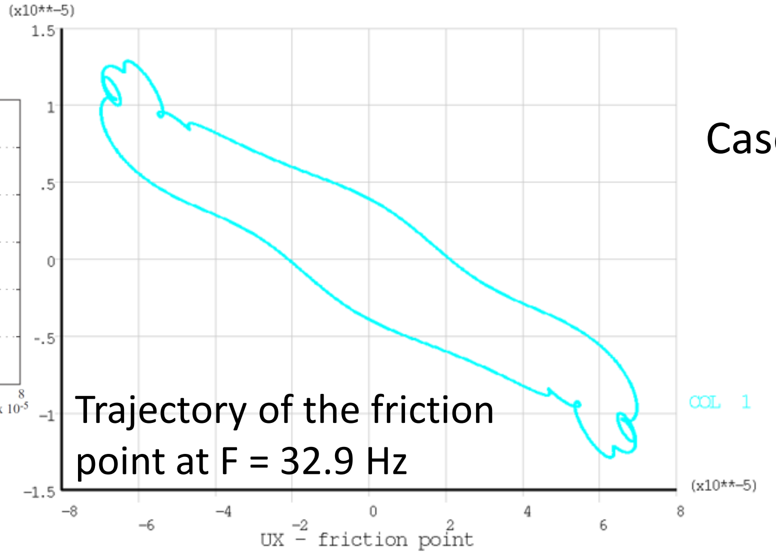
- Case 1: FX=20N, FY=15N
  - Exhibits true 2D friction behavior
  - Almost no stick-slip behavior (pure stick or pure slip only)
  - Low number of harmonics needed (NH=3)
- Case 2: FX=20N, FY=2N
  - Could be modeled with 1D friction
  - Comparison to 1D friction shows good matching
  - Stick-slip behavior on almost the whole frequency range
  - High number of harmonics needed (NH=15)





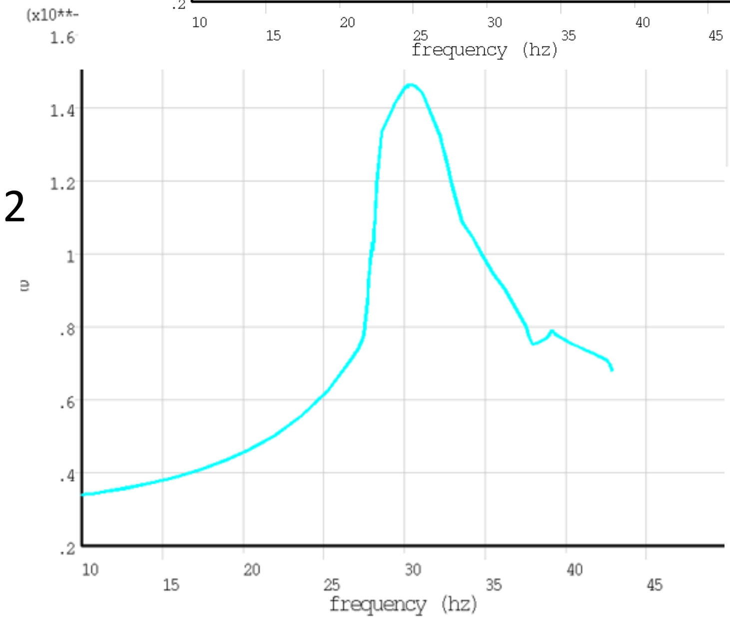
UX @ tip of the beam

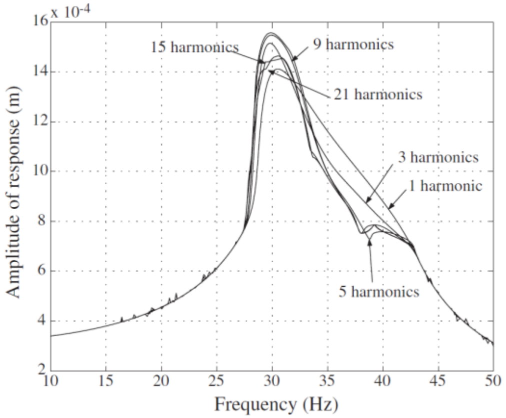




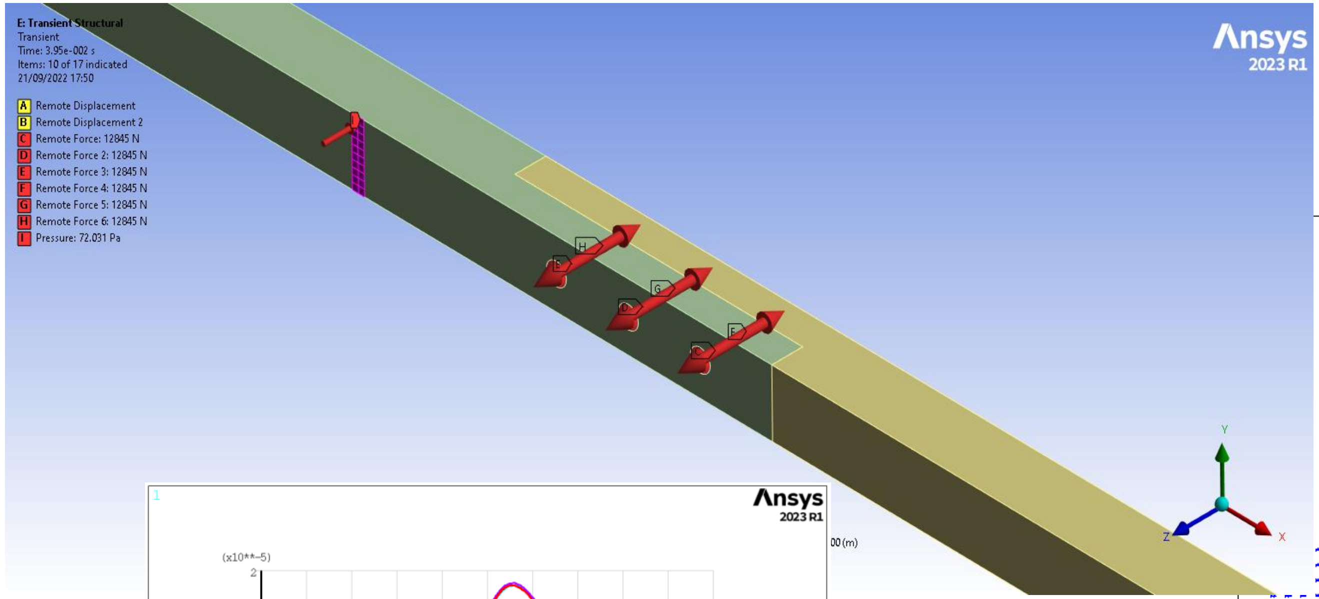
Trajectory of the friction point at F = 32.9 Hz

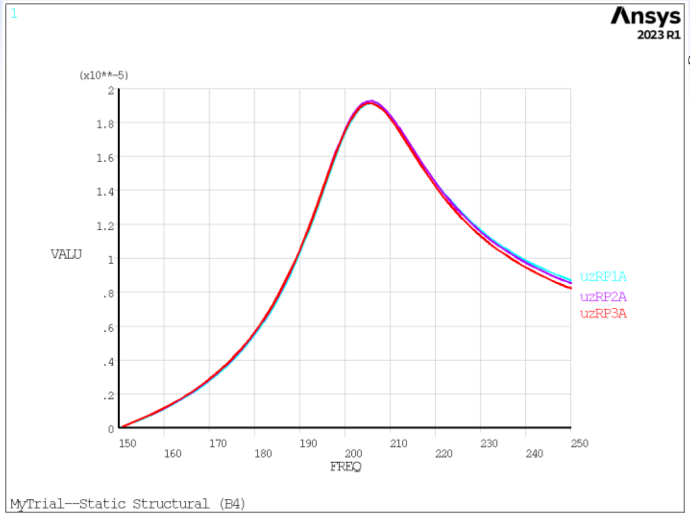
Case 2



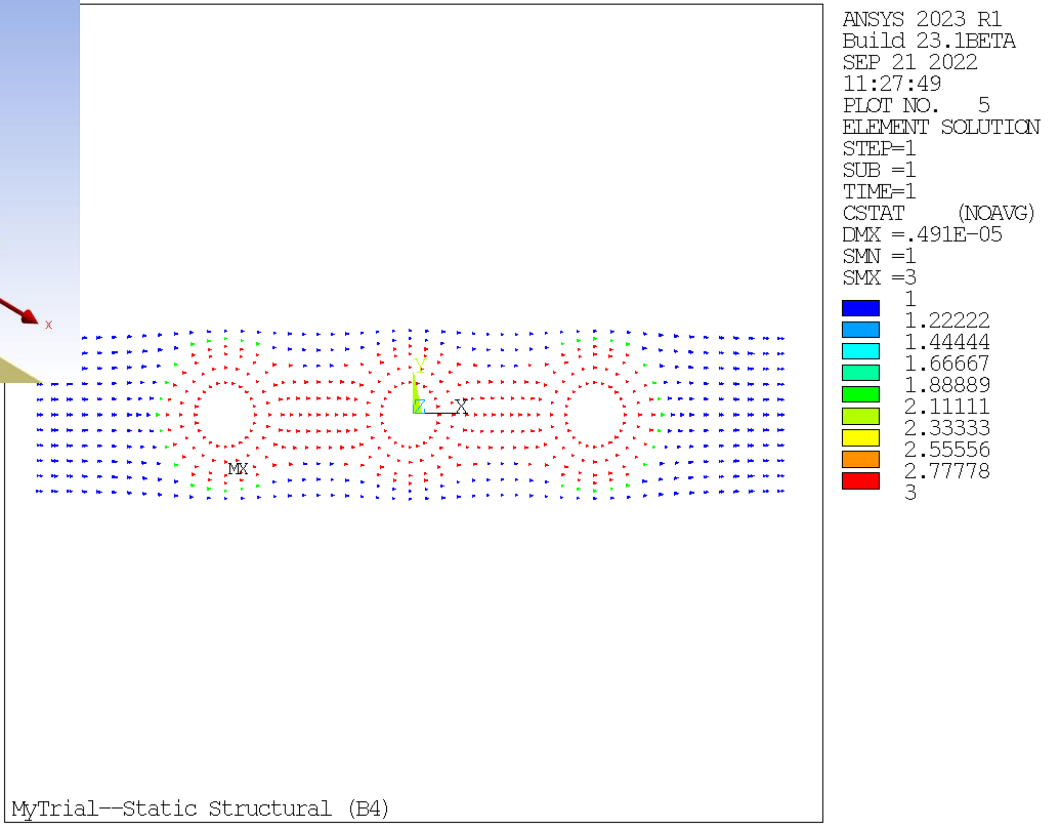


2023R1 Bolted Beam Assembly – HBM Customer Benchmark





- Prestressed CMS superelement
- Large number of node-to-node contact elements at interface

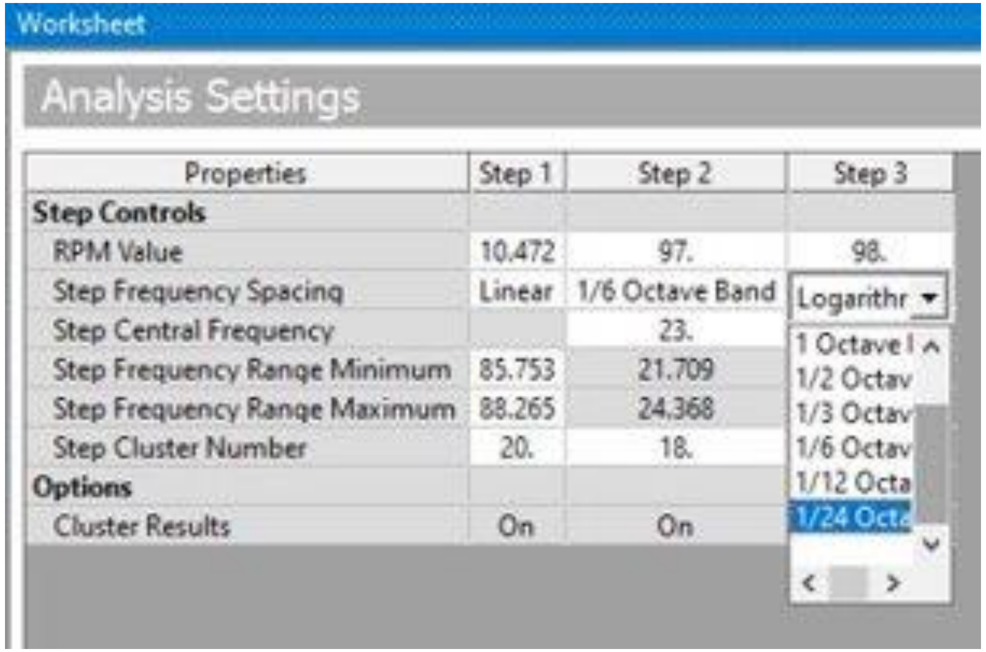


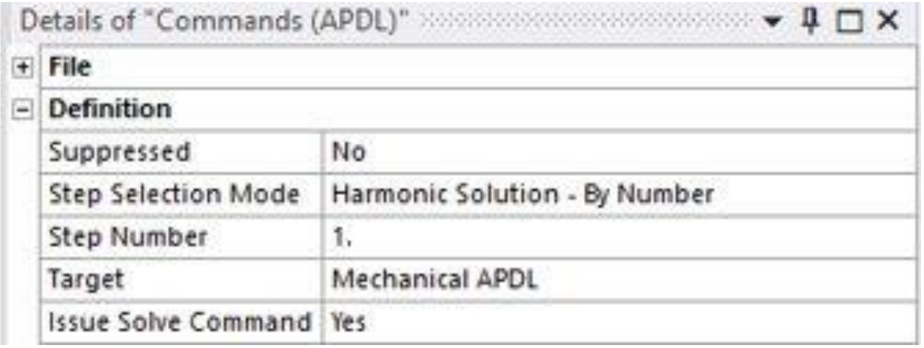
Miscellaneous: Solver

- Mode Superposition Analyses: MCF file generation setting
  - Generate MCF file by default for MSUP harmonic and MSUP transient analyses,
  - Clean up local files for distributed ANSYS.

Miscellaneous: Multi-Steps Harmonic

- Octave Band option for Harmonic Response and Harmonic Acoustics
- Step based Command Snippet for multi-steps (Harmonic Response, Harmonic Acoustics and Coupled Field Harmonic)
- Remove beta flag for multi-steps Coupled Field Harmonic
- Remove beta flag for enforced motion and force applied on vertex

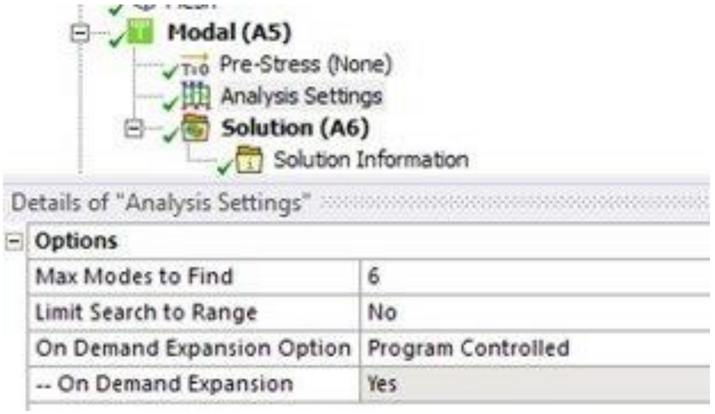




Miscellaneous: On Demand Expansion

- Program Controlled Option On Demand Expansion:

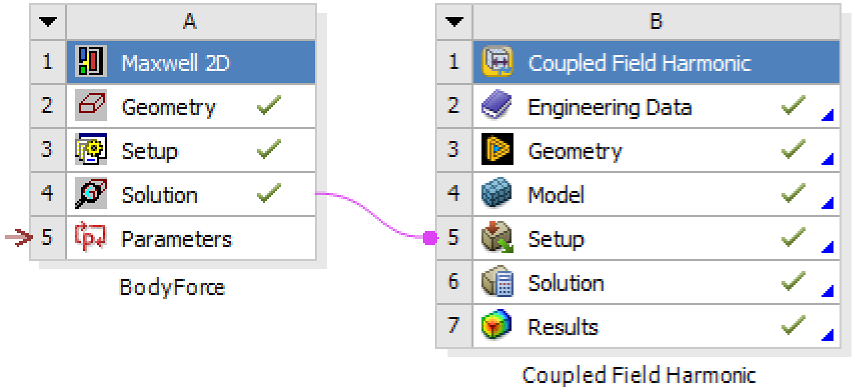
Allow to activate On Demand Expansion automatically for supported scenarios

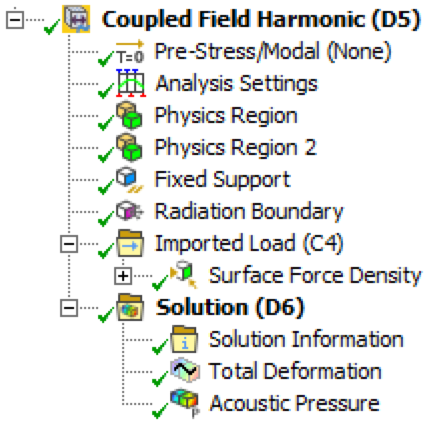


Miscellaneous: Integration

- Maxwell to Harmonic Coupled Field Coupling (Beta):

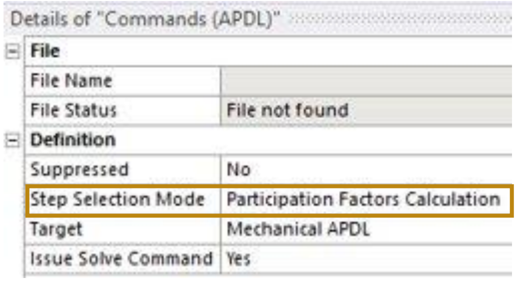
Surface Force Density, Body Force Density and Remote Loads can be transferred from Maxwell to Harmonic Coupled Field

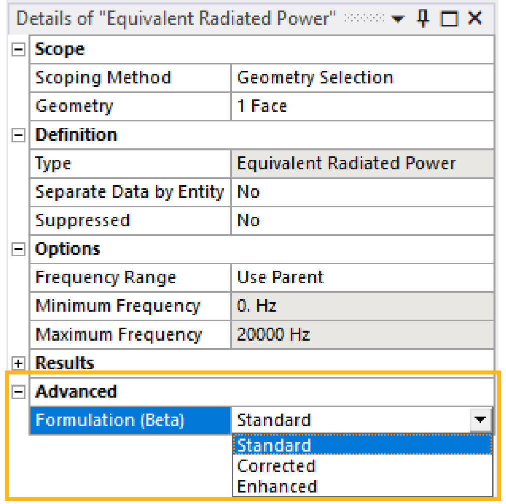


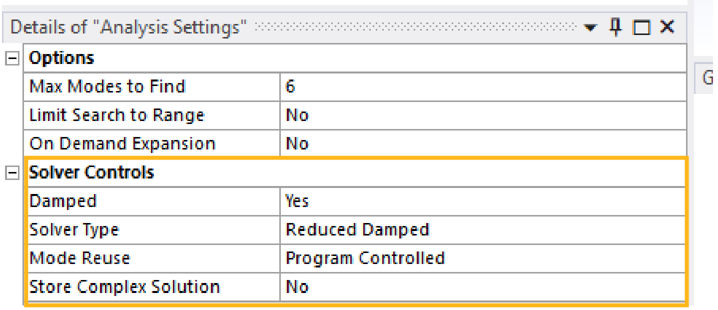


Miscellaneous

- Cyclic: Improve results when loading is applied on Cyclic Axis
- Response Spectrum: Option to send Command Snippet before the PFact command
- Option on symmetry group to send High/Low components name in CYCLIC command (beta)
- Additional ERP formulation options (beta)
- Enforced Motion with Reduced Damped solver (beta)







