DRD TECHNOLOGY

DRD Webinar:
Mechanical 2025 R2 Updates
28 August 2025

Alex Austin – Structural Team Manager

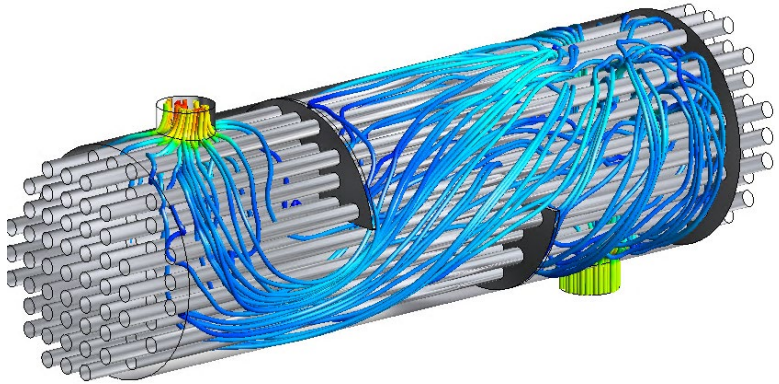
DRD

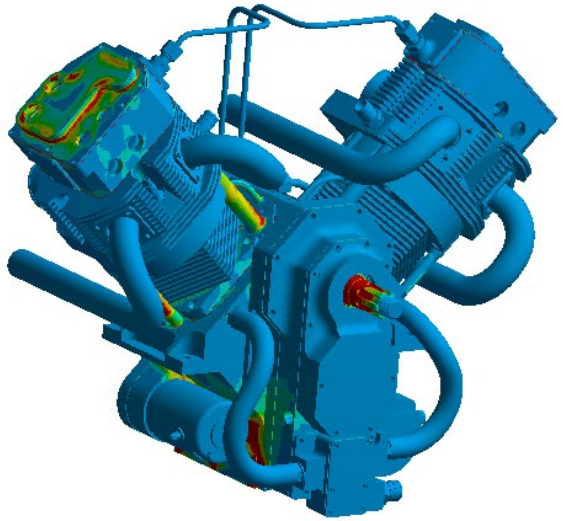
Agenda

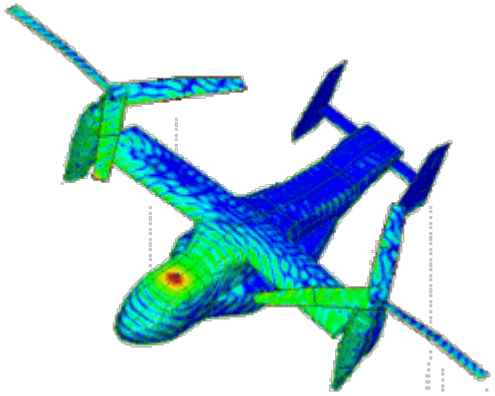
- Introductions
- Mechanical 2025 R2 Updates
- What’s New
- Wrap-Up
  - Webinar Content
  - DRD On-Demand Training Content

Mission Statement

DRD Technology helps engineering teams accelerate product development. With in-house expertise spanning the entire range of physics, we ensure customers succeed when using Ansys simulation tools for virtual prototyping and design verification.





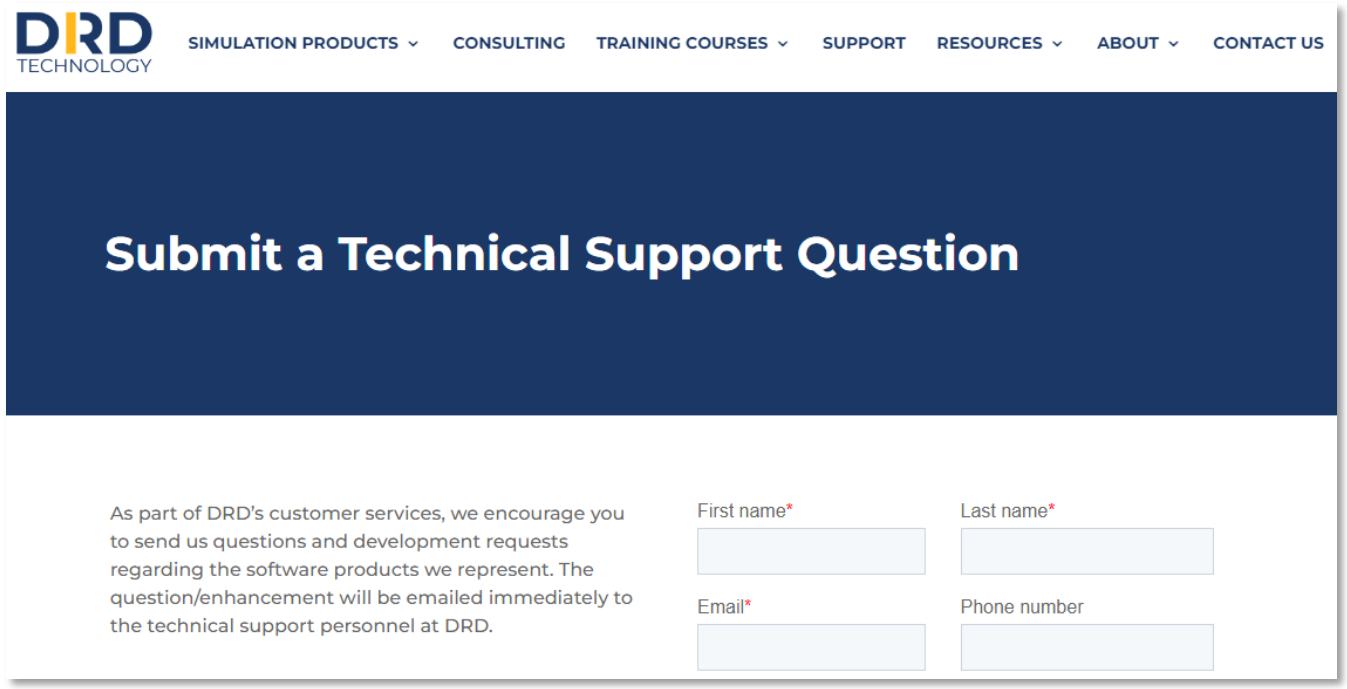


Ansys part of SYNOPSYS® | ELITE CHANNEL PARTNER

Technical Support Contact Coordinates

Support:
(918) 743-3013 x1
support@drd.com

Or through our website at
www.drd.com



For more than five years, I have worked closely with DRD Technology to execute tactical and strategic initiatives here at EaglePicher due to our unprecedented growth. We've been very happy with DRD and will continue to work with them as our business partner for using Ansys tools effectively and efficiently.

- Doug Austin
Director of Research and Development
EaglePicher™ Technologies, LLC

AnsysGPT Virtual Assistant

- A **virtual assistant** designed to assist users with Ansys specific knowledge.
- Provides **high quality responses in minutes** by tapping into a vast database of Ansys knowledge.
- Holds conversations in **multiple languages**.
- **Scalable & secure** deployment to customers on Ansys Innovation Space.
- Built using Azure OpenAI Service.

AI-Driven Innovation, Built into Mechanical

Ansys Engineering Copilot is a secure, robust, and reliable virtual assistant that leverages 50 years of technical support expertise, to be a central Copilot for your engineering work.

- ✓ **Convenience:** AI assistant directly integrated into your native Ansys simulation products.
- ✓ **Expertise:** Leverage 50 years of Ansys technical knowledge and expertise..
- ✓ **Learning :** Access guided and peer-to-peer learning and support.
- ✓ **Support :** Customers can create and track support cases

Available directly in Ansys Mechanical

Powering Innovation That Drives Human Advancement

Mechanical Meshing Development

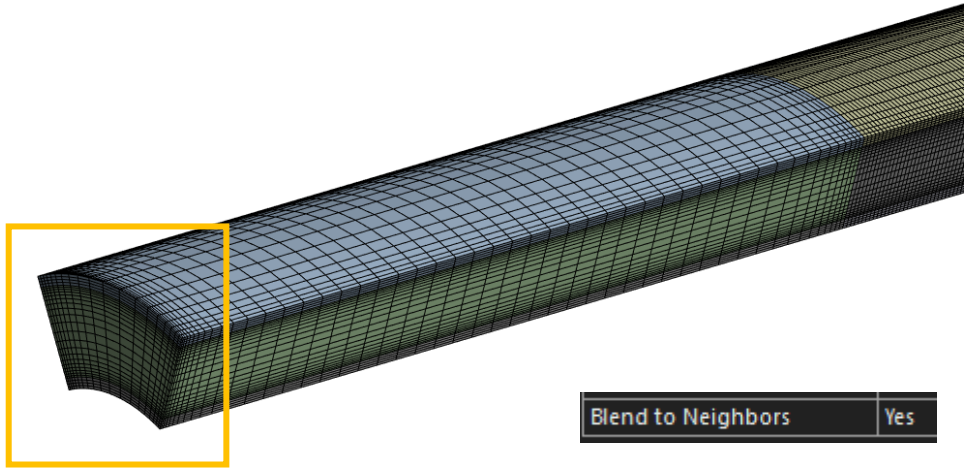
Mechanical Meshing Enhancement Targets in 2025R2

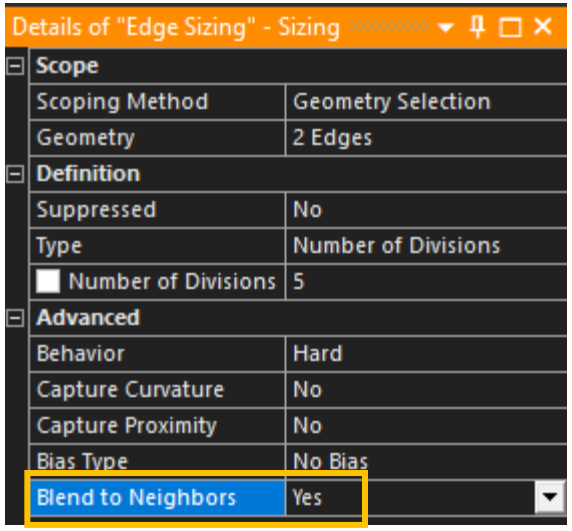
- **Meshing for Electronics**
  - Release from Beta: Fast Stacker Mesh Workflow for Electronics-Reliability hex meshing of assemblies with solder balls
    - Improved Global Settings/Usability; Improved robustness, Improved warning/error/diagnostics; Support side face seed mesh; MZ in workflow improvements for solderball; Mesh Replicate and Merge Nodes for fast meshing of similar solderballs; Multi-threaded surface meshing; Quad Layer Control support
    - Beta use of Template to create solderball geometry+mesh
- **NVH and Mesh Workflows**
  - FSI workflow enhancements
  - Enhanced tet meshing in workflow to support direct meshing (instead of wrapping i.e. topology/surface mesh clean-up steps)
  - Release of Beta options for acoustics workflow – e.g. microphone placements, Size control enhancements, size field visibility in Domain Browser
  - Many usability enhancements for Mesh Workflows
- **Morphing (Beta)**
  - Direct Morphing mesh workflow
    - I/O for meshes from External Model and/or Mesh Methods via Clone Mesh
      - Passing of Named Selections as labels for morphing
    - Morph control improvements with previews for control scopes
    - Mesh Edit/Morph Control application to mesh setup with BCs etc intact
- **General Tet Meshing**
  - Performance improvements
  - Auto-Map Fillets release
- **Welds and Shell Meshing**
  - Automatic (Prime Mesh)
    - Removal of limitations, improved sizing, …
    - Geometry projection
    - Improved interoperability with other Methods
    - Prime Quad Dominant becomes default sheet body (shell) method
- **Hex Meshing**
  - General performance/robustness improvements for ThinSweep and Medial Axis as well as multi-body meshing
  - Smoothing in MZ Method (Beta)
  - Released from Beta: Blend to Neighbors (Edge Sizing) Control; Suppress Topology for edges/vertices with MZ; Show axisymmetric bodies
  - Enhanced Mesh Workflows for Hex Meshing (Multizone volume and surface enhancements, New Hex Dominant Method, Auto-splitting,…)
- **Meshing Performance**
  - Parallel in-part body-group meshing (Beta)
- **Usability**
  - Pull bodies renamed
  - Improved contact creation with contact detection

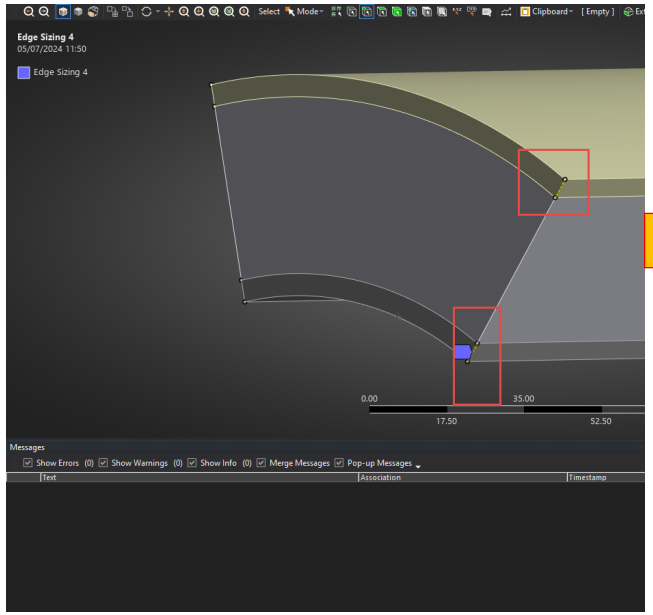
Mesh Sizing

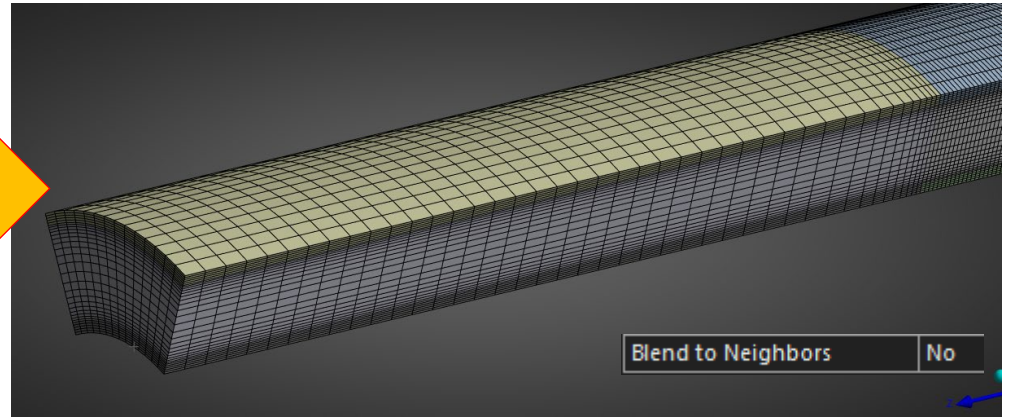
Edge Sizing: Blend to Neighbors

- In Edge Sizing disable “Blend to Neighbors “to avoid unwanted size biasing due to size field on the source/target faces
  - This was fixed as yes in the past with no ability to de-activate which helps attaining anisotropic mesh



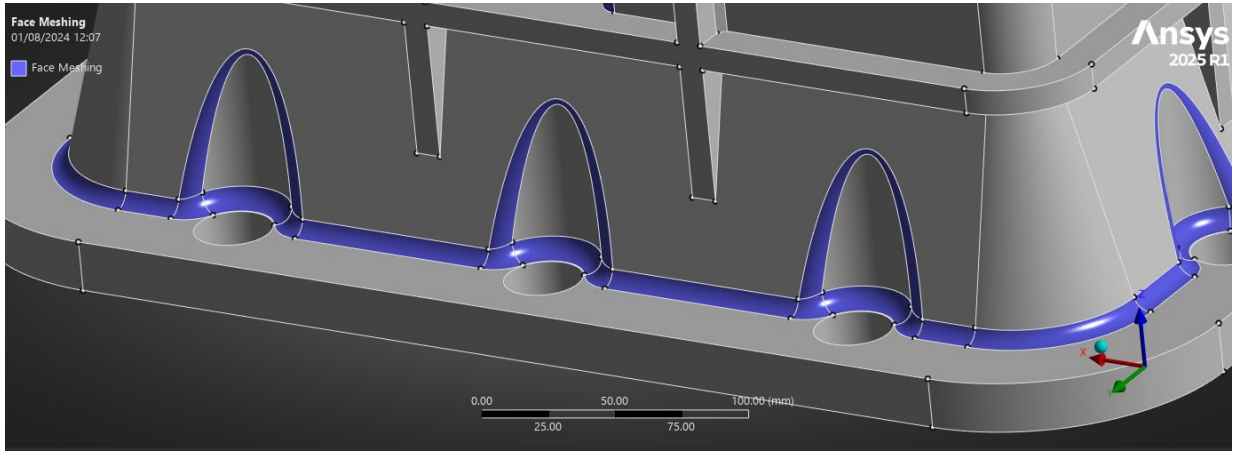


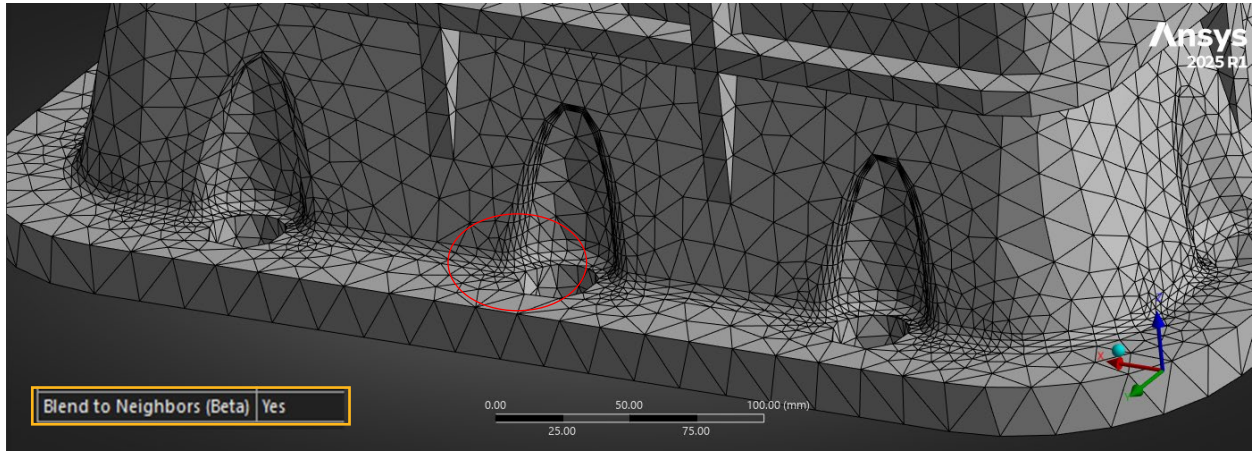


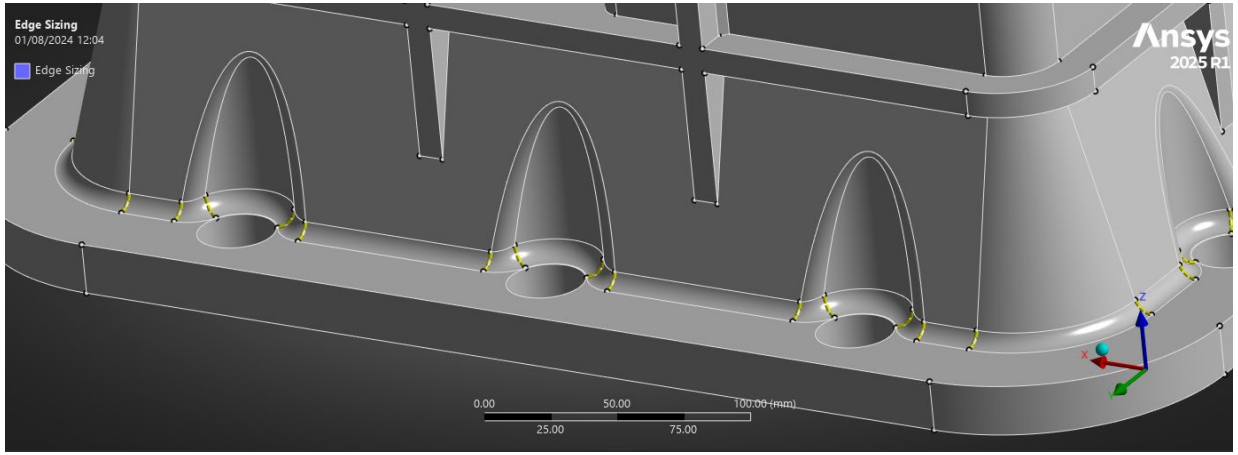


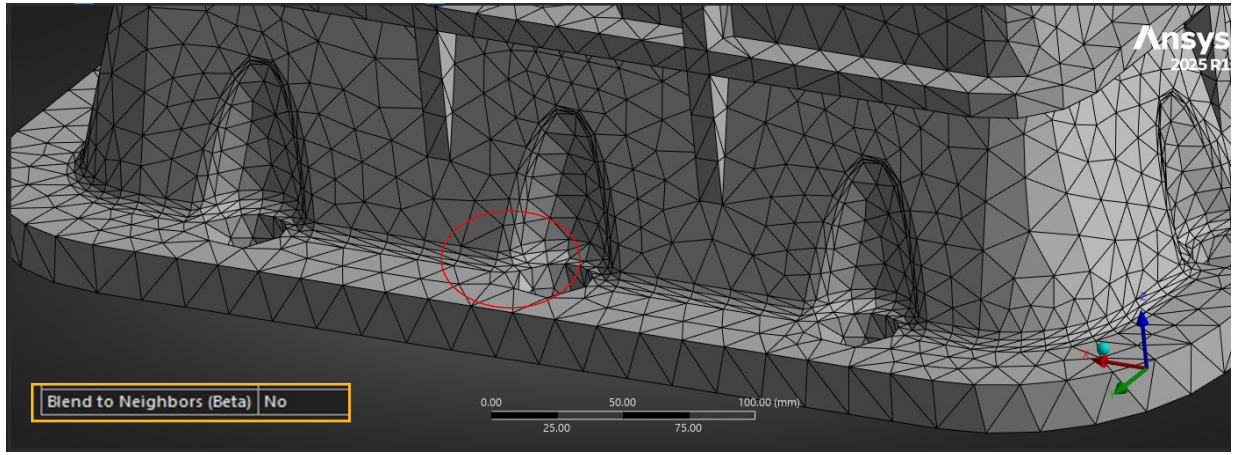
Edge Sizing: Blend to Neighbors (Release from Beta)

