DRD TECHNOLOGY

What's New in Ansys 2025R1 Structures Webinar

Alex Austin – DRD's Structures Team Lead

12 February 2025 @9AM CT

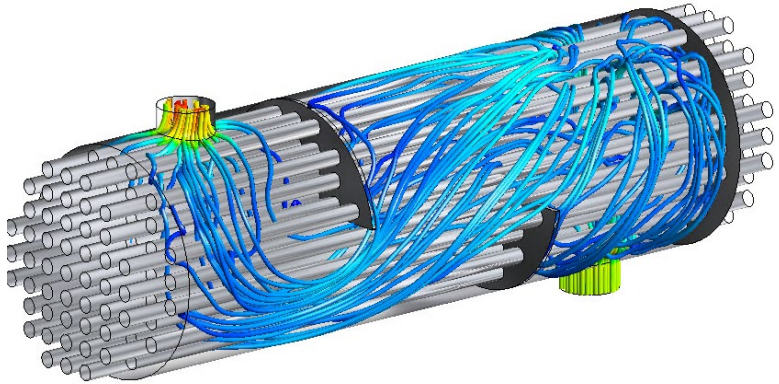
DRD

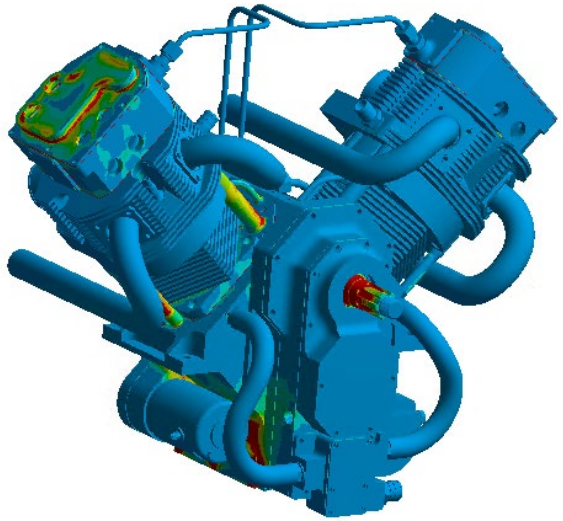
Agenda

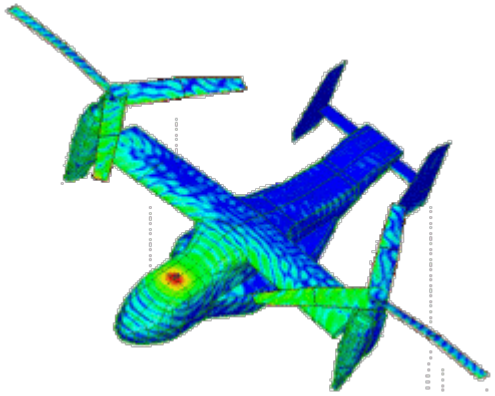
- What’s New – High Level
- Core Features
- Post Graphics
- Meshing
- Fracture
- Materials
- Noise, Vibration, Harshness (NVH)
- LS-Dyna

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.







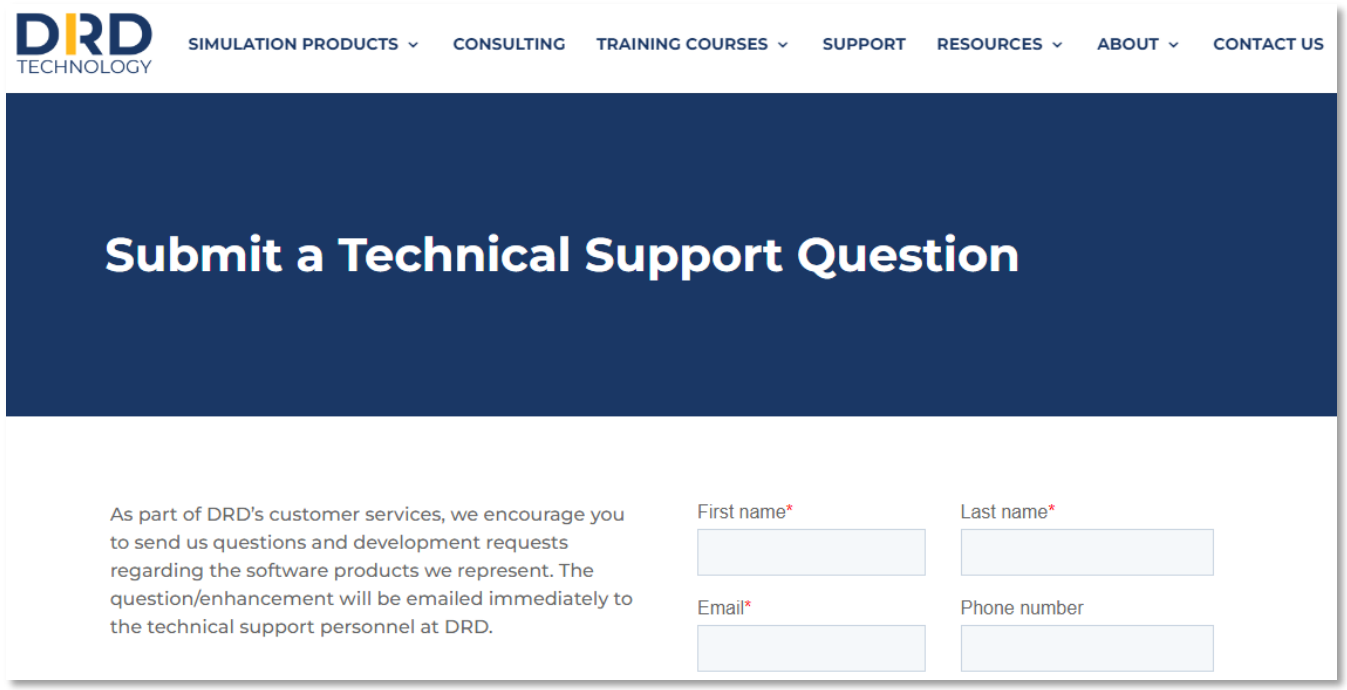
Ansys

CERTIFIED ELITE CHANNEL PARTNER

Technical Support Contact Coordinates

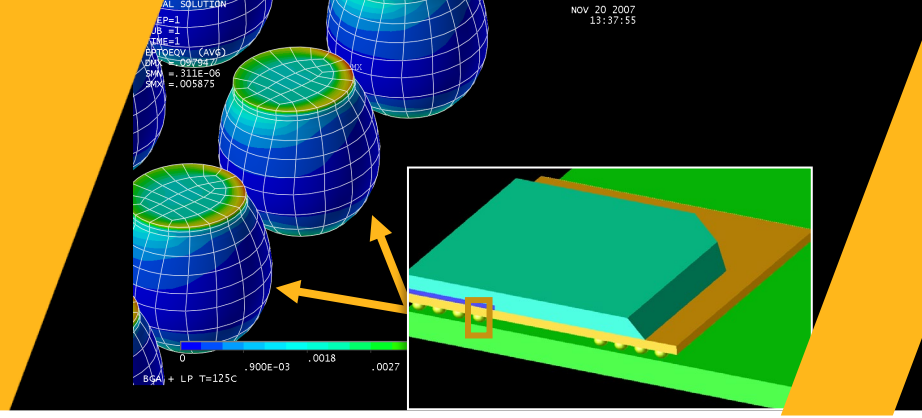
Support:
(918) 743-3013 x1
support@drd.com

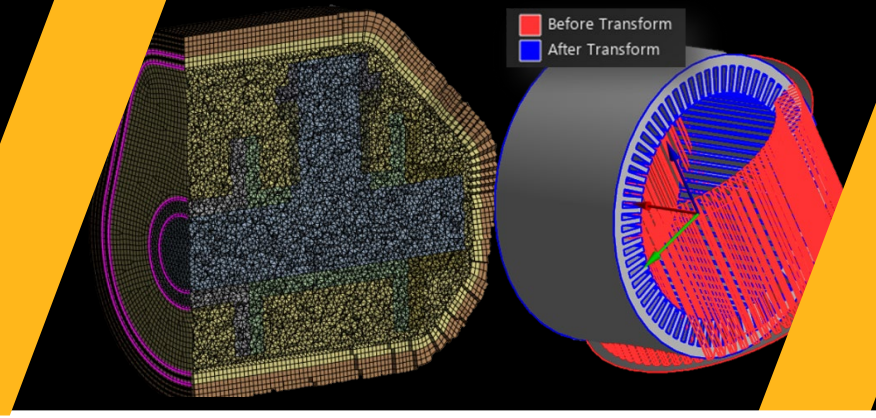
Or through our website at
www.drd.com

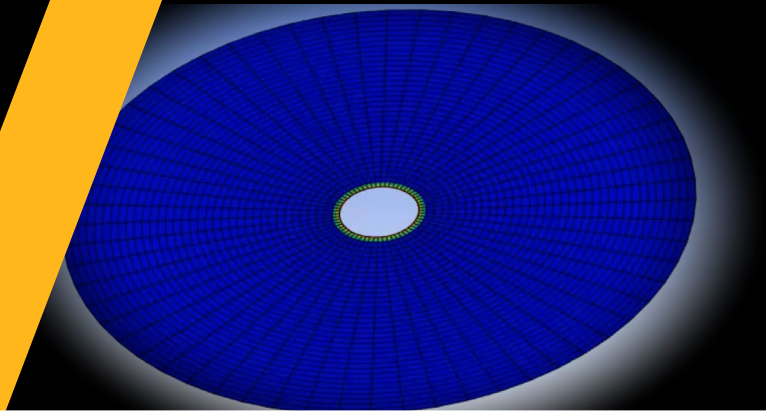


For more than five years, I have worked closely with DRD Technology to execute tactical and strategic initiatives here at EaglePicher due to our unprecedented growth. We've been very happy with DRD and will continue to work with them as our business partner for using Ansys tools effectively and efficiently.

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





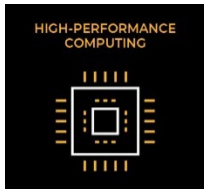


Advanced Solver Capabilities

- Go-to solution for rapid, reliable results on the most demanding simulations.
  - Major GPU and HPC solving enhancements, with the MAPDL GPU-accelerated direct solver now 2-6x faster than competing solutions and the iterative solver 2-3x faster than CPU-only versions
  - Multi-GPU support, enhanced solvers for symmetric/nonsymmetric matrices, and a new "mixed" equation solver that supports more GPU cards while reducing memory requirements by up to 25%

Benefits

- These upgrades enable rapid, efficient handling of large, complex models, making Ansys Mechanical the preferred solution for high-performance simulations in demanding engineering applications.

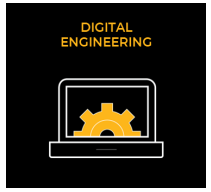


One-Stop-Shop

- The only fully integrated solution provider for NVH
  - Advances include a 10x faster FRF calculator, Vibro-Acoustics mapping, optimized meshing, and Mode Contribution analysis
  - Enhanced full-vehicle durability capabilities, featuring advanced solver competencies, improved performance and scaling, and integrated DesignLife support for comprehensive durability analysis

Benefits

- As the only fully integrated solution provider for NVH, Ansys offers efficient full-vehicle structural and durability simulation, advancing toward a unified Crash-Durability-NVH model with a fully automated, user-friendly NVH workflow.



Advanced Solver Capabilities

- Consistent powerful advancements to best-in-class solvers and solutions.
  - PolymerFEM's MCalibration for accurate material models to capture real-world behavior in simulations
  - New SMART for customizable crack growth analysis with complex loading scenarios and ACCS RTM for accurate simulation of composite material infusion processes

Benefits

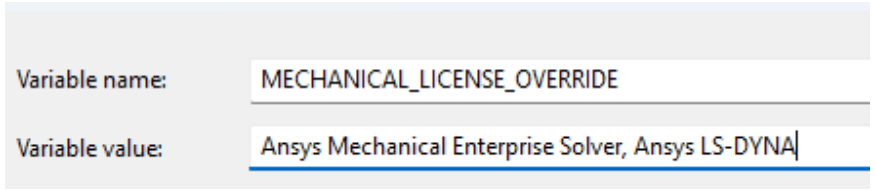
- These advanced tools enhance Mechanical's unmatched precision and adaptability, helping engineers deliver safer, optimized designs across various applications and industries.