Miscellaneous

- Supported when the **Solver Type** property is set to either **Direct**, **Unsymmetric**, or **Subspace**
- Max Modes to Find property will accept the value of 0 to find all modes within the range. This entry to find all modes is valid only when Limit Search to range is set to Yes
- The modopt command will send "all" option to compute all modes as shown below for one case

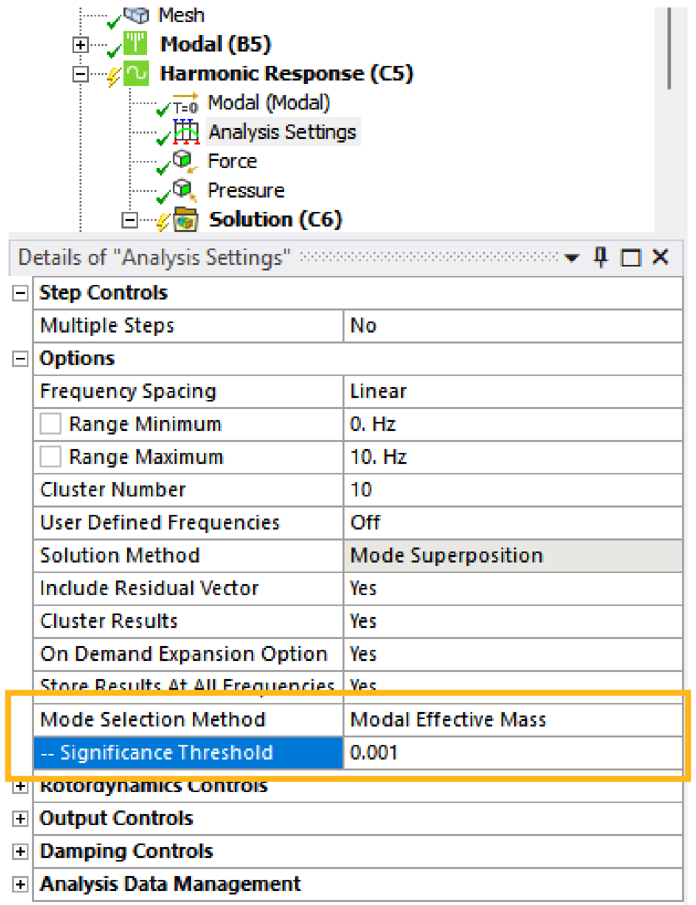
```
*get,_wallbsol,active,,time,wall
/solu
antype,2                    ! modal analysis
_thickRatio=0.667   ! Ratio of thick parts in the model
modopt,lanb,all,0.,150.
outres,erase
outres,all,none
outres,nsol,all
```

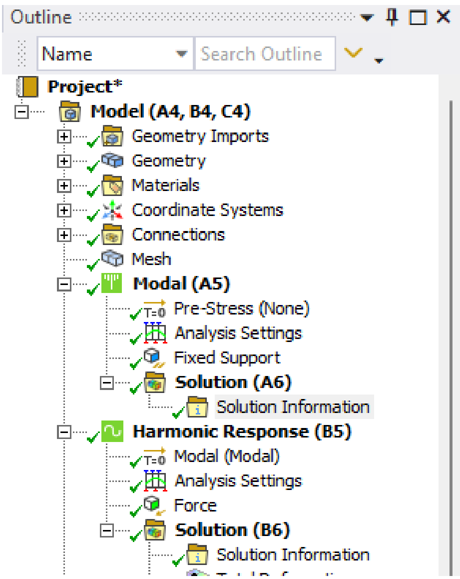
Details of "Analysis Settings"

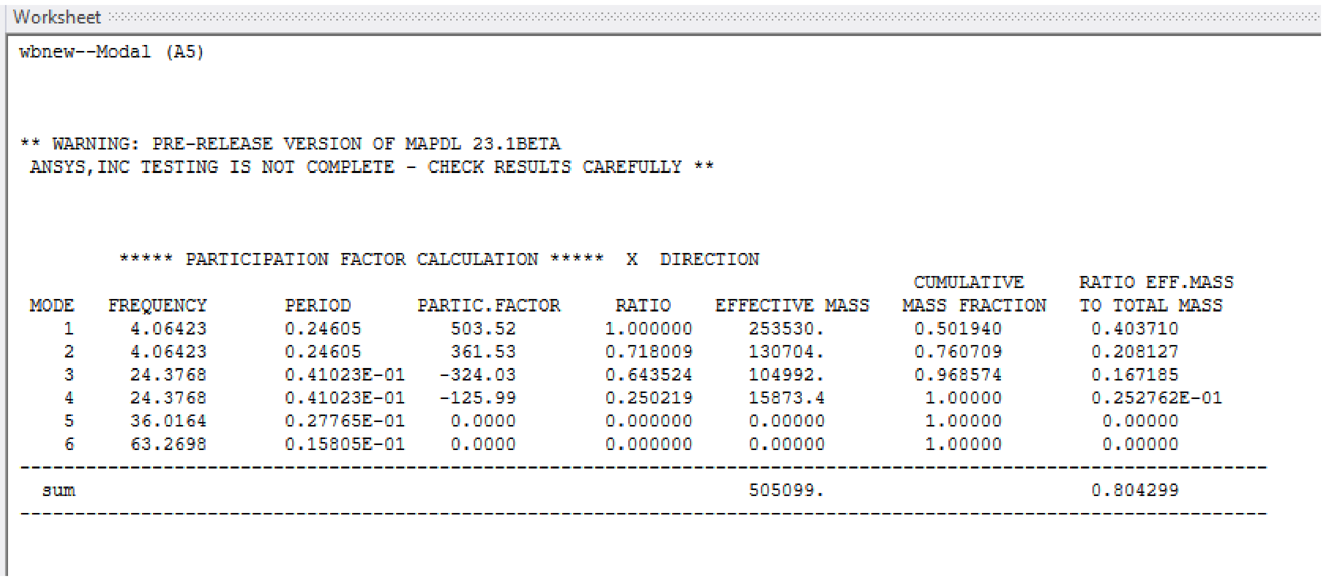
Options	
Max Modes to Find	All
Limit Search to Range	Yes
Range Minimum	0. Hz
Range Maximum	150. Hz
On Demand Expansion Option	No
Solver Controls	
Damped	No
Solver Type	Program Controlled
Rotordynamics Controls	
Advanced	
Output Controls	
Analysis Data Management	

Miscellaneous

- Modal Effective Mass based Mode Selection Method is supported for Linked MSUP Harmonic and Transient analysis
- When the Mode Selection Method is set to Modal Effective Mass, the Significance Threshold can be specified (default is 0.001). This will enable Harmonic analysis to select modes where the modal effective mass to total mass exceeds this level






Worksheet

wbnew--Modal (A5)

** WARNING: PRE-RELEASE VERSION OF MAPDL 23.1BETA
ANSYS,INC TESTING IS NOT COMPLETE - CHECK RESULTS CAREFULLY **

***** PARTICIPATION FACTOR CALCULATION ***** X DIRECTION

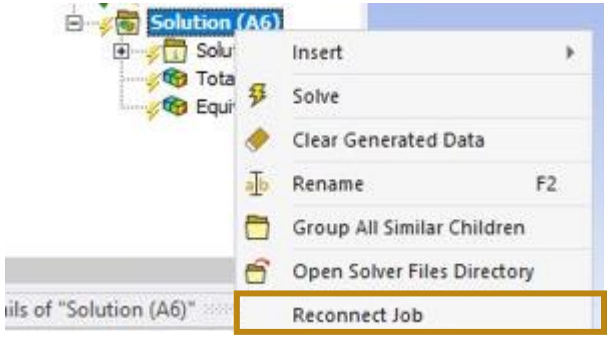
MODE	FREQUENCY	PERIOD	PARTIC.FACTOR	RATIO	EFFECTIVE MASS	CUMULATIVE MASS FRACTION	RATIO EFF.MASS TO TOTAL MASS
1	4.06423	0.24605	503.52	1.000000	253530.	0.501940	0.403710
2	4.06423	0.24605	361.53	0.718009	130704.	0.760709	0.208127
3	24.3768	0.41023E-01	-324.03	0.643524	104992.	0.968574	0.167185
4	24.3768	0.41023E-01	-125.99	0.250219	15873.4	1.00000	0.252762E-01
5	36.0164	0.27765E-01	0.0000	0.000000	0.00000	1.00000	0.00000
6	63.2698	0.15805E-01	0.0000	0.000000	0.00000	1.00000	0.00000
sum					505099.		0.804299

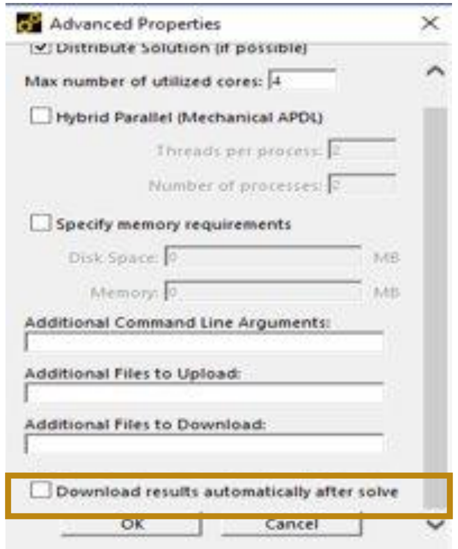
Distributed Compute Services (DCS)

Ansys

Distributed Compute Services (DCS) enhancements

- Automatic download of results through an option
- Reconnect option to a recently disconnected DCS job

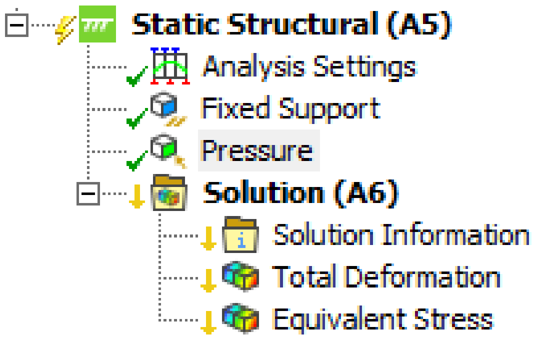




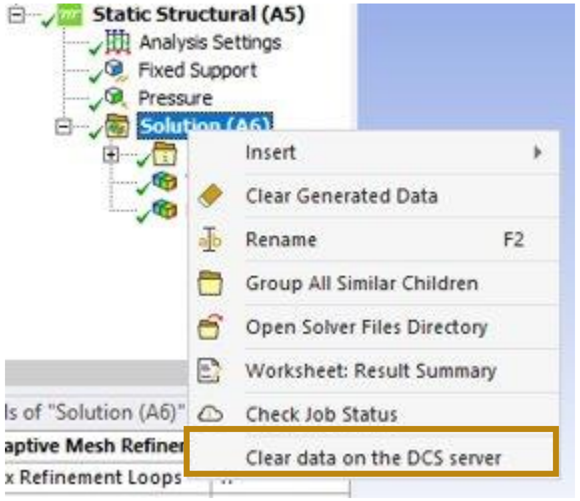
- Progress bar to indicate the live percentage of input files being uploaded when submitting the job.

Distributed Compute Services (DCS) enhancements

- Introduce new solving state to signal the download of results which may not be relevant anymore



- Clear generated data option to delete the generated files and better management of job

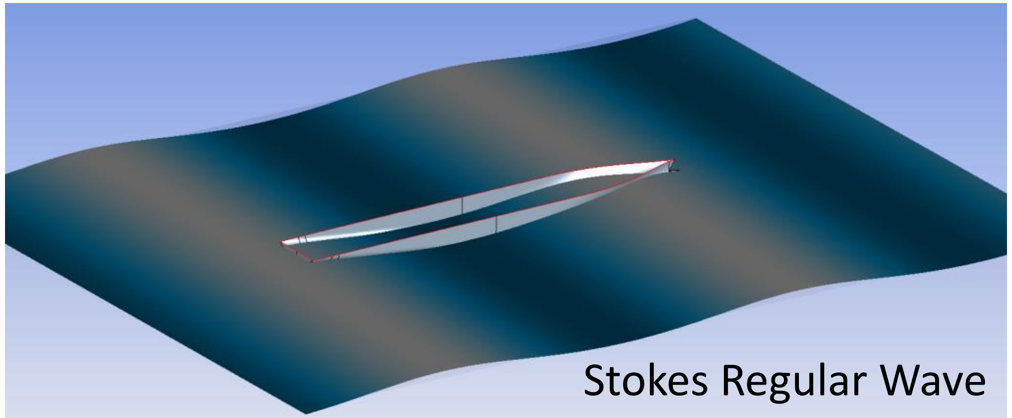


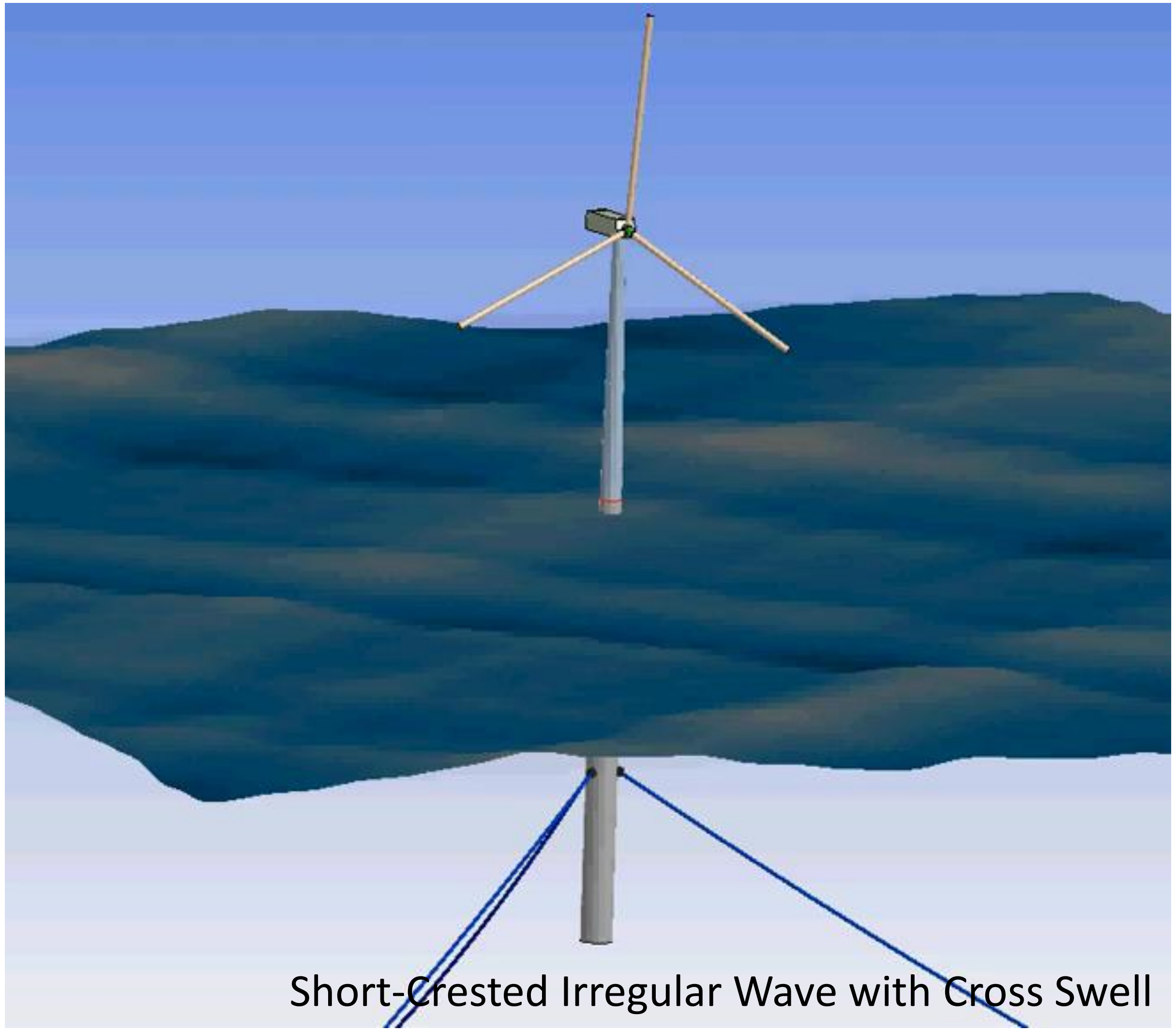
- Enable CMS use pass on DCS

Hydrodynamics

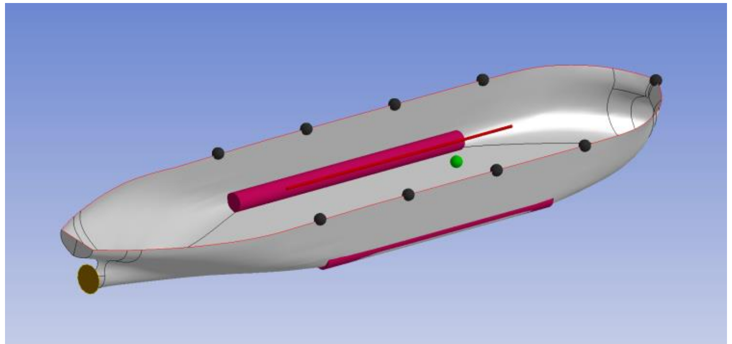
Time Domain Wave Surface Animation

- Animation results in time domain Hydrodynamic Response systems now display the incident wave surface
  - Regular or irregular waves
  - Long- or short-crested wave spectra, including cross swell
  - Imported wave height time histories
  - Deep-water or finite-depth formulations
  - Adjustable color palette and resolution


Stokes Regular Wave


Short-Crested Irregular Wave with Cross Swell

Part Transforms in Aqwa Workbench

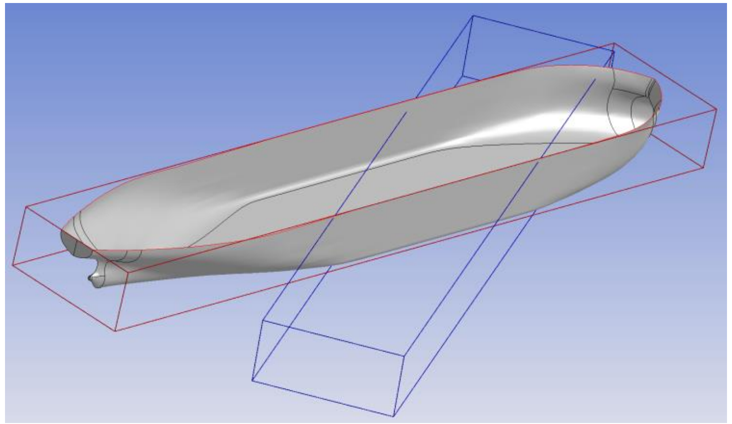


Initial position

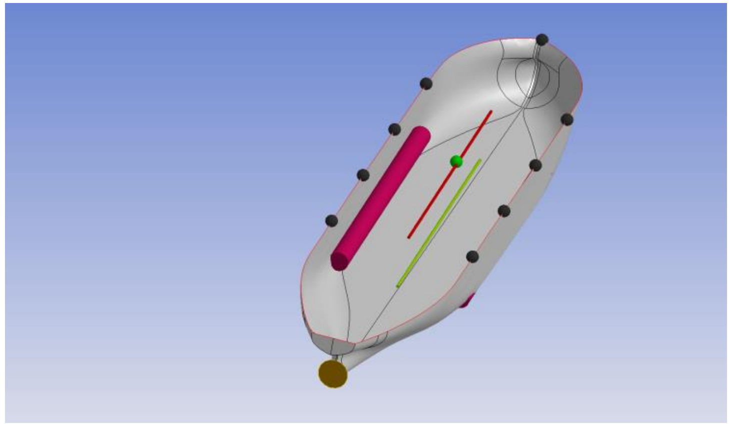
- Apply a **Part Transform** to modify the Part **position or orientation** directly in Aqwa Workbench
  - No need to modify and re-import from the geometry editor
  - Part features (Point Mass, Connection Points etc) also transformed
  - Perform parametric studies of e.g. vessel draft/trim more easily
- Define an Internal Tank by **Fluid Volume** so that Internal Tank geometry can be transformed consistently