Allows faster change to anisotropic, stretched elements in fillets



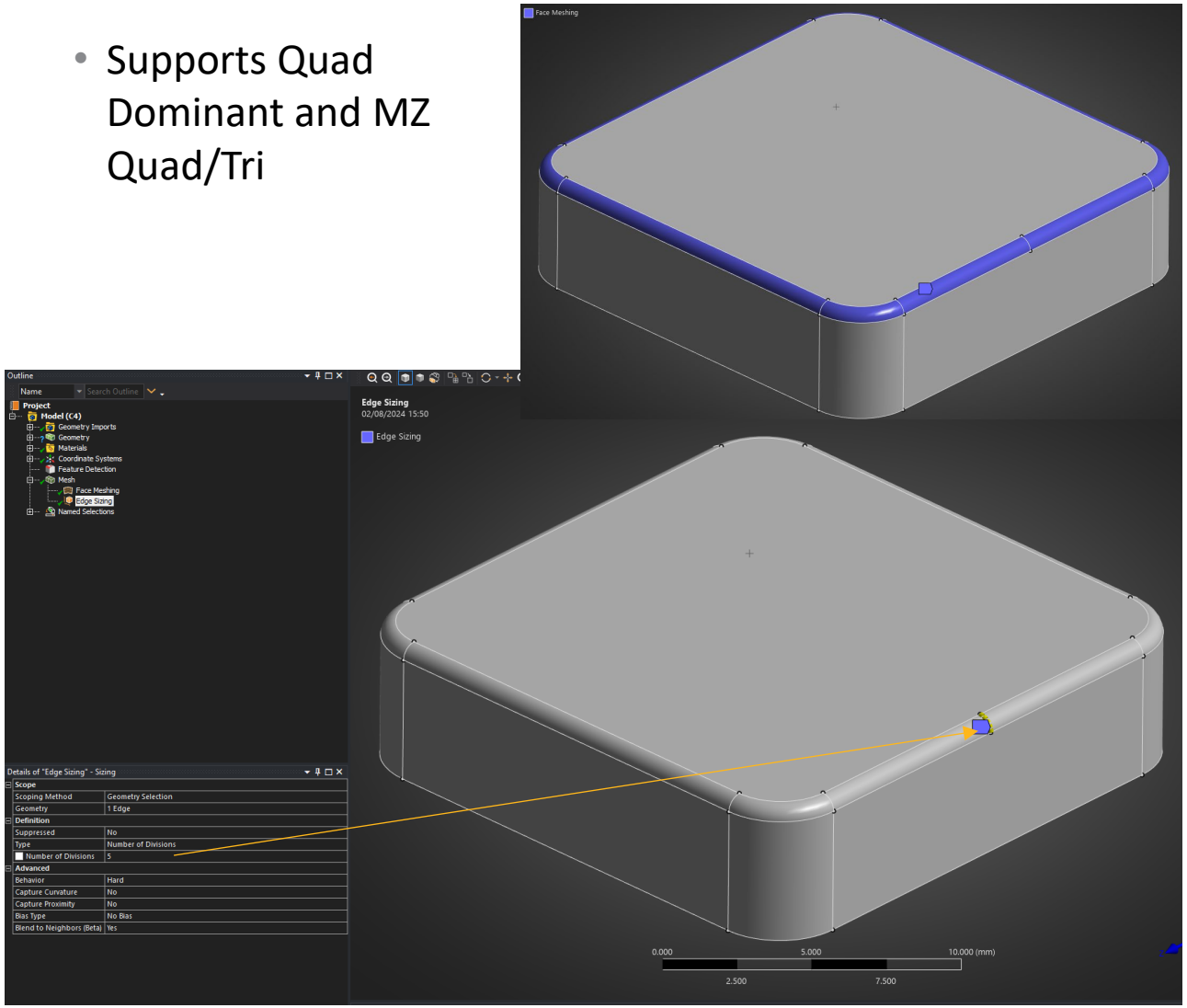


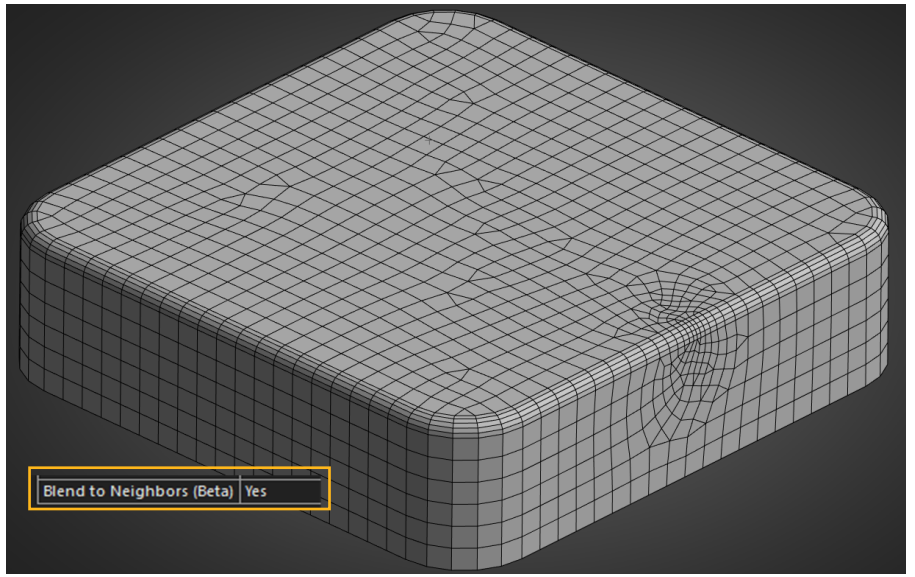


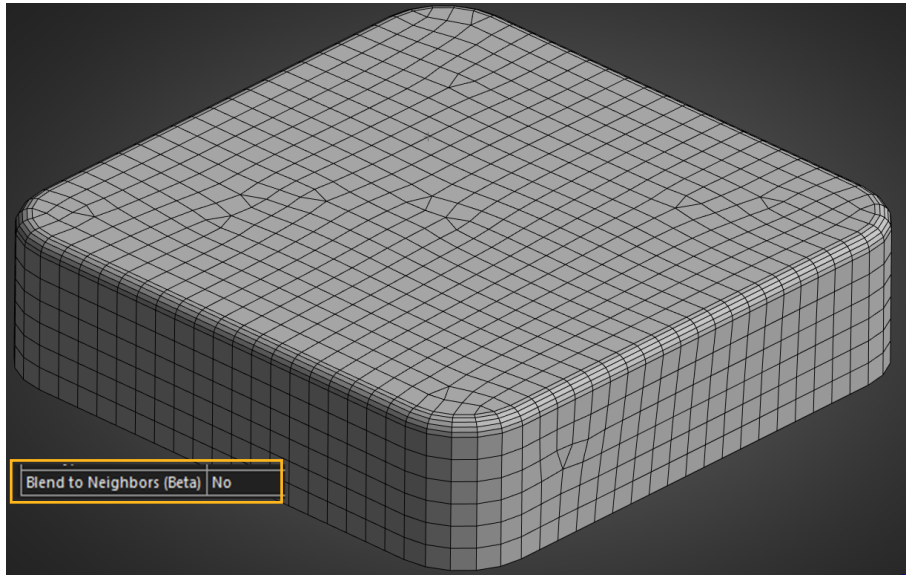


Edge Sizing: Blend to Neighbors for Shell/Mapping

- Supports Quad Dominant and MZ Quad/Tri



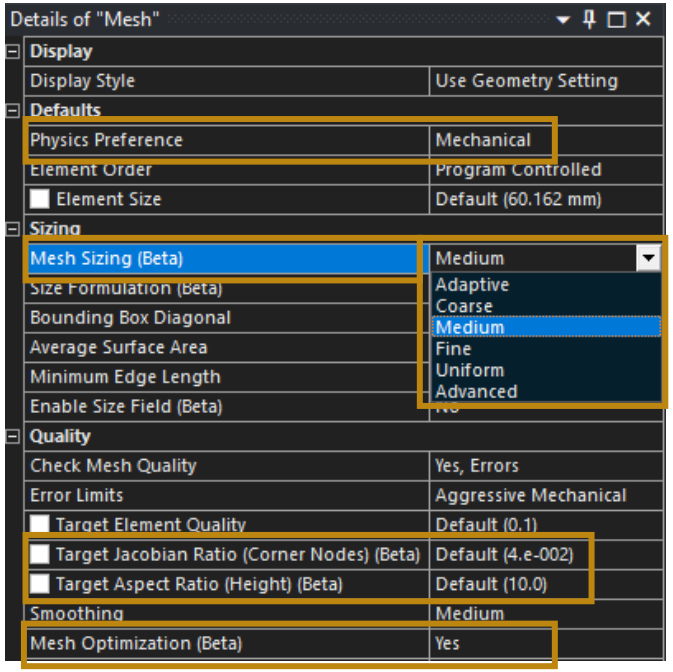




New Simplified Settings for Mesh Sizing (β)

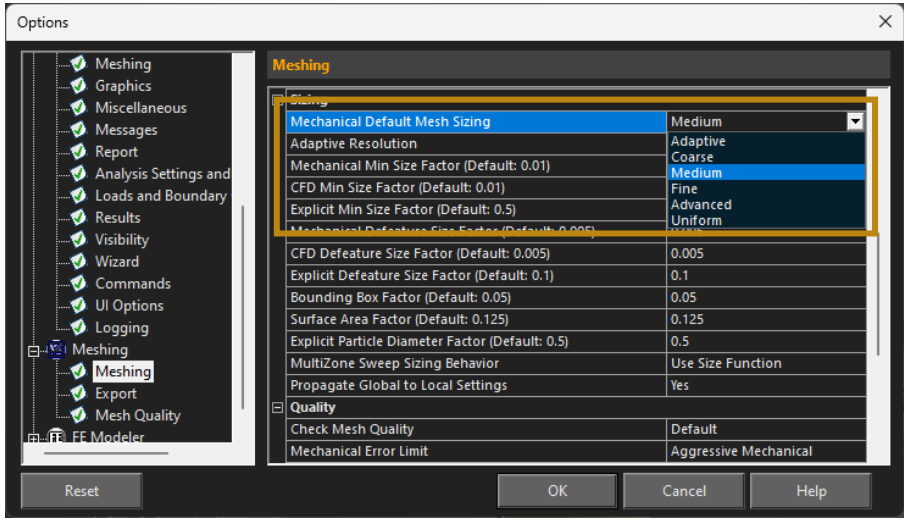
- If Beta is enabled, 2 new dropdowns option appears for Mechanical Physics Preference
  - Mesh Sizing
    - Coarse
    - Medium
    - Fine
    - Uniform
    - Advanced
    - Adaptive

Simple controls with low # of inputs for novice users, aiming for high quality



  - Coarse/Medium/Fine/Uniform hide the detailed settings and modify Curvature and Defeaturing parameters to alter the level of refinement from the Element Size
  - Advanced gives access to all the controls (curvature angle, growth rates etc, equivalent to previous “Adaptive=No” behaviour)
  - Adaptive sets Adaptive Sizing
- Mesh optimization (Beta)
  - This is enabled by default if beta is on and is only visible for Mech Physics preference
  - It enables many of the strong features developed for Explicit meshing including Aspect Ratio targeting including mapped meshing of cylinder faces where possible
  - Target JR And AR become available

Users can set their preference for Default Mesh Sizing in Options



Coarse/Med/Fine/Uniform Settings (β)

- Element Size remains calculated based on Bounding box factor by default as usual
  - Default Element Size is 0.05*Assembly Bounding Box Diagonal
- The new sizing formulation changes only the Defeaturing Size, curvature refinement parameters, Growth Rate and Max (volume) size
- As we crank up the refinement we will see
  - Holes/fillets etc being resolved better
  - More gradual growth in the mesh
  - Defeaturing being less aggressive to keep more features
    - For bad geometry this may cause failures and user may need to add local sizing controls with local defeaturing sizes to successfully mesh

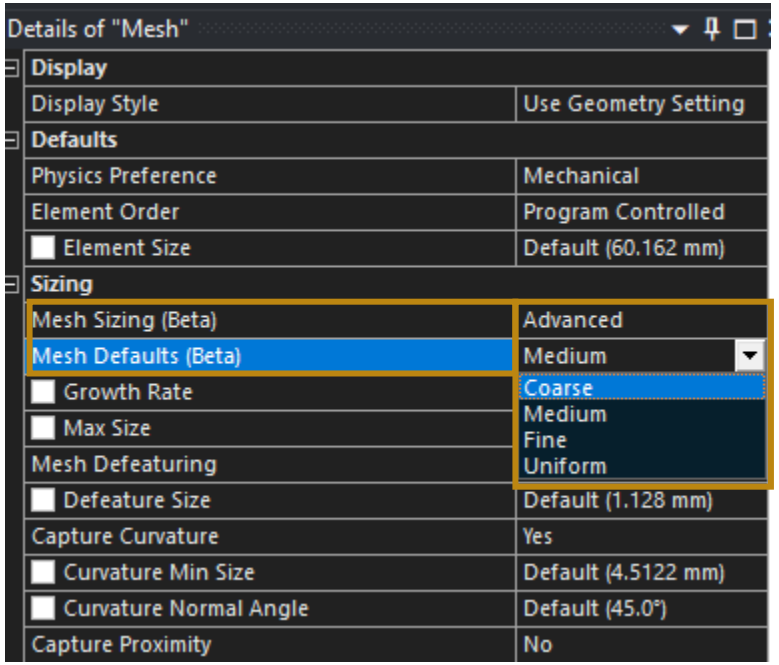
	Coarse	Medium	Fine	Uniform
Growth Rate	1.8	1.65	1.5	1.2
Defeature Size Factor	0.025	0.01875	0.0125	0.1
Capture Curvature	Yes	Yes	Yes	No
Curvature Min Size Factor	0.1	0.075	0.05	N/A
Curvature Normal Angle	60	45	30	N/A
Max Size Factor	2	2	2	1
Mesh Optimization	Yes	Yes	Yes	Yes

Factor → Multiplied by Element Size

Target will be to replace “Adaptive” as default with “Medium” in 26R1

Using Mesh Sizing = Advanced (β)

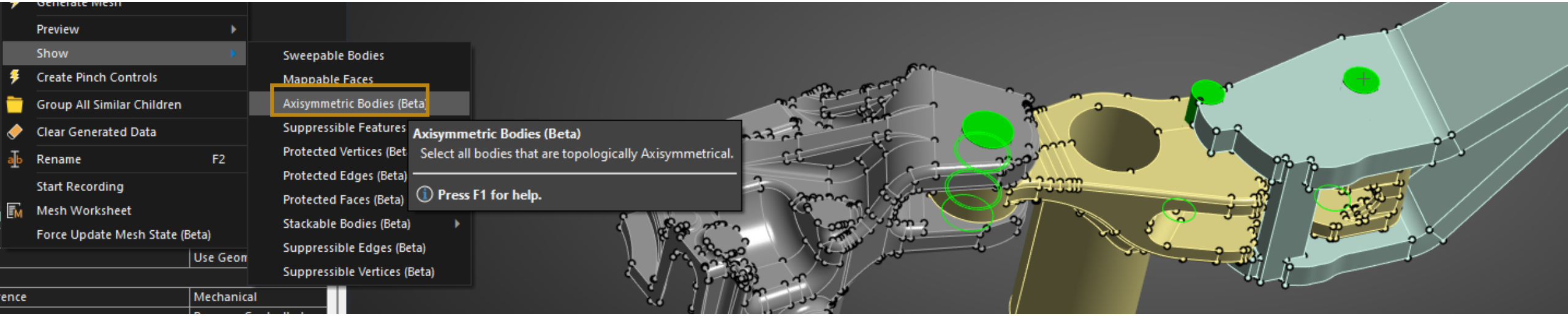
- If a user chooses coarse/med/fine/uniform but then wishes to tweak the mesh they can go to Advanced
- Advanced will populate the global controls based on whatever they were just using
  - If user was using Medium, the Advanced controls will take the medium params, etc
  - User can then tweak a control e.g. increase defeature size or lower the curvature angle, to get their desired mesh
  - If a user wants to set the default values to coarse/medium/fine/uniform while in Mesh Sizing =Advanced mode they can choose it in the Mesh Defaults dropdown



Hex Meshing

Show Axisymmetric Bodies

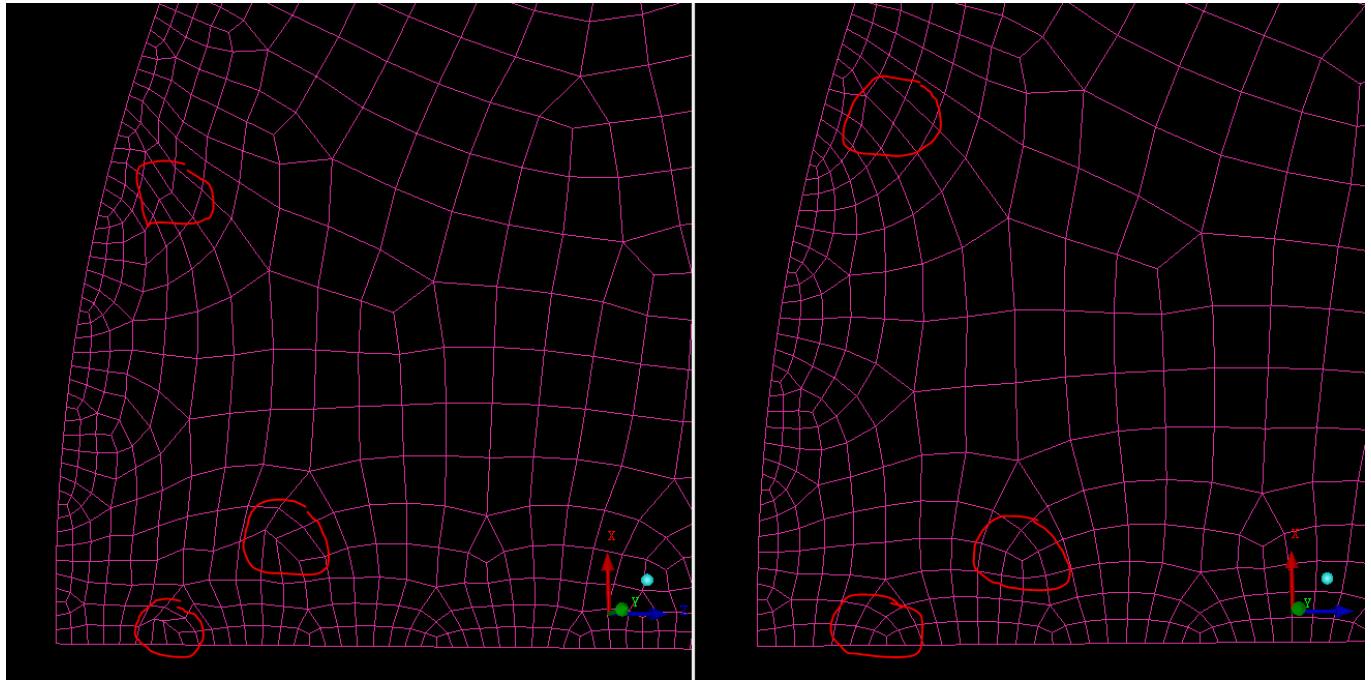
- Find Axisymmetric bodies and place them into MultiZone or Sweep/Axi-symmetric Methods to get automated hex meshing
- Aim to detect **and** automate hex meshing of such bodies in 26R1 without any user input



Smoothing Algorithm for MultiZone Paver Improved

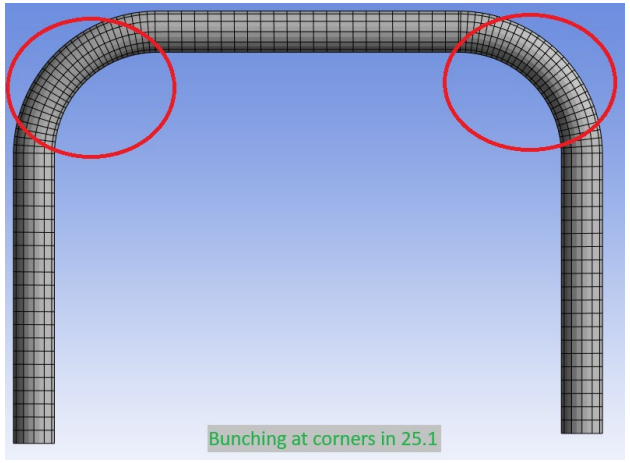
- Pave meshes were sometimes not optimal with some small kinks, elements in the surface mesh leading to sub-optimal quality in the swept volume mesh