Powering Innovation That Drives Human Advancement

Core Features 2025R1

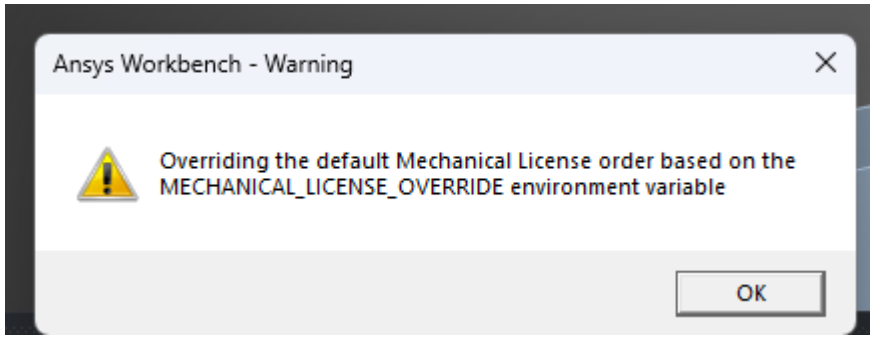
Override Default License Ordering

- The user can override the default License Ordering used by Mechanical, by using a system Environment Variable, **MECHANICAL_LICENSE_OVERRIDE**
- The variable value is a comma-separated entry.
- This variable is read, and the overriding of Licenses is applied when Mechanical is opened for the first time after a new installation.
- A warning message is shown when Mechanical opens, indicating the **MECHANICAL_LICENSE_OVERRIDE** has been read to change the license order.