Details	
Details of Part Transform	
Name	Part Transform
Activity	Not Suppressed
Reference Point Definition	
Reference Point	Manual Definition
Reference Point X	11.88 m
Reference Point Y	0.0 m
Reference Point Z	10.4 m
Move Reference Point to Center of Gravity	Click to Apply
Transform Definition	
Translation X	10 m
Translation Y	-20 m
Translation Z	-2 m
Rotation about Reference Point X	-3°
Rotation about Reference Point Y	2°
Rotation about Reference Point Z	30°

Transform definition



Transform preview



Applied Part Transform

Workbench Additive

Ansys

Improved Automatic Distortion Compensation

More user controls w.r.t to output geometry

- Improvements in re-faceting the input stl mesh needed for compensation
- User controls for re-faceting operation
- Option to output stl at any/all iteration points
- "Zero Deformation at Base and Z Gap" are new options added to help with convergence. When enabled, the scaling of the deformations at nodes below the Z Gap will vary linearly from 0 at the baseplate to the 1 at the Z Gap

Preview of faceted geometry

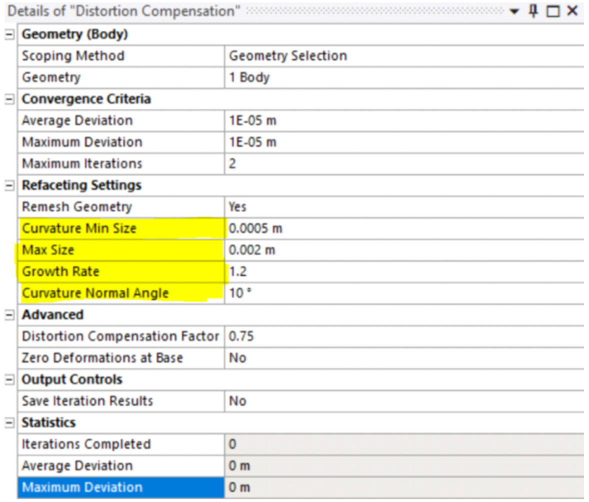
- This new version allows the user to preview the faceted geometry that will be used for distortion compensation

Preview of the compensation convergence

- The convergence plot visualizes the iteration data like average and maximum deviations against the respective criteria and the iteration number. This will give the user a real time visualization of the performance of distortion compensation with selected parameters

Improved Usability

- Compensation enabled in conjunction with spring-back and cut-off simulations


Details of "Distortion Compensation"

Geometry (Body)	
Scoping Method	Geometry Selection
Geometry	1 Body
Convergence Criteria	
Average Deviation	1E-05 m
Maximum Deviation	1E-05 m
Maximum Iterations	2
Refaceting Settings	
Remesh Geometry	Yes
Curvature Min Size	0.0005 m
Max Size	0.002 m
Growth Rate	1.2
Curvature Normal Angle	10 °
Advanced	
Distortion Compensation Factor	0.75
Zero Deformations at Base	No
Output Controls	
Save Iteration Results	No
Statistics	
Iterations Completed	0
Average Deviation	0 m
Maximum Deviation	0 m

Fig.1 New re-faceting settings

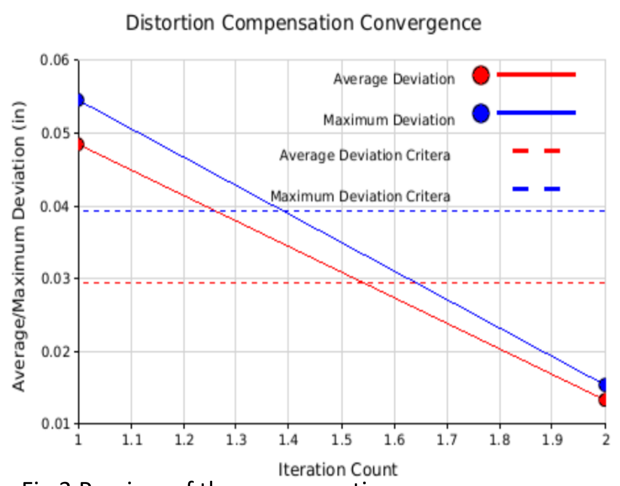


Fig.3 Preview of the compensation convergence

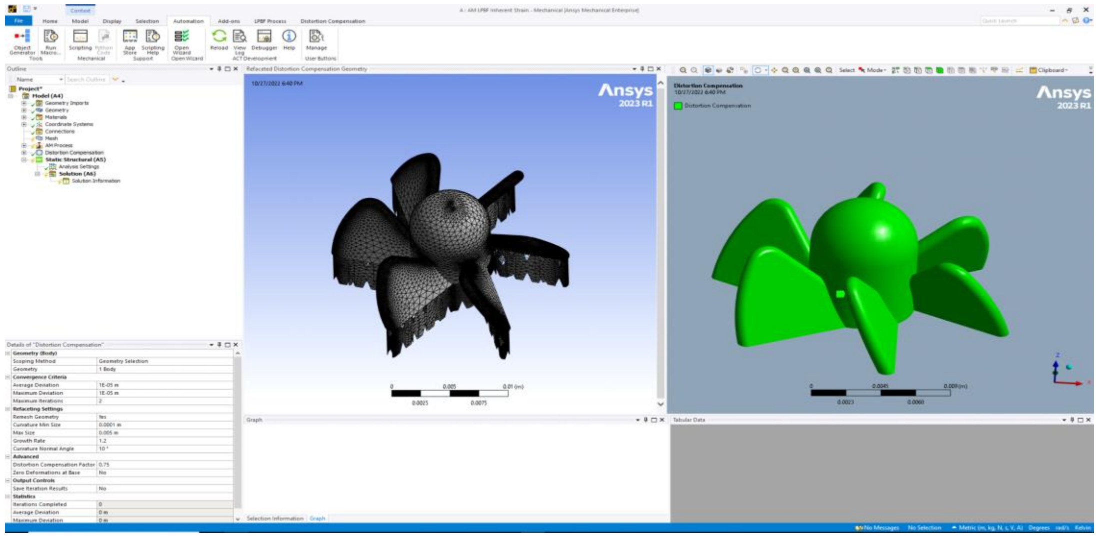


Fig.2 Preview of faceted geometry that will be used for compensation

Other Improved Capabilities into Mechanical Additive

Other new capabilities

- Read build files from various manufactures for scan pattern and ML thermal strain simulations in WB Mechanical
- Spring-back/Cut-off simulations are improved with directional removal
- Post-processing results include high strain
- Majorly improved Additive Wizard now features
  - Scan pattern, thermal strain along with earlier assumed strain
  - Option to choose build file for a given machine manufacturer
  - Add high strain result option
  - Option to enable Directional Cutoff step
- Adaptive mesh coarsening using Octree
  - For inherent strain simulations, mesh is coarsened adaptively away from laser source to reduce element count which in-turn helps reduce solver time.
  - Element count reduced majorly in bulk areas of the geometries.

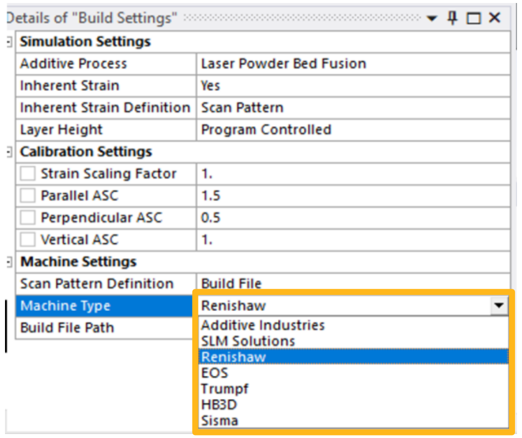


Fig.1 build file reader

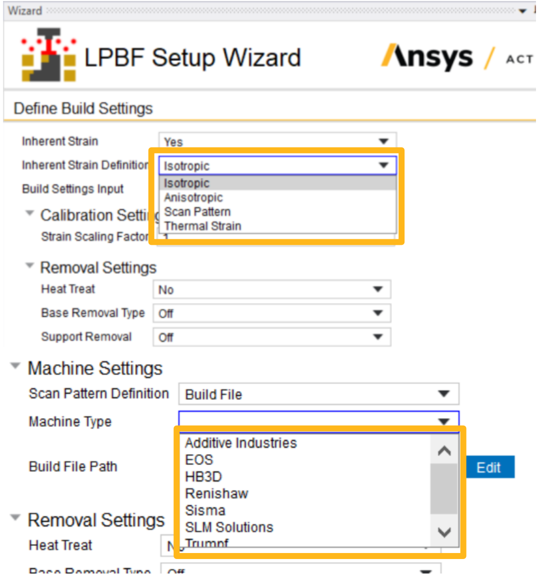
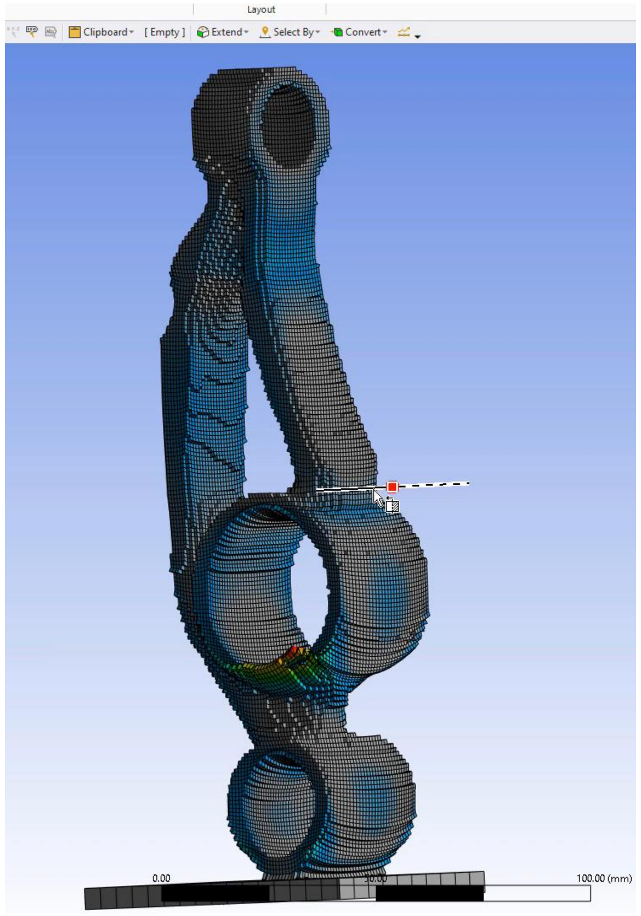


Fig.2 Improved AM setup wizard



Video.1 Adaptive mesh coarsening using Octree

Automatic Distortion Calibration Wizard

Theme category

- Ease of use/Usability

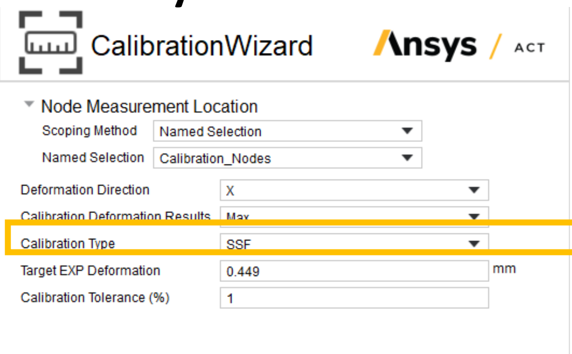
Customer pain points

- LPBF distortion simulation calibration workflow
  - Users manually run distortion simulations by applying iteratively calculated Strain Scaling Factors (SSF) and Anisotropic Strain Coefficients (ASC) to match simulation results with experimental target values within a given tolerance. Such manual processes are repeated by utilizing spreadsheet calculation table

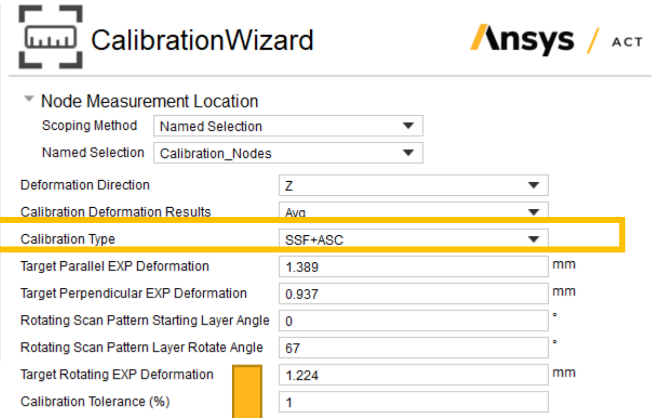
Value prop

- Streamline and automate the manual distortion calibration workflow for all LPBF distortion simulation modes via the new automatic distortion calibration wizard
- More calibration part types are added to calibration geometry library

SSF only calibration mode



SSF + ASC calibration mode



DX module automatically optimizes calibration factor(s)

1	Name		P1 - Build Settings Thermal Strain Scaling Factor	P2 - Calibration Deformation Maximum (mm)
2	1	DP 0	1	0.65576
3	2	DP 1	0.002	0.0018668
4	3	DP 2	0.684	0.47439
5	4	DP 3	0.647	0.45507
6	5	DP 4	0.638	0.45054

1	Name		P1 - Build Settings Strain Scaling Factor	P2 - Build Settings Parallel ASC	P3 - Build Settings Perpendicular ASC	P4 - Build Settings Start Layer Angle (degree)	P5 - Build Settings Layer Rotation Angle (degree)	P6 - Calibration Deformation Average (mm)
2	1	DP 0	1	1.5	0.5	0	0	1.534
3	2	DP 1	1	1.5	0.5	90	0	0.75164
4	3	DP 2	1.018	1.363	0.637	0	0	1.5204
5	4	DP 3	1.018	1.363	0.637	90	0	0.92262
6	5	DP 4	1.005	1.305	0.695	0	0	1.5133
7	6	DP 5	1.005	1.305	0.695	90	0	0.97746
8	7	DP 6	1.05	1.267	0.733	0	0	1.5001
9	8	DP 7	1.05	1.267	0.733	90	0	1.0604
10	9	DP 8	0.002	1.363	0.637	0	0	0.0034021
11	10	DP 9	0.002	1.363	0.637	90	0	0.0018064
12	11	DP 10	1.018	1.015	0.985	0	0	1.3372
13	12	DP 11	1.018	1.015	0.985	90	0	1.3065
14	13	DP 12	0.954	1.286	0.714	0	0	1.5054
15	14	DP 13	0.954	1.286	0.714	90	0	0.95099
16	15	DP 14	0.903	1.275	0.725	0	0	1.4601
17	16	DP 15	0.903	1.275	0.725	90	0	0.91338
18	17	DP 16	0.885	1.357	0.643	0	0	1.5038
19	18	DP 17	0.885	1.357	0.643	90	0	0.8129
20	19	DP 18	0.889	1.237	0.763	0	0	1.406
21	20	DP 19	0.889	1.237	0.763	90	0	0.93801
22	21	DP 20	0.882	1.231	0.769	0	0	1.3908
23	22	DP 21	0.882	1.231	0.769	90	0	0.93684
24	23	DP 22	1	1.231	0.769	0	67	1.3063
25	24	DP 23	0.882	1.231	0.769	0	67	1.1678
26	25	DP 24	0.002	1.231	0.769	0	67	0.002594
27	26	DP 25	0.937	1.231	0.769	0	67	1.2346

Ansys

Improved DED Simulation

Introduce ***Clustering Settings*** object

- A table which allows import, export, and modifications of machine parameters for each cluster during a DED simulation. Great for optimizing process settings and improving printing quality (e.g., reduce distortion and localized overheating, etc.)

Cluster NS	Deposition Rate[mm³/s]	Cluster Preheat Temperature[°C]	Dwell Time[s]
el_loop_01	20	23	0
el_loop_02	20	23	0
el_loop_03	20	23	0
el_loop_04	20	23	0
el_loop_05	20	23	0
el_loop_06	20	23	0
el_loop_07	20	23	0
el_loop_08	20	23	0
el_loop_09	20	23	0
el_loop_10	20	23	0
el_loop_11	20	23	0
el_loop_12	20	23	0
el_loop_13	20	23	0
el_loop_14	20	23	0
el_loop_15	20	23	0
el_loop_16	20	23	0
el_loop_17	20	23	0
el_loop_18	20	23	0
el_loop_19	20	23	0
el_loop_20	20	23	0

Go to page: 1 Show rows: 20 1-20 of 48

Fig.1 Clustering Settings table

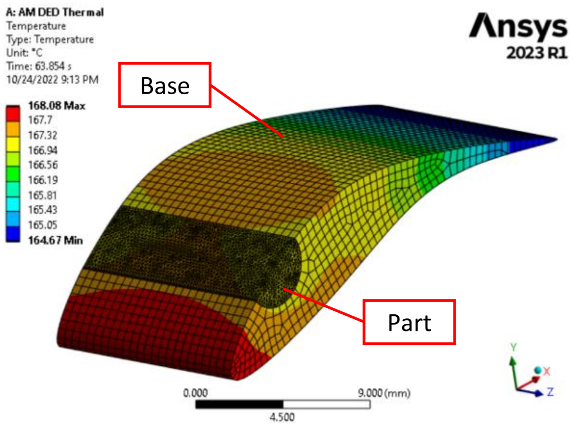


Fig.3 DED simulation w/ non-planar base plate

Improved Gcode reader performance

- Gcode cluster generation has been optimized to give up to 40x by optimizing the clustering algorithm

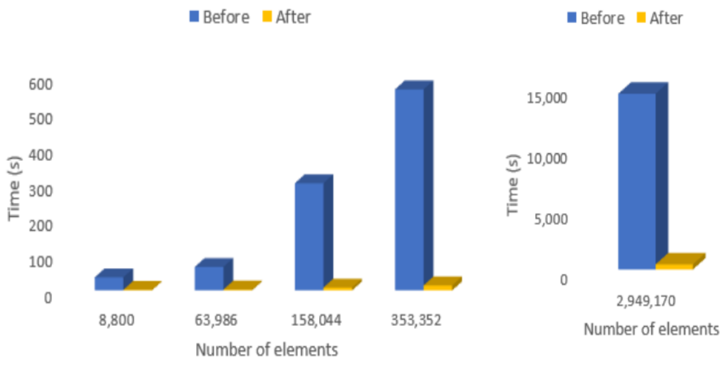


Fig.2 Gcode clustering cost before and after improvements

Simulation w/ non-planar base plate

- Enable DED simulation to consider parts building on non-planar base plates with improved contact generation workflow in wizard. It also provides a way to simulate DED repairing applications

Improved distortion prediction algorithm

- The simulation takes into account of geometry true shape after large deformation to offer a better match to the reality

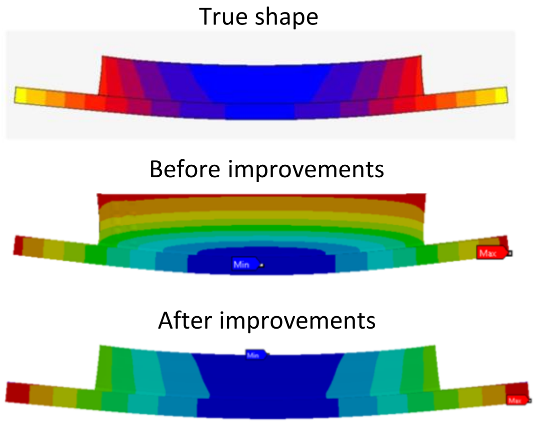


Fig.4 Distortion prediction algorithm improvements

Explicit Simulation

Explicit Dynamics

Explicit Dynamics – Per Material Erosion Controls

- Four new erosion criteria have been added to Engineering Data to be added to materials for Explicit Dynamics analyses
- Allows each body in Mechanical to have a different erosion criteria
- Finer degree of control over behaviour of bodies in Mechanical
- Material based erosion controls work in combination with global erosion controls

⊟ Erosion
- Erode on Plastic Strain
- Erode on Failure
- Erode on Geometric Strain
- Erode on Timestep

Properties of Outline Row 4: COPPER

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	8900	kg m^-3		
4	⊞ Multilinear Isotropic Hardening	Tabular			
7	Specific Heat Constant Pressure, C_p	1E-12	J kg^-1 C^-1		
8	Shear Modulus	4.64E+10	Pa		
9	⊞ Shock EOS Linear				
14	⊟ Erode on Plastic Strain				
15	Erosion Strain	0.44			

LS-DYNA

Ansys

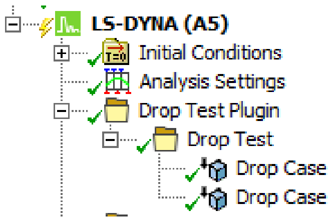
LS-DYNA Prep

Enhancements

Drop Case Setup

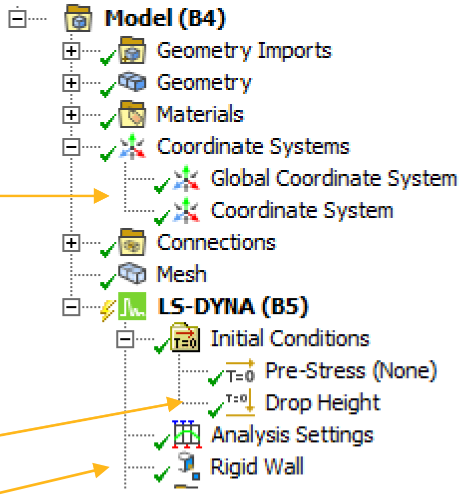
- Drop Test Plugin

LS-DYNA Analysis



Details of "Drop Case"

Definition	
Define By	Vector with 2 Points
Point 1	
Define By	CG
Point 1	Click to Change
Origin X	1.25 m
Origin Y	0.5 m
Origin Z	0.5 m
Point 2	
Point 2	Click to Change
Origin X	1 m
Origin Y	1 m
Origin Z	0 m



Details of "Drop Test"

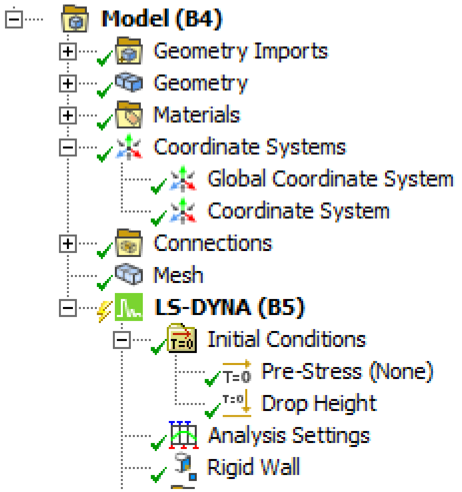
Drop Load	
Define By	Drop Height
Drop Height	2 m
Rigid Wall	
Friction	0.1
Gap	1E-05 m
Note	
note	Model needs to be meshed before creating ...