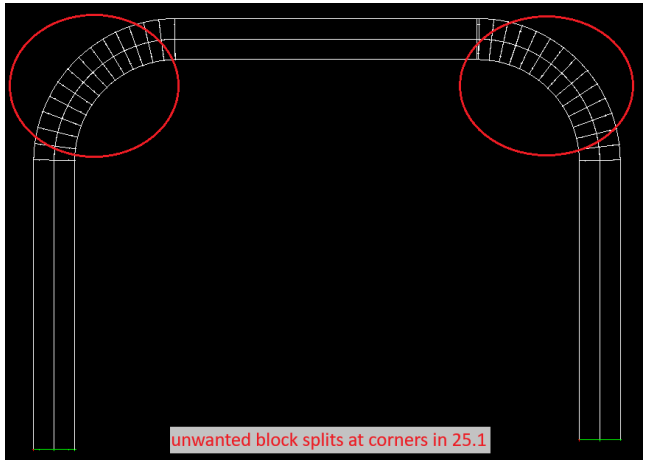
- Default MultiZone results for meshes with paved mesh are now significantly improved

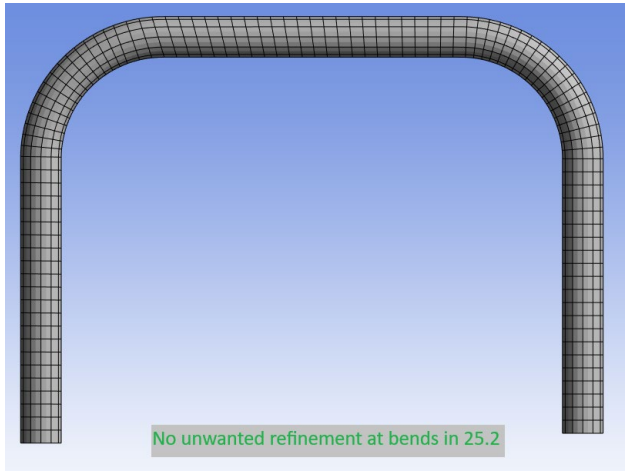


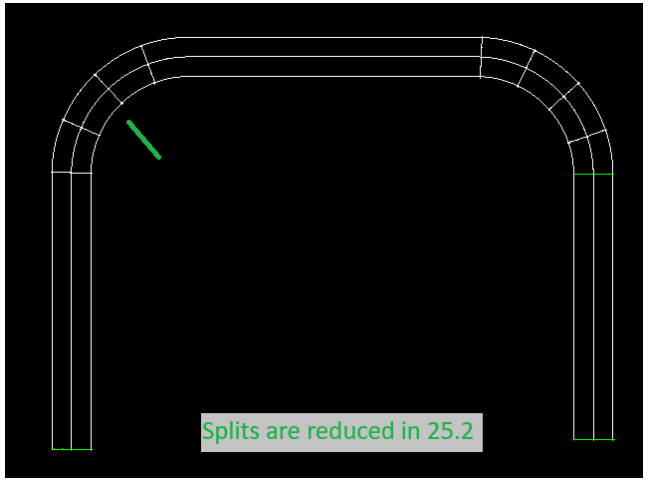
Multizone Axis-Sweep Enhancements

- Medial Axis Decomposition is renamed to “Axis-Sweep” and can be used to sweep along a complex path or around a central axis (for axis-symmetric bodies)
- Medial Axis has been improved to avoid splits along the sweeping direction at turns when sweeping along complex paths







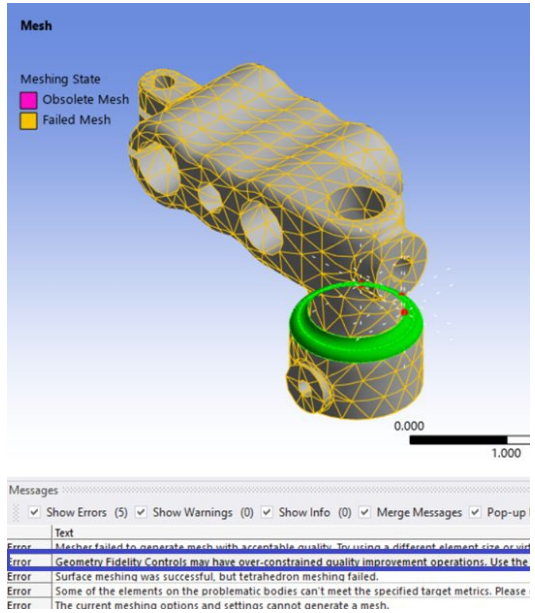


Tet Meshing

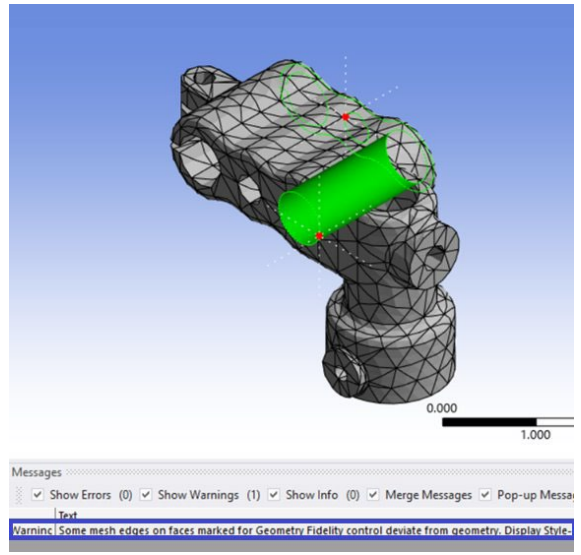
Improvements to Geometry Fidelity

- Geometry Fidelity controls allow users to scope faces where nodes must lie on geometry e.g. contact faces
- Previously, this added constraint could cause meshing failure due to quality checks
- In 2025R2, enhancements have been made to adhere to Geometry Fidelity controls and ensure good quality mesh is retained
- Some very small tolerance differences may be seen between mesh and geometry that can be visualized with the contour Display Style → Deviation from Geometry

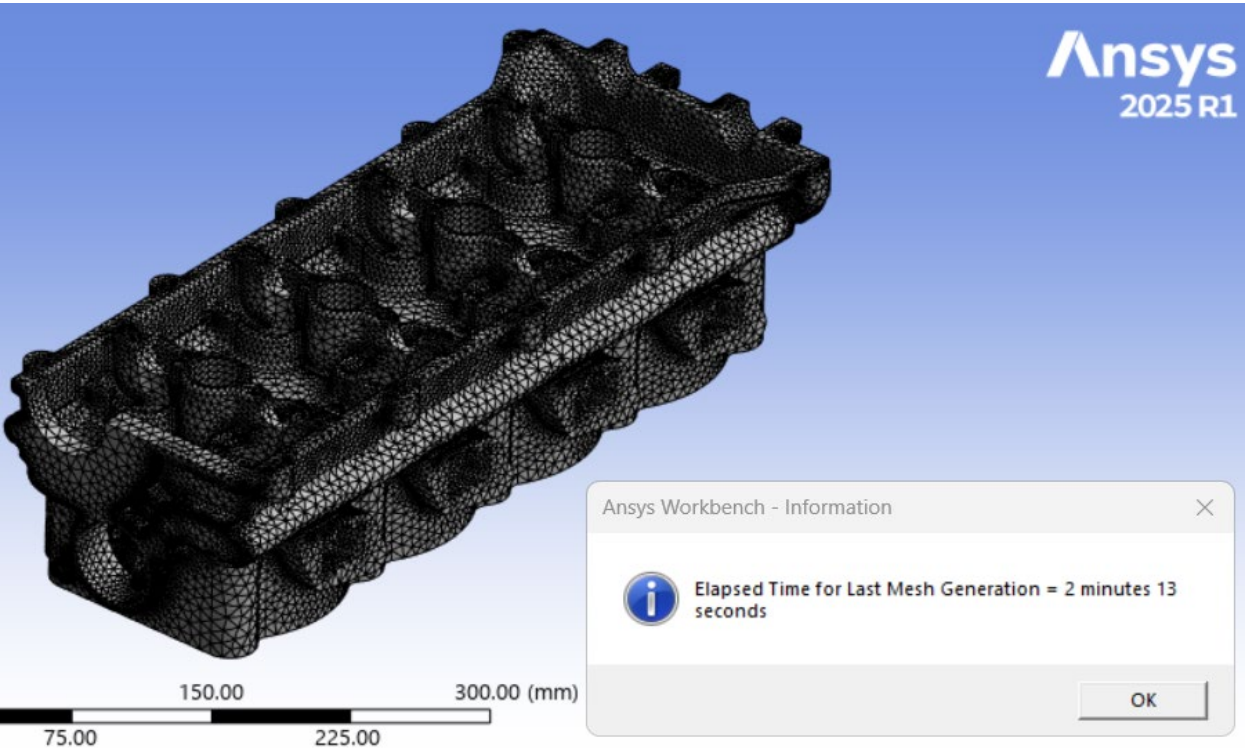
2025R1 – Fails due to bad quality

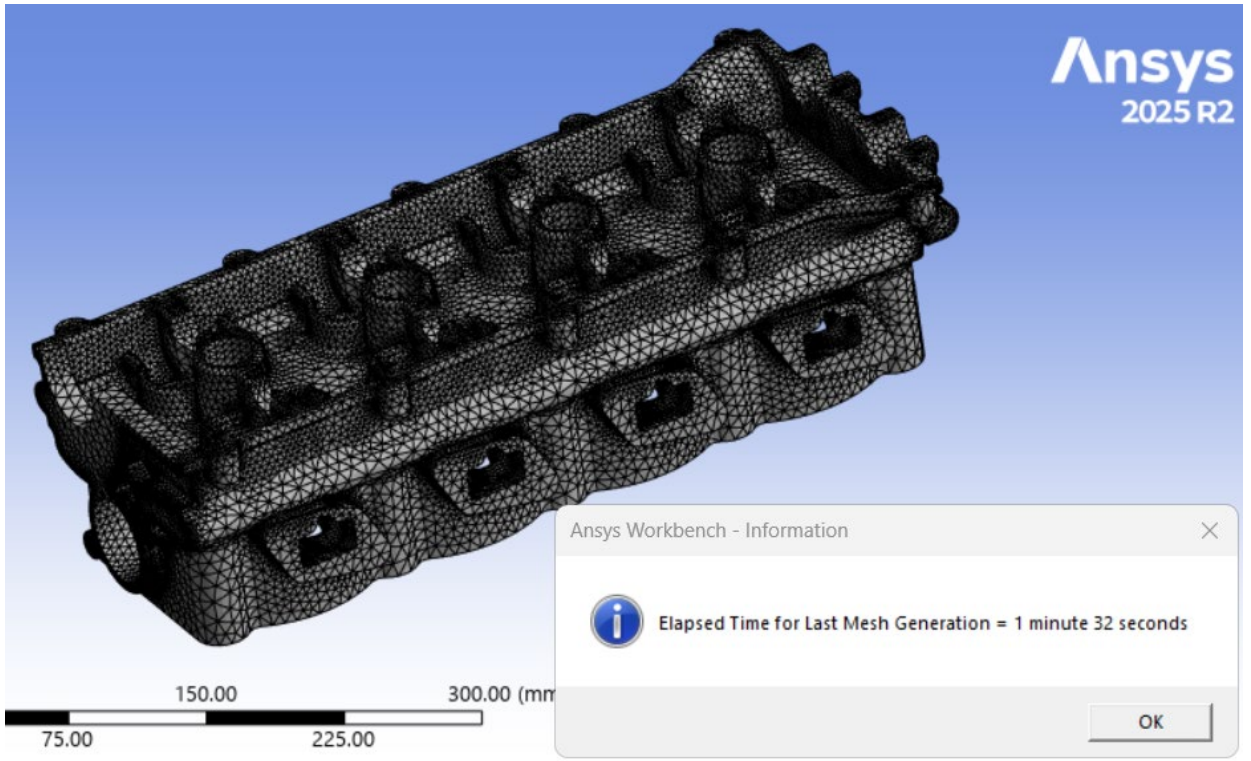


2025R2



Tet Mesh Performance – Generic Engine Block





Mesh Time = 2 min 13 secs
Memory = 1547 MB
Element Count = 667546

Mesh Time = 1 min 32 secs
Memory = 1464 MB
Element Count = 665545

30.8%
Speed-up

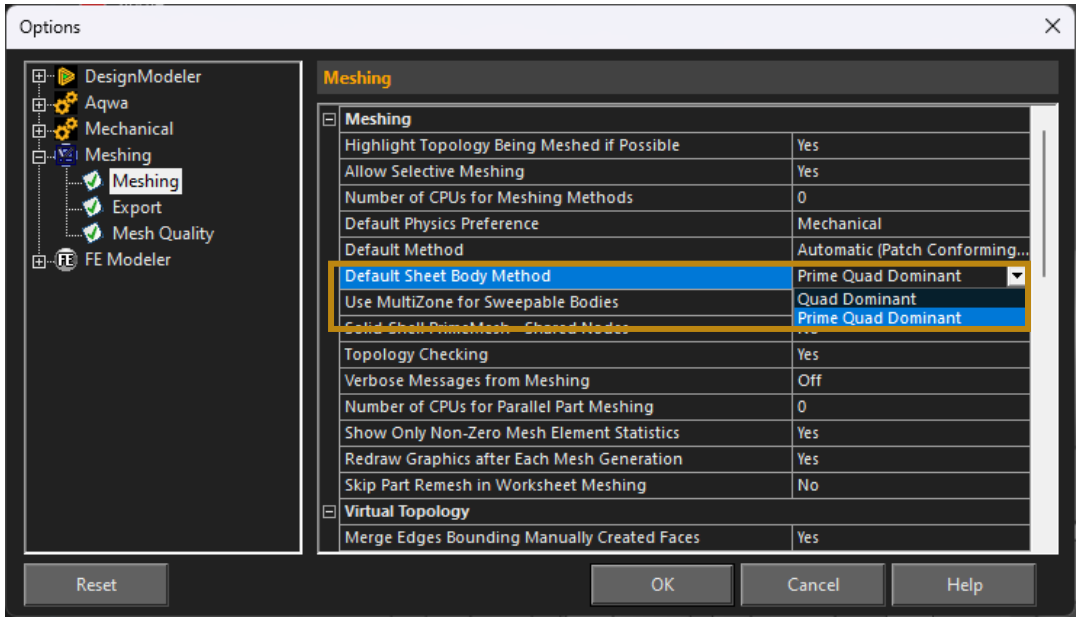
Tet Meshing Performance Data – 2025R1 Vs 2025R2

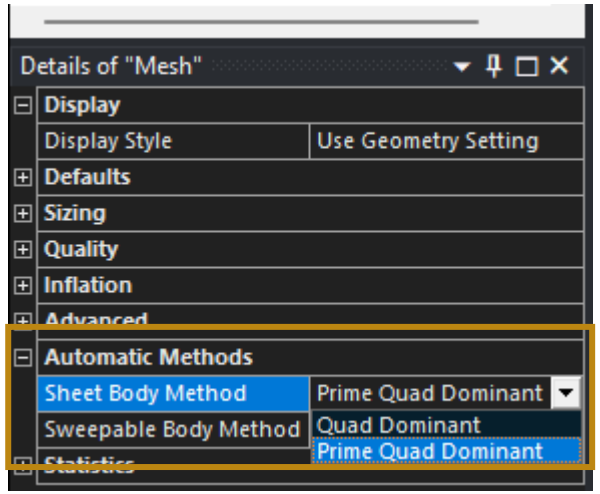
Model	Elements		Memory (MB)		Mesh Time (s)		% Reduction in Mesh Time
	25.1	25.2	25.1	25.2	25.1	25.2	
Customer Model A	400348	398100	2297	2365	308	150	51.3%
Customer Model B	515079	509582	2351	2057	407	234	42.5 %
Customer Model C	480847	475288	2979	2145	286	141	50.7%
Generic-Model	667546	665870	1547	1486	133	89	33.1%

Welds and Shell Meshing

Prime Method as Default

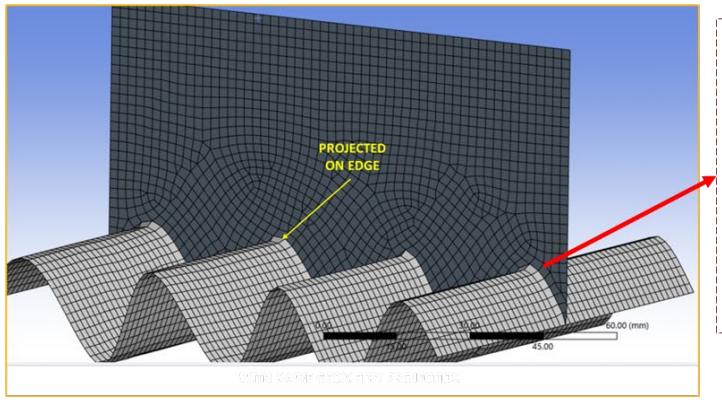
- Sheet bodies will be meshed with Prime Quad Dominant by default
  - Improved quality and performance
  - Allows capabilities to Weld and Connect bodies in Mechanical
- User can switch back to previous default
  - Option to make this default “Quad Dominant”

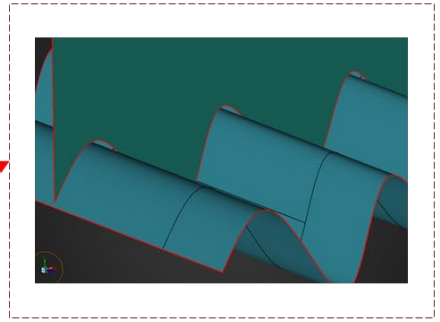




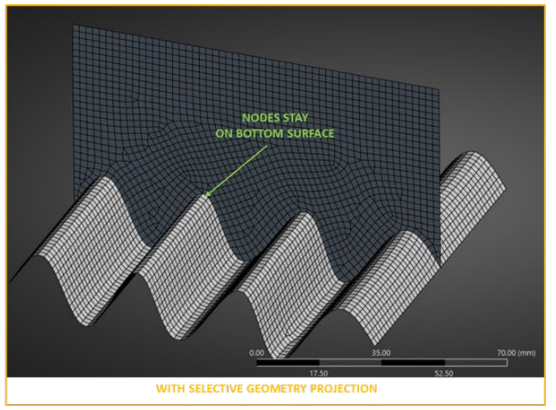
Accurate Geometry Recovery for Prime Quad Dominant

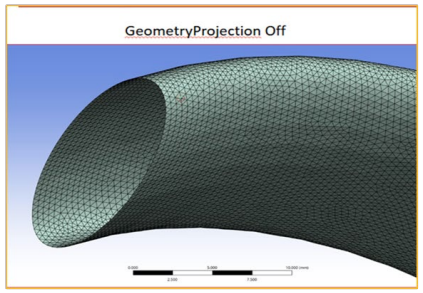
- Now Prime Quad Dominant places mesh nodes directly on the geometric curves and surfaces
- Avoiding reliance on good graphical faceting for all cases
- Smart projection is utilised in areas where geometry projection is not warranted e.g. where welds exist or there are large/tolerant gaps between true geometric curves and surfaces