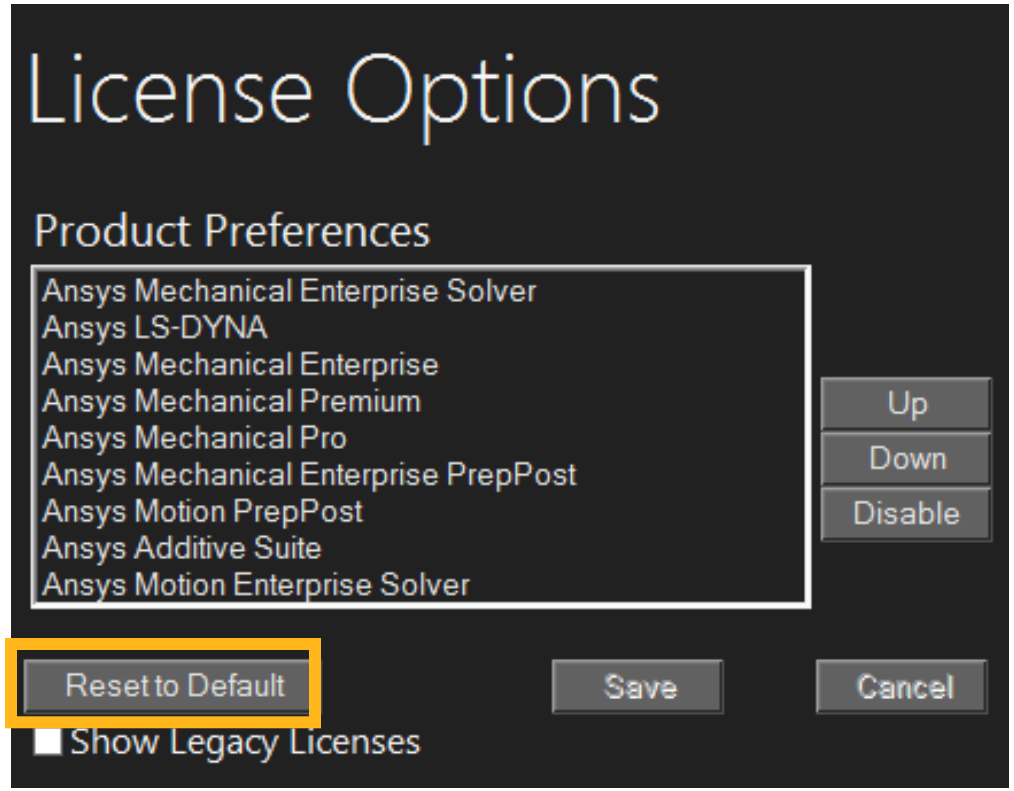
New License Ordering

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<Licenses>

  <LicenseInfo Active="1" LicenseName="Ansys Mechanical Enterprise Solver"/>

  <LicenseInfo Active="1" LicenseName="Ansys LS-DYNA"/>

  <LicenseInfo Active="1" LicenseName="Ansys Mechanical Enterprise"/>

  <LicenseInfo Active="1" LicenseName="Ansys Mechanical Premium"/>
```



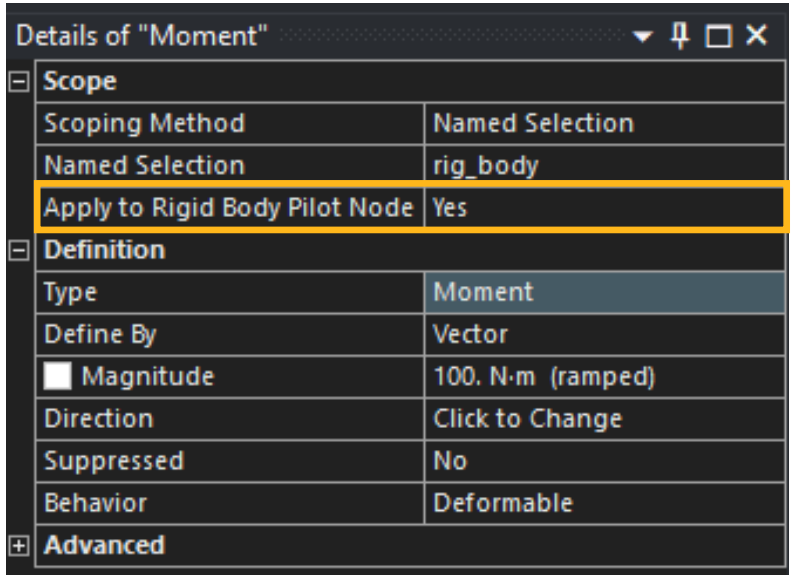
Override Default License Ordering

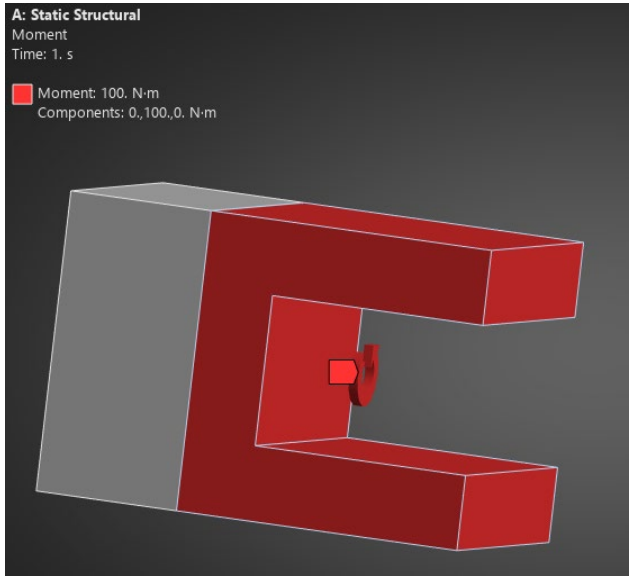
- The Licensing page in Mechanical will reflect the new License order.
- The ‘Reset to Default’ when **MECHANICAL_LICENSE_OVERRIDE** is present will give priority to the licenses in the environment variable.



Apply remote loads on rigid body pilot node

- Remote loads can create an unwanted torsion if pilot node is created on the face or edge of rigid body
- New property **Apply to Rigid Body Pilot Node** can be used to apply the remote load at the centroid of the scoped rigid body to overcome this
- Supported loads: Remote Force, Remote Displacement, Moment





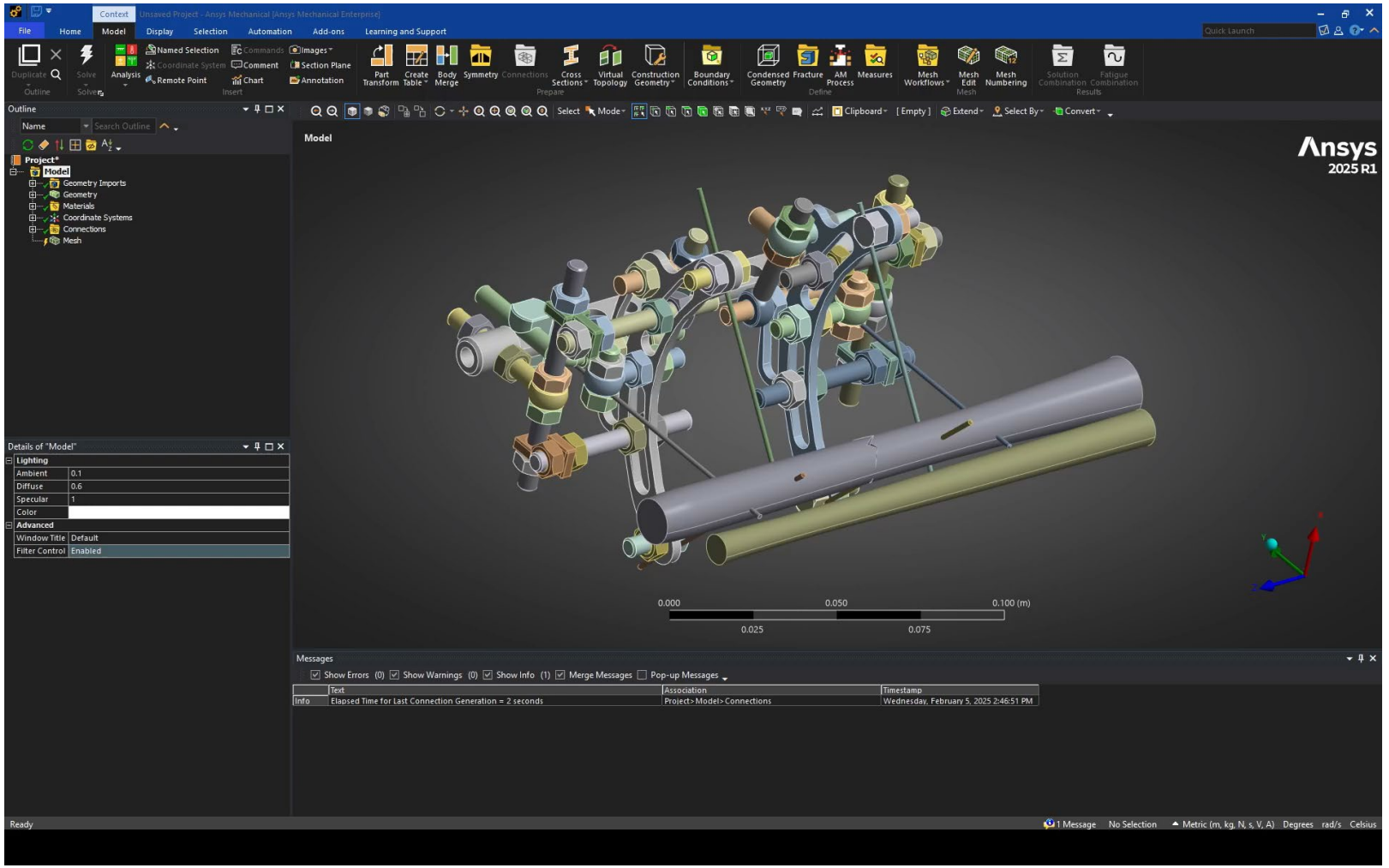
Powering Innovation That Drives Human Advancement

Post Graphics Features 2025R1

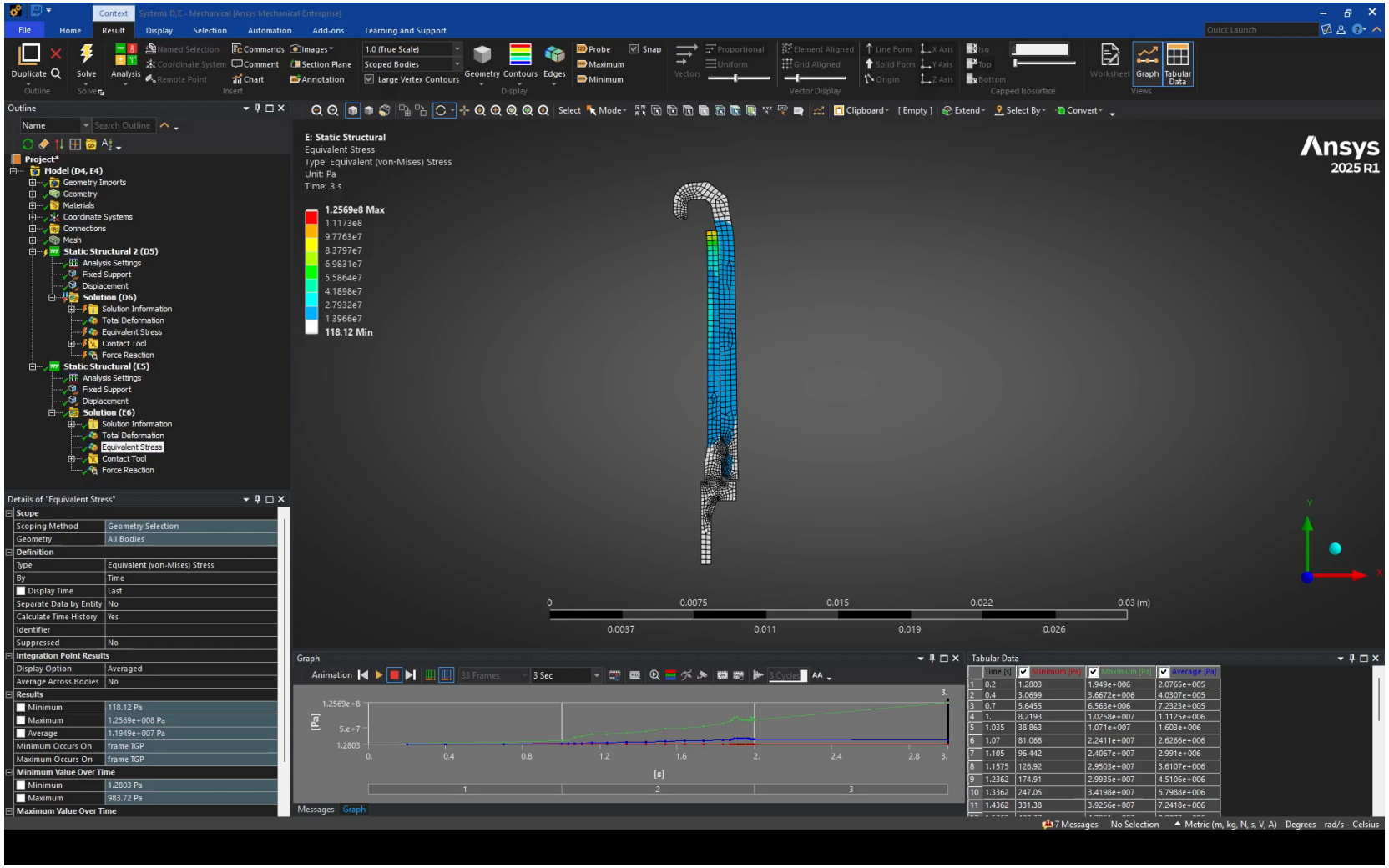
General Post Enhancements

- Nodal scoping for Response PSD probes
  - Previously could only be scoped to vertices, now can be scoped directly to nodes.
- Display Element Normals for Mesh
- MP4 animation export can be played on browsers
- Ruler display preference
  - New option to change the number of ruler subdivisions while keeping the length of each subdivision the same
- Image To Clipboard enhancements
  - additional options like **Current Display (default)** and **Use Image Export Settings**.
- Custom Dimension Animation Exports
  - Now when exporting animations from the UI, you will be presented with an option to customize the dimensions of the video

Ruler Display Preference



Animation Export Preferences



Powering Innovation That Drives Human Advancement

Mechanical Meshing 2025R1

Mechanical Meshing Enhancements in 2025R1

- **Meshing for Electronics**
  - Fast Stacker Mesh Workflow (beta) for Electronics-Reliability hex meshing of assemblies with solder balls
    - Connected mesh without need for Share Topology in Geometry editor
    - Multizone (including Medial Axis) support for solder ball assemblies
    - Mesh copy utilities for solder ball patterned mesh performance
    - Stacker robustness enhancements for 2.5D (vector-extruded geometries)
    - Surface meshing improvements – auto-map on ring shaped faces
    - Mesh Workflow support for Quad Layers and MultiZone surface meshing
    - Detection of stackable bodies and diagnostics tools

- **NVH and Mesh Workflows**