\+

- Use of the Center of Mass (CG) when determining drop direction
- Offset of the Rigid Wall along the drop direction
- Multiple cases in a single Analysis

Drop Case Setup

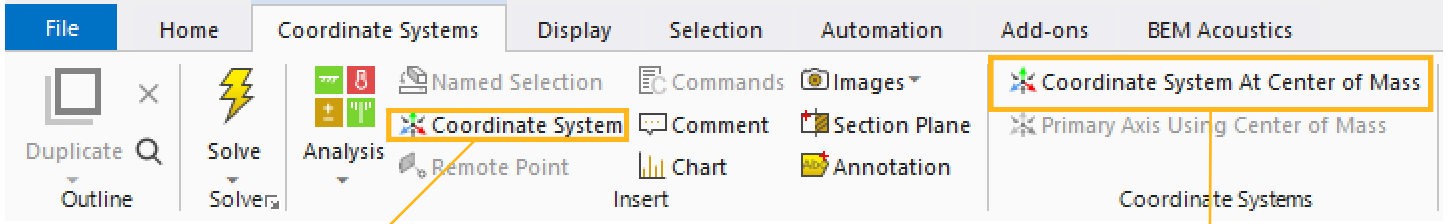
LS-DYNA Analysis

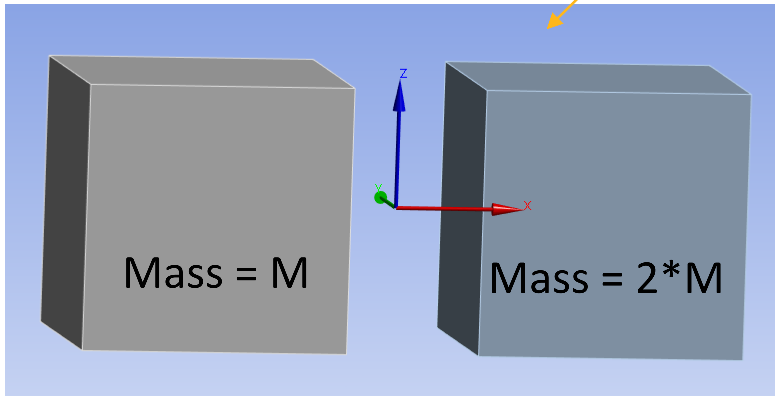


- Use of the Center of Mass (CG) when determining drop direction
- Offset of the Rigid Wall along the drop direction
- Multiple cases in a single Analysis

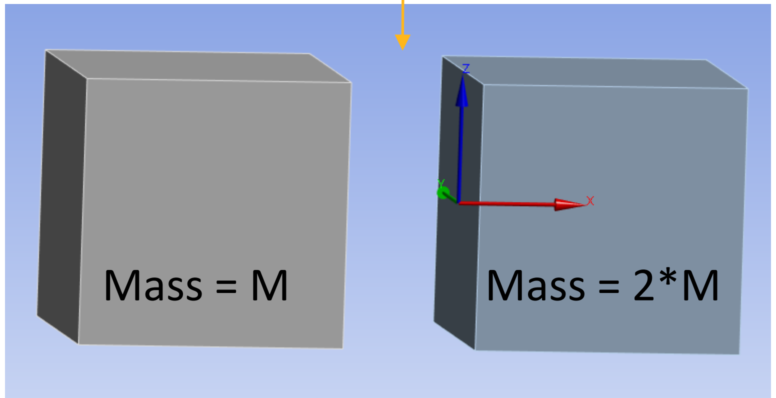
Coordinate System Definition

- New option to insert a coordinate system with the origin positioned at the center of mass
- Default center of mass calculation uses all unsuppressed bodies and point masses





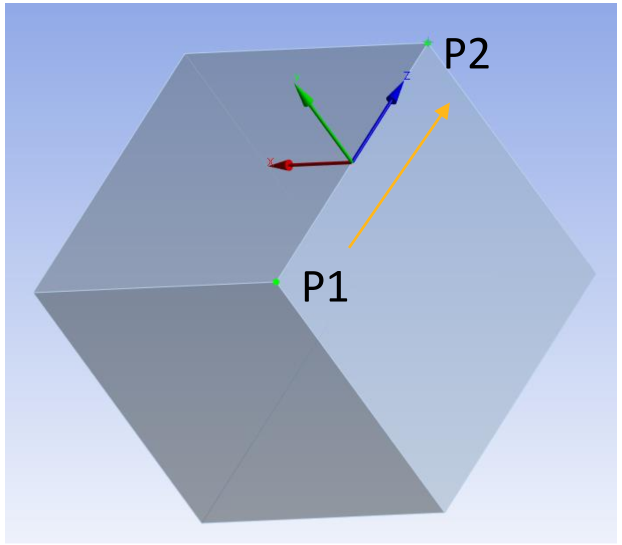
Coordinate System (Origin @ Center of Geometry)



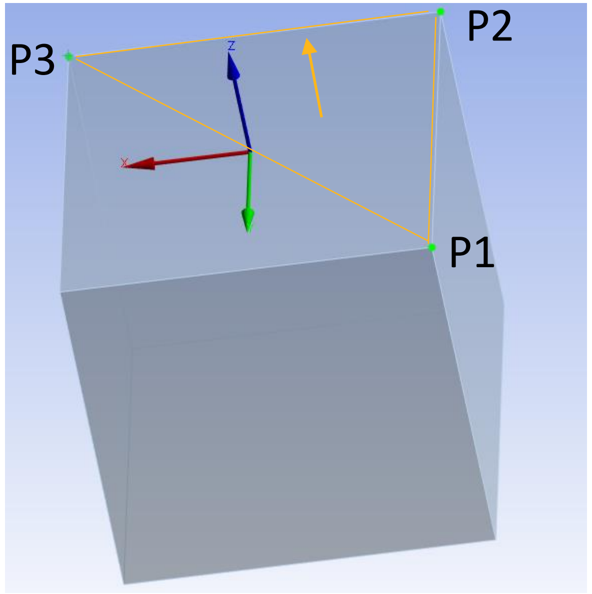
Coordinate System (Origin @ Center of Mass)

Coordinate System Definition

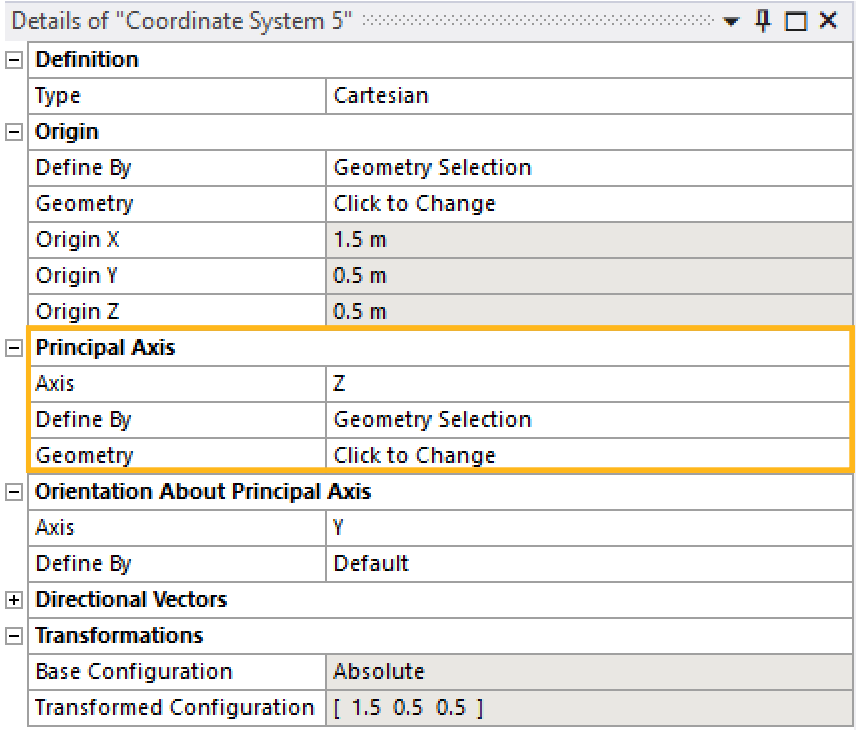
- Summary of existing coordinate system functionality
- Defining the principal axis direction using a 2 or 3 point selection



Principal Axis Alignment
Using 2 Points



Principal Axis Alignment
Using 3 Points



Details of "Coordinate System 5"

Definition	
Type	Cartesian
Origin	
Define By	Geometry Selection
Geometry	Click to Change
Origin X	1.5 m
Origin Y	0.5 m
Origin Z	0.5 m
Principal Axis	
Axis	Z
Define By	Geometry Selection
Geometry	Click to Change
Orientation About Principal Axis	
Axis	Y
Define By	Default
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[1.5 0.5 0.5]

Coordinate System Definition

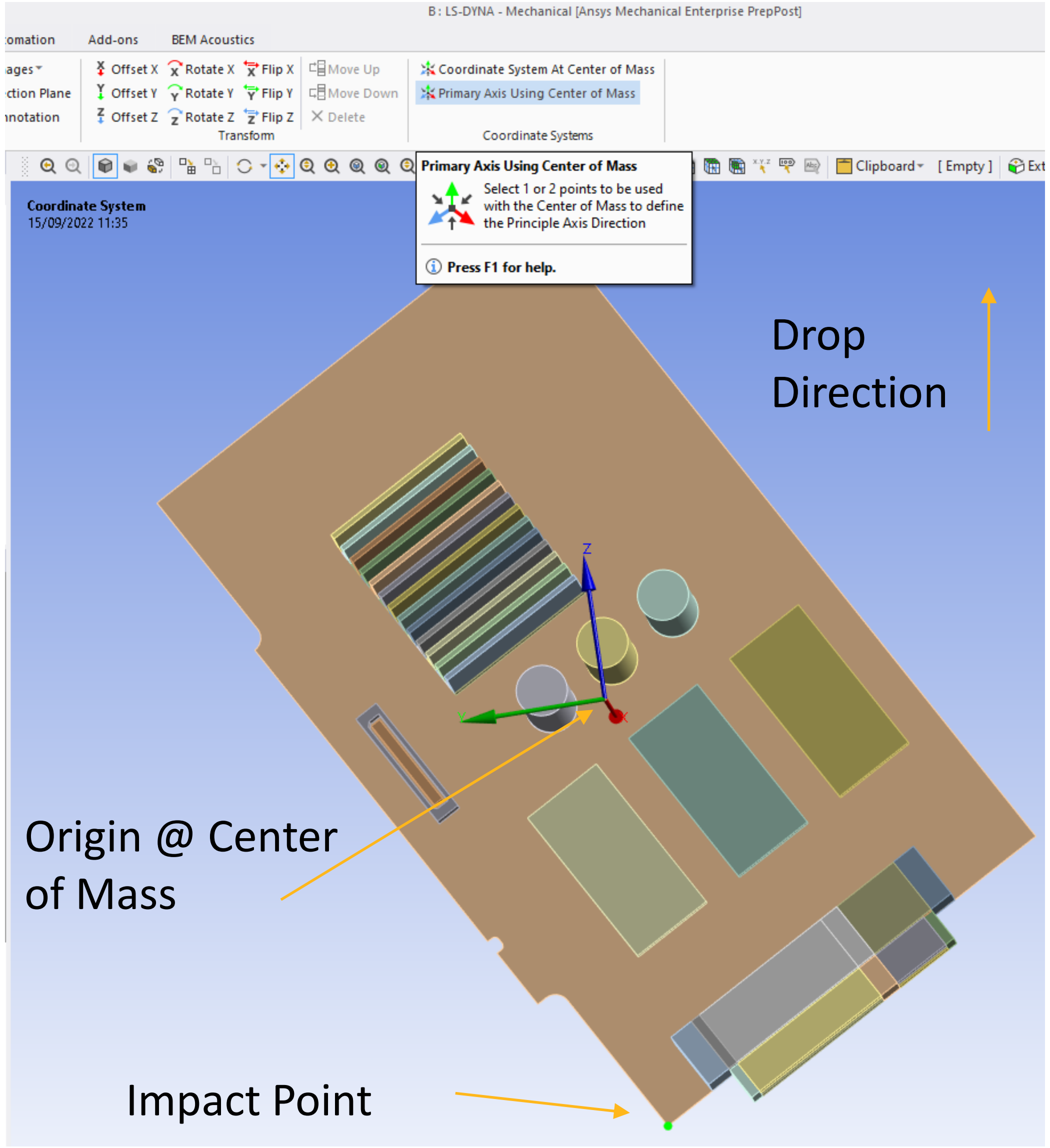
- Step 1 – Insert coordinate system with origin at the center of mass
- Step 2 – Select 1 or 2 points that will be used with the center of mass to define the drop direction
- **Center of mass calculation and coordinate system creation can also be scripted allowing additional customization**

Default center of mass calculation (all active bodies and point masses)

```
CoM = Model.CenterOfMass()
print(CoM)
```

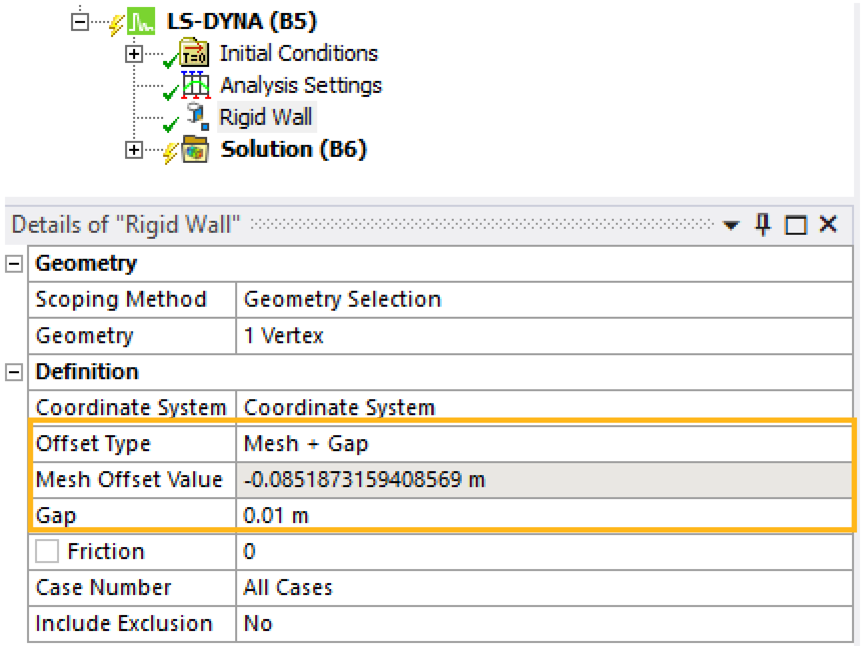
Custom center of mass calculation with user defined lists of bodies and point masses

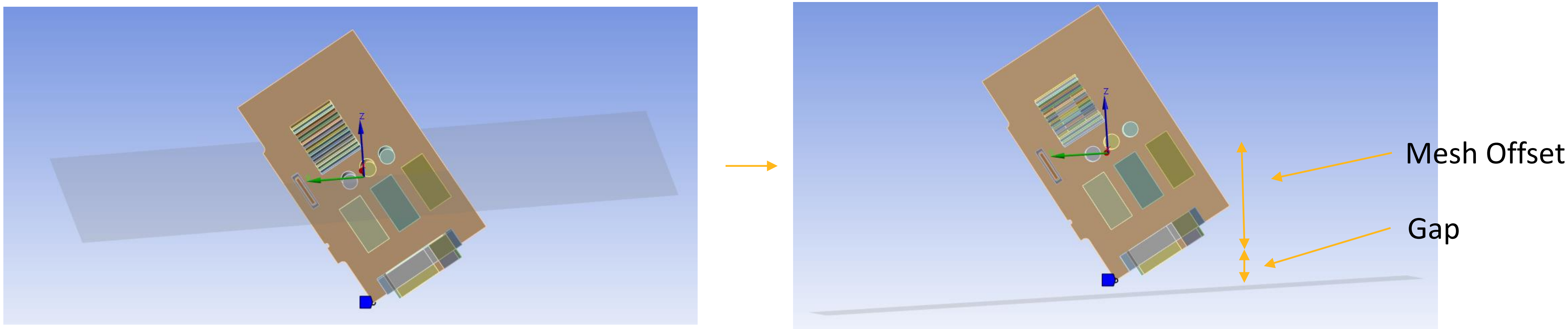
```
geometry = Model.Geometry

b = geometry.GetBodies()
bodies = []
for obj in b:
if not obj.Suppressed: # Criteria for including Body
bodies.Add(obj.GetGeoBody())

pm = geometry.GetChildren(DataModelObjectCategory.PointMass, True)
point_masses = []
for obj in pm:
if not obj.Suppressed: # Criteria for including Point Mass
point_masses.Add(obj)

CoM = geometry.CenterOfMass(bodies, point_masses)
print(CoM)
```



Drop Plane Definition

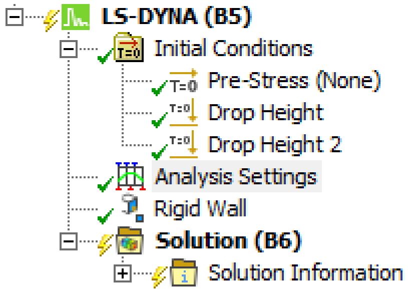
- Rigid Wall
  - Infinite plane (x, y, z = 0) of the selected coordinate system, with the normal to the plane given by the positive z-axis.
- Offset Type (None, Mesh, Gap, Mesh + Gap)
  - Specifies an offset of the Rigid Wall plane along the z-axis
  - The Mesh options offsets the plane to the furthest node in the negative z- direction