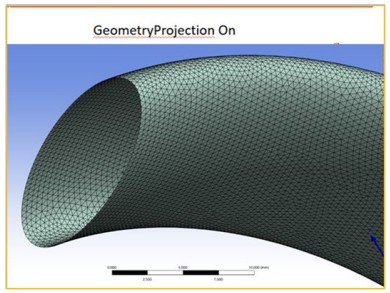


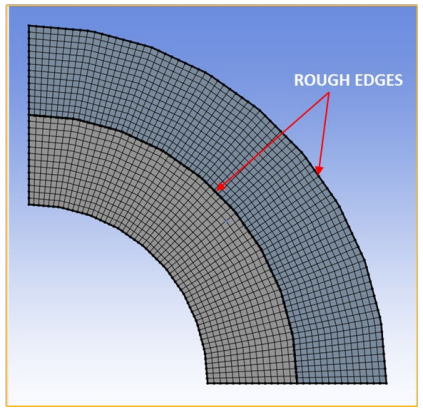


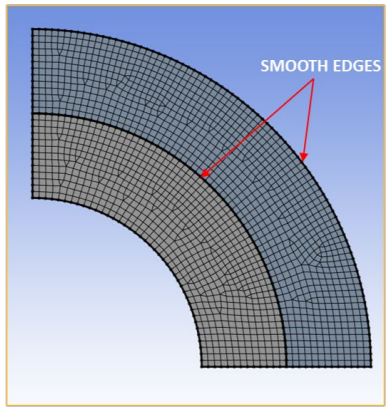
Gap Between Bodies

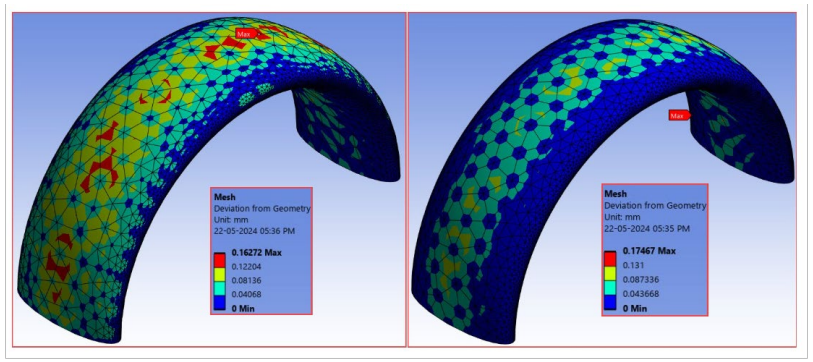






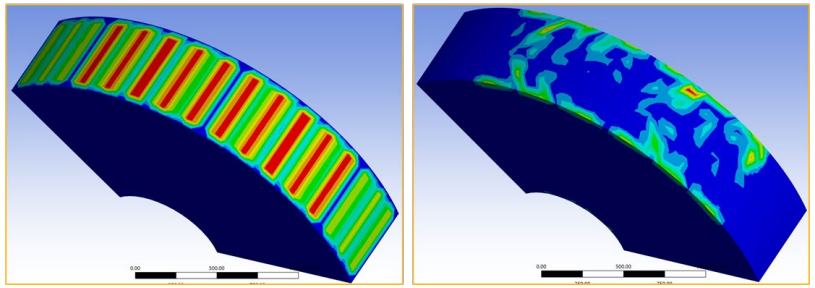






Geometry Projection OFF

Geometry Projection ON



Mesh Deviation Contours Vs CAD

General Usability Improvements

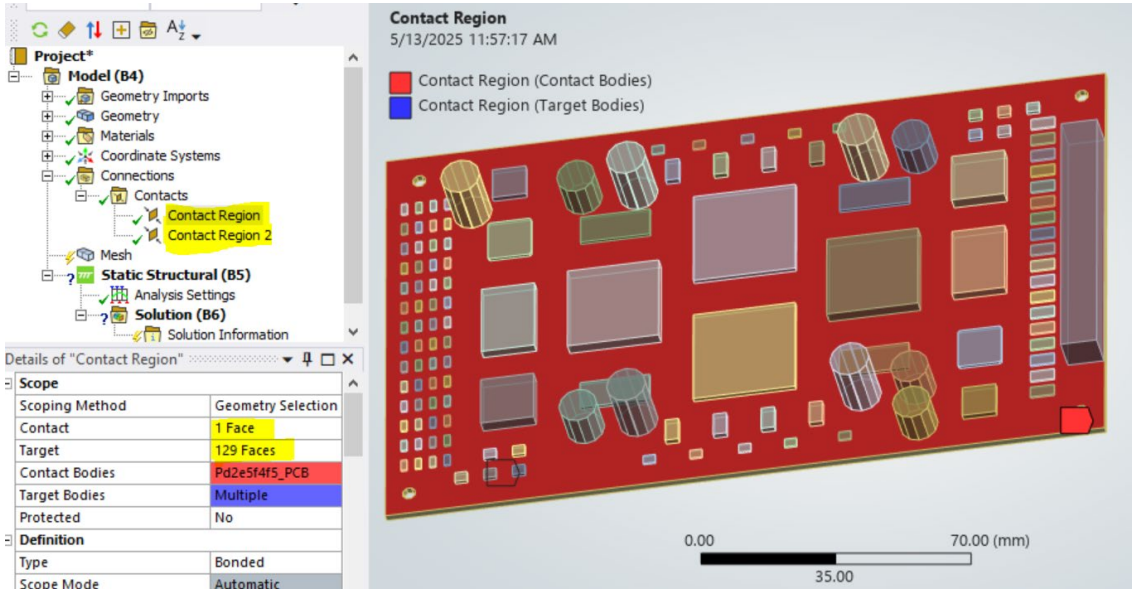
Proximity Detection Group By Faces for Contact Improvements

Objective

Grouping contacts by faces, helps in **Contact Reduction & Constraint Cleanup**

Two Key Goals

- **Reduce Number of Contacts**
  1. Significant improvement observed in **Electronics models**
  2. Example: PCB boards with many small components on a large face
  3. Leads to **fewer contacts**, improving performance and analysis time
- **Remove Overlapping Constraints**
  1. Aim to merge contact face definition sharing edges in **multibody parts** when writing MAPDL input file**.**
  2. Relevant for **engine and gas turbine models**
  3. Helps eliminate **redundant constraints in solver**



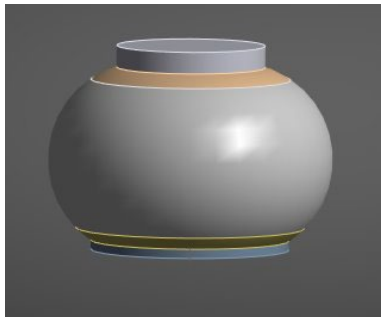
There are a total of 221 contacts detected in this model. Group By Faces merges them into 2 contact regions based on common contact faces.

Meshing Performance

Parallel Body Group Meshing within Parts (β)

- Parallel Part meshing has been in Mechanical for many years and allows fast meshing of disconnected (non-shared) parts across threads/cores
- Within a multi-body part with share topology, parallel meshing is not implemented for Mesh Methods
- New in 25R2, when BETA is enabled, bodies sent to the mesher together will be
  - Sorted into clusters based on connectivity
  - Disconnected clusters will be sent to separate threads for meshing
  - This will allow incremental meshing using body selection and Mesh Worksheet ordering to make use of multiple cores to complete meshing faster
  - A hidden option is available to disable this if somehow it interferes with other beta tests

- 989 Solder Balls
- Each has 5 connected bodies
- Meshed with MultiZone



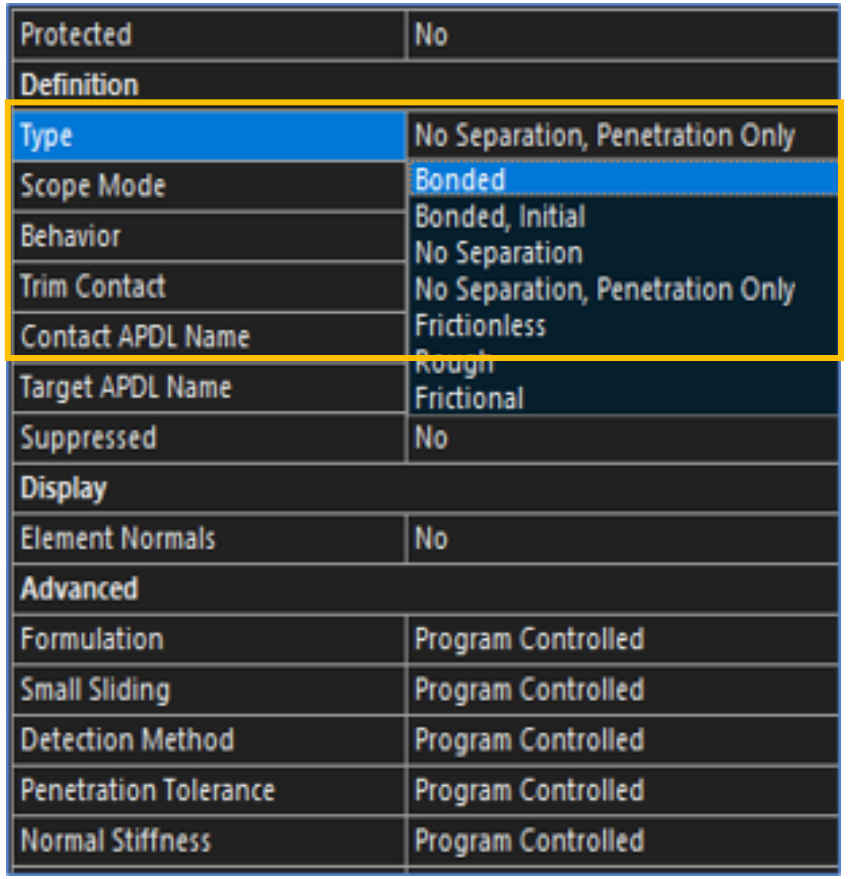
	Time taken	Speed-up
Serial Code	15min 46s	1
Parallel Body Meshing	3min 33s	4.44

Powering Innovation That Drives Human Advancement

Contacts & Connections

Contact Type Options

- The Contact Region object has two new options for the Type property:
  - **Bonded, Initial:**
    - Models bonded contact, but the contact detection for the initial state is maintained through the analysis.
    - Closed initial state will remain attached, and open initial state will remain open, even if deformation brings the contact pair close together.
  - **No Separation, Penetration Only:**
    - The contact is established only when these surface(s) are either touching or penetrated.
    - No contact with gap

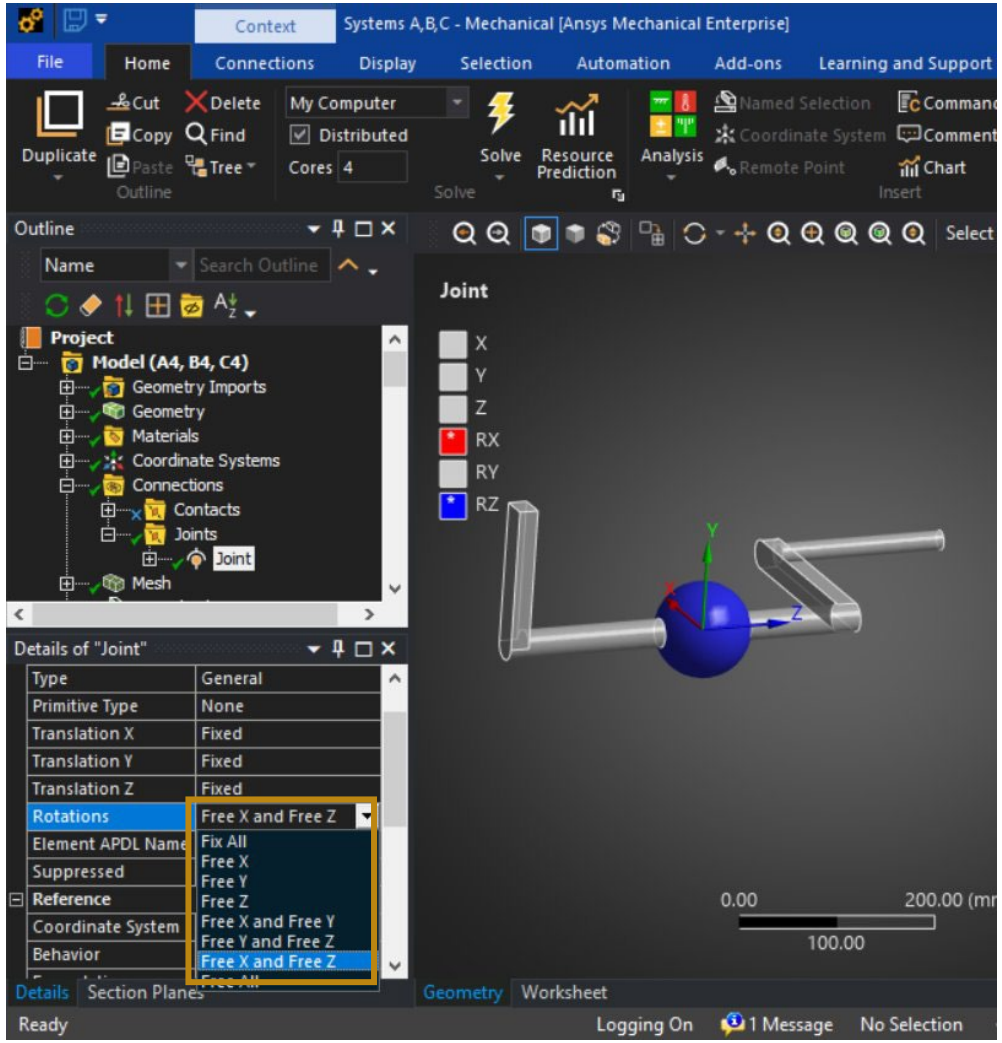


Performance Improvement on MPC Contact

- Performance of MPC bonded contact overall in static/transient assembly and mode expansion pass are improved.
- Contact pair-splitting operation (CNCHECK,DMP and CNCHECK,SPLIT) are producing more evenly distributed split MPC pairs across domains

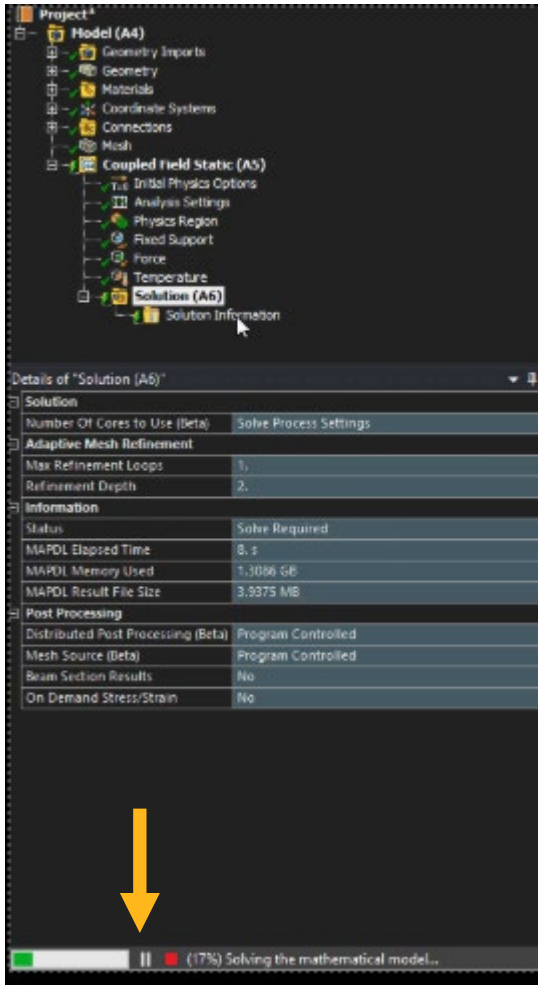
General Joint: Additional Rotational Constraint Options

- The **Rotations** property for the **General** joint type has three new options: **Free X and Free Y**, **Free Y and Free Z**, and **Free X and Free Z**.
- These options help in further constraining the rotational degrees of freedom of the model in all six degrees of freedom.



Coupled Field Interrupt Solution

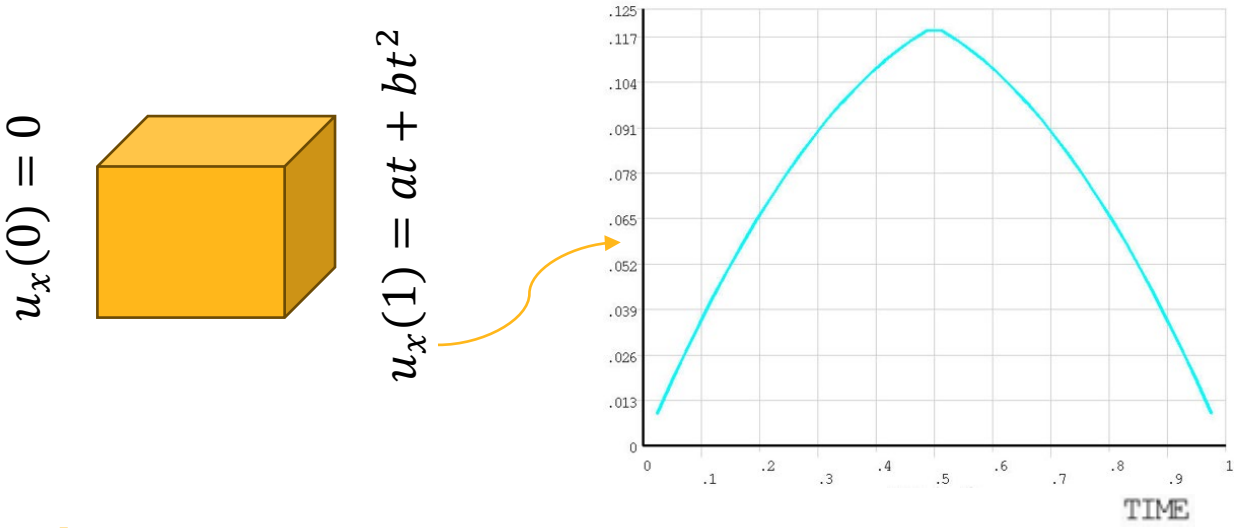
- Interrupt button is now supported in Coupled field Static and Transient analyses
- For couplings with restart controls, excluding Pure Acoustics and Pure Electrostatic physics.
- Solutions will resume from the interruption point, and loads can restart if modified afterward.

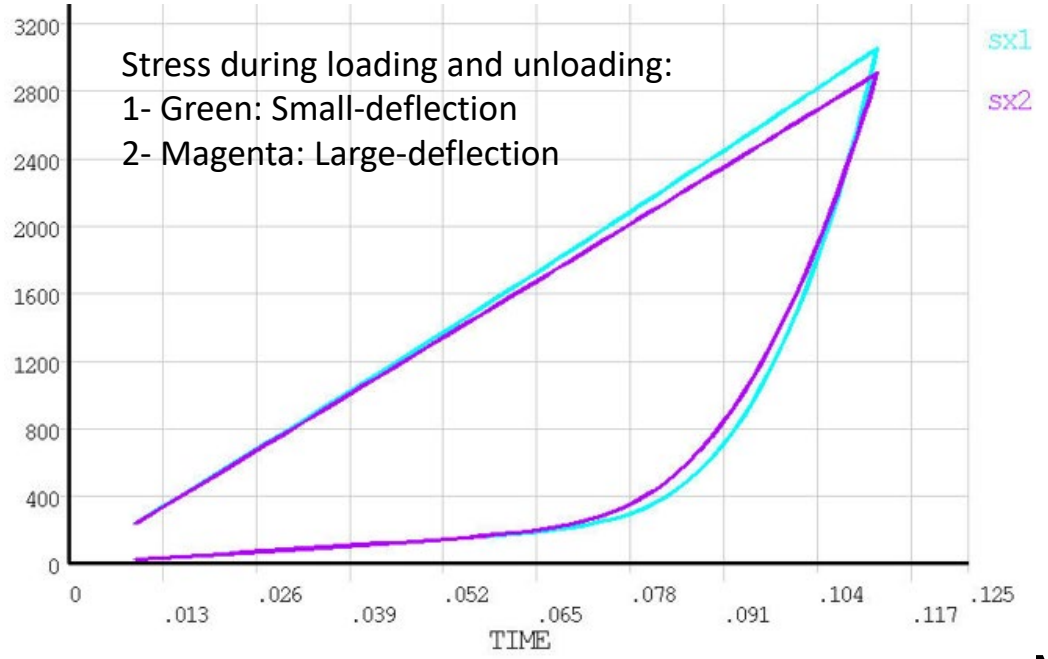


Small-deflection Hyperelasticty

- Hyperelasticity in small-deflection analysis (NLGEOM, off) is now supported
- Hyperelastic materials in small-deflection static, small-deflection transient, modal and harmonic analyses are now allowed
- Different behavior in tension and compression, or stress softening due to Mullin's effect are supported

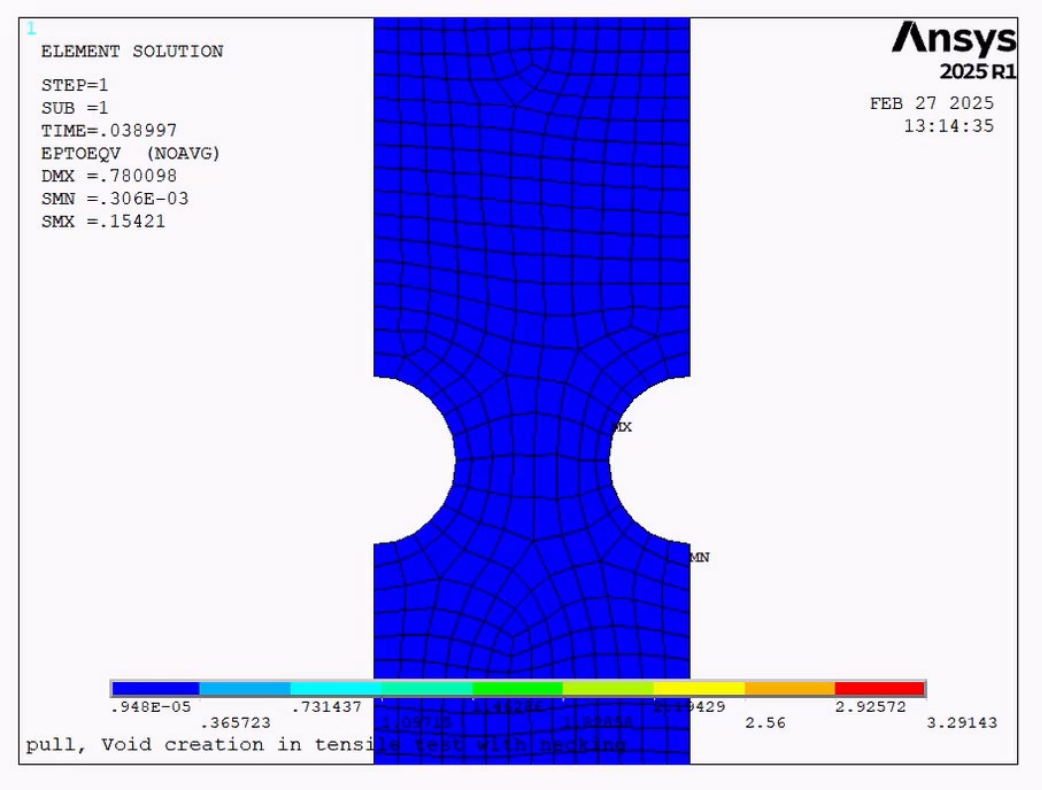
Example:
Neo-Hookean material with Mullins effect from
a modified Ogden-Roxburgh damage function
(TB,HYPER,,,,NEO + TB,CDM,,,,PSE2)