  - FSI workflows enhancements
  - New workflow steps to support direct meshing (instead of wrapping i.e. topology/surface mesh clean-up steps)
  - New options for acoustics workflow – microphone placements (beta)
  - Size control enhancements, size field visibility in Domain Browser
  - Usability enhancements for Mesh Workflows
  - Enhance Quality Worksheet inside Mesh Workflow to allow quantitative mesh quality tables

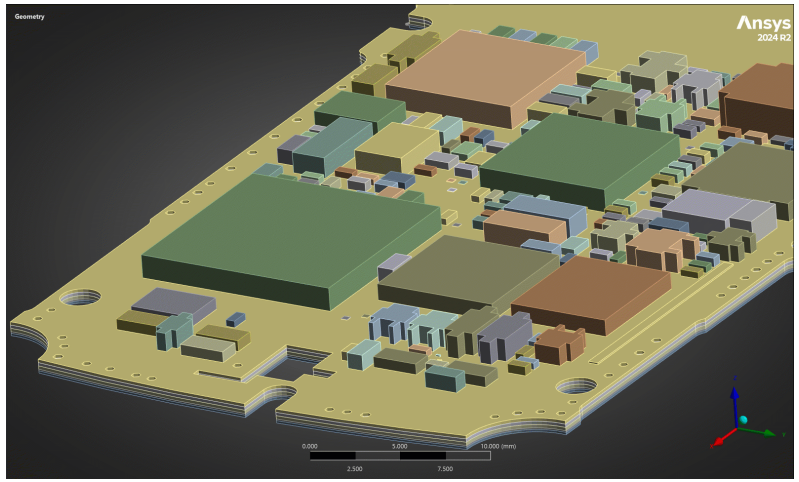
- **General Tet Meshing**
  - Auto-Map Fillets (Beta)

- **Welds and Shell Meshing**
  - Automatic (PrimeMesh)
    - Quad Meshing enhanced on thin faces
    - Curvature calculation enhanced
    - Defeaturing enhancements

- **Hex Meshing**
  - ThinSweep enhancements
  - Released from Beta: MZ Medial Axis Decomposition, MZ Prime Quad for Surface Mesh Method
  - Beta release: Blend to Neighbors (Edge Sizing) Control; Suppress Topology for edges/vertices with MZ; Show axisymmetric bodies

- **Usability, Automation and Performance**
  - Enhanced Quality Worksheet
  - Enhanced Mesh Worksheet
  - Enhanced Mesh Copy Control
  - New global **Automatic Methods** option in Mesh Details
    - **Sheet Body Method**: Quad Dominant or PrimeMesh
    - **Sweepable Body Method**: Sweep or MultiZone

Meshing for Electronics



New Mesh Workflow Approach

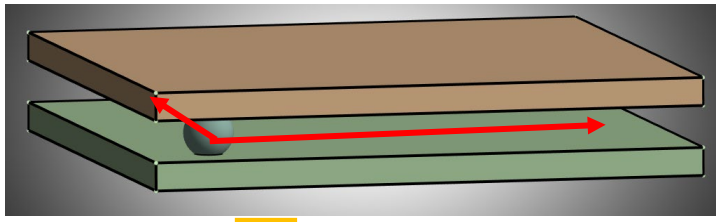
Beta release in 25R1

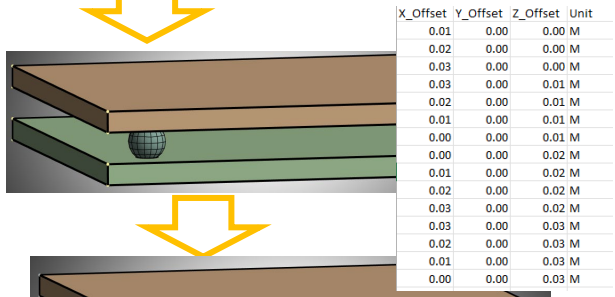
Input to Workflow:

1. Geometry of assembly with **single** solder ball
2. **No Share Topology or imprint requirement**
3. CSV file containing delta vectors from the single solder ball location. N vectors for N balls.

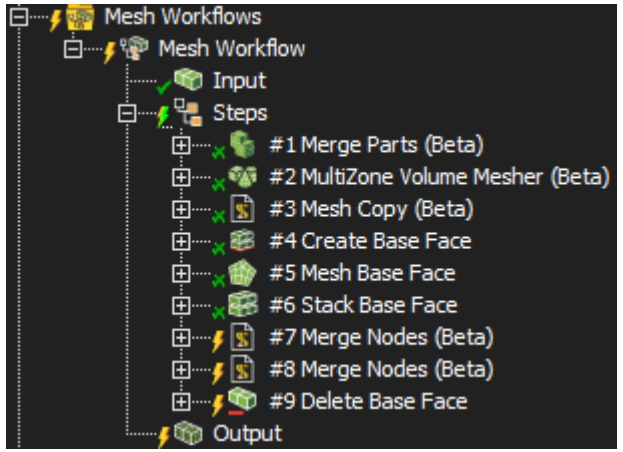
Workflow:

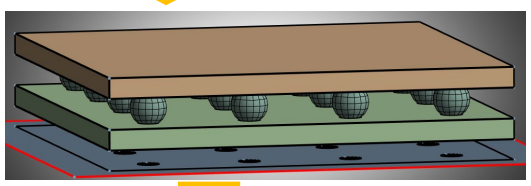
1. Generate mesh for primary solder ball (MultiZone)
2. Copy solder ball meshes to vectors in CSV file
3. Mesh boards with solder ball contacts – i.e. Stacker mesh with seeded faces from (2)
4. Merge nodes between solder balls and boards

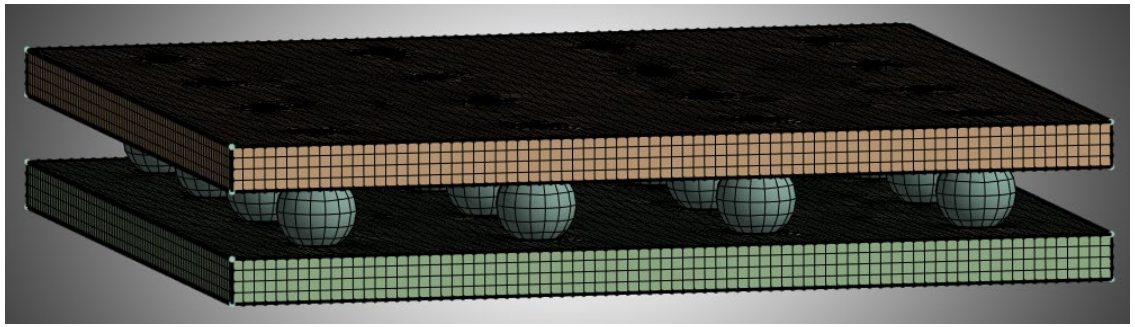




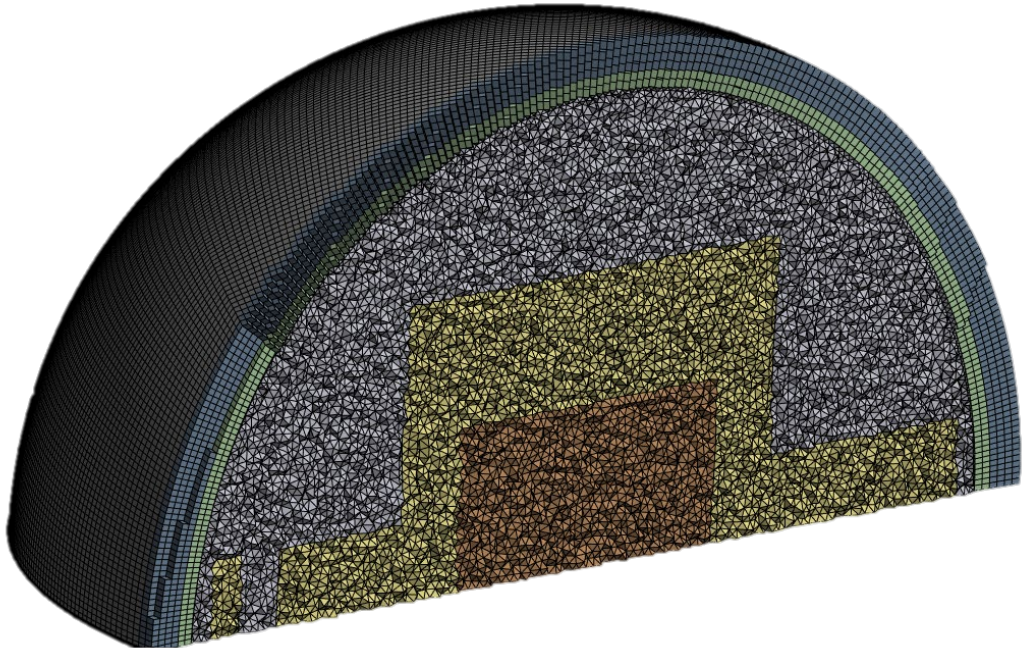
X_Offset	Y_Offset	Z_Offset	Unit
0.01	0.00	0.00	M
0.02	0.00	0.00	M
0.03	0.00	0.00	M
0.03	0.00	0.01	M
0.02	0.00	0.01	M
0.01	0.00	0.01	M
0.00	0.00	0.01	M
0.00	0.00	0.02	M
0.01	0.00	0.02	M
0.02	0.00	0.02	M
0.03	0.00	0.02	M
0.03	0.00	0.03	M
0.02	0.00	0.03	M
0.01	0.00	0.03	M
0.00	0.00	0.03	M



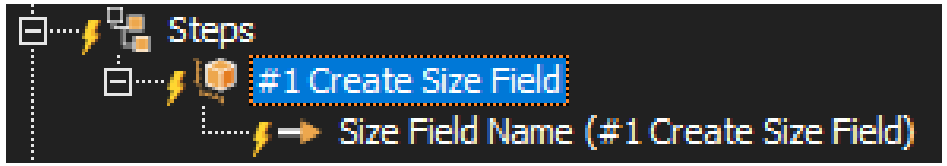


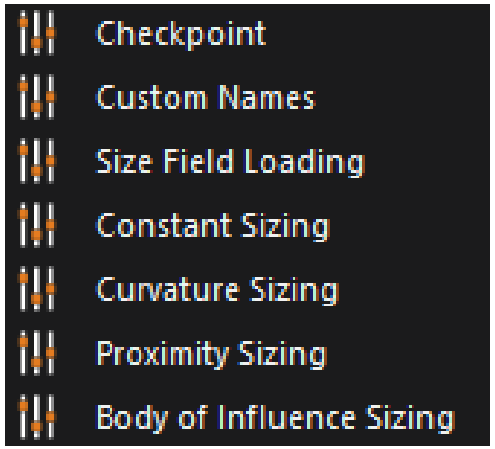


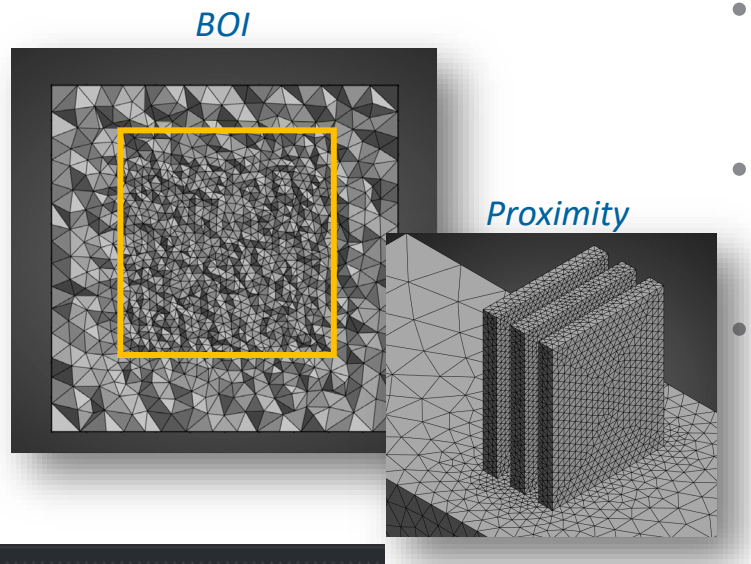
Mesh Workflows and NVH Meshing

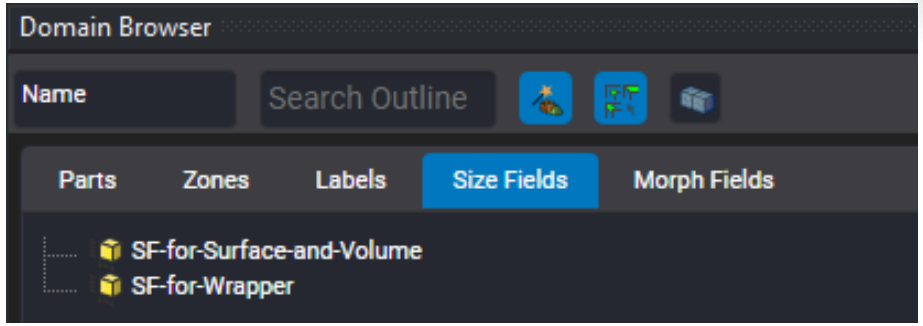


Workflow Step: Create Size Field (Released from Beta)



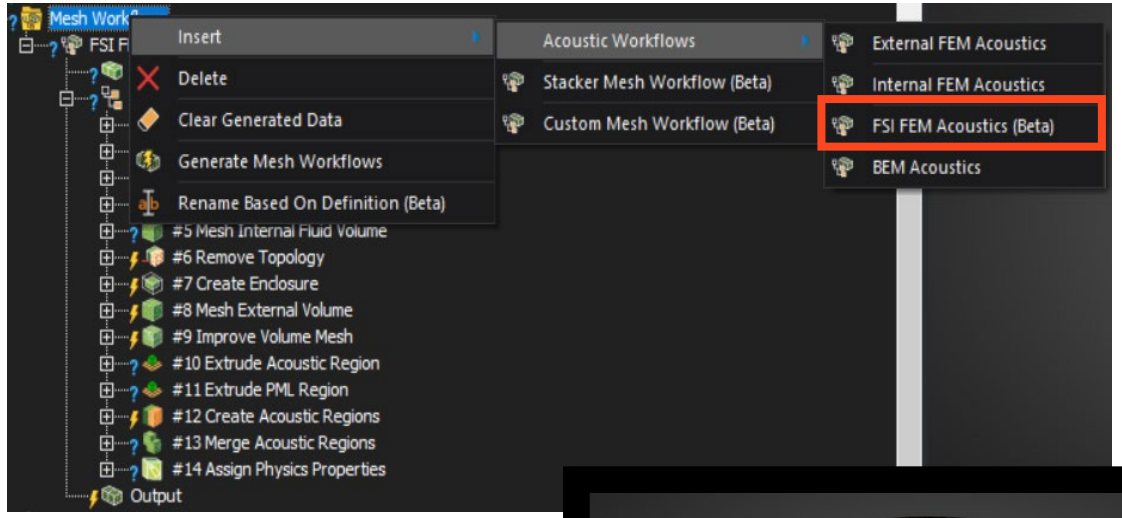


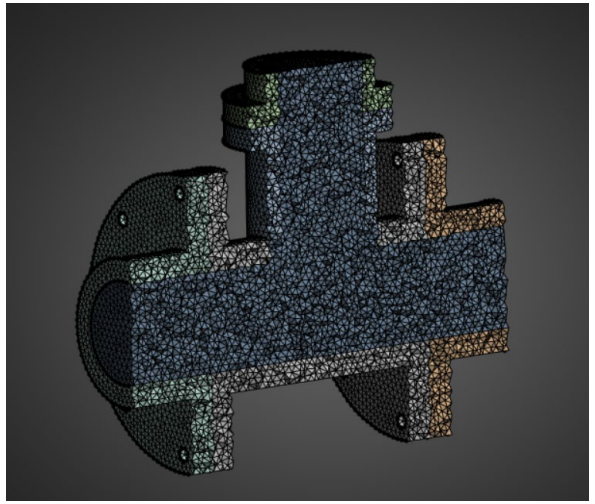


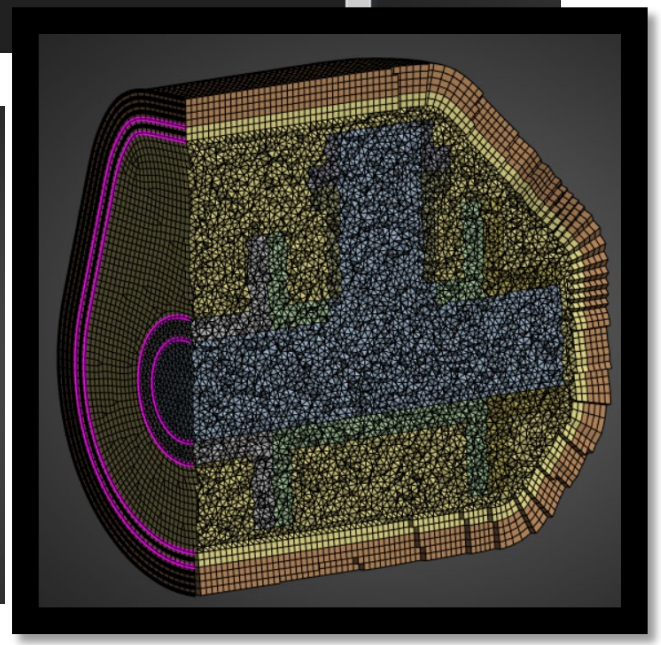