Multiple Case Definition

- “Number of Cases” > 0 passes the CASE command line argument to the solver
- The solver splits the single input file into separate cases and solves each case producing a individual set of output files for each case
- Supported Objects:
  - Initial Conditions
  - Rigid Wall
- Keywords are wrapped in *CASE_BEGIN_n \ *CASE_END_n



etails of "Analysis Settings"

Step Controls	
End Time	0.001 s
Time Step Safety Factor	0.9
Maximum Number Of Cycles	10000000
Automatic Mass Scaling	No
Number of Cases	2

Details of "Drop Height"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Input Type	Drop Height
Pre-Stress Environment	None Available
Define By	Drop Height
Drop Height	2. m
Impact Velocity	6.2631 m/s
Coordinate System	Coordinate System 2
Direction	-Z Direction
Case Number	1.
Suppressed	No

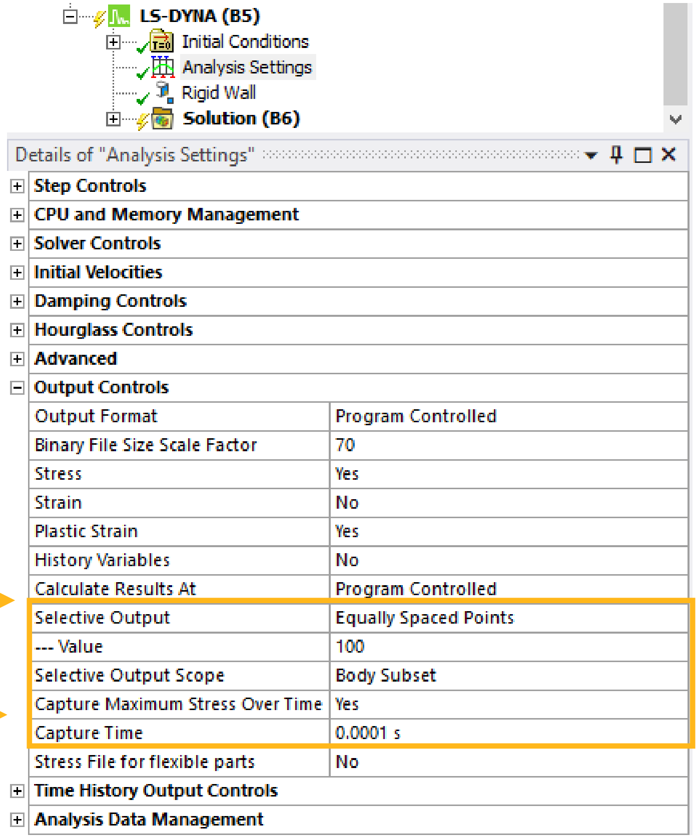
Details of "Rigid Wall"

Geometry	
Scoping Method	All Bodies
Definition	
Coordinate System	Coordinate System 2
Offset Type	Mesh + Gap
Mesh Offset Value	-1.43614065647125 m
Gap	0.05 m
Friction	0
Case Number	All Cases
Include Exclusion	No

Additional Output Settings

- Selective Output (D3Part)
  - Named selection of bodies
  - Output frequency (Equally Spaced Points, Time)

- Maximum Stress Over Time (D3Max)
  - Output frequency (Time)



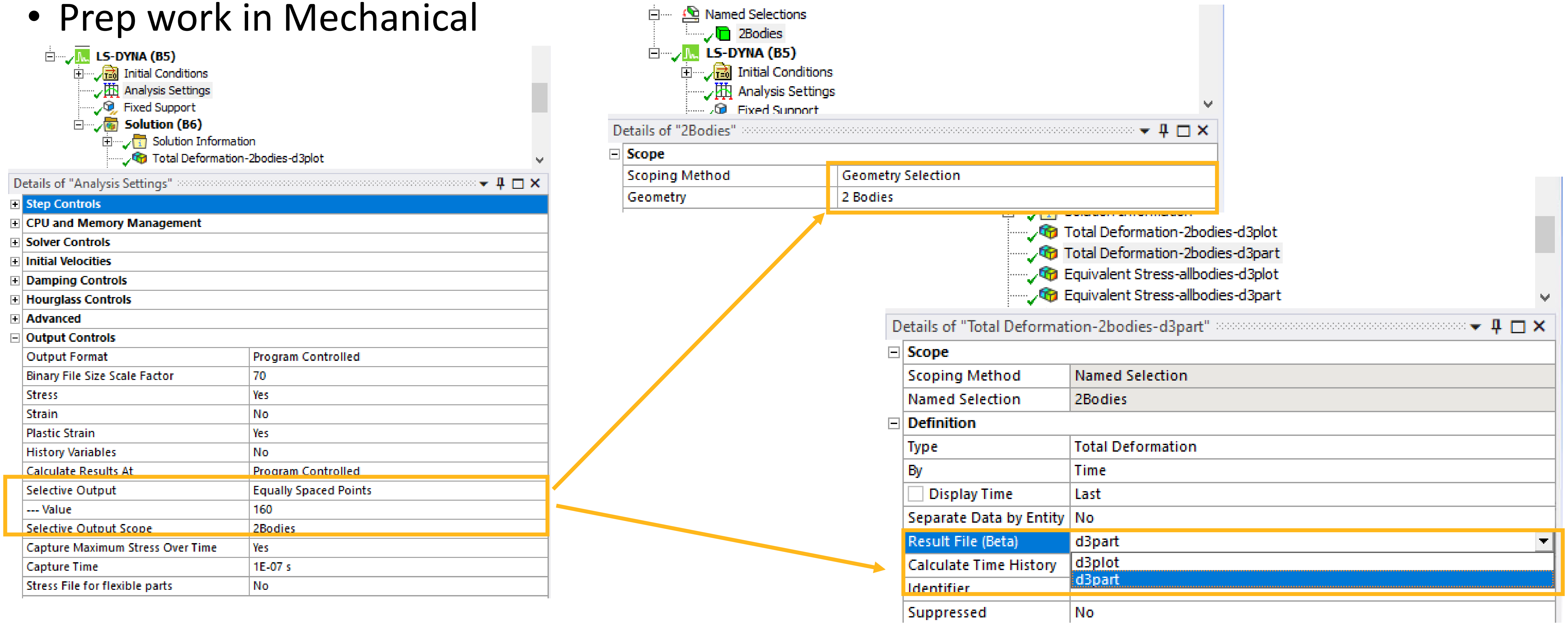
Dimensionally Reduced Rigid Body Behavior

- Dimensionally reduced behavior for rigid body meshing is now supported in the LS-DYNA system.
- In that approach , only the surfaces used in contact and boundary conditions are meshed.
- This is the approach supported by the MAPDL solver
- Now a same rigid body behavior can be shared between implicit simulations and explicit simulations in the same model

LS-DYNA D3Part, D3Max & Case Postprocessing

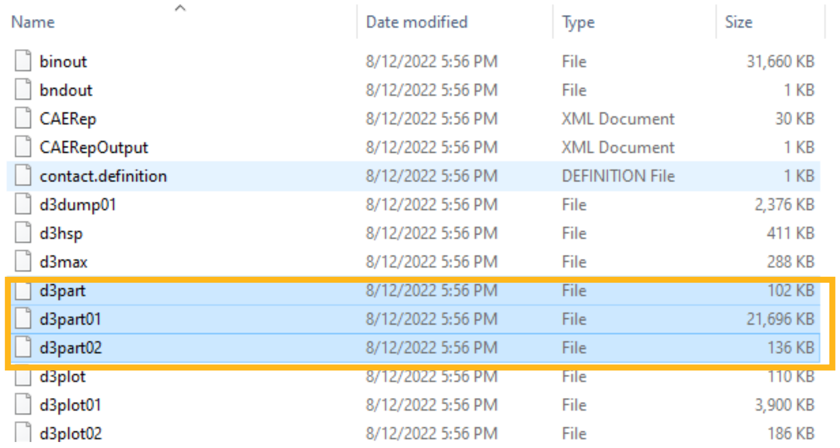
D3Part

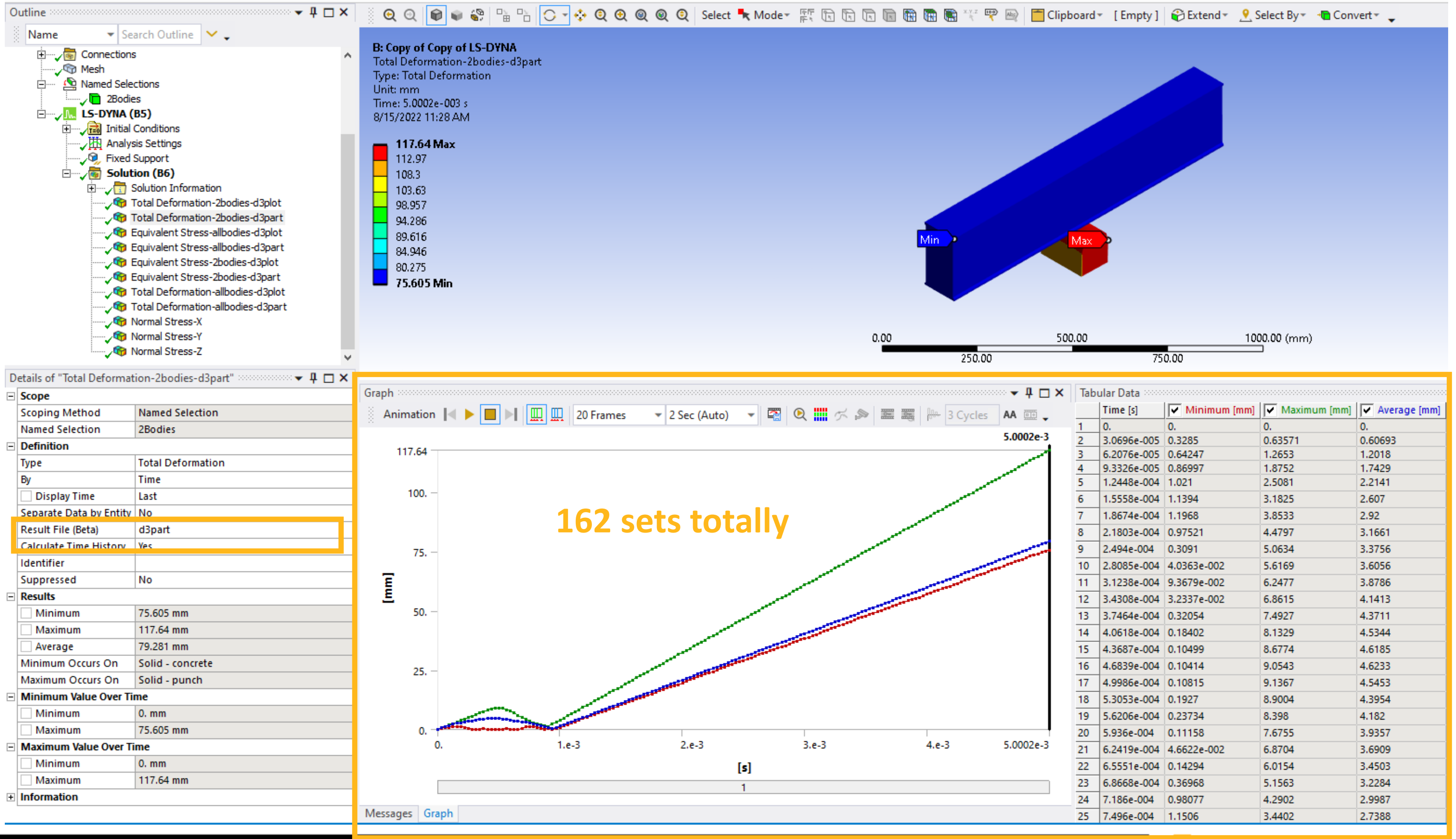
- Allow to capture **more** results for a **subset** of the geometry, using *Named Selections* for scoping
- Prep work in Mechanical



D3Part

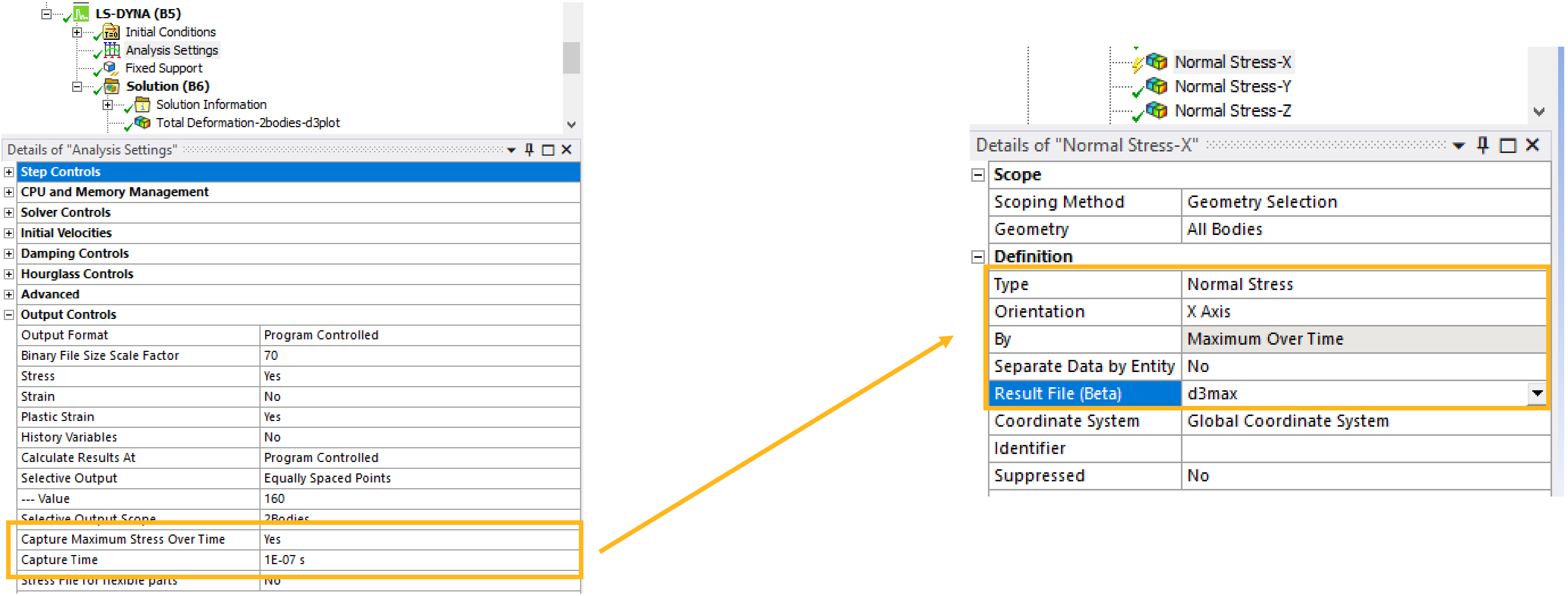
- Postprocessing





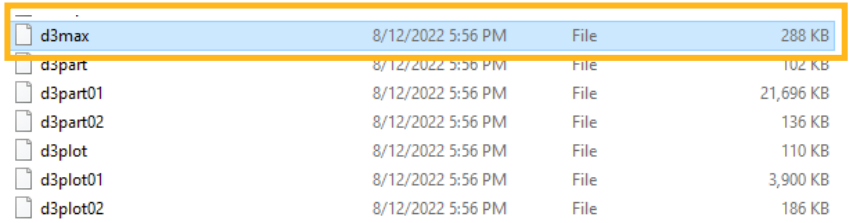
D3Max

- Only support **Stress** result
- Contain a single time and be active when **Maximum Over Time** is set to Yes
- Pre work in Mechanical

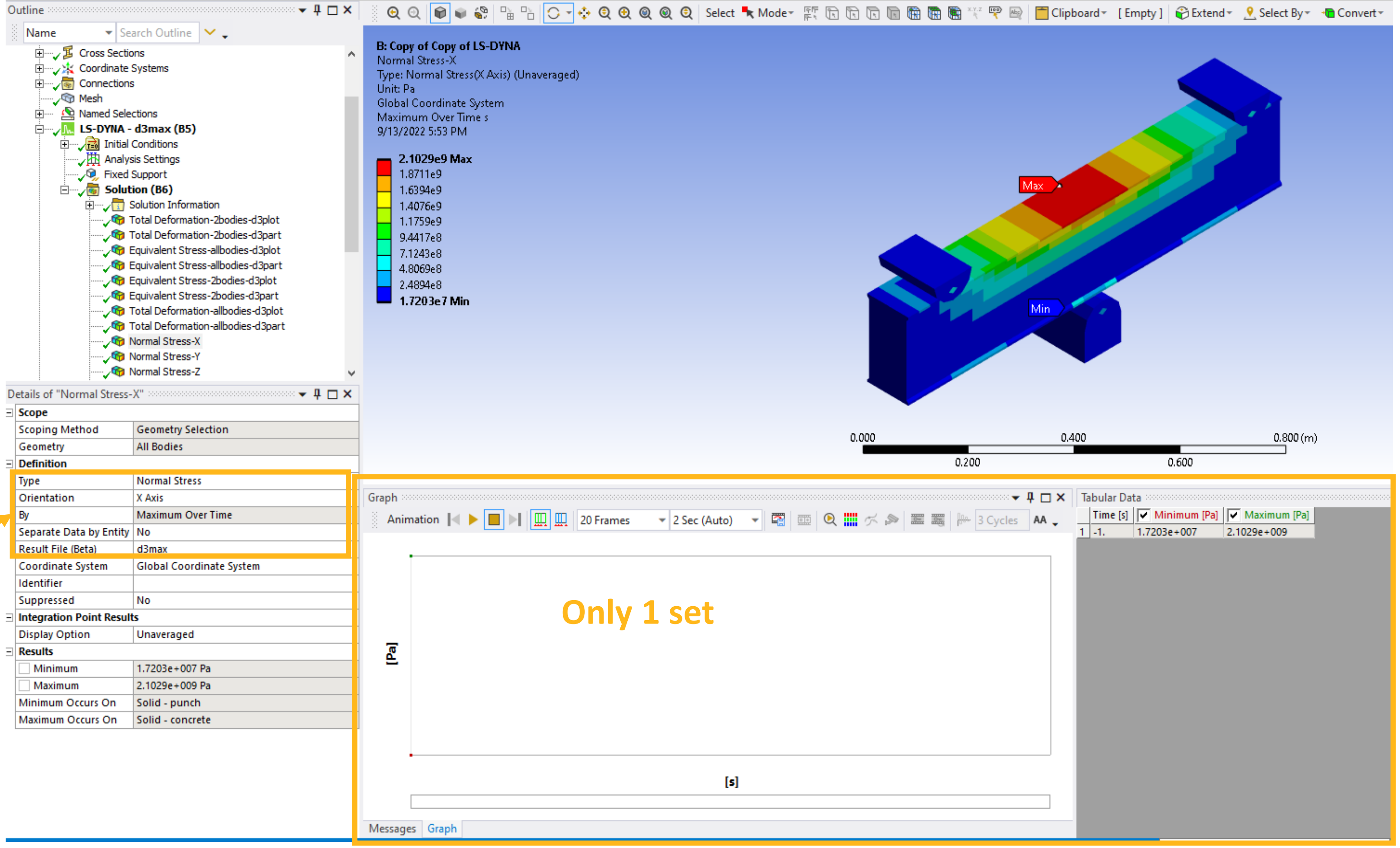


D3Max

- Postprocessing



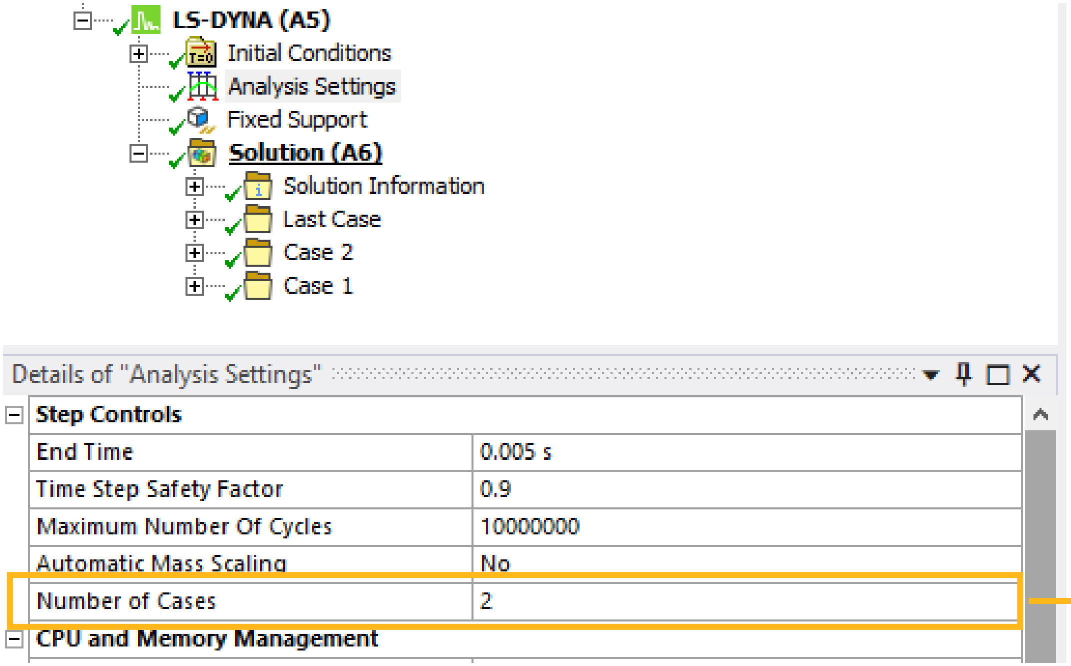
Maximum Over Time will become read-only when the d3max file is selected