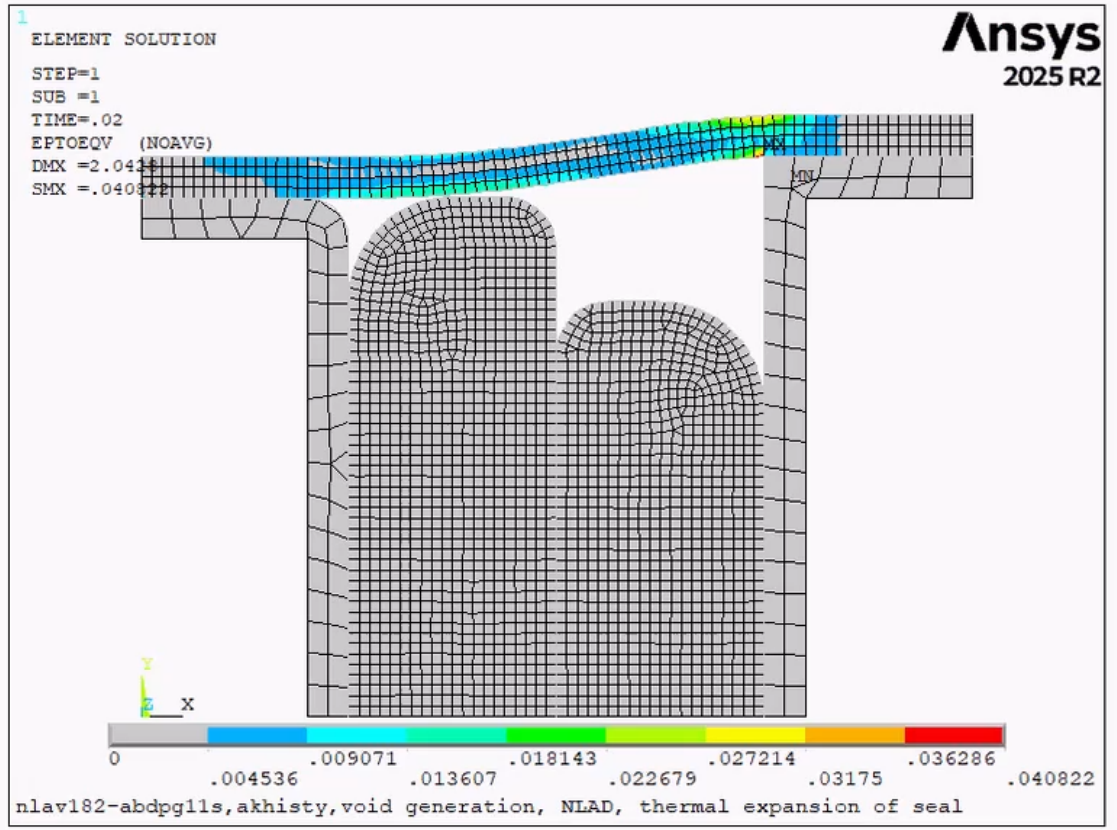
Nonlinear Adaptivity: Material Removal

- **Use case:**
  - Ductile damage and failure is usually caused by macroscale void initiation followed by void coalescence leading to crack formations and eventual failure
- **Applications**:
  - Void nucleation in computer chip silicon substrate interface
  - Void nucleation and coalescence leading to cracks in Lithium-ion solid state batteries.
- **Objective:**
  - Model ductile damage/failure using **NLAD** in an implicit Solver.
  - Material removal by introducing voids.



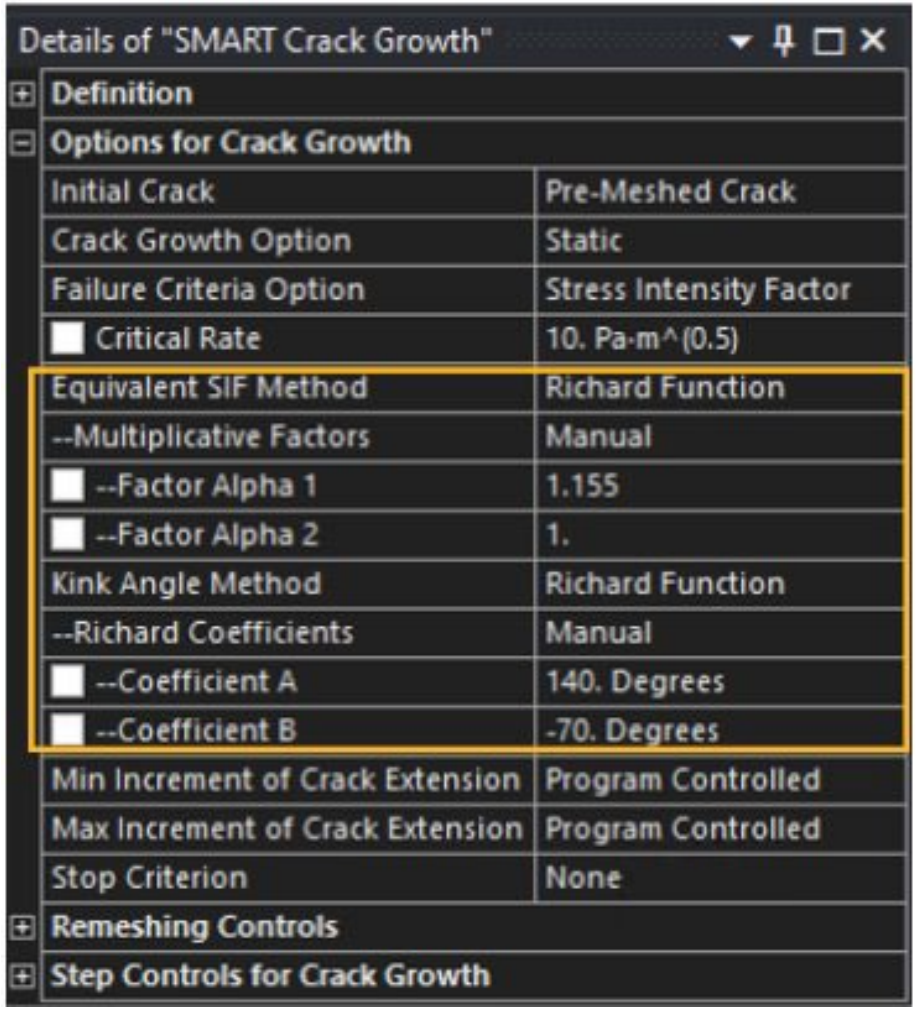
Nonlinear Adaptivity: Material Removal

- Remove material in a deformed body by inserting an elliptical void when certain user-defined criteria are satisfied
- Principal strain criterion and equivalent strain criterion are supported
- Applicable to 2D analysis only.
- Static and Quasi-static procedures are supported.
- All material types supported by NLAD framework
- Single void per substep only
- Generating new contact elements in the void to prevent physical interpenetration and continuation of existing contact where necessary
- Residual rebalancing to ensure equilibriation and continuation of the solution.



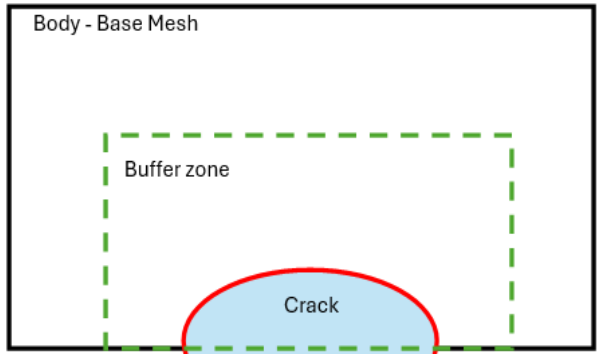
Equivalent SIF Methods for SMART Crack Growth

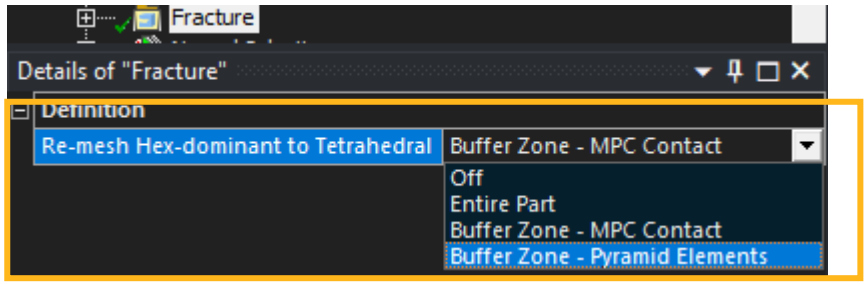
- New Equivalent SIF Method property
  - Max Tangential Stress Criterion
  - Richard Function
  - Pook criterion
  - Empirical Function
  - Program Controlled

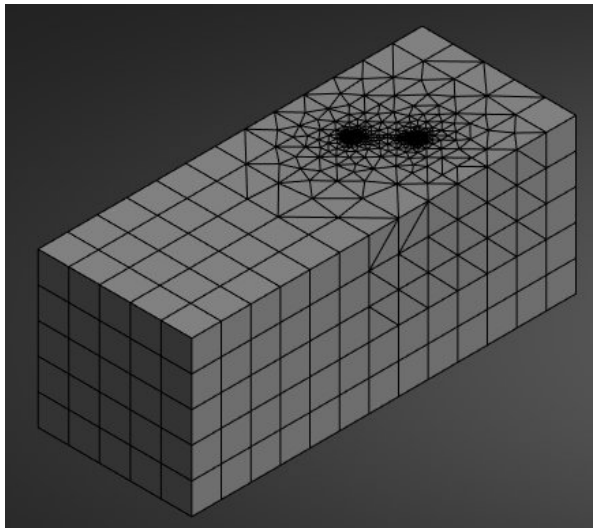


Hex-Dominant to Tetrahedrons conversion within Crack Buffer Zone

- Fracture options for connecting the converted buffer zone (tet) to the base mesh (hex)
- **Buffer Zone - MPC Contact**: Use this option to create a bonded MPC contact pair at the mesh interface between the hex-dominant elements and the tetrahedrons.
- **Buffer Zone - Pyramid Elements**: Use this option to create pyramid elements at the mesh interface between the hex-dominant elements and the tetrahedrons.

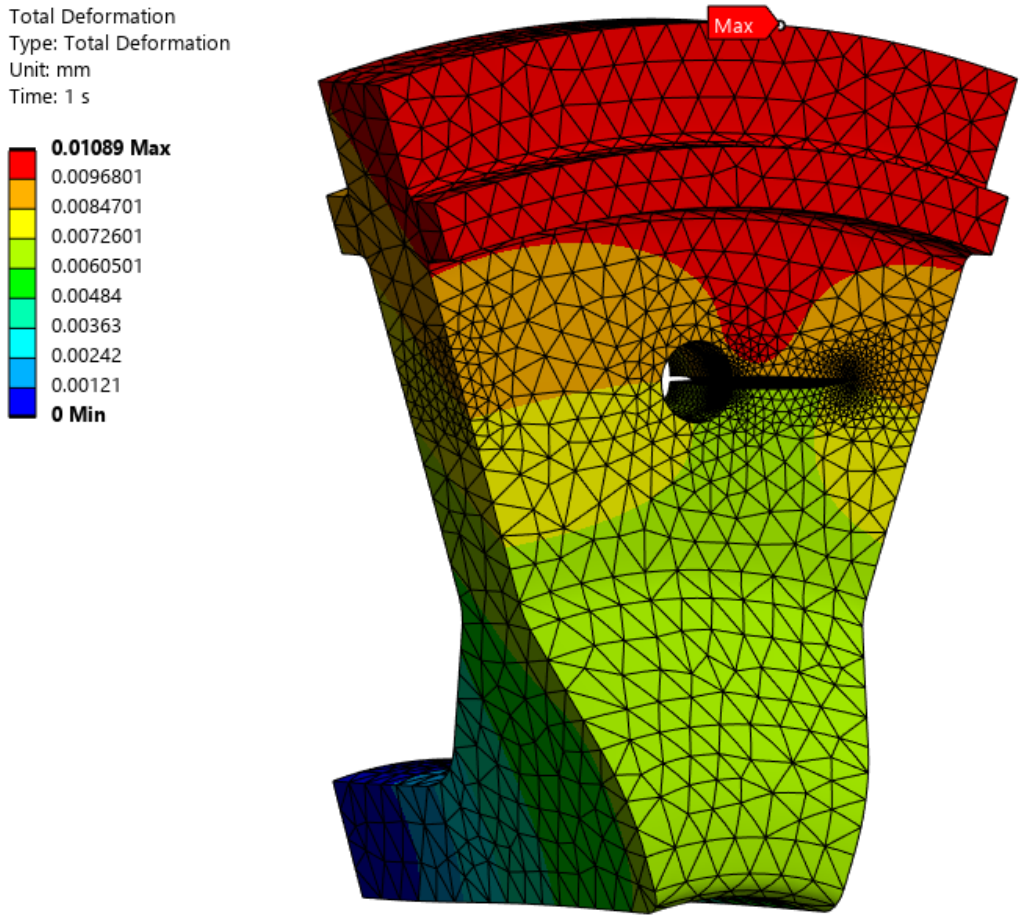






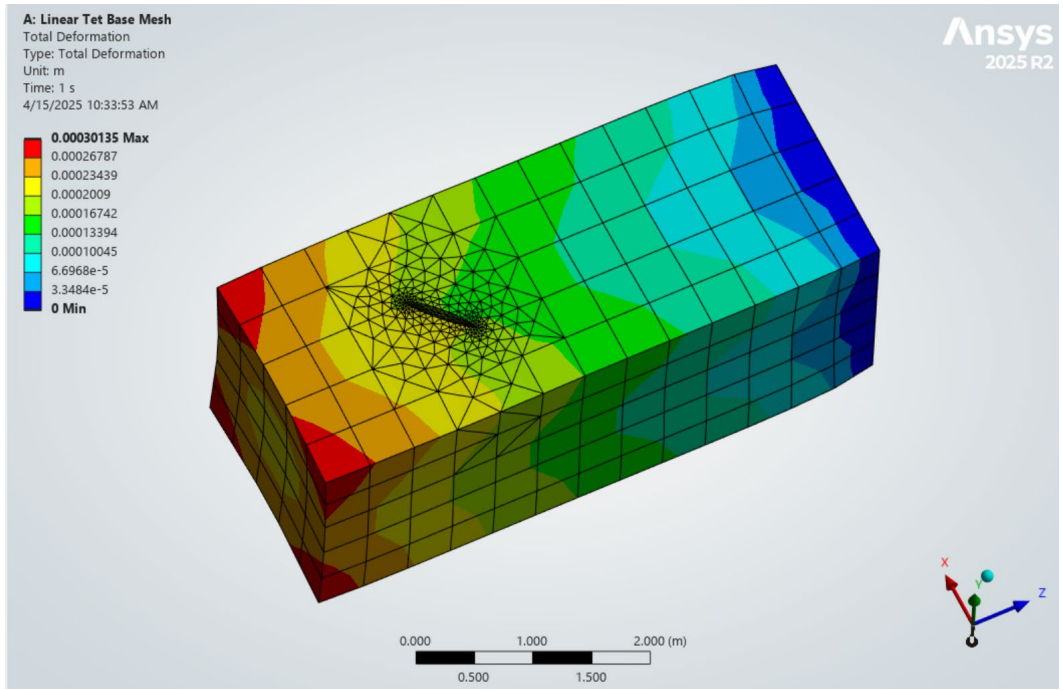
SMART crack-growth with Cyclic Symmetry

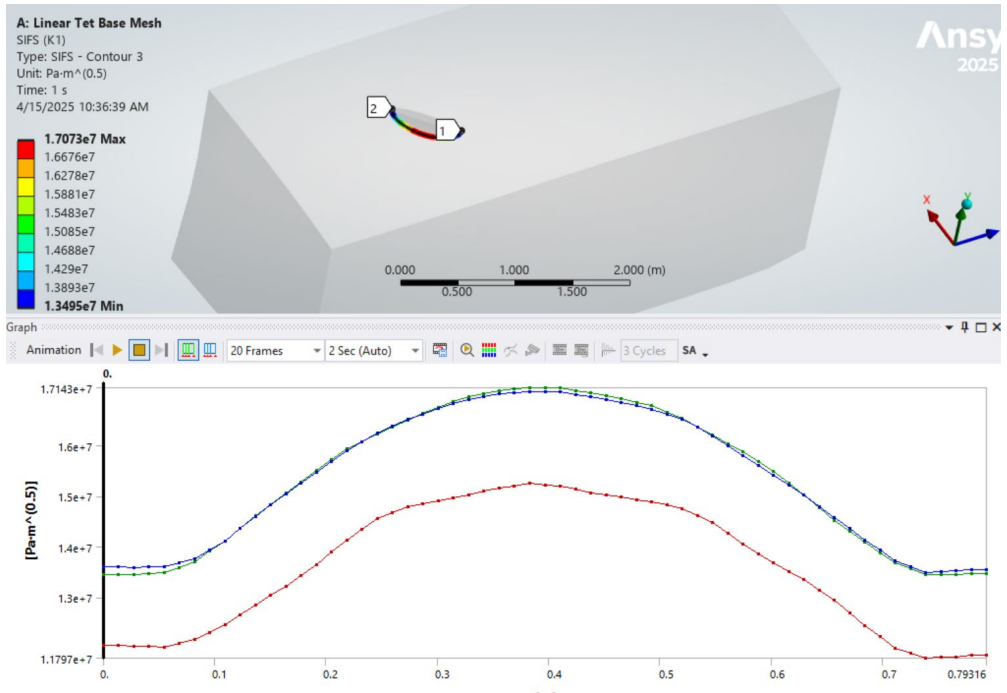
- Static or fatigue crack-growth
- Matching mesh is maintained on the cyclic surface pair during crack-growth remeshing
- Cyclic constraint equations are automatically updated during remeshing
- Limitations:
  - Crack cannot cut into cyclic boundary
  - Multi-stage Cyc. Sym. (CECYCMS) only
  - Single stage only



Fracture Parameter Calculation in Mixed Mesh

- Support fracture parameter calculation in mesh containing mixed element types and/or orders
- Possible to insert and grow cracks in legacy hex-dominated models
- Reduction of computations in growing cracks in meshes with mixed element types and/or orders





J-integral Supports More Material Models

- New supported material models:
  - Ramberg-Osgood material model
  - Bilinear kinematic hardening of elastic-plastic material
  - Nonlinear kinematic hardening of elastic-plastic material
- J-integral calculation is suitable only for monotonic loading of elastic-plastic material models (including isotropic and kinematic hardening elastic-plastic materials)

Parameter and Component names

- The upper limit of user defined parameter name is increased to 240 characters (was 32)
- Adding support for special characters for component names:

  Dots (.), minus (-), forward slash (/), colon (:), semicolon (;), Pound (#), Ampersand (&), Left parentheses ({) Right parentheses (}), Plus (+), Less than (<), equal to (=), greater than (>), Question mark (?), At symbol (@), Back slash (\), Bar (|), Tilda (~), Hat symbol (^), Open braces ([), Close braces (]), Accent (`), and single space

  - Single space only in the middle of component names, and not in the beginning or end

  - No consecutive spaces in the middle of component names

HPC

Distributed Memory Parallel Enhancements

- **MPI library support**
  - Upgraded to Intel MPI 2021 Update 14 version on Windows and Linux
    - Improves performance, scalability and robustness
    - Linux clusters using (older) Mellanox Infiniband 4.x → (older) Intel MPI 2018 is automatically chosen
  - Microsoft MPI v10.0 version is unchanged at this release on Windows
  - Open MPI v4.0.5 version is unchanged at this release on Linux
    - IMPORTANT: Open MPI is default when running on AMD processors