- Computes a volumetric size field as input into various meshing steps
- Supports sizing controls including: Constant Size, Curvature, Proximity, Body of Influence
- Insert Size Field outcome to obtain Size Field name after execution
- The meshers that support size field are
  - Size Field Wrapper
  - Wrapper Specific Surface Mesher
  - Size Field Surface Mesher
  - Size Field Volume Mesher
  - MultiZone Volume Mesher (β)
- Domain Browser is updated to see/pick Size Fields available

FSI FEM Acoustics Workflow (β)





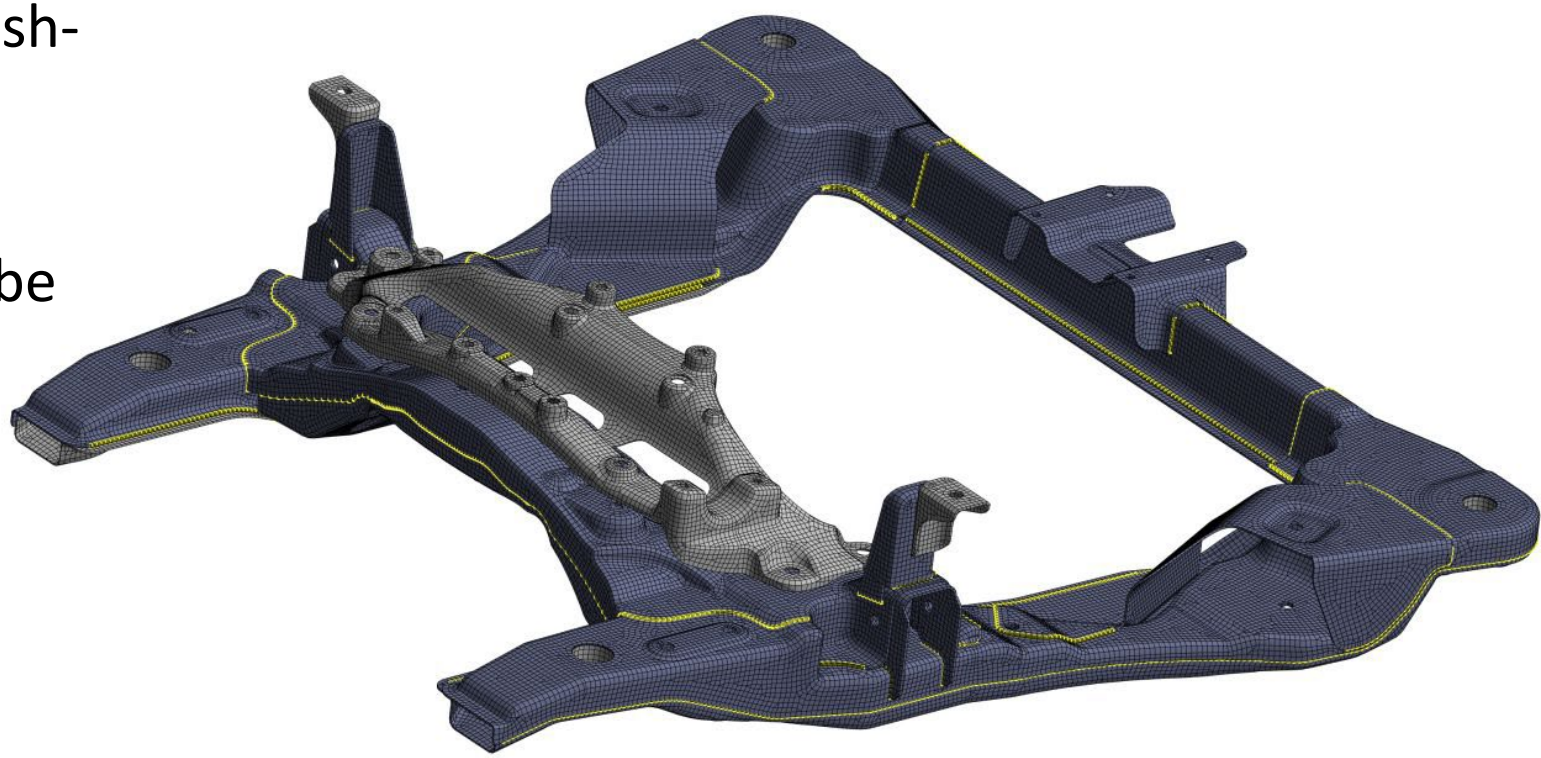


- FSI FEM Acoustics Template supports multibody part with shared topology (“watertight” geometry as input)
- Internal acoustic fluid regions are defined by material points and meshed under single operation
- Once structural parts mesh is generated, user can follow same procedure for External FEM Acoustics workflow
- More optional steps can be inserted to improve the robustness of surface&volume meshing (e.g. topology auto repair, improve surface mesh steps)

Welds and Shell Meshing

Shell Meshing with PrimeMesh

- Automatic (PrimeMesh) Method
  - Fast, high-performant, high-quality quad dominant mesher
  - Optionally, use with Connect (mesh-based share topology) and Weld controls
  - Optionally, scope solid bodies to be considered for Weld-based connections to Sheet bodies



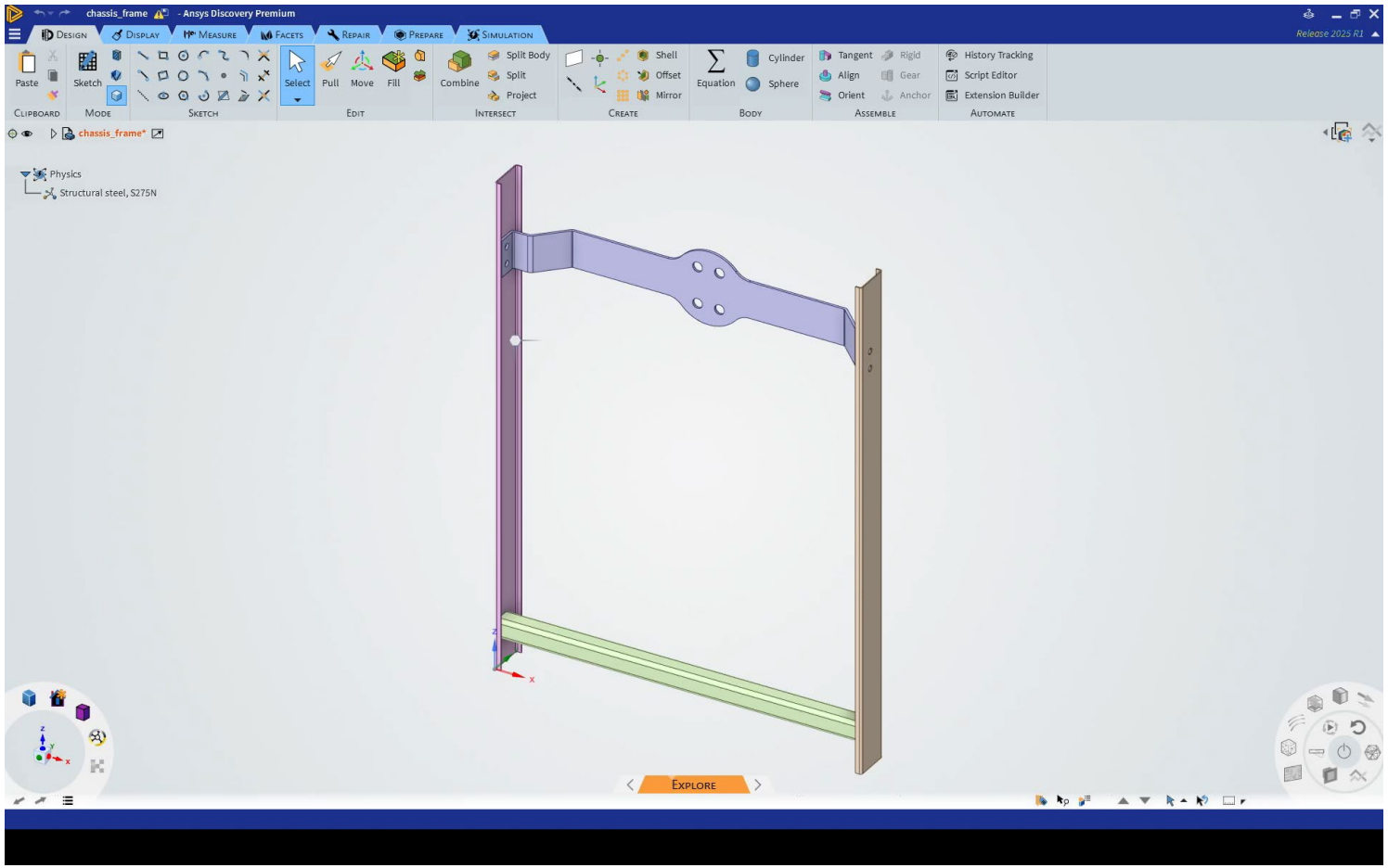
Objective

Streamline weldment analysis with a simple weld tool in Discovery Premium, coupled with ease of meshing in Mechanical Pro

Applicability

This workflow is applicable to sheet metal structures simulated with shell finite elements.

Pre-Recorded Demonstration



Benefits

1. Remove need to capture weld geometry directly in CAD
2. Remove complexity of shared topology
3. Allow highly parallelized meshing

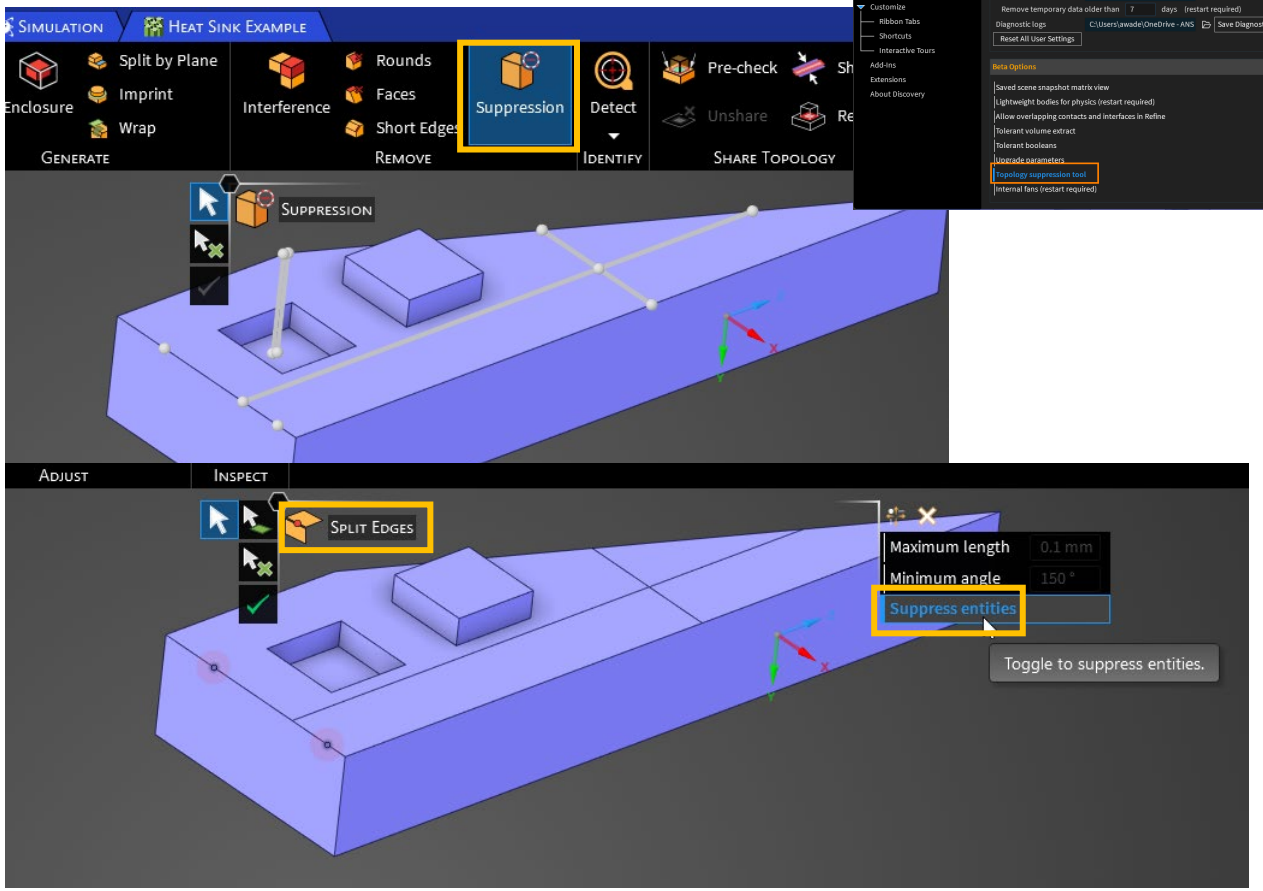
Batch Connections: Deprecated

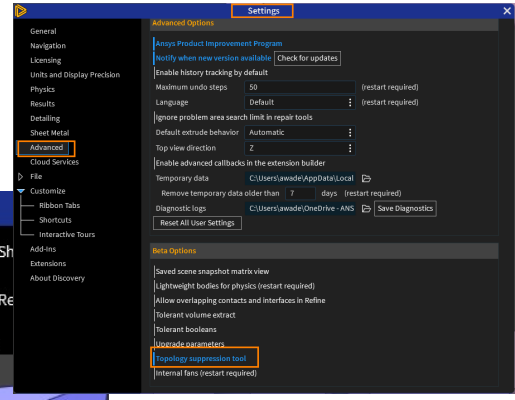
- Batch Connections is now deprecated and replaced with the Automatic (PrimeMesh) Method and Connect control
- Resumed legacy databases with Batch Connections Enabled will auto-update to having the Automatic (PrimeMesh) Method scoped to all bodies along with a Connect control
- Warning message is issued to users about this change

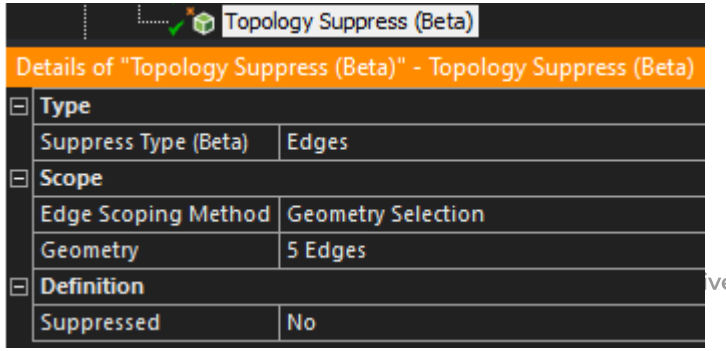
Hex and Map Meshing

Discovery Marking of Edges/Vertices to Suppress

- User can mark edges/vertices to suppress during MZ meshing manually or through the repair tools (e.g. Split Edges) in Discovery 25R1 (β)
- In Mechanical user can “Show Suppressible Edges & Vertices” (β) which will highlight and place these entities into NS group
- New Topology Suppress (β) control will allow suppression of these entities for MultiZone Algorithm

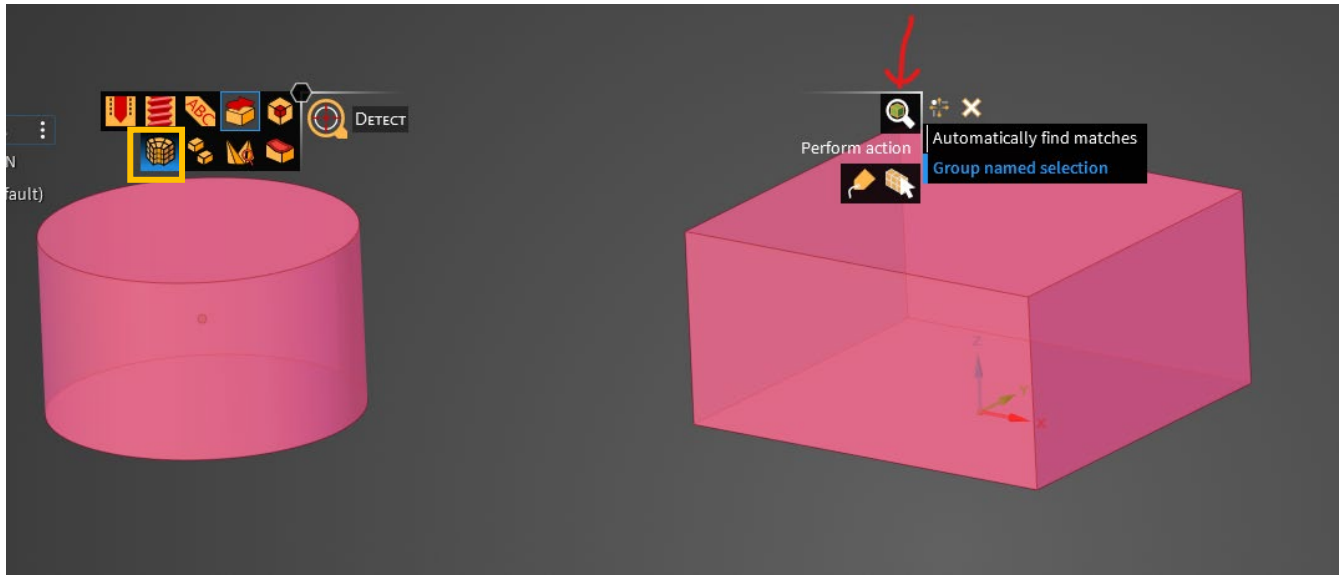