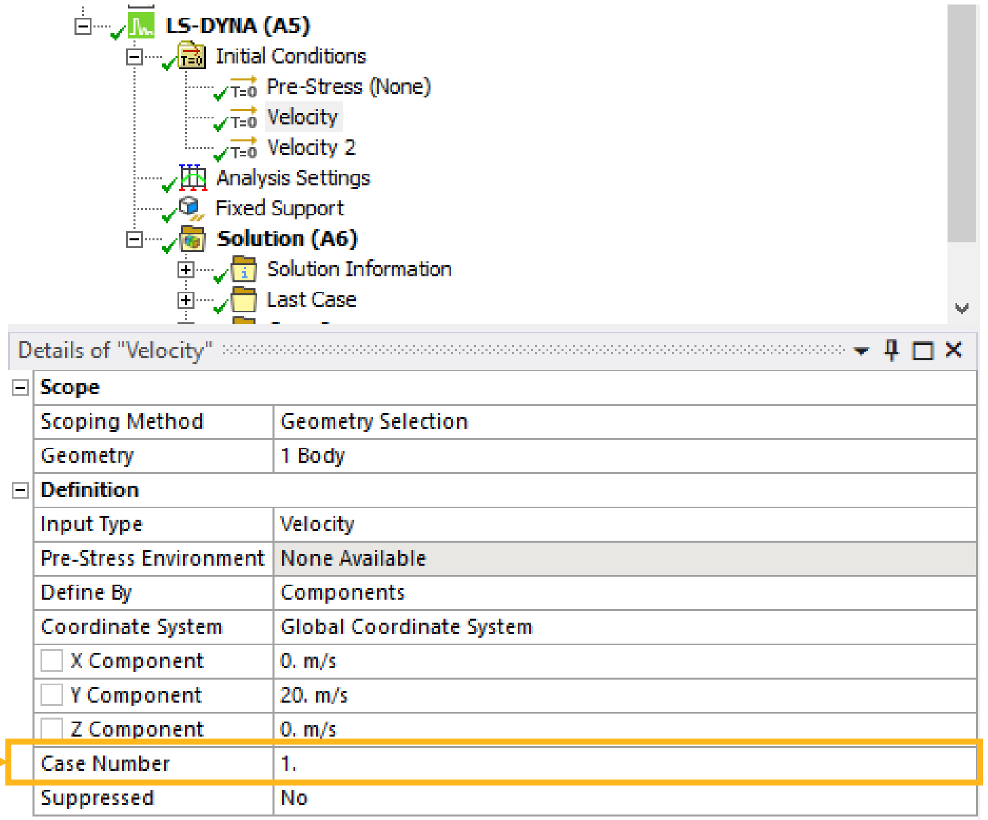


Ansys

Case

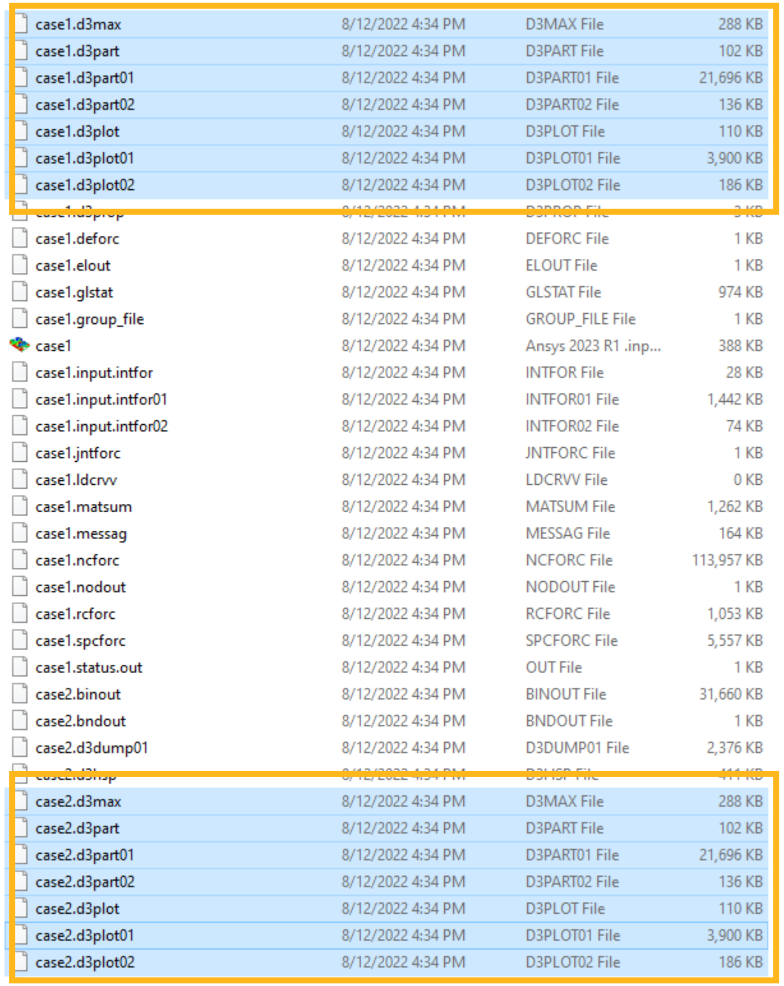
- Provides a way of running multiple LS-DYNA analyses (or cases) sequentially by submitting a single input file.
- Prep work in Mechanical GUI

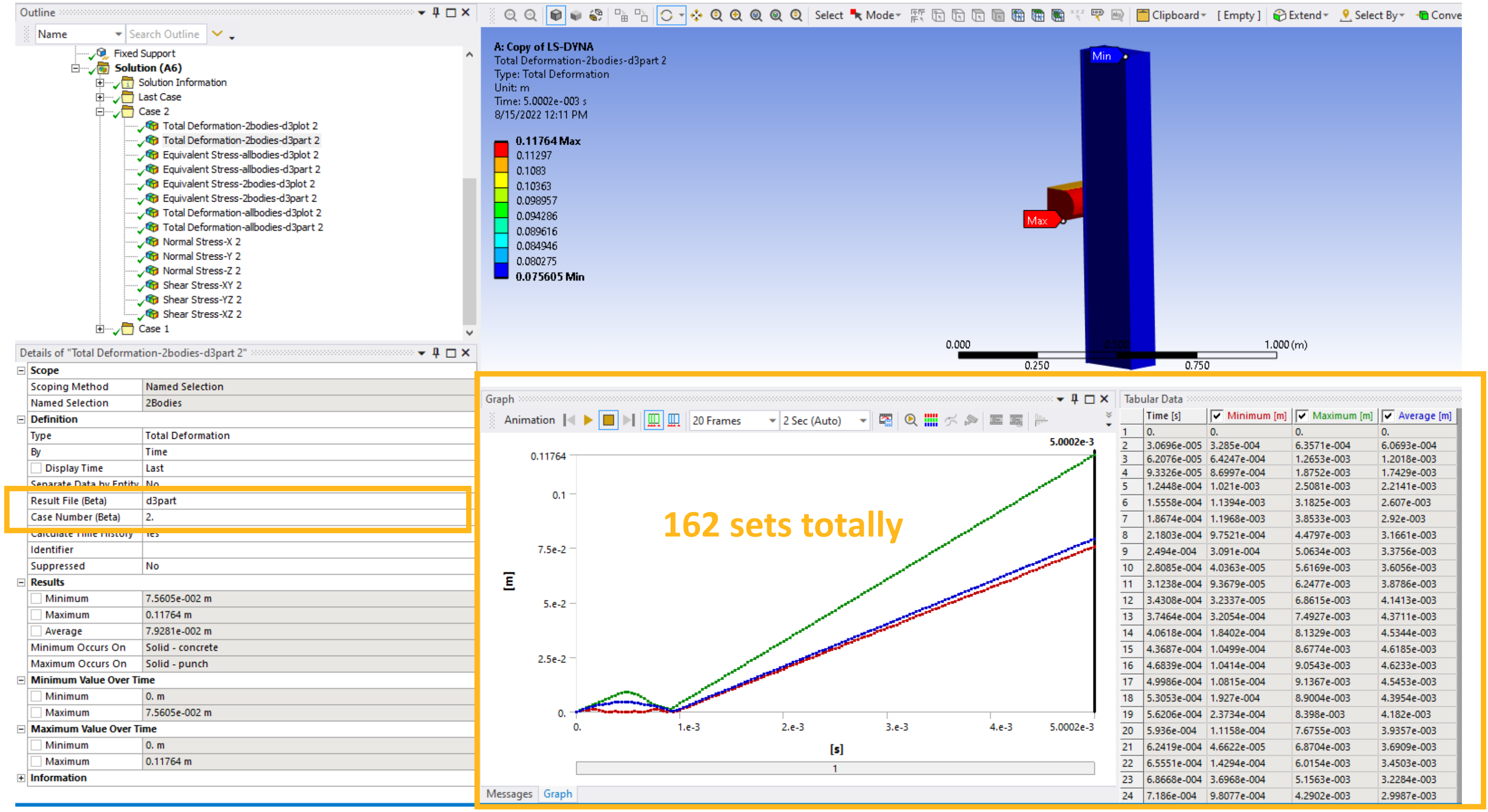




Case

- Postprocessing





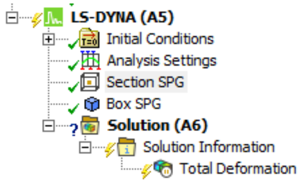
Ansys

LS-DYNA other solvers

SPG : Smoothed Particle Galerkin

- LS-DYNA now supports support Smoothed Particle Galerkin (SPG)

- SPG is particularly useful to simulate destructive manufacturing such as riveting, screwing, drilling and machining.
- SPG is only available for solid elements



Details of "Section SPG"

Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Method	SPG
Formulation	Program Controlled
LS-DYNA ID	47
Type	Section Solid SPG
SPG Controls	
Dilation X	1.6
Dilation Y	1.6
Dilation Z	1.6
Kernel Function	Program Controlled
Kernel Scheme	Program Controlled
Time Steps for Displacement Smoothing	15
Smoothing Scheme for Momentum	Program Controlled
Advanced SPG Controls	
Bond Failure Mechanism	Program Controlled
Critical Failure Value	10000000000
Critical Relative Deformation	10000000000
Option of Stabilization	Program Controlled
Quadrature Factor	1
Self-Contact Indicator	Program Controlled
Box	Box SPG
Particle-to-Particle Damping Coefficient	-0.001

SECTION_SOLID_SPG

SECID	ELFORM=47	AET					
DX	DY	DZ	ISPLINE	KERNEL		SMSTEP	
IDAM	FS	STRETCH	ITB	MSFAC	ISC	BOXID	PDAMP

ISPG : Incompressible Smoothed Particle Galerkin

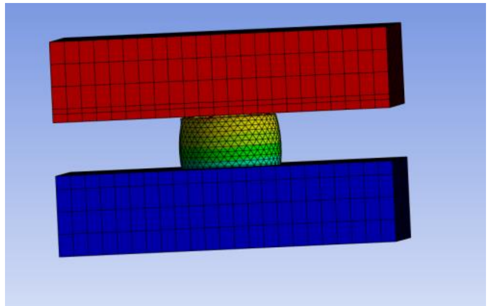
- The LS-DYNA system now supports Incompressible Smoothed Particle Galerkin (ISPG)
- ISPG is suitable for the simulation of incompressible free surface fluid flow.
- For instance, the simulation of shape evolution of solder joints in electronic equipment during the reflow process.
- ISPG is only available for tetra elements

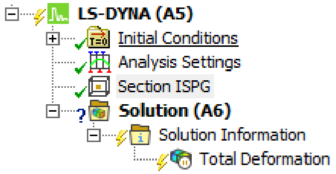
SECTION_FPD

SECID	**ELFORM**						
DX	**DY**	**DZ**					
MCVISC							

MAT_IFPD

MID	**RO**	**MU**	**GAMMA**				





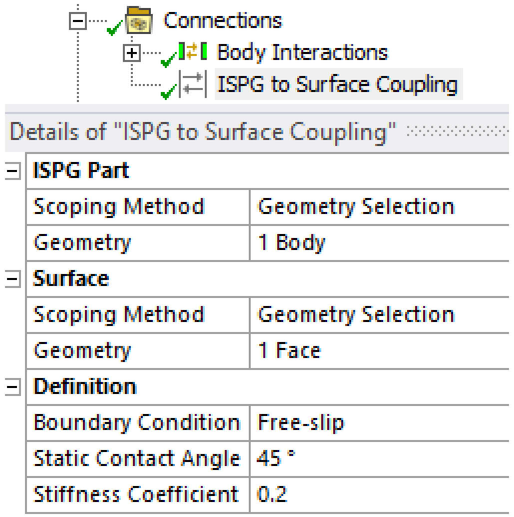
Details of "Section ISPG"

Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Method	ISPG
Formulation	Program Controlled
LS-DYNA ID	49
Type	Section FPD
ISPG Controls	
Dilation X	1.6
Dilation Y	1.6
Dilation Z	1.6
Relaxation Parameter	0.8
ISPG Material Controls	
Density	1 kg/m^3
Dynamic Viscosity	1 Pa·s
Surface Tension	1 kg/s^2

Ansys

ISPG To Surface Coupling

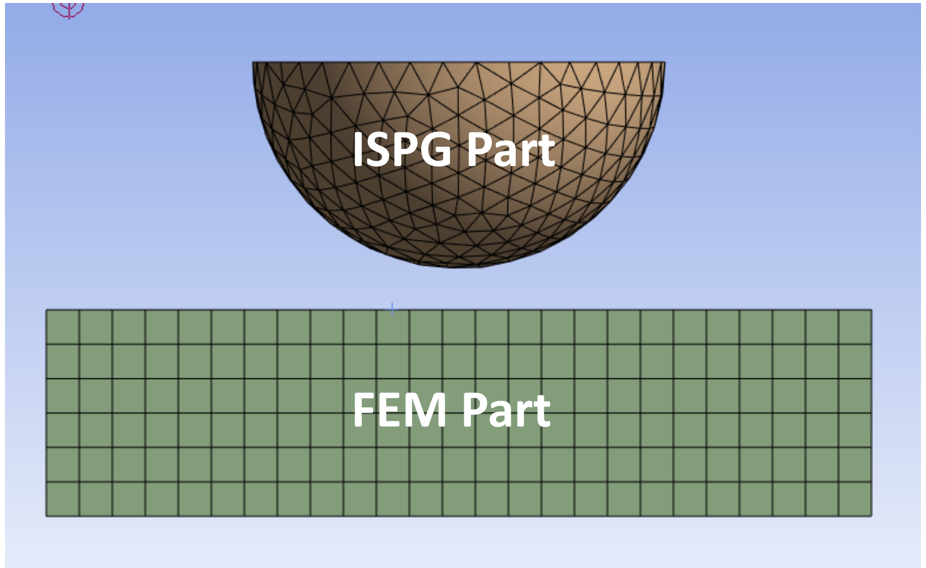
- Defines contact between ISPG bodies and other finite element bodies



**DEFINE_FP_TO_SURFACE_COUPLING*

SLAVE	**MASTER**	**STYPE**	**MTYPE**				
SBC	**SCA**				**SFPN**		

SCA : *Static Contact Angle* between the ISPG and FE part
SFPN : *Stiffness coefficient* along the normal direction of the contact interface
SBC : Type of Boundary Condition : **Free- Slip** *or* **Non-Slip** boundary

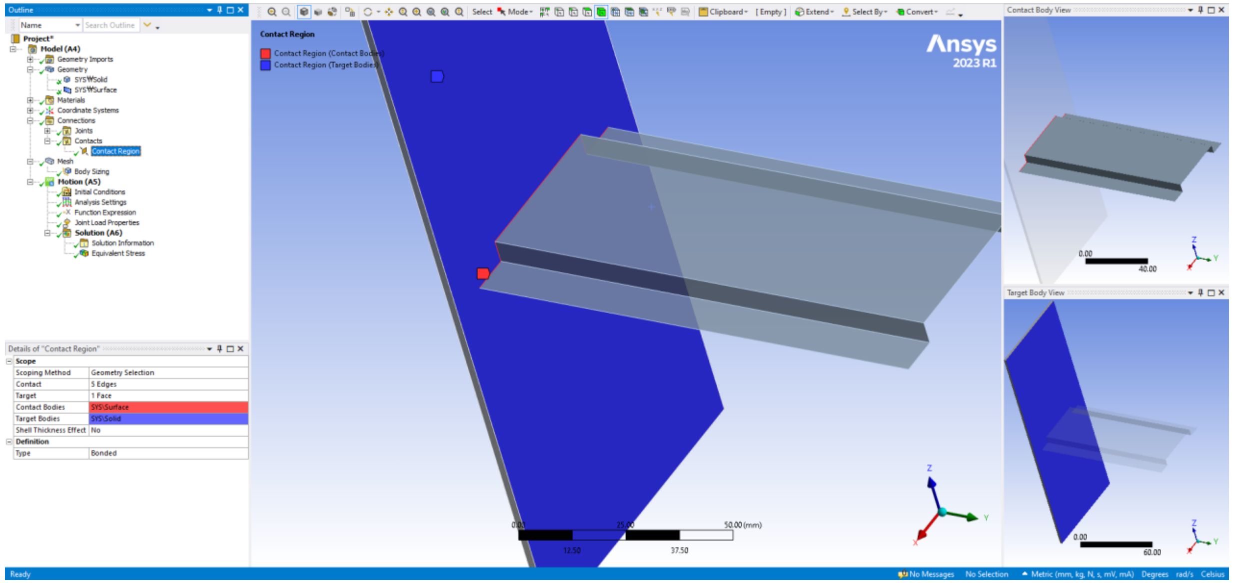


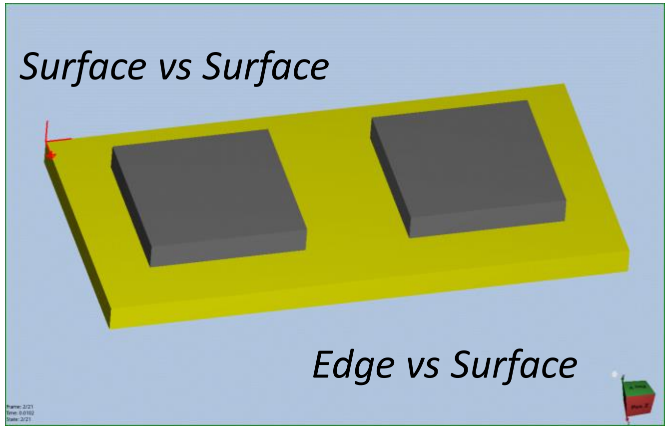
Multibody Dynamics

Mechanical Motion

Edge Contact

- Flexible body edge contact is available when contact region scoping contains edges.
  - If users have a structural model that contains Edge Contacts, dynamic analysis can be performed easily by converting the system to Motion.