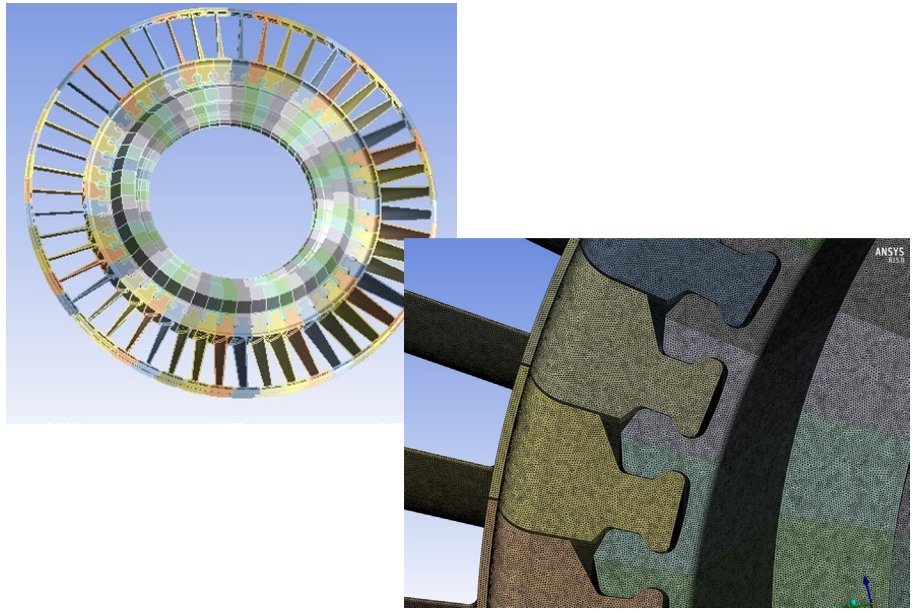
Math Library Enhancements

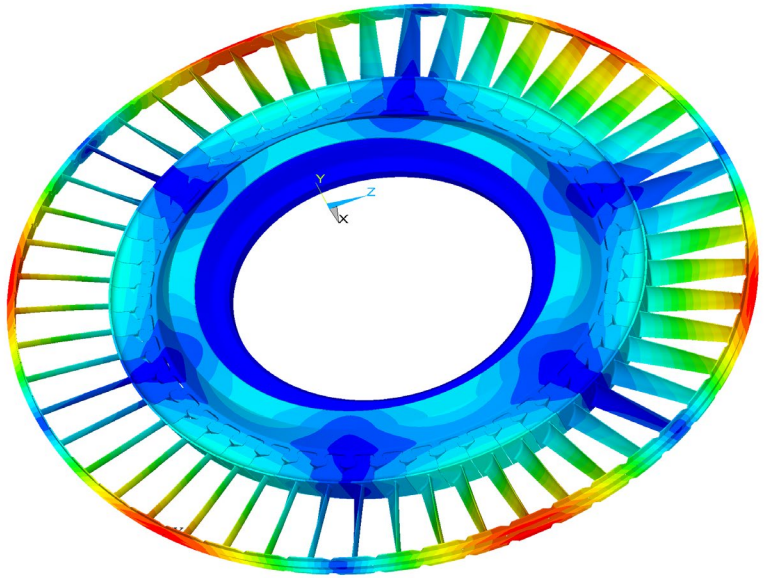
- **Intel MKL libraries upgraded from 2023.1 to 2024.2**
  – Optimized performance on the latest Intel architectures

- **AMD AOCL libraries upgraded from 4.2.1 to 5.0.1**
  – Optimized performance on the latest AMD architectures

Sparse Solver Enhancements

- **Sparse solver in MAPDL is extremely robust and fast**
  - 2024 R1 → 1 Billion DOF turbine blade model → successfully solved on MicroConsult HPC cluster
  - 2025 R2 → 1.5 Billion DOF turbine blade model → successfully solved on MicroConsult HPC cluster



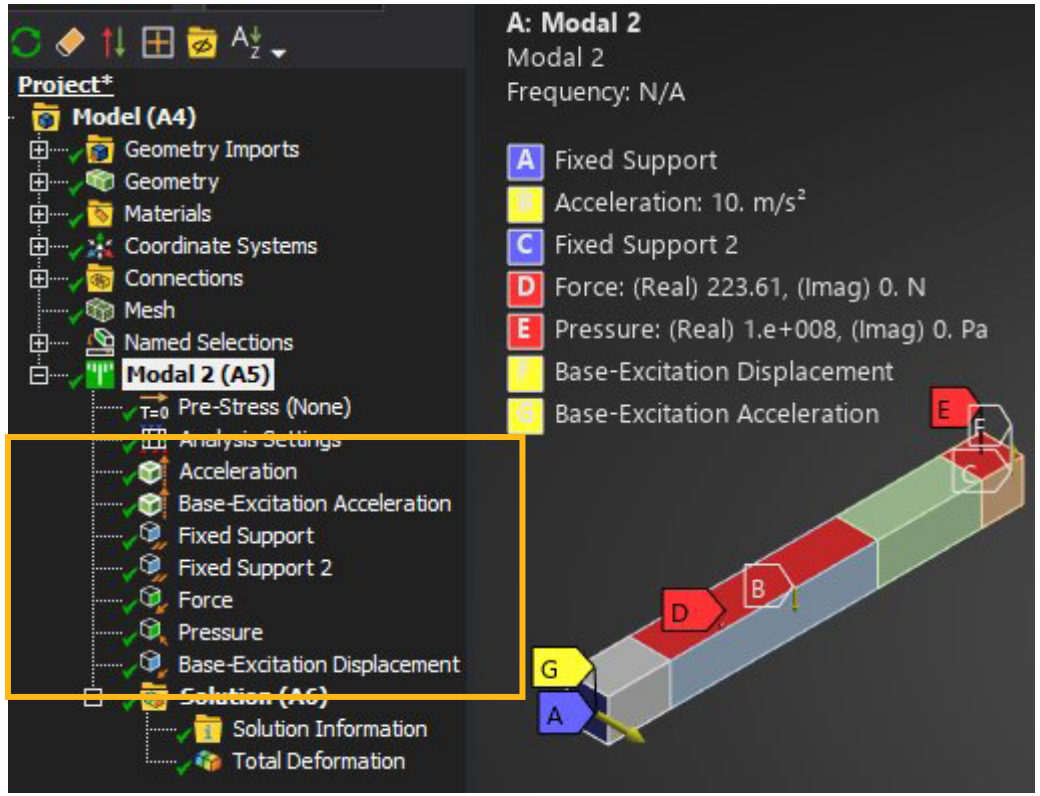


Mechanical Workflow

Load Vector, Residual Vector and Base Excitation in Modal Analysis

Load Vector and Base excitation in Modal analysis

- Mechanical supports Force, Pressure and Acceleration loads in Modal analysis which generates load vector and can then be applied and scaled in the downstream MSUP Harmonic analysis
- Mechanical supports Displacement and Acceleration as Base excitation in Modal analysis which can then be applied in the downstream MSUP Harmonic analysis



Load Vector controls and Options in Modal analysis

- Load vector number is automatically assigned to loads using Program Controlled default option in Modal analysis. It is displayed as read only field after solve. Users can use Manual option to group loads to one load vector
- Modal analysis now support Include Residual Vector property to calculate residual vectors which can then be used in MSUP analysis. Pre-Stress Modal analysis supports Load Control property to control loads transferred from pre-stress analysis

Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
ID (Beta)	151
Type	Force
Define By	Components : Real – Imaginary
Applied By	Surface Effect
Coordinate System	Global Coordinate System
X Component - Real	100. N
X Component - Imag	0. N
Y Component - Real	200. N
Y Component - Imag	0. N
Z Component - Real	0. N
Z Component - Imag	0. N
Suppressed	No
Load Vector Controls	
Load Vector Assignment	Program Controlled
--X Load Vector Number - Real	1
--Y Load Vector Number - Real	2

Details of "Analysis Settings"

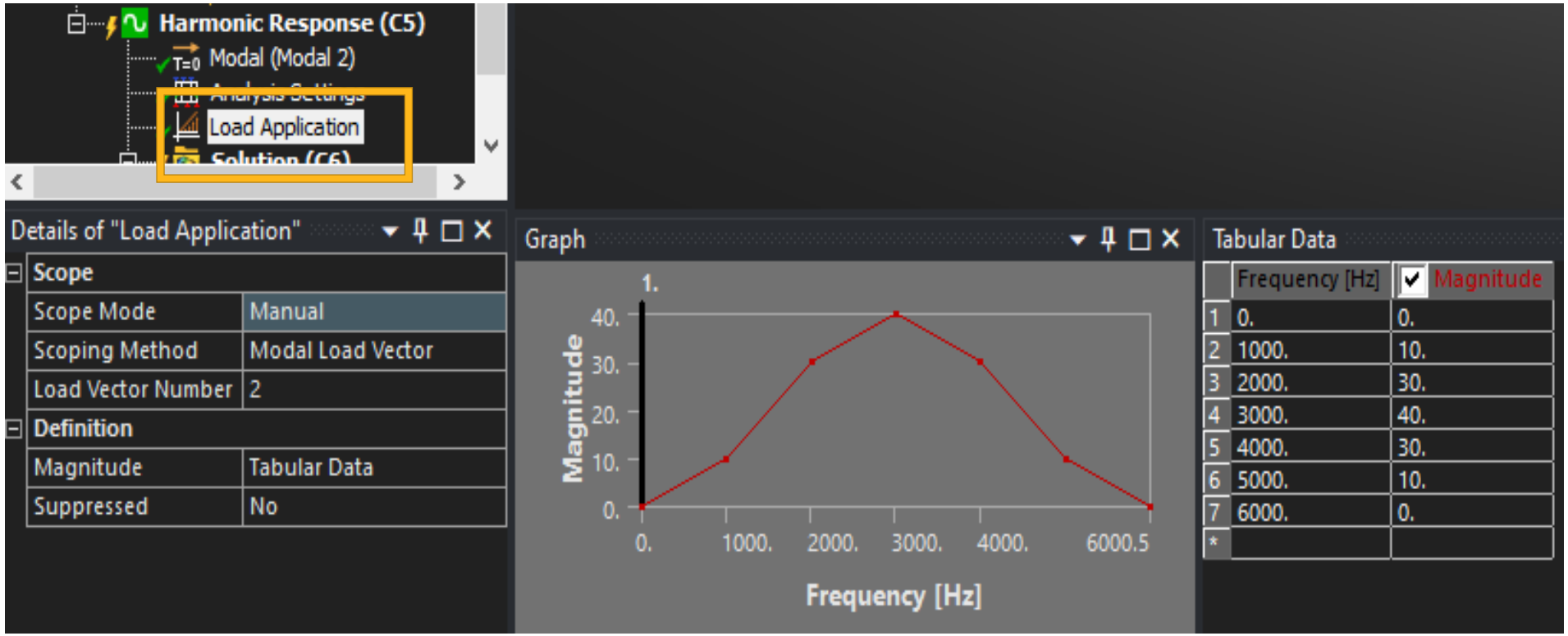
Options	
Max Modes to Find	6
Limit Search to Range	No
Include Residual Vector	Yes
On Demand Expansion Option	No
Spin Softening	Program Controlled

Details of "Pre-Stress (Static Structural)"

Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status
Newton-Raphson Option	Program Controlled
Load Control	Keep All Displacements As Zero

Load application in MSUP Harmonic analysis

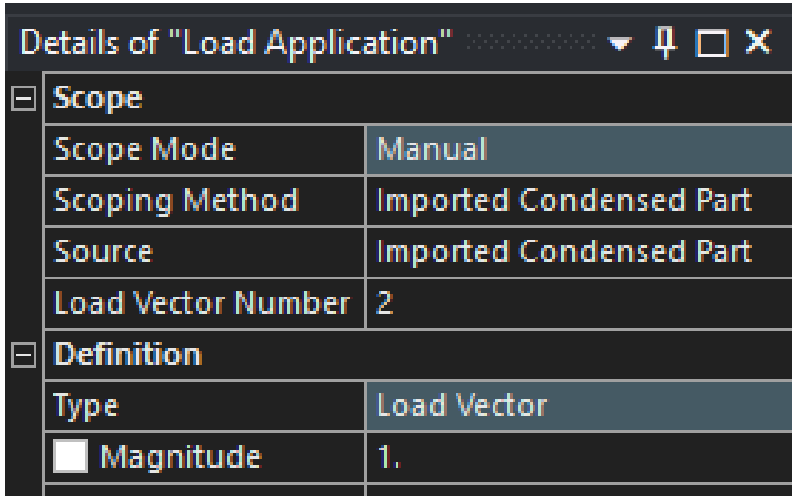
- Load application object is then added in MSUP harmonic analysis to apply and scale the load vector added in Modal analysis

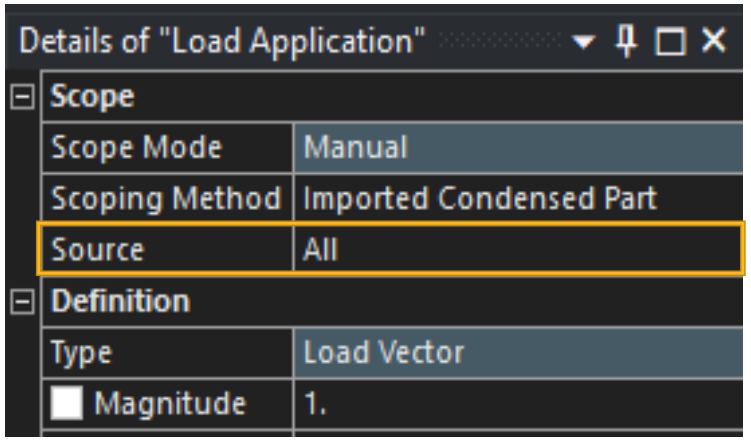


Load application for Imported Condensed Parts

- **Load Application** object with **Scoping Method** set to **Imported Condensed Part** scales load vectors included with Imported Condensed Parts.

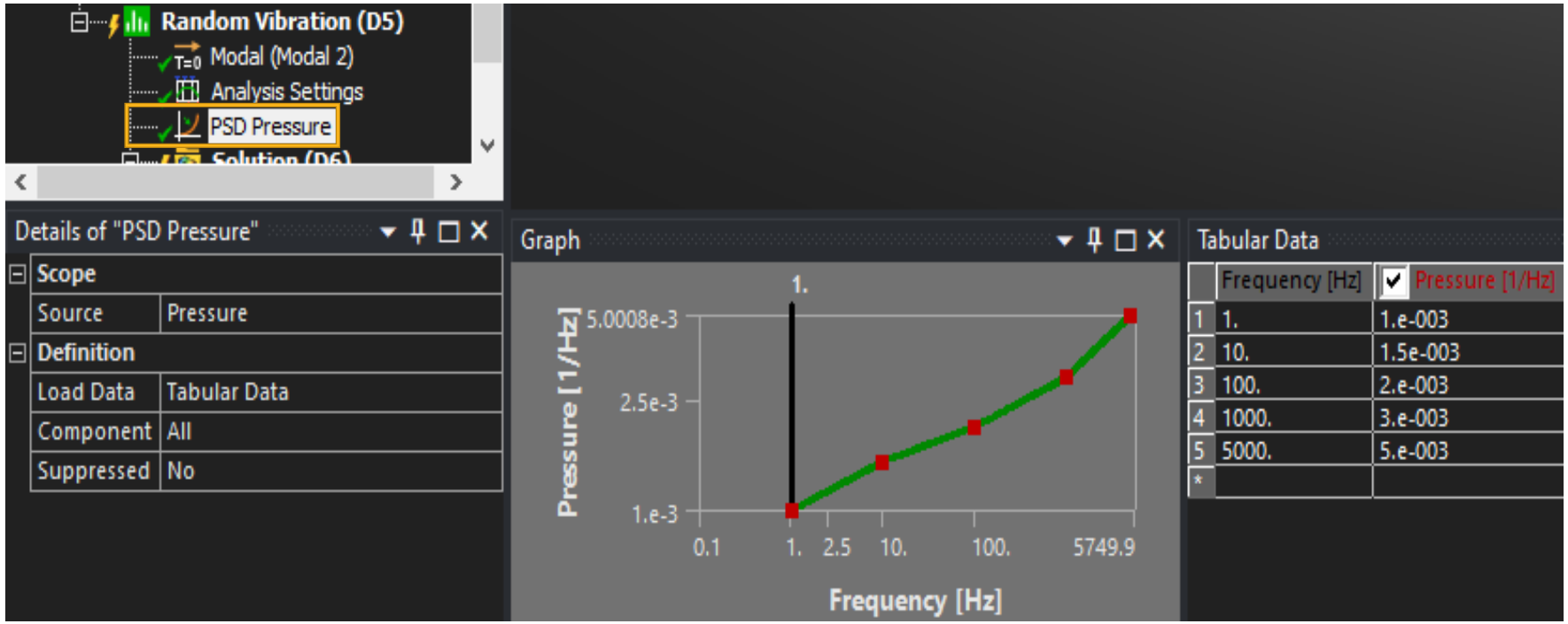
- All Imported Condensed Part load vectors can be scaled via one **Load Application** object by setting **Source** property to **All**.





PSD pressure in Random Vibration analysis

- PSD pressure can be applied and scaled using the load added in Random Vibration analysis which uses the Pressure load as load vector added in the linked Modal analysis



Mechanical Workflow

Data Handling & Storage

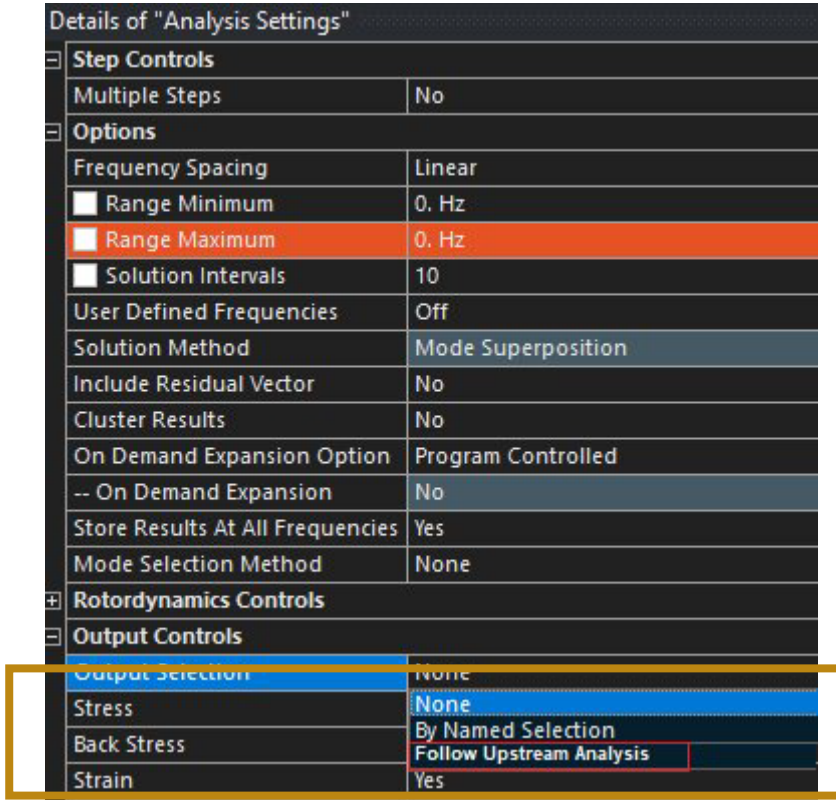
Output Selection – On Mode File By Named Selection

- The By Named Selection On Mode File option is only valid for Modal and Harmonic Response analysis when on demand expansion is active, when all other Output Control properties are set to No and no modal restart is performed. This option enables you to filter the content of the mode file and only write the results for the nodes of the selected named selections

Output Controls	
Output Selection	On Mode File By Named Selection
-- Named Selection	1 Object
Stress	No
Back Stress	No
Strain	No
Contact Data	No
Nodal Forces	No
Volume and Energy	No
Euler Angles	No
Calculate Reactions	No
General Miscellaneous	No

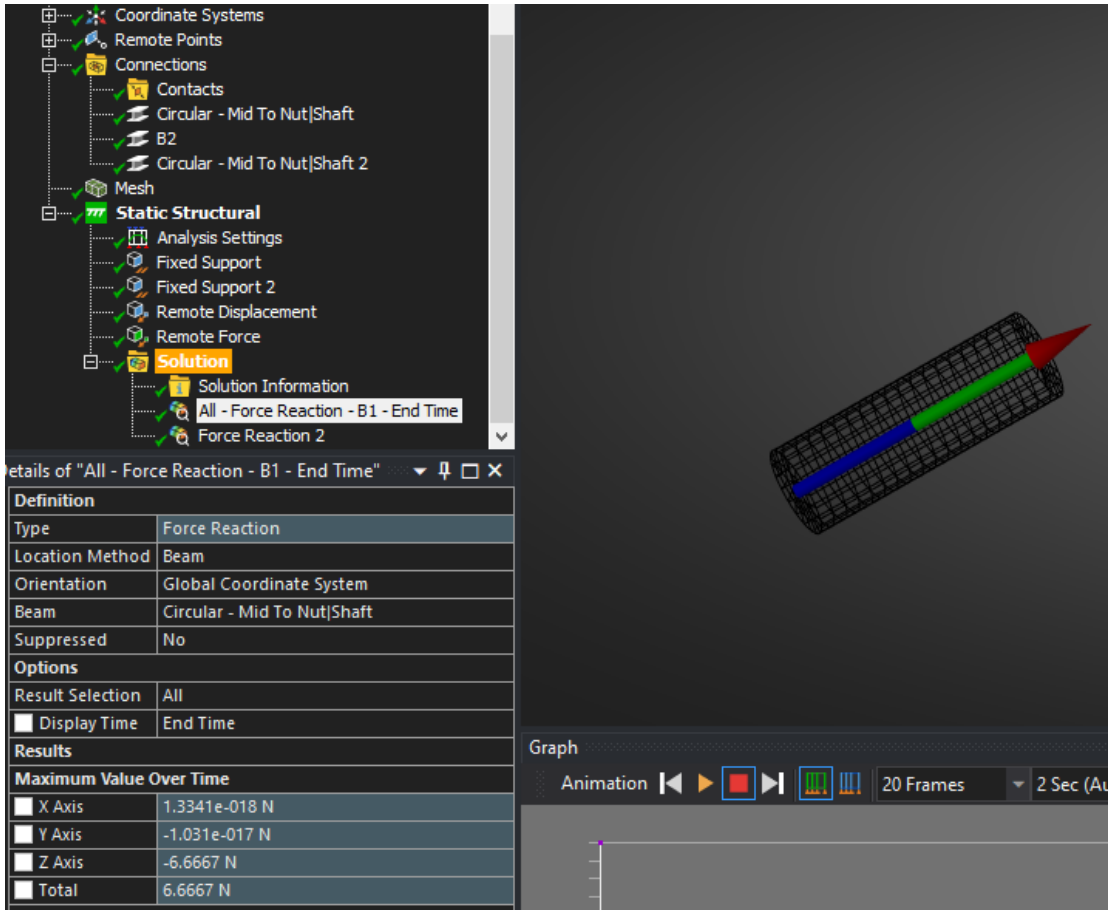
Output Selection – Follow Upstream Analysis