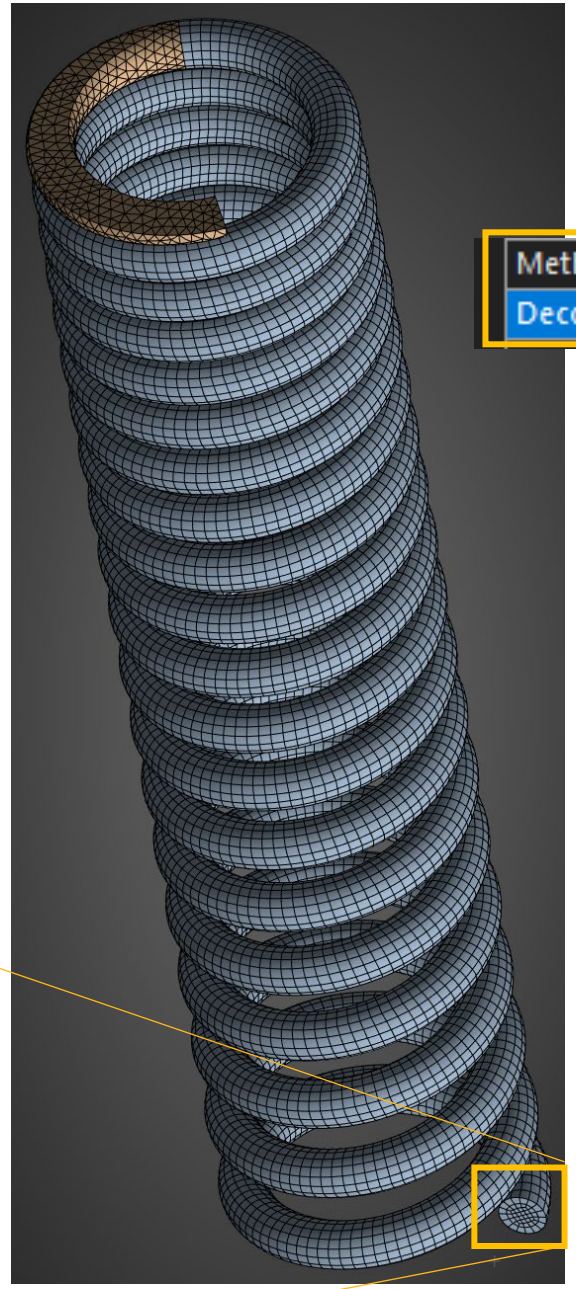
Discovery Detection of Sweepable Bodies

- Goal is to give users ability to slice and dice geometry and know with more certainly that they are hex meshable before moving to the Flagship product for hex meshing
- New tool first exposed under “Detect” to show sweepable bodies
- Planned: color-by-sweepable during decomposition for users to assist geometry preparation for hex meshing (based on feedback regarding performance etc)
- WIP with algorithm still improving
  - Feedback needed on performance for real-life models



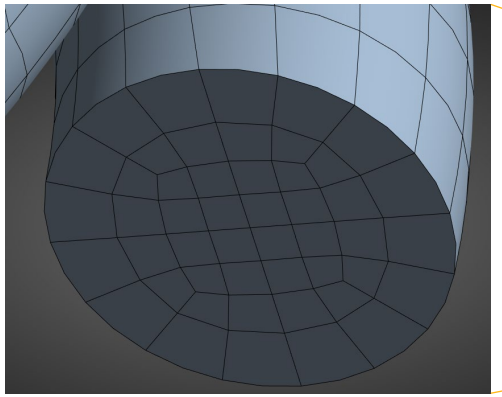
MultiZone Decomposition Type

- Medial Axis Decomposition
  - Released from Beta in 25R1
  - Decomposition method for sweeping along a complex path or for axi-symmetric bodies
  - Many robustness issues and limitations fixed
  - Support for inflation has been added
  - Support inside Prime for Mesh Workflows has also been added as β

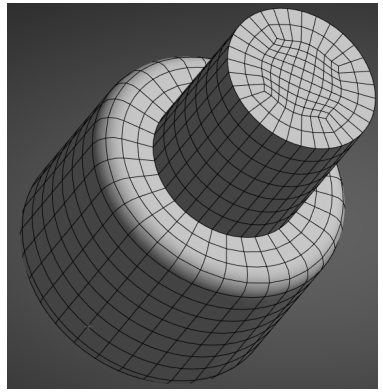


Method	MultiZone
Decomposition Type	Medial Axis

Face Meshing on Source



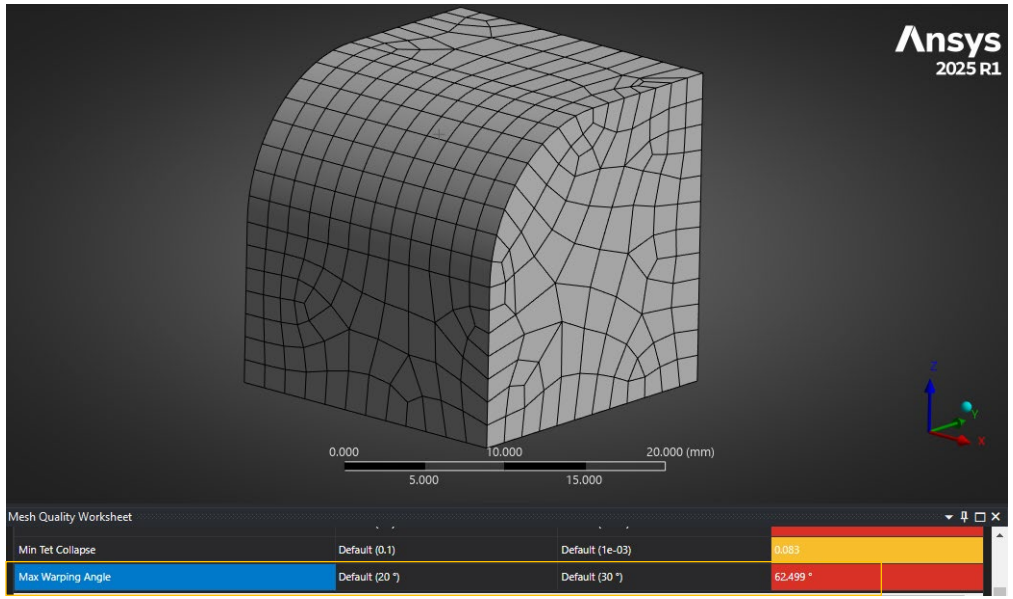
Axisymmetric Body

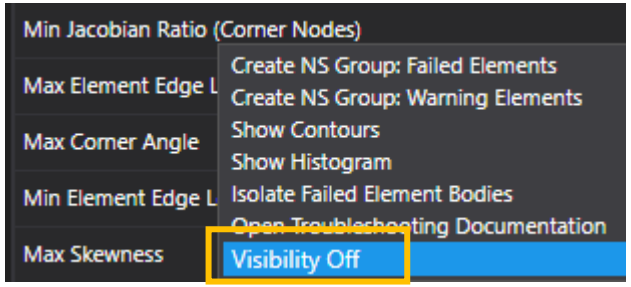


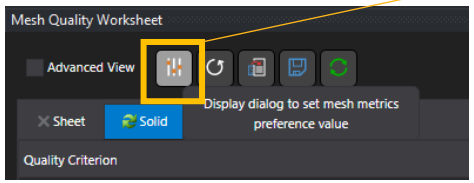
Usability, Automation and Performance

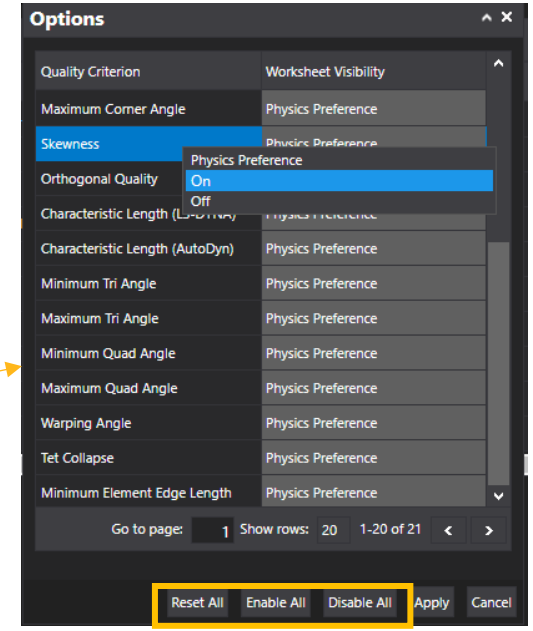
Quality and Quality Worksheet

- Changed name of Aspect Ratio (Explicit) to Aspect Ratio (Height)
  - Formulation was requested by Explicit simulation engineers but is generally applicable and is based on element height rather than shortest edge length
- Support added for Warping Angle for Solid elements with quad faces (hex, wedge, pyramid)
- Ability to quickly hide quality metrics of no interest by clicking then RMB → Visibility Off
  - This will be stored for future preference
- Ability to set visible quality criteria to default or set all as on/off quickly in the Options



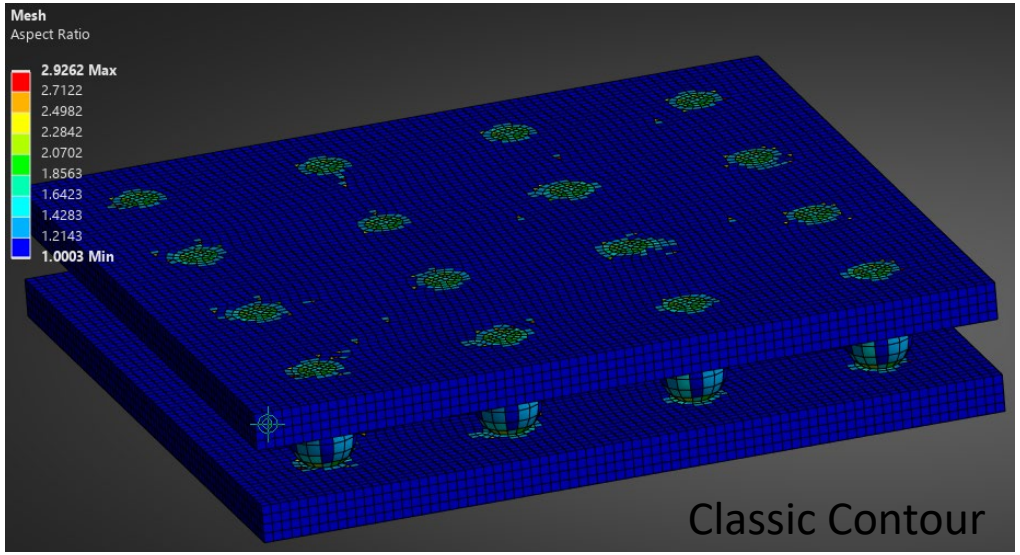


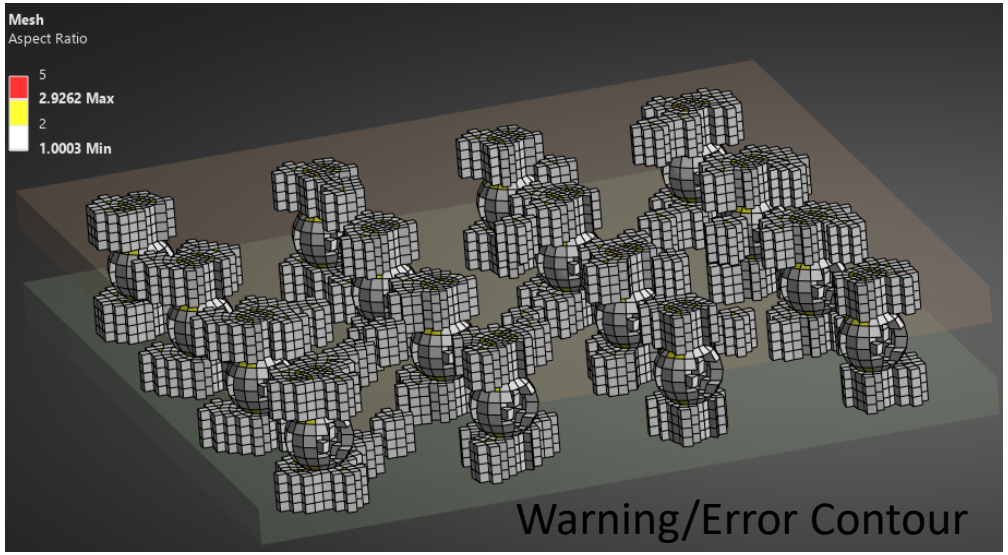




Quality Contour Plotting

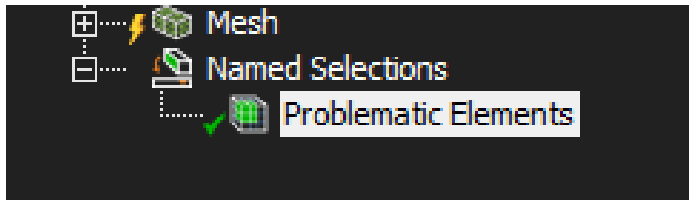
- Previously, the “Warning/Error” contours were only available when user sets “Check Mesh Quality” to Quality Worksheet
- New icon in Quality Worksheet to switch between “Classic” and “Warning/Error” contours





Failed Element Diagnostic from Messages

- When mesh fails due to shape checking i.e. quality is bad, error messages give a new RMB option to “**Show Problematic Elements**”
- Creates a NS group containing the elements that exceed documented error limits so user can take remedial actions



Problematic Elements

Generate

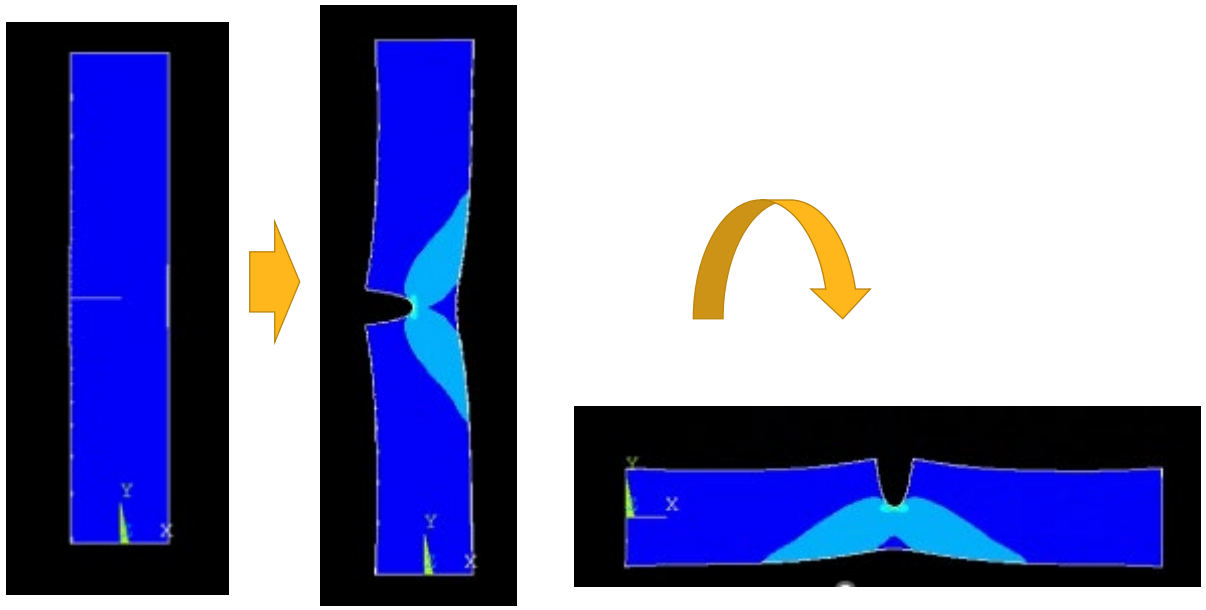
Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

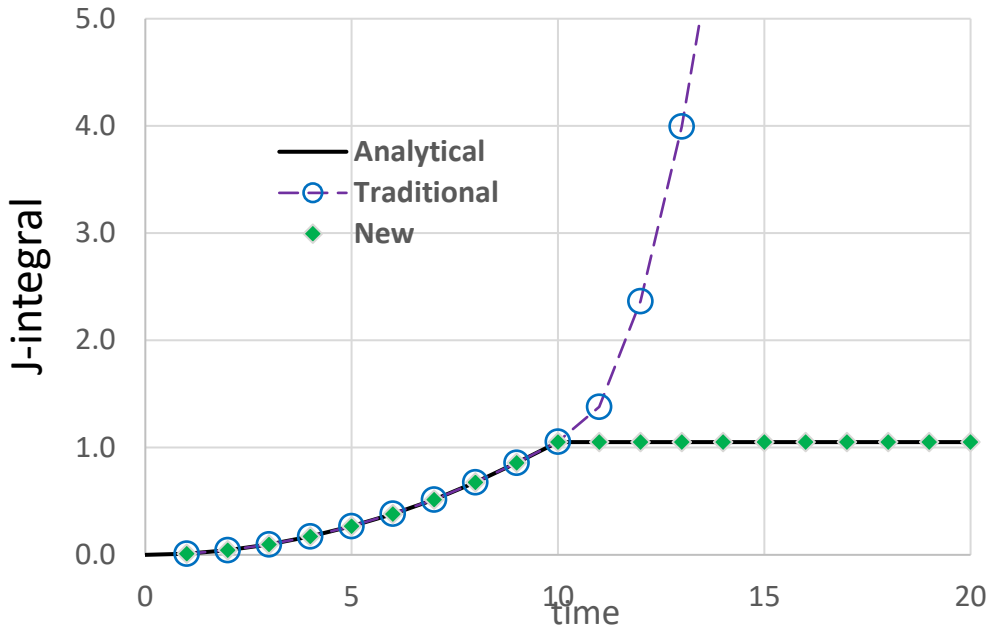
	Action	Entity Type	Criterion	Operator	Units	Value