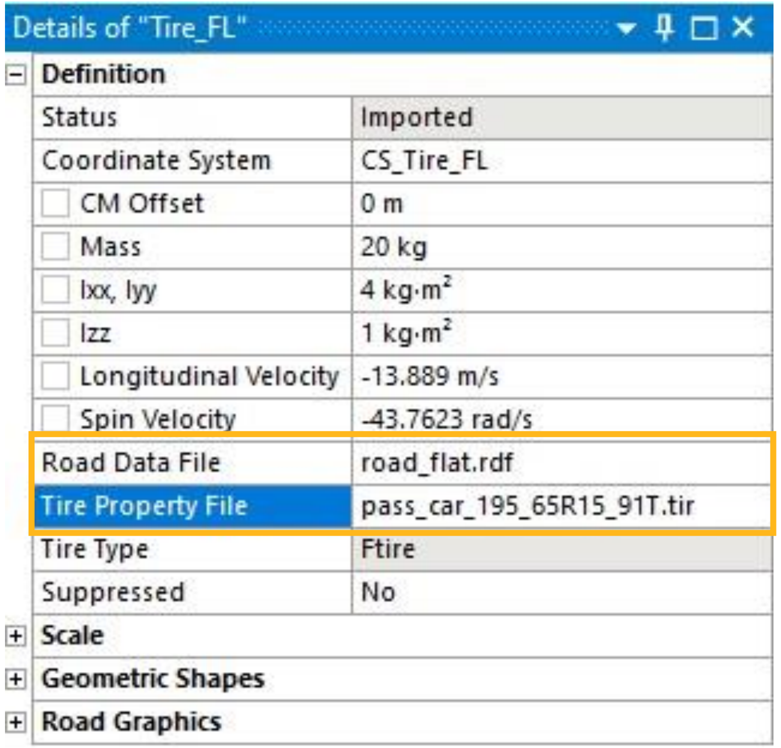


		Contact/Base	
		Face	Edge
Target / Action	Face	O	O
	Edge	X (O-STD)	O

< Possible combinations >

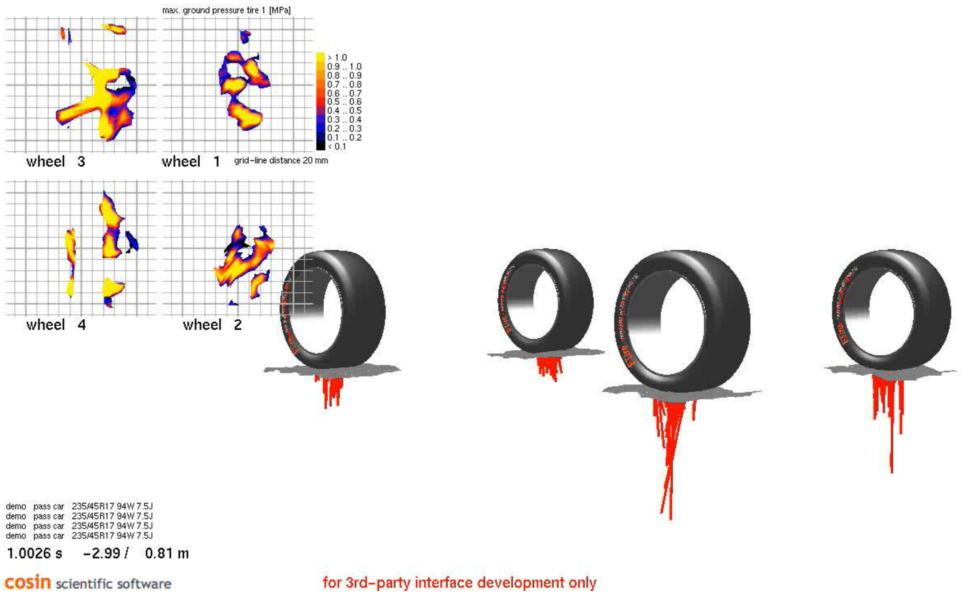
FTire Interfacing

- Motion now supports co-simulation with cosin scientific software by using the FTire format tire property file.
- RGR and CRG files can also now be used as road data files.



Details of "Tire_FL"

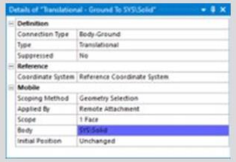
Definition	
Status	Imported
Coordinate System	CS_Tire_FL
CM Offset	0 m
Mass	20 kg
Ixx, Iyy	4 kg·m²
Izz	1 kg·m²
Longitudinal Velocity	-13.889 m/s
Spin Velocity	-43.7623 rad/s
Road Data File	road_flat.rdf
Tire Property File	pass_car_195_65R15_91T.tir
Tire Type	Ftire
Suppressed	No
Scale	
Geometric Shapes	
Road Graphics	

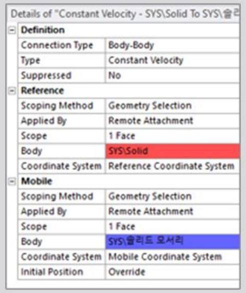
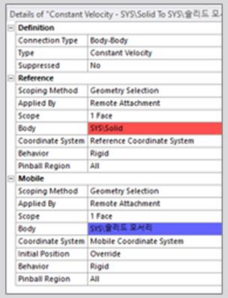


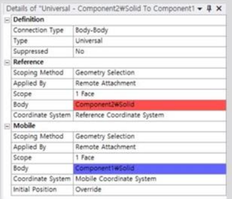
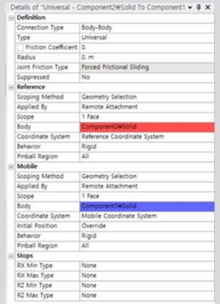
< Visualization of FTire simulation >

Joint Consistency

- Universal Joint, Constant Velocity Joint and Translational Joint, that were not using consistent degrees of freedom between Motion and Rigid Dynamics have been made consistent.

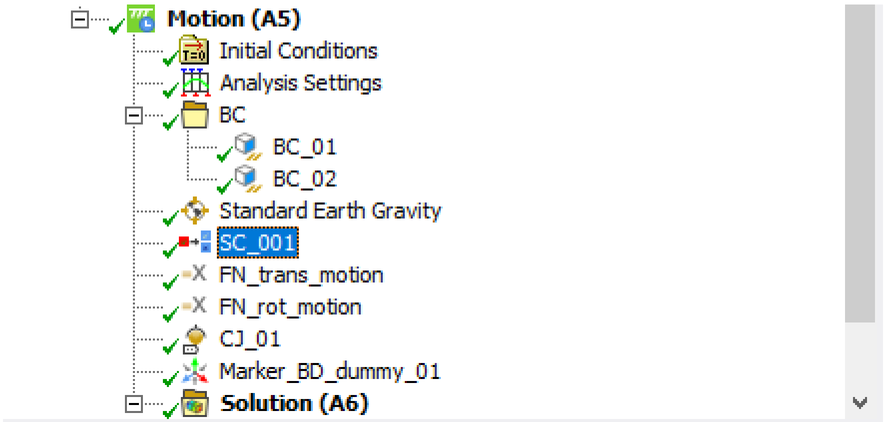
Translational Joint	Motion	RBD
Details		
Axis of the translation	X Axis (but use Z Axis in Motion Solver)	X Axis

Constant Velocity Joint	Motion	RBD
Details		
Axis of the rotation	Z Axis	Y Axis
Orthogonal Axis	X Axis	X Axis

Universal Joint	Motion	RBD
Details		
Cross-shaft orientation of Reference coordinate	X Axis	Z Axis
Cross-shaft orientation of Reference coordinate	Z Axis	Z Axis

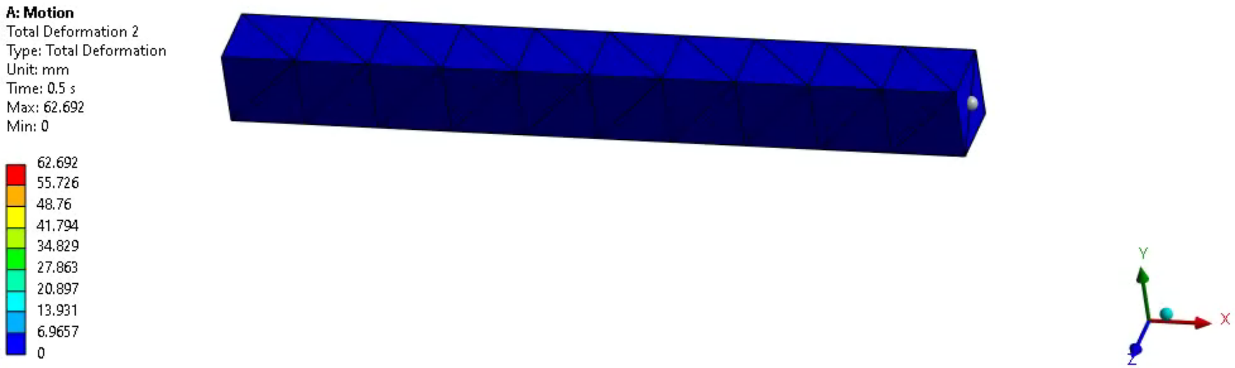
Simulation Scenario with BC

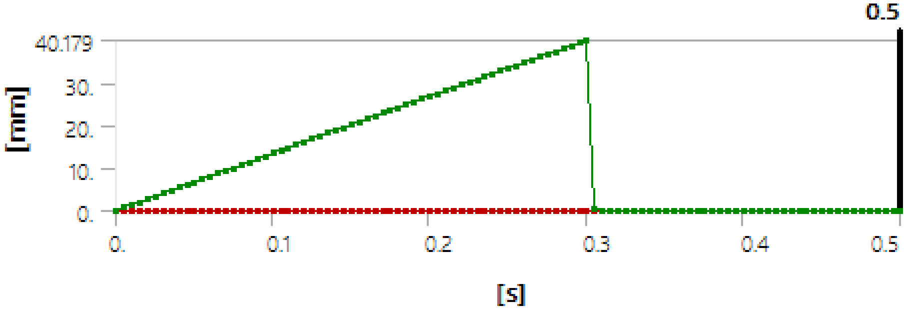
- Simulation Scenario is available to DEACTIVATE or ACTIVATE Boundary Condition while running simulation according to the user-defined criterion.



Details of "SC_001"

Definition	
Sensor Type	Time
Sign	>=
Reference	0.3
Event Type	DEACTIVATE
Target	BC_02
Suppressed	No

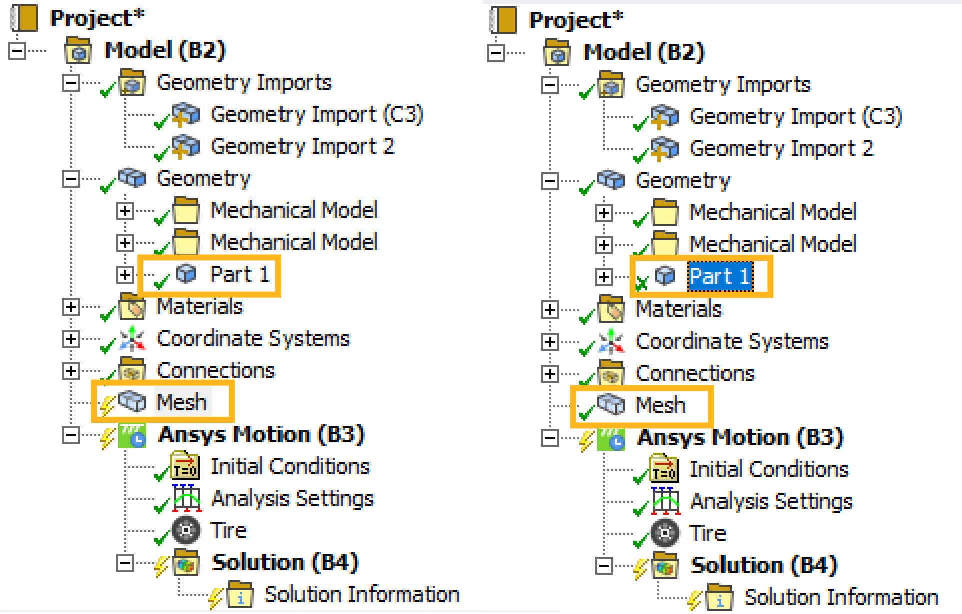




< Deactivate BC relative to sphere at 0.3 s >

Mesh Automation

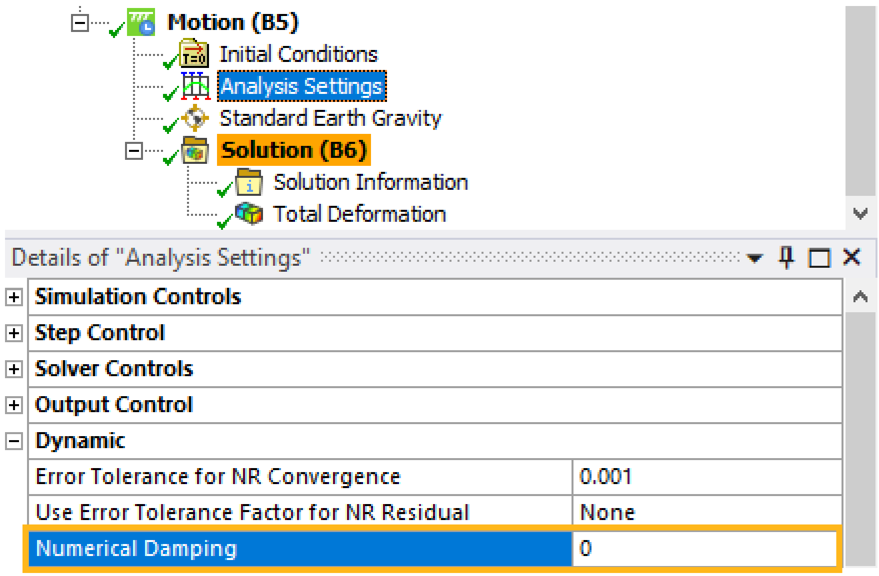
- In assembly model, users should generate mesh for imported bodies to make the model ready to solve. It is difficult for users to notice and cumbersome to mesh manually.
- Now mesh will be generated after importing the bodies. So, it is possible to use geometry importing capability in Motion without considering this limitation anymore.
- Example: After creating a geometry for Tire object.
  - Previously in this case, the state of mesh was ***Not Solved***.
  - From 2023 R1, it will change to ***Solved***.

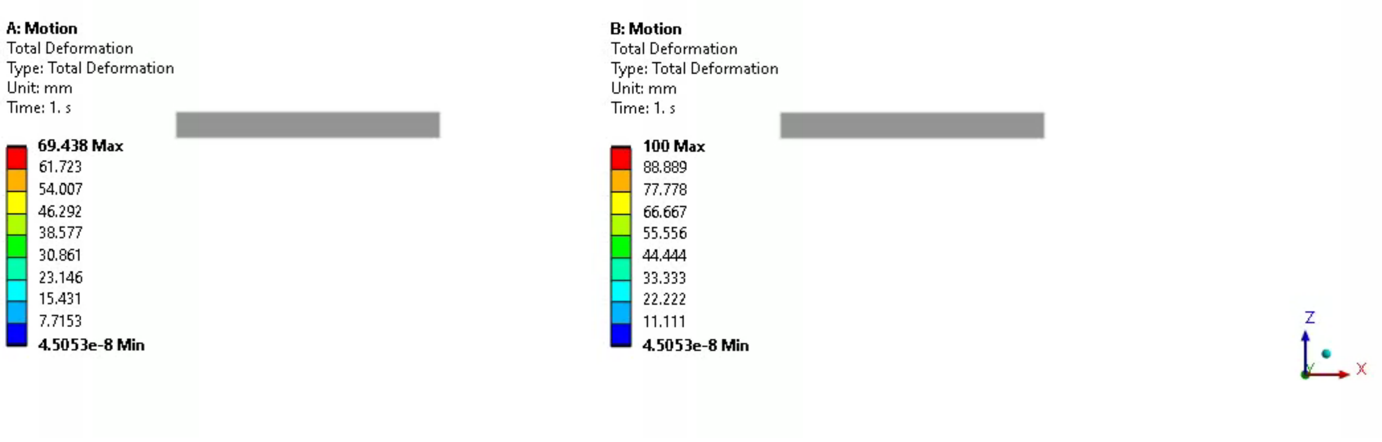


< Previous vs. 2023 R1 >

Numerical Damping

- It is now possible to input Numerical Damping within the range of values between zero and less than 4.
- Example: Pendulum simulation with Numerical Damping of 3.99 and 0.



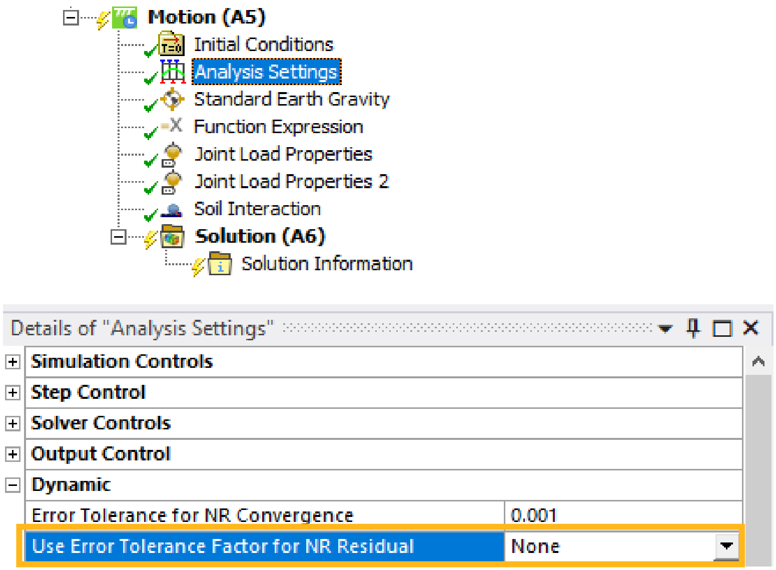


< Numerical Damping 3.99 vs. 0 >

-> It can be seen that the smaller the value, the more freely the movement is.