- To avoid having to redefine output controls in linked analyses (Harmonic & Transient), we add an entry in Output Selection as illustrated :
- Activating this will reuse the output controls of upstream modal and make downstream ones read-only.
- This option can be activated by default when we create a harmonic or transient analysis linked to a modal that already uses OnDemand Expansion
- If By Named Selection or On Mode File By Named Selection is used in upstream modal, it is also used in downstream analysis



Force Reaction Probe: Beam Scoping

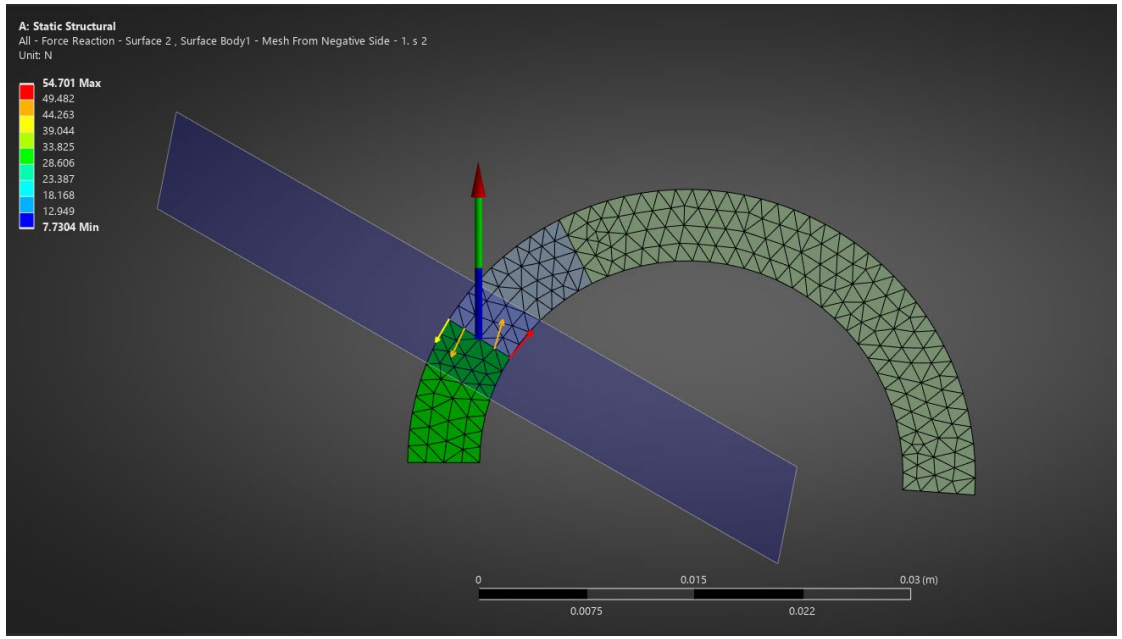
- The Force Reaction probe can now use Location method = Beam
- Body-Body beam will display the element-nodal (FSUM) solution for beam node I.
- This probe is supported in the following analyses:
  - Static
  - Transient Full/MSUP
  - Modal
  - Harmonic Response Full/MSUP
- The reaction solution for this probe is controlled by the ESOL option of the OUTRES command. To enable the output, set both the Calculate Reactions and Nodal Forces properties of the Output Controls (Analysis Settings) to Yes.

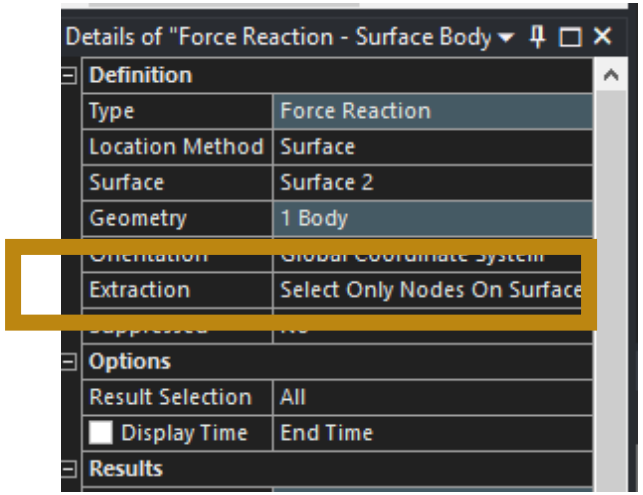


New Extraction Option for Reaction Force on Surface Plane

- “Select Only Nodes on Surface” is a new extraction option that determines the scoping for the force reaction based on nodes that lie directly on the surface.
- Existing options provided an element-based scoping based on which elements intersected the surface. This did not always provide the expected reaction sum at the exact surface.
- Mirrors MAPDL ability to select nodes that lie within a specific plane on a coordinate system:

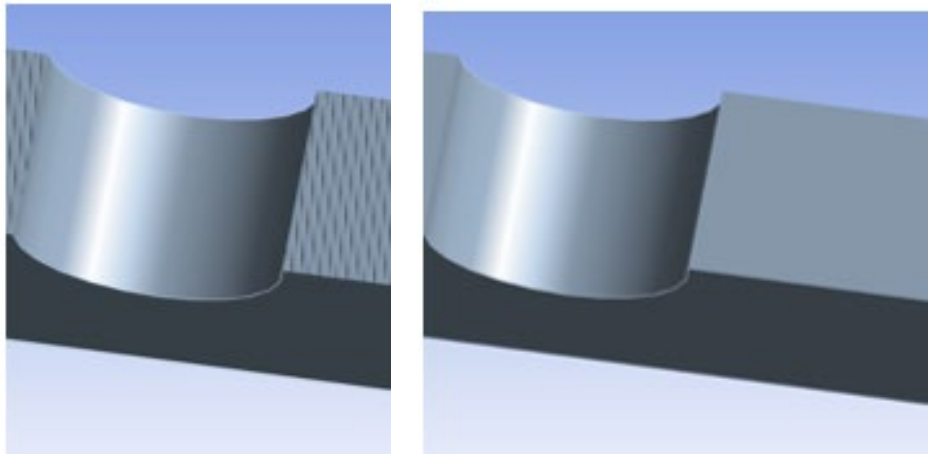
```
CSYS, myCSYS

NSEL, R, LOC, Z, 0
```





General Graphics Improvements

- Faster UI response switching between contacts, especially with many bodies (>10k)
- Accelerated Animation can now be used to generate video exports
  - Significantly faster export and potentially higher quality
- Section planes can be created by any plane of coordinate system (previously only XY)
- Section plane X-hatching toggle On/Off:



Accelerated Animation Export

- Accelerated Animation can now be used to generate video exports
- In previous releases, Standard Animation was always used for video export, regardless of whether Standard or Accelerated Animation was being used for onscreen animation
- This process is automatic – the method used for video exports will always be the method that is currently being used for onscreen animation
  - As a reminder, you can see what method will be used by looking for the “SA” or “AA” icon in the animation toolbar
- Accelerated Animation video exports can be significantly faster than Standard Animation exports, and in some cases have higher visual quality
  - For ex. on one heavy test model, it was observed that animation export used to take ~90 s but now takes ~15 s

Performance Improvement On Switching Between Contact Region Objects

- Examples:
  - For a model with 9000 bodies, switching between contacts is faster now with ~0.75s compared to previous versions which takes about ~1.9s.
  - For a model with 71624 bodies, switching between contacts is faster now with ~4.3s compared to previous versions which takes about ~36s.
- So, the models with large number of bodies have a huge performance impact compared to models with few number of bodies.

Expanded Anti-Aliasing (MSAA) Usage

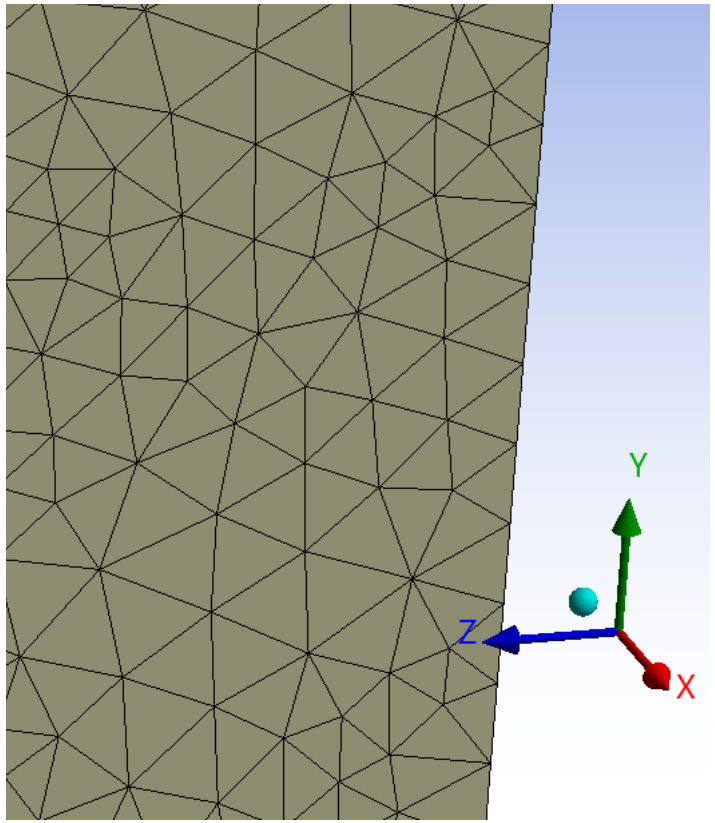
- In previous releases, the user preference “Options > Mechanical > Graphics > Default Graphics Options > Anti-Aliasing (MSAA)” affected the graphics in only the main Geometry window and was only available on Windows

- This option, which enhances the visual quality of the graphics, is now available on Linux and now affects the graphics of various other scenarios, including image exports, animation exports, print preview, report preview, and the Body Views windows

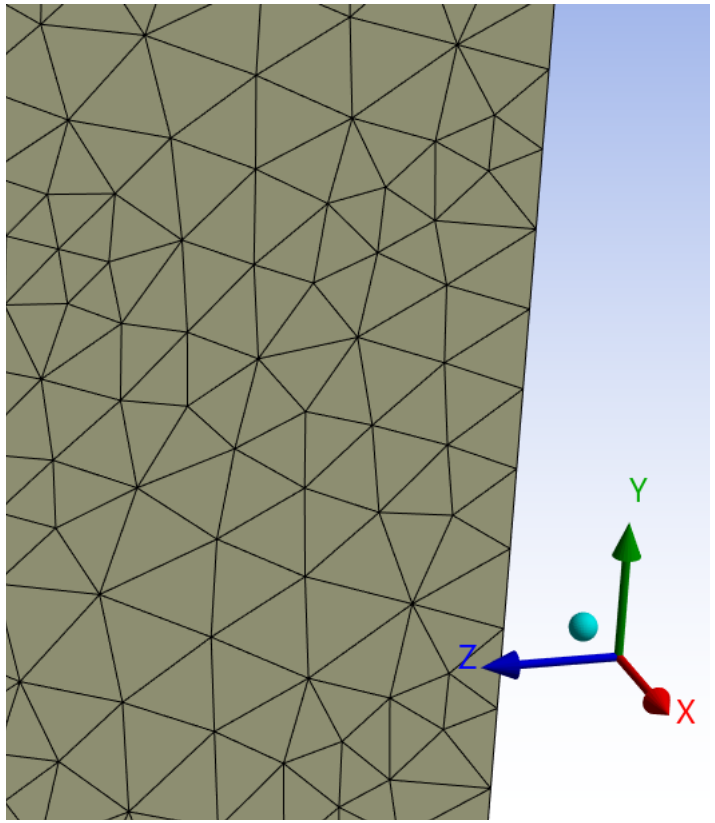
Example of Anti-Aliasing Visual Enhancement

- Anti-aliasing can visually enhance graphics by “smoothing out” lines and edges

- As an example, notice the differences between the element edges and the triad in these two images
  - To best see the difference, view this PowerPoint in presentation mode



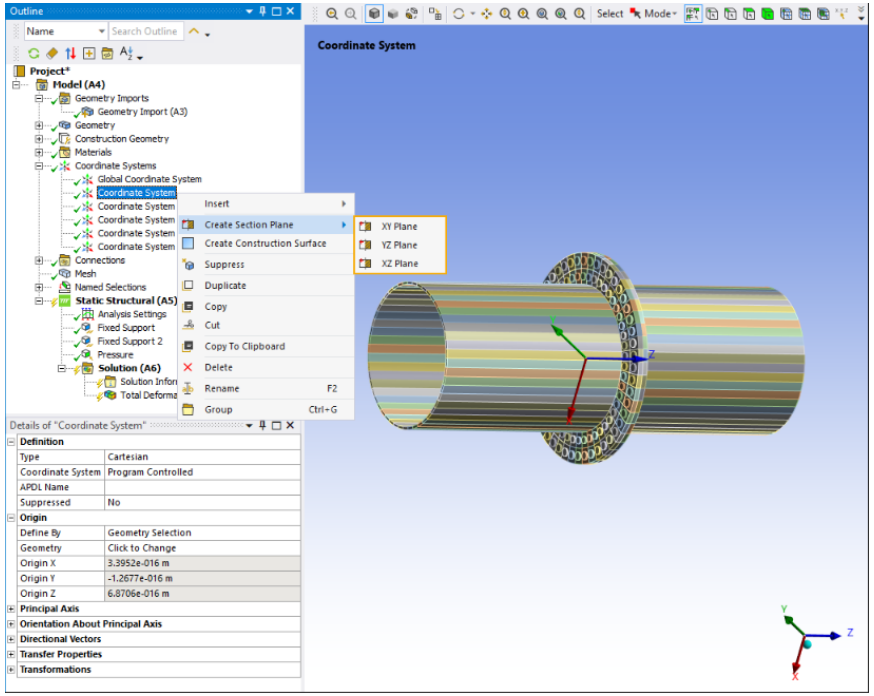
1x MSAA (Off)

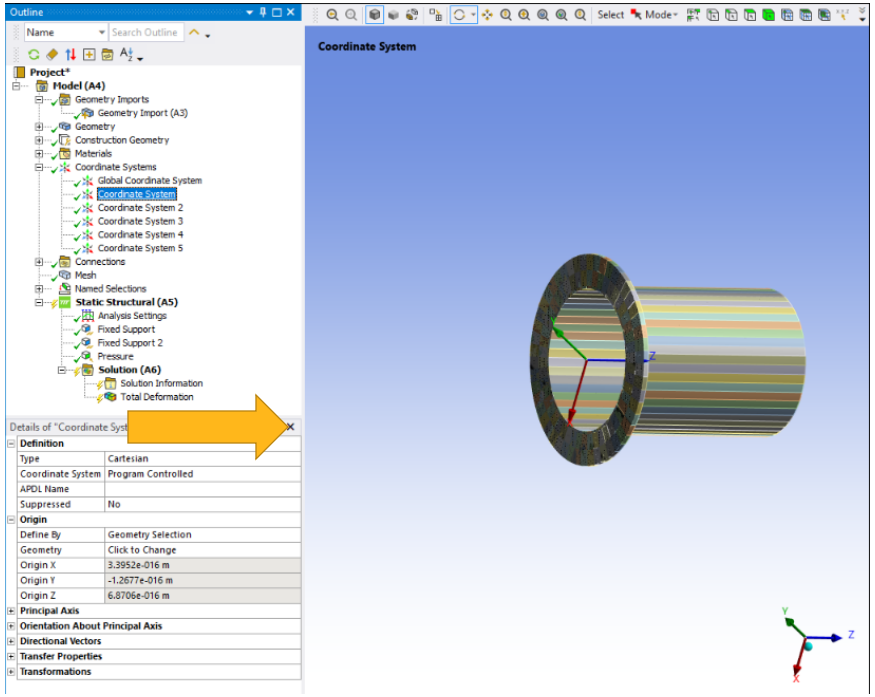


8x MSAA (High)

Section planes can now be created based on Coordinate Systems

- Select a desired *Coordinate Systems* object in the Outline pane.
- Right-click the object, select *Create Section Plane*, and then select the desired plane upon which to create the Section Plane.
- There are following options to choose from
  - XY Plane
  - YZ Plane
  - XZ Plane

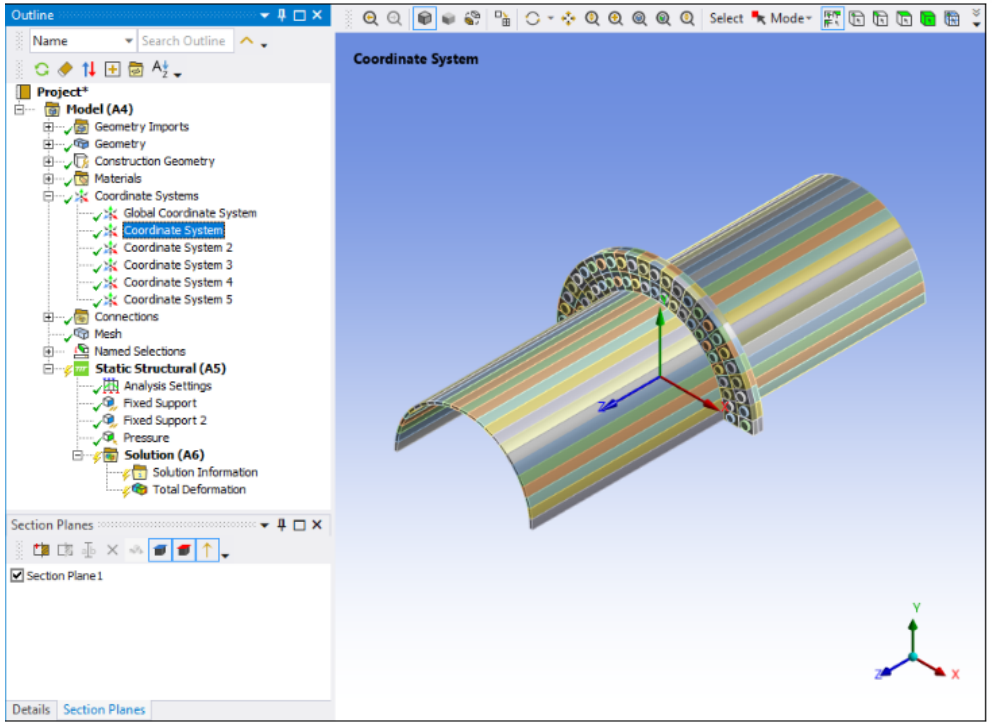




Section Plane Scripting

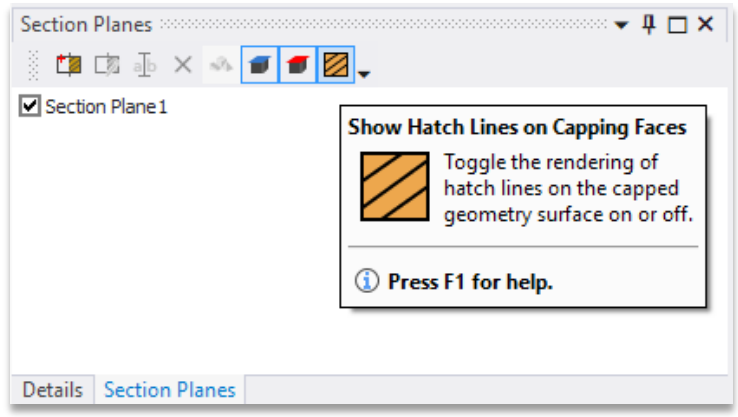
- *CreateSectionPlane()* function is added to take in the coordinate system and an optional parameter that enables user to select the plane orientation from a standard set of planes viz- (XY, YZ, XZ)
- Following code snippet demonstrates the usage of the said function by creating a section plane across the XZ Plane of the selected coordinate system.