☑	Add	Mesh Element	Element Quality	Less Than	N/A	2.e-002
☑	Add	Mesh Element	Jacobian Ratio (Corner ...	Less Than	N/A	2.5e-002
☑	Add	Mesh Element	Jacobian Ratio (Gauss ...	Less Than	N/A	2.5e-002

Fracture

J-integral calculation to support large deflection

- New J-integral formulation supports large deflection and eliminates limitation of small-strain deformation assumption





Benchmark problem: Single Edge Crack in

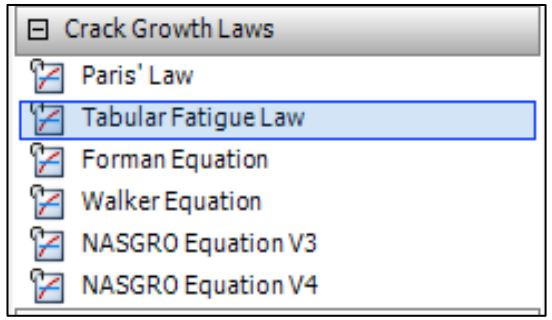
1. Firstly, purely pull the model in the Y-direction
2. Secondly, purely rotate the model with 90 degrees.

Comparison between traditional and new J-integral results

Fatigue Crack Growth Laws and Crack-Closure Functions Supported

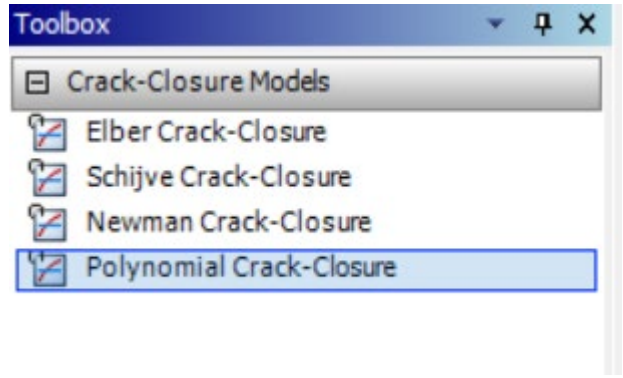
Mechanical now supports these Fatigue Crack Growth laws, in addition to Paris' Law:

- **Walker Equation**
- **Forman Equation**
- **Tabular Fatigue Law**
- **NASGRO Equation V3**
- **NASGRO Equation V4**



Paris' Law and the Tabular Fatigue Law, support these Crack-Closure functions:

- **Elber Crack-Closure**
- **Schijve Crack-Closure**
- **Newman Crack-Closure**
- **Polynomial Crack-Closure**

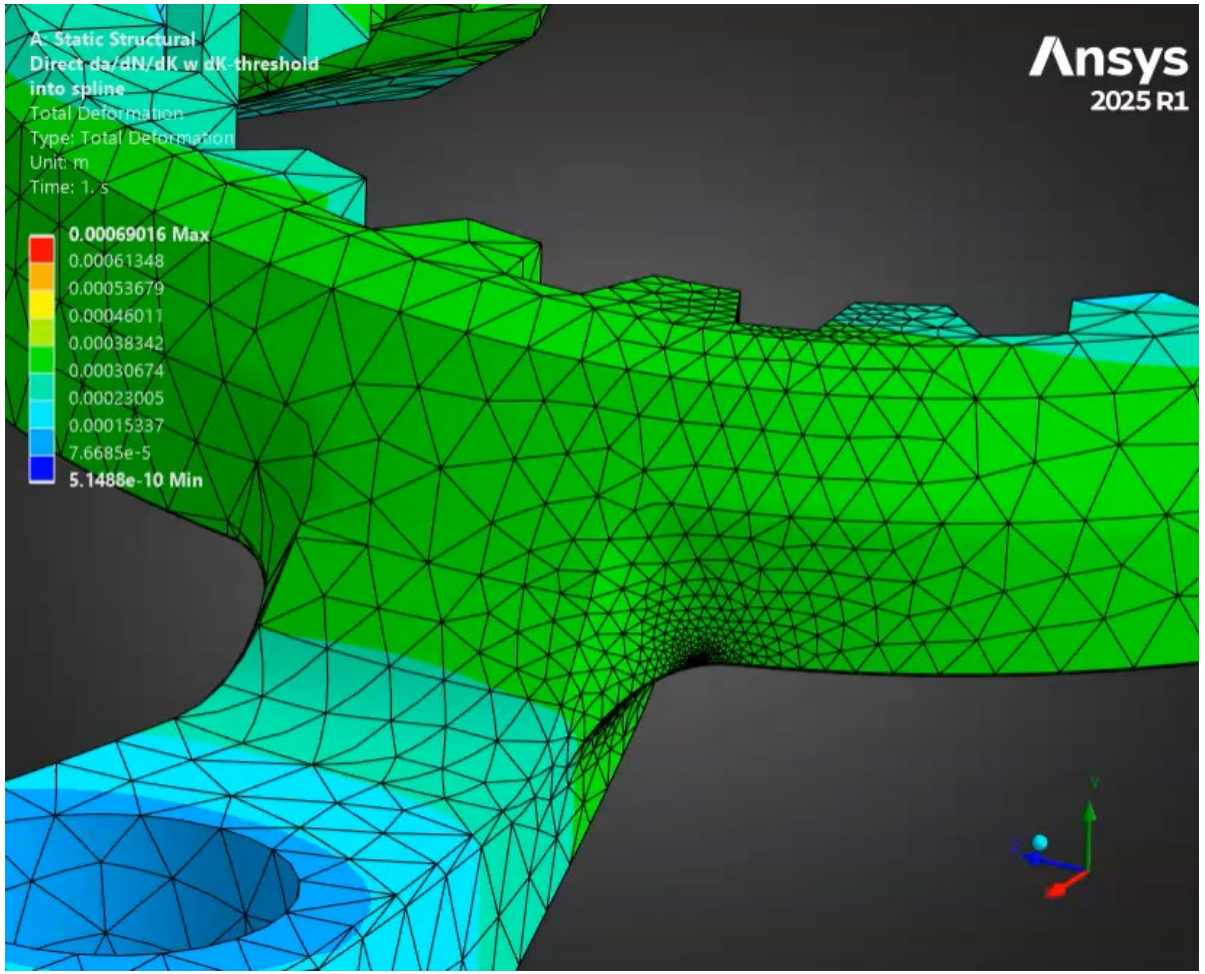


Tabular Crack Growth Input Example

13	Tabular Fatigue Law				
14	Crack Growth Constants	Tabular			
15	Scale	1			
16	Offset	0	m cycle^-1		

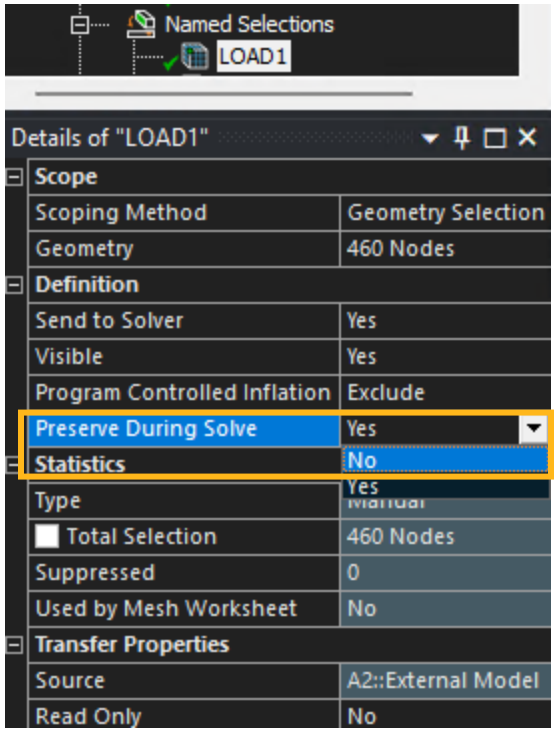
Table of Properties Row 13: Tabular Fatigue Law

	A	B
1	Stress Intensity Factor range (Pa m^0.5)	Crack Growth Rate (m cycle^-1)
2	8.12E+06	1E-10
3	8.13E+06	1E-09
4	8.3E+06	2E-09
5	8.55E+06	3E-09
6	8.8E+06	4E-09
7	9E+06	5E-09
8	9.22E+06	6E-09
9	9.4E+06	7E-09
10	9.7E+06	1E-08
11	1.06E+07	2E-08
12	1.21E+07	4E-08
13	1.34E+07	6E-08



Restart analysis support with SMART Crack Growth

- Restart analysis is now supported with SMART Crack Growth
- Tabular Temperature loads are also supported
- During restart analysis with SMART Crack Growth, any load changes must be mapped to the re-meshed model.
- To achieve this, a property called **Preserve During Solve** is exposed on the nodal named selections like Element based named selections. Component is updated with new mesh
- Pressure, Force and Displacements loads should be scoped to nodal named selections with this Preserve During Solve set to Yes, for multiple load step analysis, if the load is modified/de-activated in follow on load steps

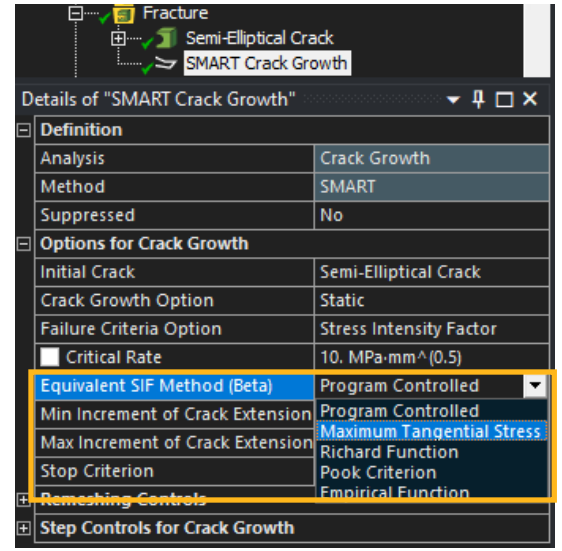


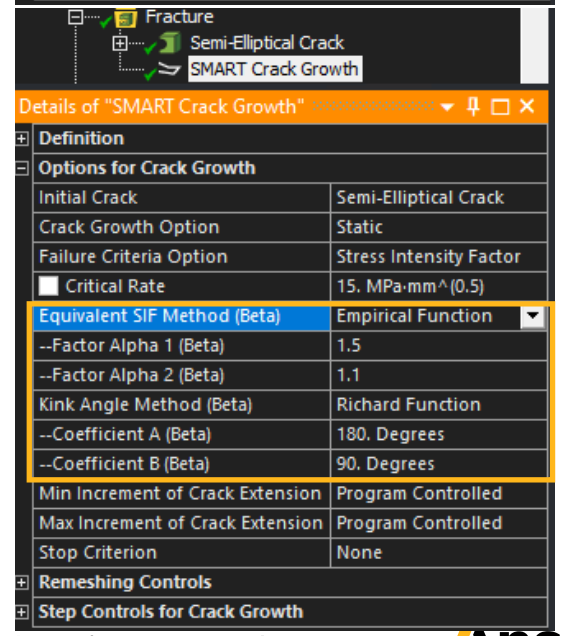
Mixed mode crack growth based on Equivalent SIF Calculation (Beta)

- For a mixed mode fracture analysis, it is important to include SIFS(K3) in fracture calculations
- Based on literature, MAPDL provides multiple methods to include SIFS(K3) and calculate Equivalent SIF. Available methods: **Maximum Tangential Stress, Richard Function, Pook Criterion and Empirical Function**