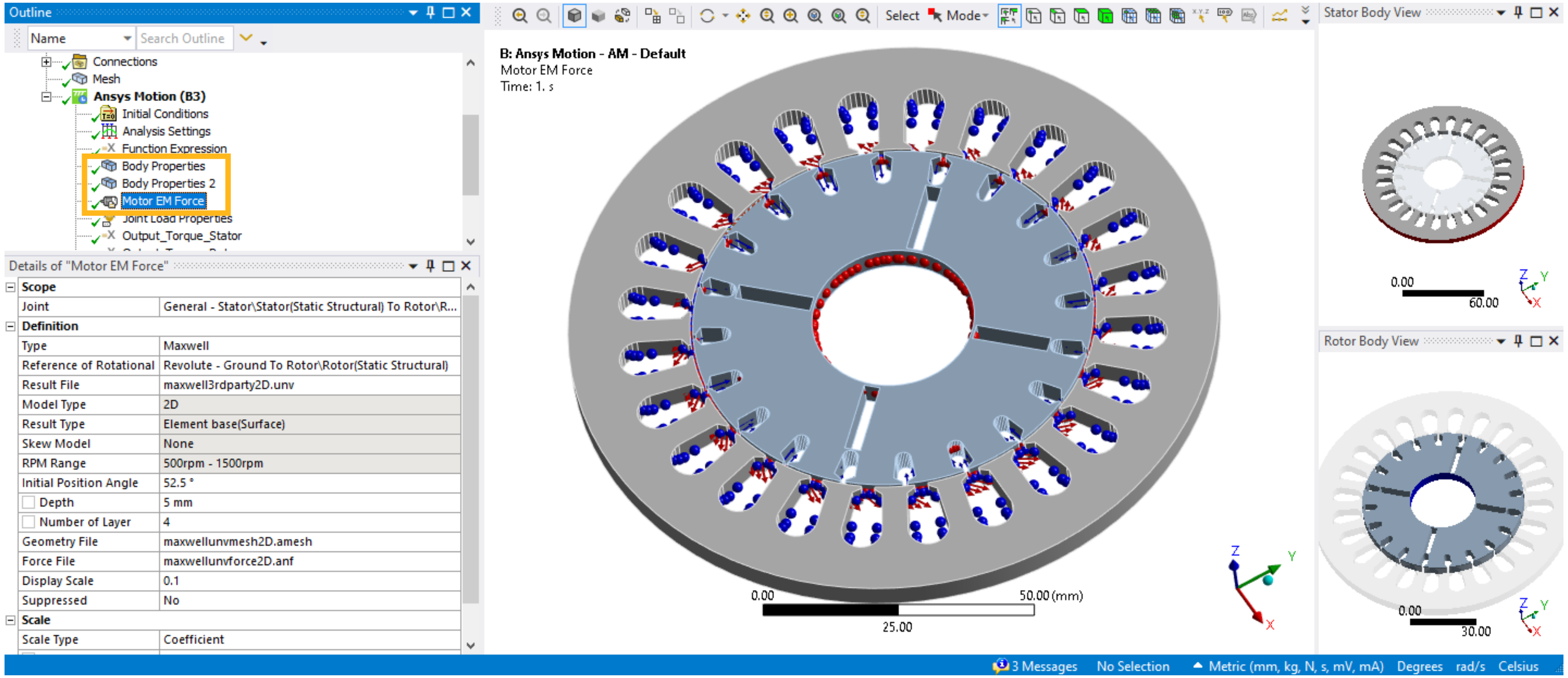
Residual Tolerance

- An option to use Error Tolerance Factor for NR Residual is now available in Analysis Settings.
- It may be helpful to clear this option if the number of NR failures is greater than number of integration failures.
- Solver provides recommendations via log file message, for better options in subsequent solutions.



Solving Speed for Motor EM Force (with modal bodies)

- By optimizing the way of managing Electromagnetic force data that is imported from Maxwell, the simulation performance has been improved up to three times over previous versions.



< Motor EM Force with modal stator and rotor >

Ansys

Ansys nCode Design Life

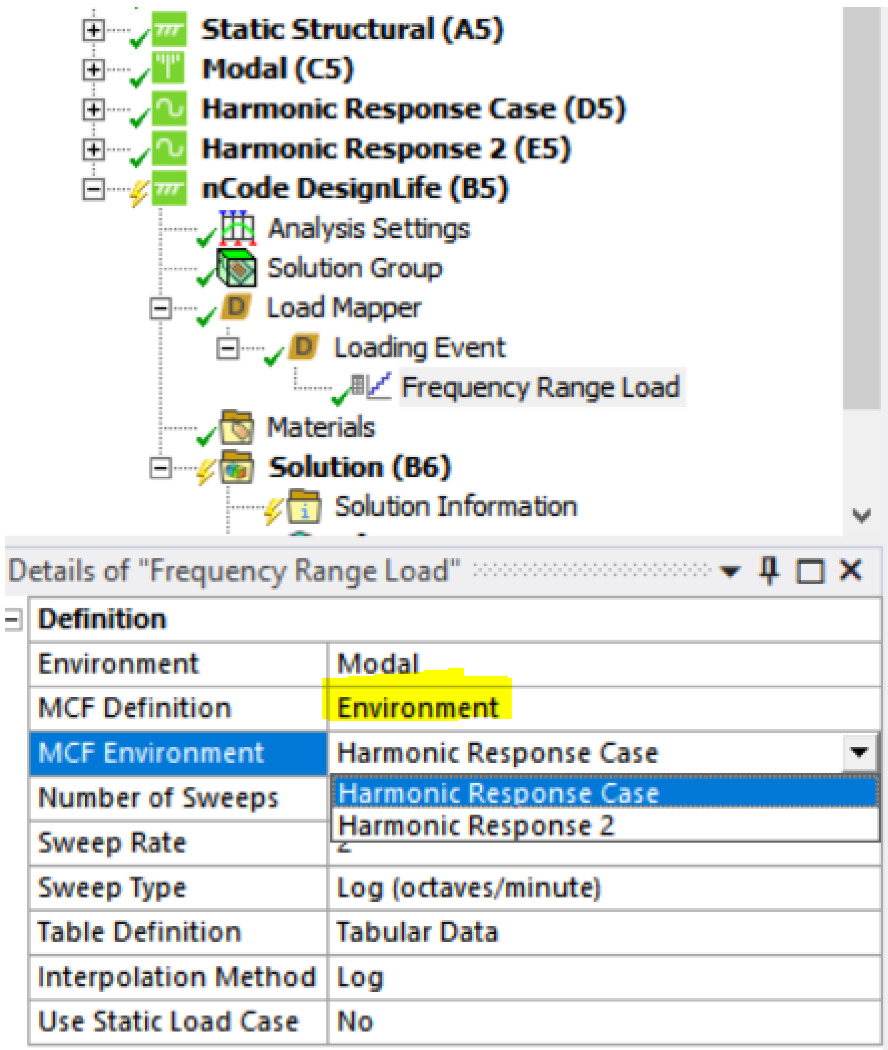
1.1 Frequency-Based Modal Superposition Vibration Fatigue Analysis

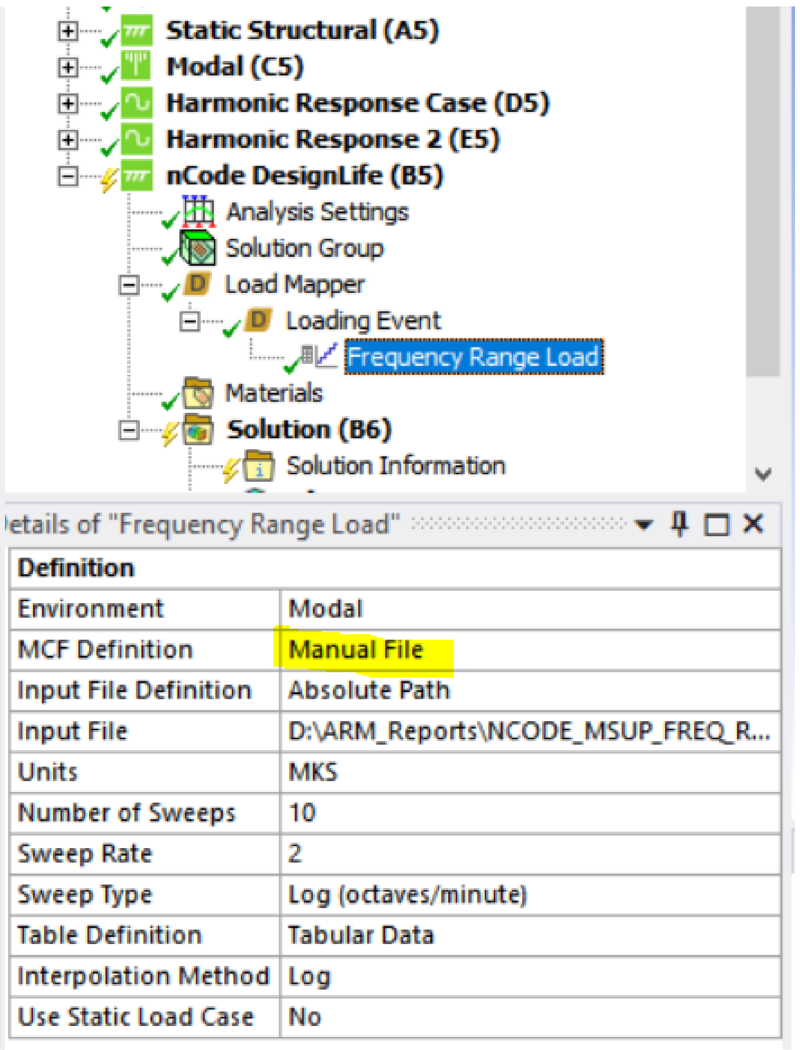
- The DesignLife add-on now supports modal superposition frequency-based vibration fatigue using modal results.

Vibration Fatigue Load	Stress	Strain	Shell Seam Weld	Solid Seam Weld
PSD Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Single Frequency Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Frequency Range Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓
Sine On Random Loading, Including: - Static Offset Case - Single and Multiple Events	✓	✓	✓	✓

1.2 Frequency-Based Modal Superposition Vibration Fatigue Analysis

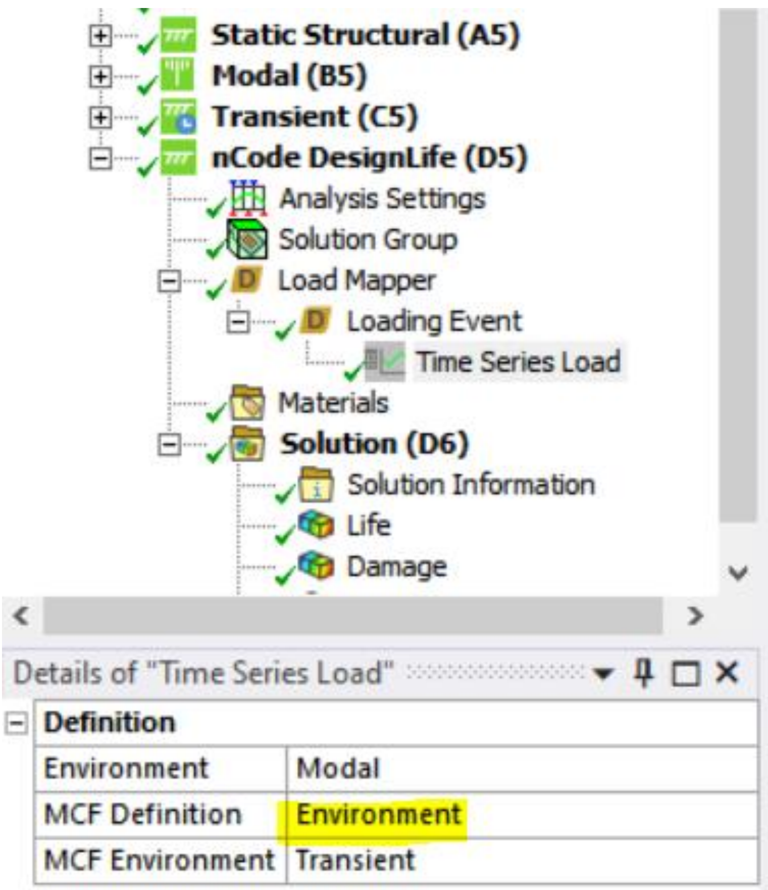
- This approach requires a modal results file and a Modal Coordinates file from a harmonic analysis.
- You can identify the mcf file location by linking the add-on to a **Harmonic** system or by importing it from an external location.
    - Modal rst + mcf file from linked Harmonic system
    - Modal rst + imported mcf file

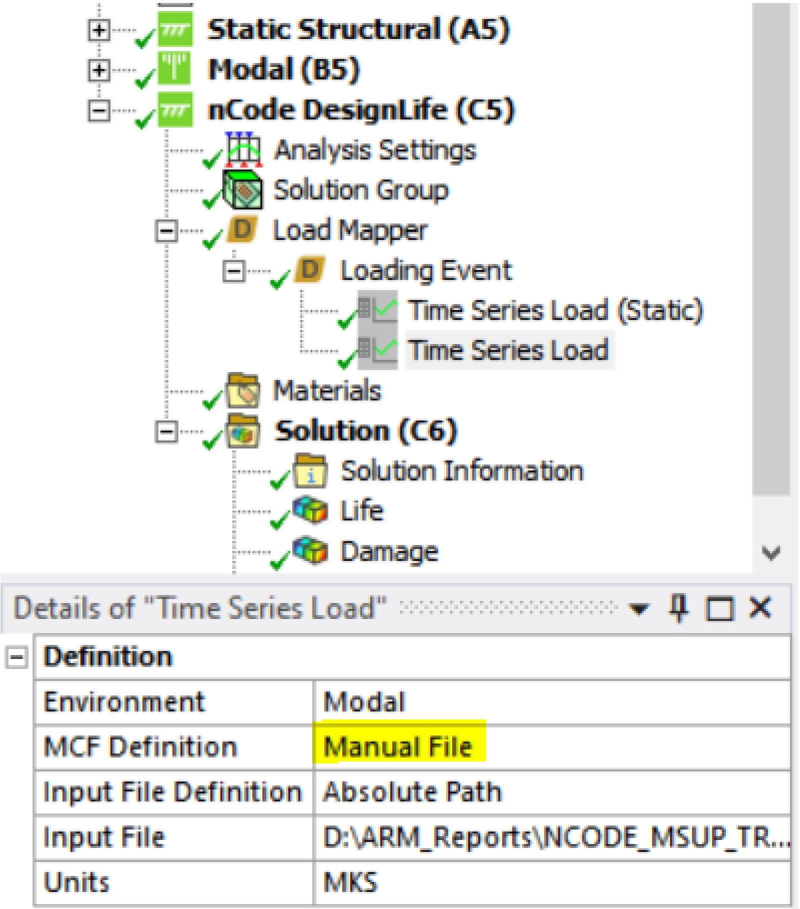




2.1 Time-Based Modal Superposition Fatigue Analysis

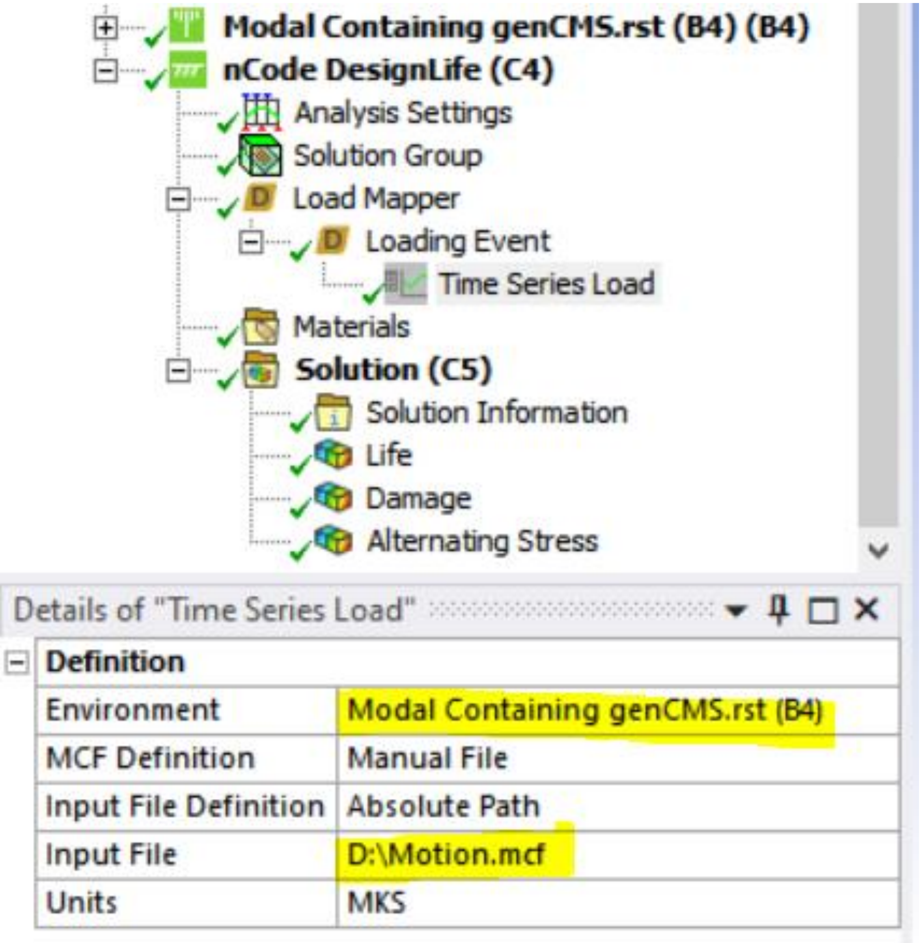
- The DesignLife add-on now supports modal superposition time-based fatigue using modal results and time series loading.
- This approach requires a modal results file and a Modal Coordinates file from a transient system. You can identify the mcf file location by linking the add-on to a **Transient** system or by importing it from an external location.
  - Modal rst + mcf file from linked Transient system
  - Modal rst + imported mcf file





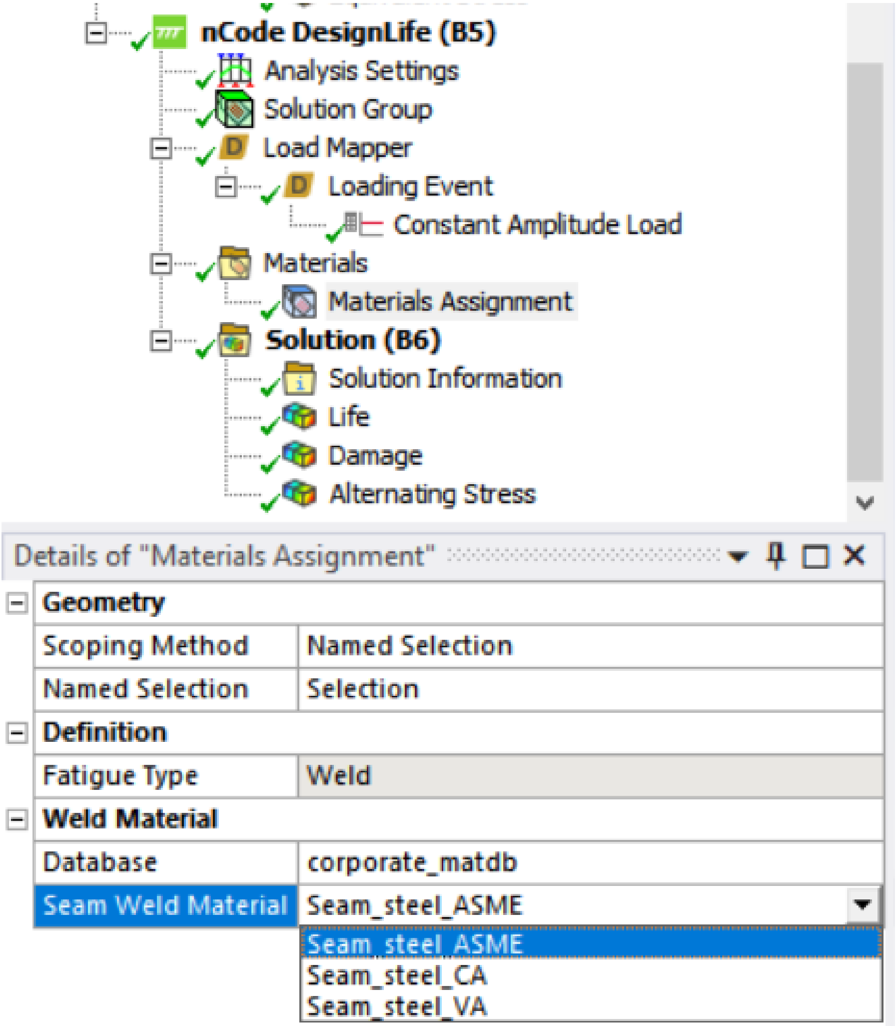
2.2 Time-Based Modal Superposition Fatigue Analysis (Motion)

- The add-on also now supports modal superposition time series fatigue using **Ansys Motion** results.
- For **Motion**, you must link the add-on to the Modal environment containing the genCMS.rst file and import the appropriate Motion mcf file.
  - Modal genCMS rst + imported Motion mcf file



3.1 Seam Weld Material Customization

- DesignLife Add-on now supports customization of the weld material for **Solid** or **Shell Seam Weld** analysis.
  - You must identify the database (*.mxd file) containing the customized weld material data.
  - Then select that material from the database.



Ansys