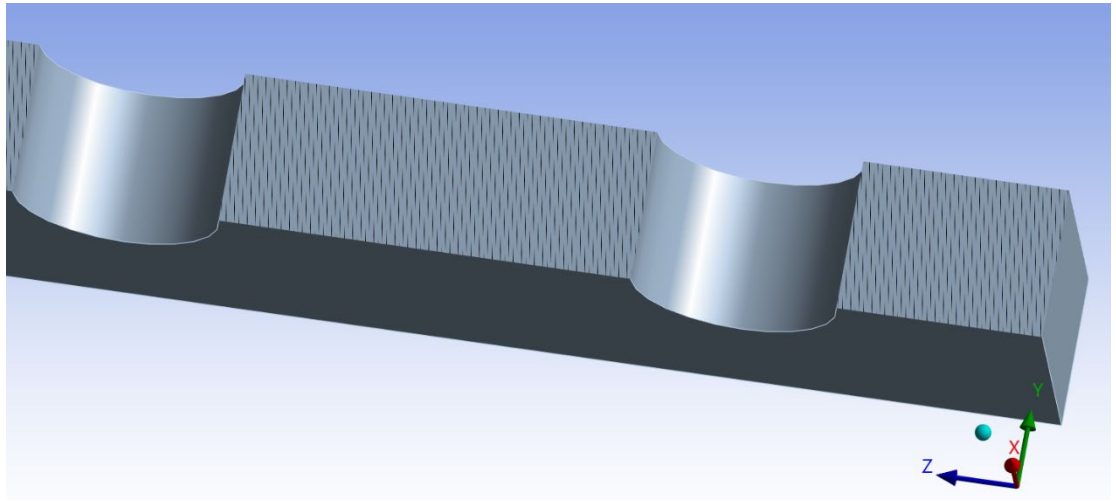
```
coord_system = Model.CoordinateSystems.Children[1]
section_plane = Graphics.SectionPlanes.CreateSectionPlane(coord_system, Ansys.Mechanical.DataModel.MechanicalEnums.Common.PlaneOrientation.PlaneOrientation_XZ)
```

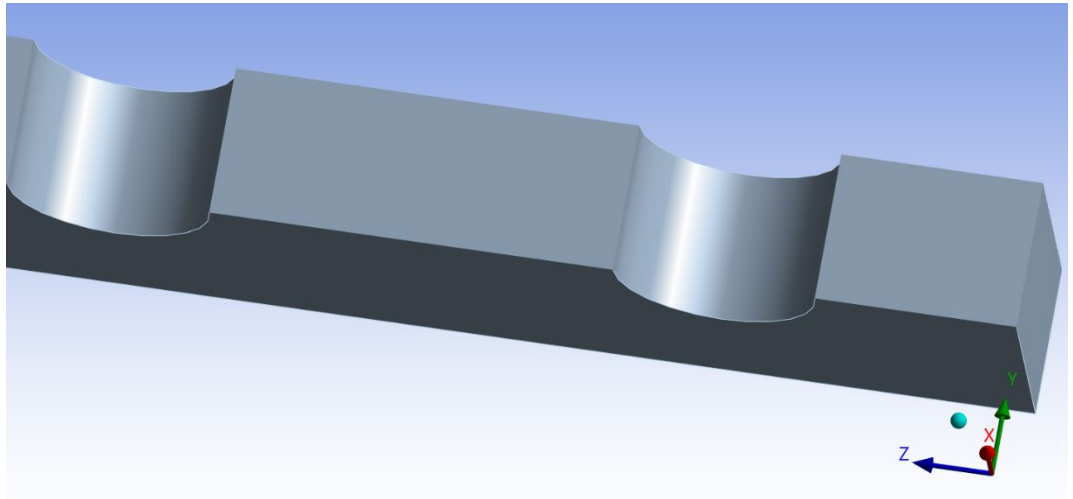


Section Plane Hatch Line Display Option

- The Section Planes toolbar now includes an option for toggling the display of cross section hatch lines, which are shown when cutting geometry with the Show Capping Faces display option enabled







Ansys LS-DYNA

2025 R2

Ansys LS-DYNA – What's New

Ansys LS-DYNA 2025 R2

Material Improvements

- UI improvements
- New Materials in Engineering Data
- Support of user defined materials
- Material Assignment

Structured ALE Improvements

- Creating SALE using Mesh Workflows
- Material Groupings
- Support of latest keywords

Battery & EM Improvements

- Micro & macro modelling for single cells
- More postprocessing options

Acoustic Improvements

- Rigid Walls and Absorption Surface added as new boundary conditions

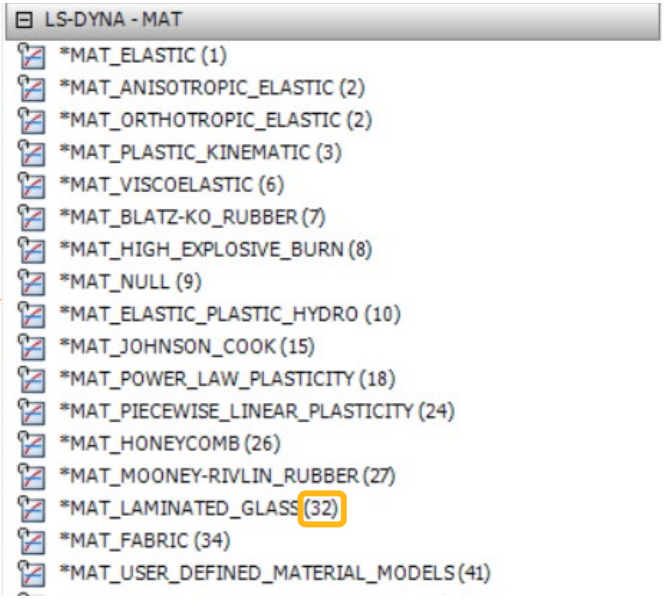
Misc Improvement

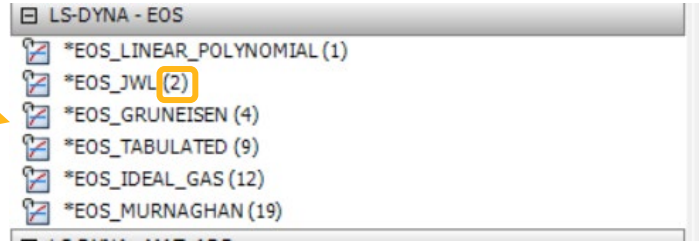
- Compatibility with HPS v1.3
- UI clean up for LS-DYNA specifics

Material Improvements

Material Improvements: GUI & UX

- The LS-DYNA specific sections of **Engineering Data** have been renamed and their names has been shortened
- The LS-DYNA material number is now appended next to the material name





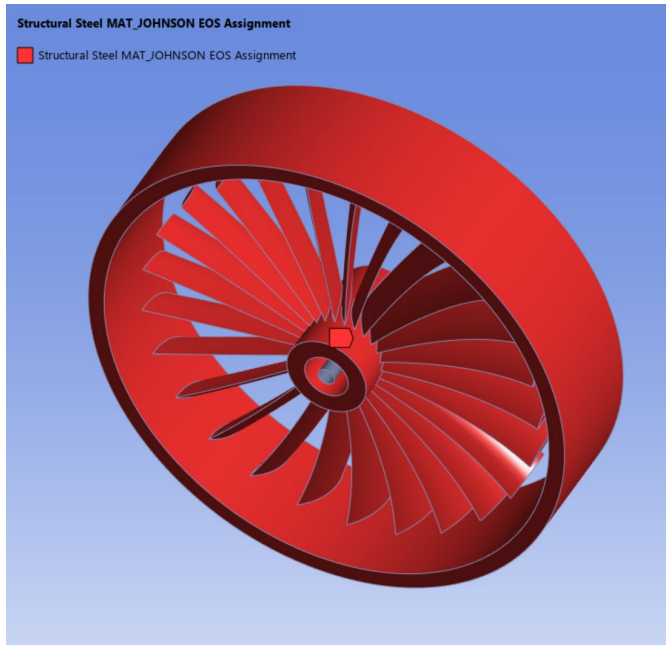
Material Improvements: New Materials exposed

- 18 further LS-DYNA materials models have been added to **Engineering Data:**

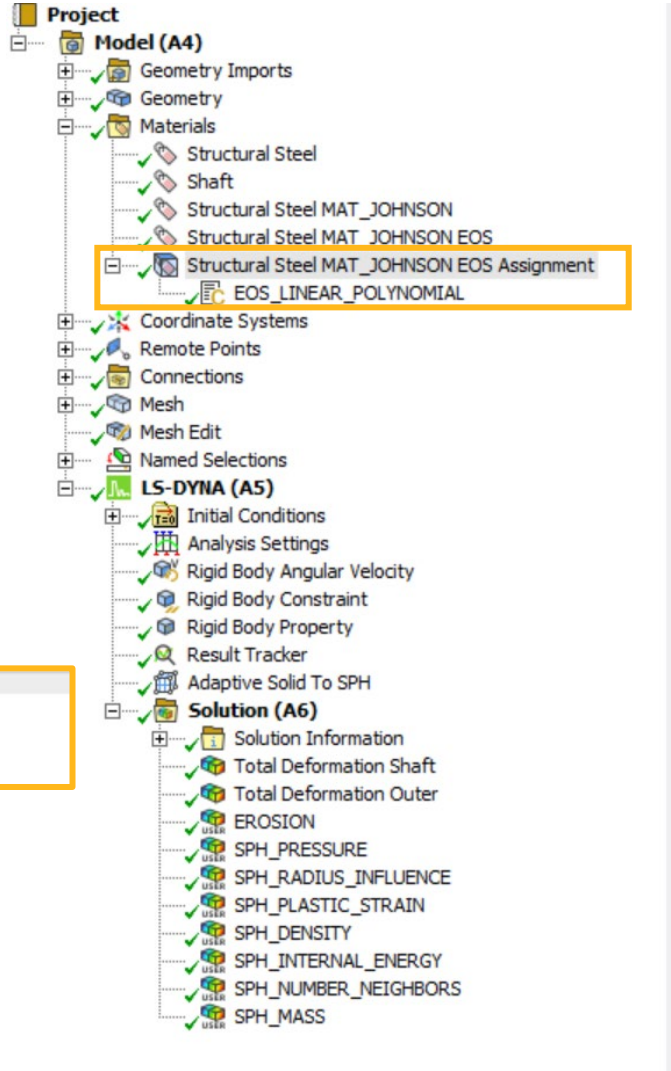
- ***MAT_VISCOELASTIC (6)**
- ***MAT_MAT_HONEYCOMB (26)**
- ***MAT_LAMINATED_GLASS (32)**
- ***MAT_LOW_DENSITY_FOAM (57)**
- ***MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM (67)**
- ***MAT_HYPERELASTIC_RUBBER_H (77)**
- ***MAT_MAT_FU_CHANG_FOAM (83)**
- ***MAT_PLASTICITY_COMPRESSION_TENSION (124)**
- ***MAT_SAMP_LIGHT (187)**
- ***MAT_ELASTIC_PLASTIC_HYDRO (10)**
- ***MAT_MOONEY-RIVLIN_RUBBER (27)**
- ***MAT_USER_DEFINED_MATERIAL_MODELS (41-50)**
- ***MAT_ELASTIC_ELASTIC_DISCRETE_BEAM (66)**
- ***MAT_CONCRETE_DAMAGE_REL3 (72)**
- ***MAT_PLASTICITY_WITH_DAMAGE (81)**
- ***MAT_MODIFIED_JOHNSON_COOK (107)**
- ***MAT_MAT_MODIFIED_HONEYCOMB (126)**
- ***MAT_PIECEWISE_LINEAR_PLASTIC_THERMAL (255)**

Material Improvements: Material Assignment

- LS-DYNA now supports **Material Assignment** for bodies
  - Only a single material card is written
  - Command snippets are supported



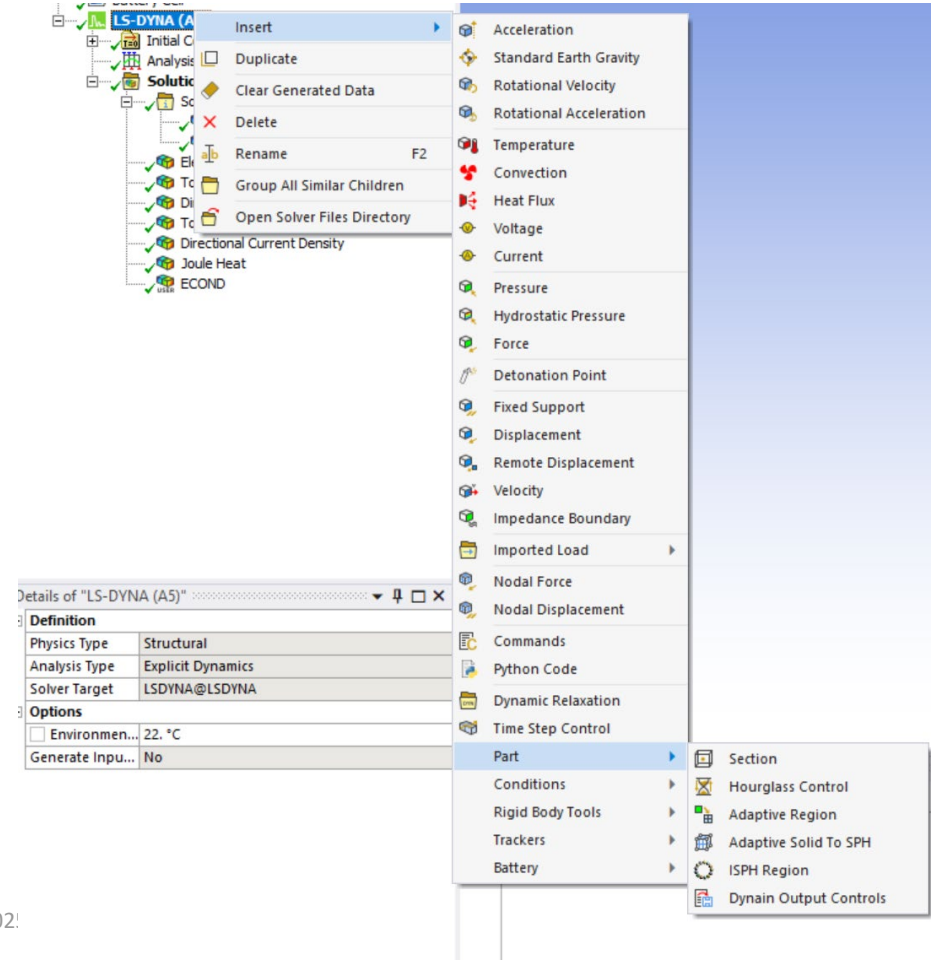

```
*EOS_LINEAR_POLYNOMIAL
$#   eosid         c0         c1         c2         c3         c4         c5         c6
         1  0.06.80000E10        0.0        0.0        0.0        0.0        0.0
$#     e0         v0
       0.0        0.0
```

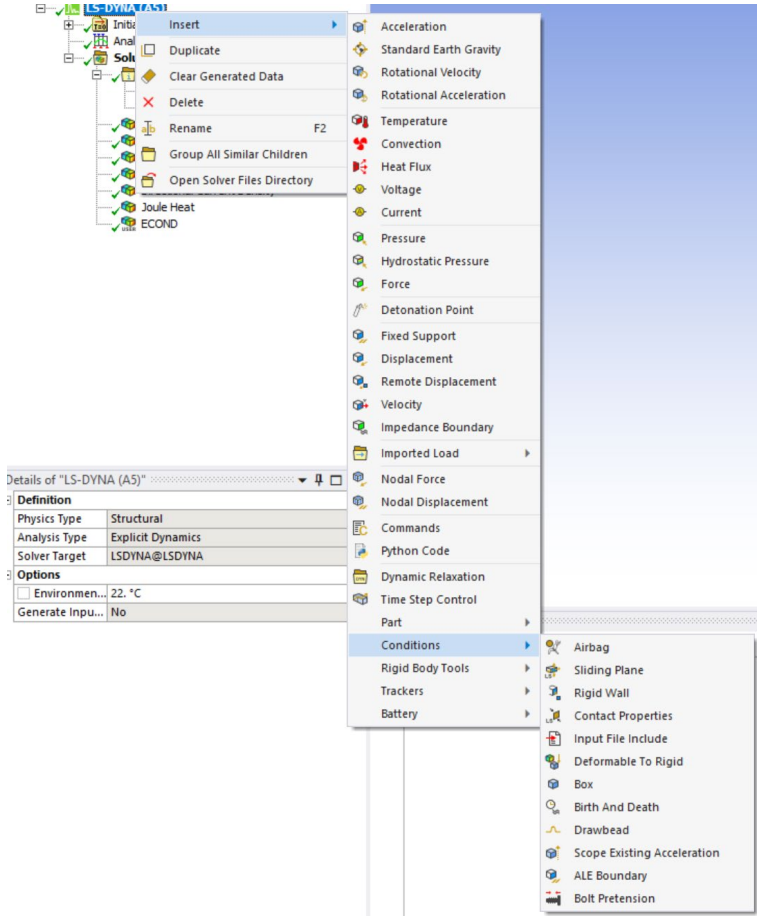


Misc Improvements

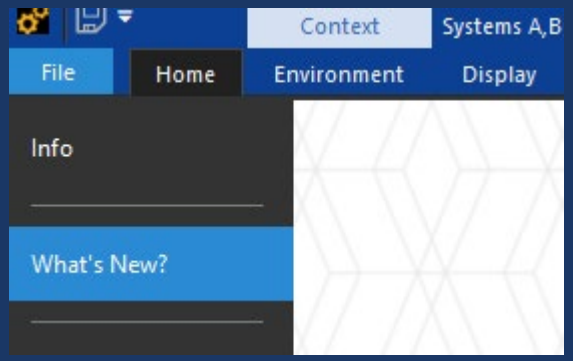
UI Improvements

- LS-DYNA Loads and Boundary Conditions have been grouped into context menu Groups



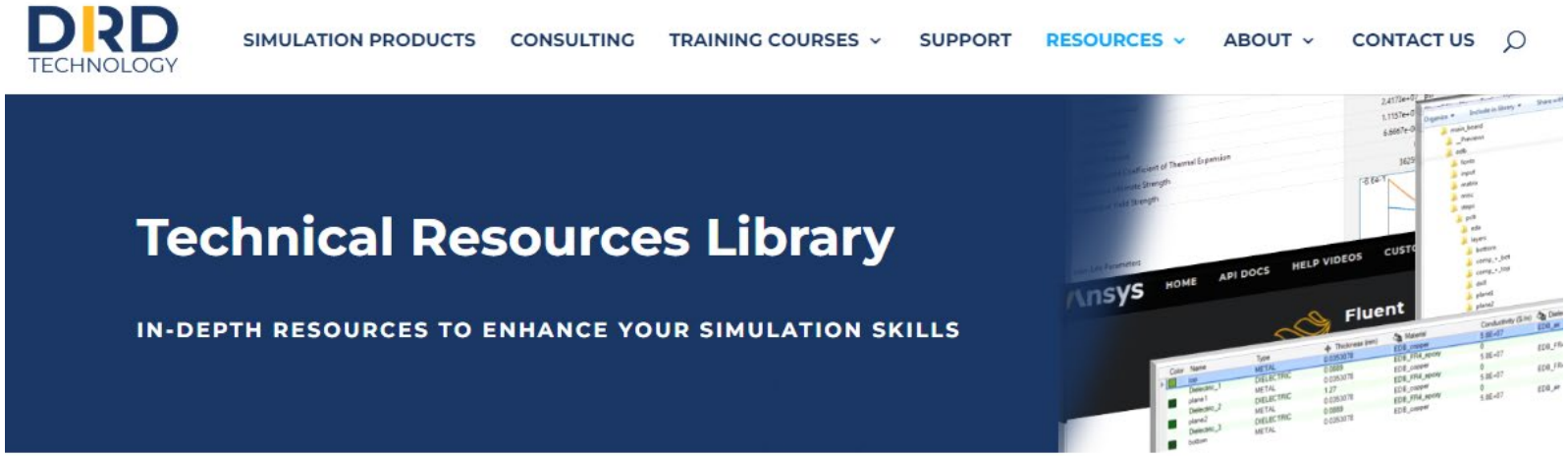


What’s New! There’s more…

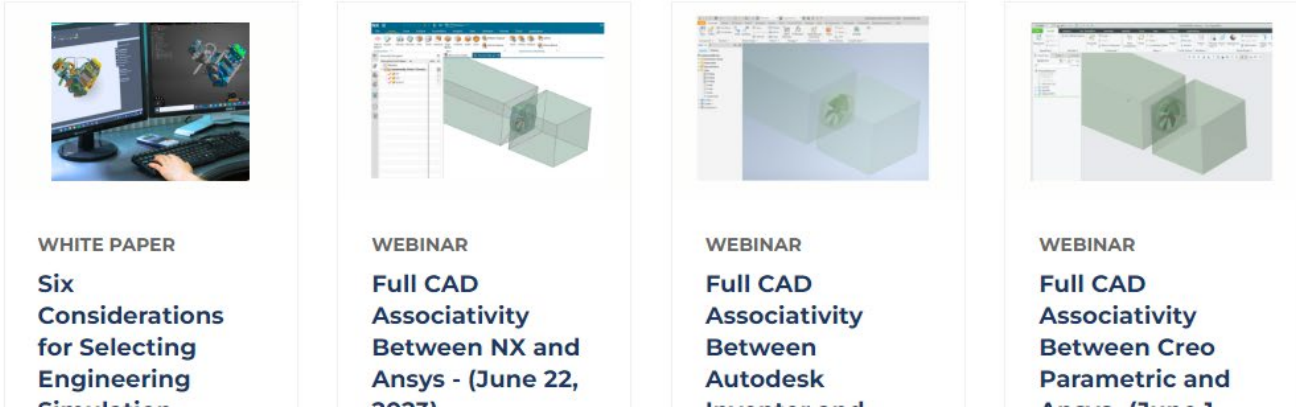


Wrap Up

The recording and slides for this webinar are in our Technical Resources Library.



If you are not on our mailing list, or are unsure if you are, please let us know at support@drd.com and we can add you!



Wrap Up

Whether you're onboarding with the Ansys platform or looking to take your simulation proficiency to the next level, we have a training course carefully designed to fit your needs. With frequent introductory and advanced courses conducted live virtually and in-person or on-demand, we offer many opportunities for you to get the training experience that best suits your needs. Additionally, since our trainings are conducted by our in-house engineering and physics experts, we have the unique opportunity to carefully listen to your requirements and further refine our custom training materials to help you continually meet your goals.

Explore our training center below.

Wrap Up

Thank you for your attention!

May I address any questions?