- Equivalent SIF Method (Beta) and Kink Angle Method (Beta) properties are exposed for SMART Crack Growth object

- **Program Controlled option uses the MAPDL default Maximum Tangential Stress option, which does not consider SIFS(K3) in calculations**

  All the other three methods consider SIFS(K3) in calculations



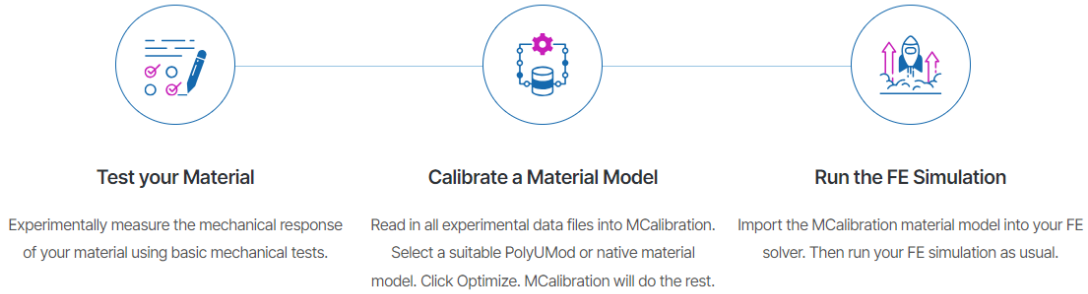


Materials

PolymerFEM Product Offerings

Ansys MCalibration

- Best in class **material model** selection and calibration tool
- Standalone, solver agonistic
- Helps adoption of complex advanced material models by streamlining the material characterization and qualification

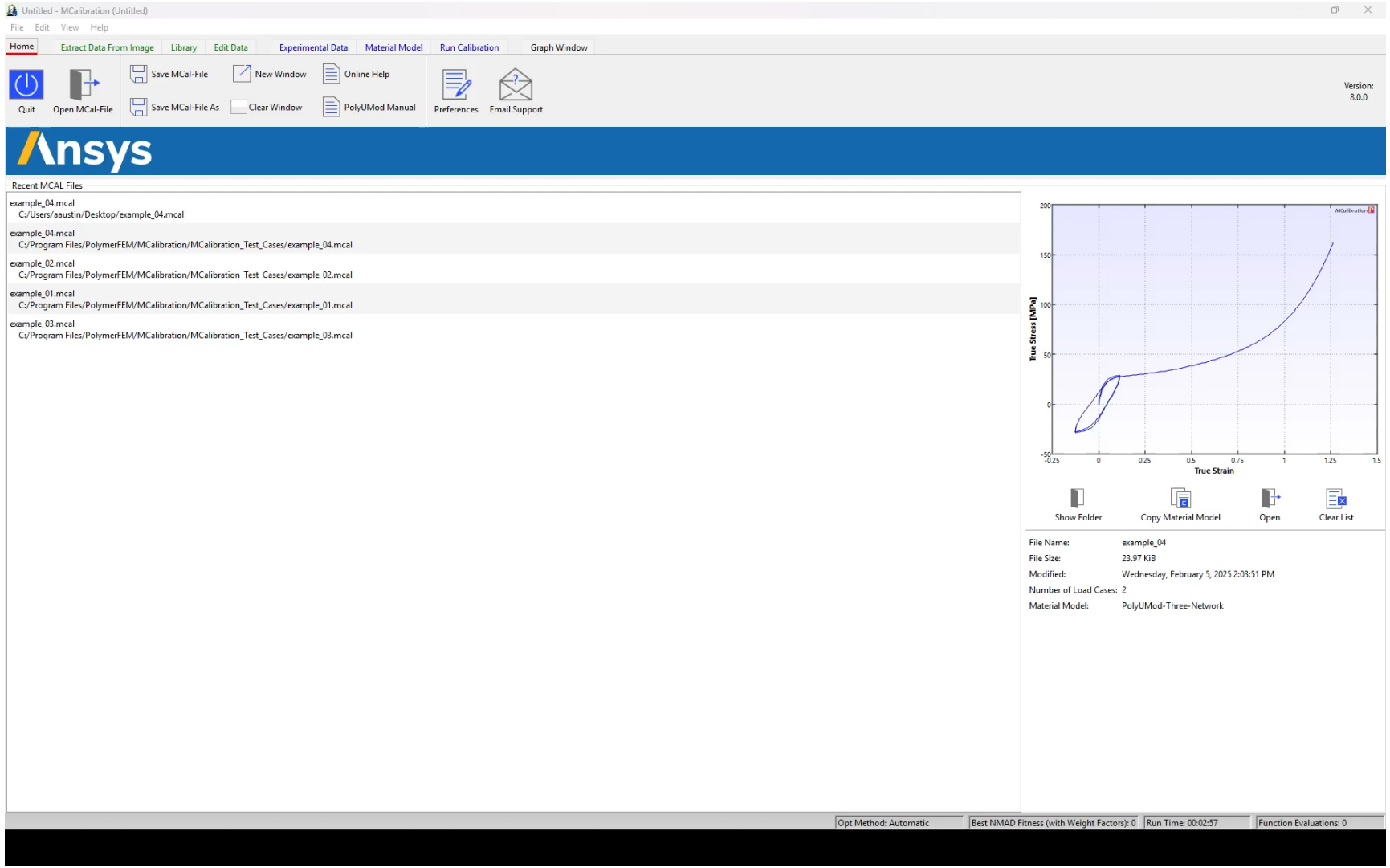


Ansys PolyUMod

- Extension of FE solvers by adding advanced user-material models for polymers, biomaterials and other non-linear materials.

Because of excessive flexibility, polymers need to be modeled with advanced analysis
(Mechanical Enterprise)

PolymerFEM: MCalibration



Noise, Vibration, Harshness (NVH)

NVH Toolkit

NVH Toolkit - MAC Calculator

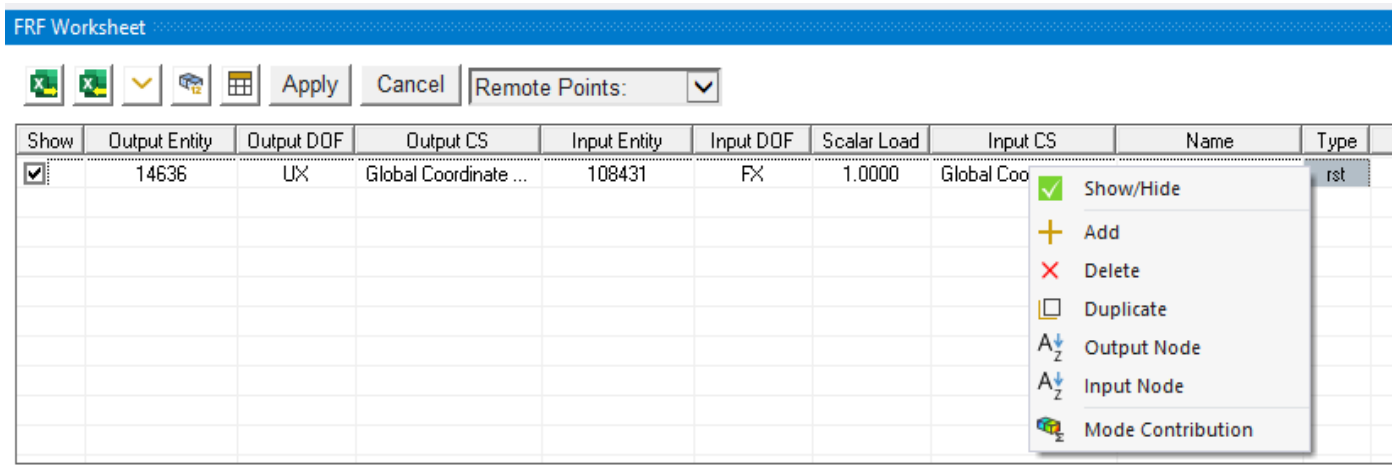
- Allow to use distributed rst files for second file.

This enhancement allows to select distributed files for MAC Calculation, which avoids merging distributed rst files. For extreme cases, where there are files of 500Gb you don't even have space in the computer to merge the distributed files, as you need to have both set of files at the same time.

- Improvements of performance of MAC Calculator calculations.
  - Matching algorithm is now 3 times faster.
  - Computing the MAC Table is now 2 times faster.

NVH Toolkit - FRF Calculator – Mode Contribution (1/2)

- Added Mode Contribution feature.
  You can now extract the mode contribution from the FRF Calculator object, by RMB in the FRF Worksheet:



- This feature allows you to understand the contribution of each mode shape of a structure to its overall dynamic reponse, and hence, allowing you to select only those that are more important to the FRF analysis. This concept is crucial because not all modes are equally significant in influencing the behavior of the structure under dynamic loading conditions

Linear Dynamics

Displacement and Velocity IC in MSUP Transient

- During Large Mass Method analysis with MSUP transient, drift in displacement is observed, when subjected to periodic loading at the base; required Displacement and velocity ICs in MSUP transient
- IC in MSUP helps us to get accurate responses for problems involving IC, just like IC in Full transient.

If displacement initial conditions are specified, they are transferred to the modal coordinates initial displacement, y_j^0, using:

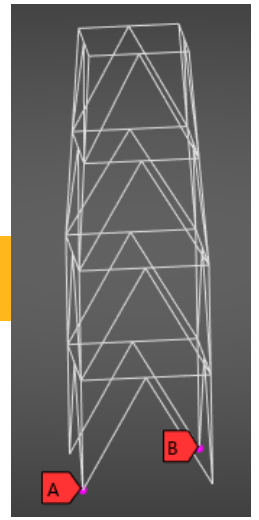
$$y_j^0 = \{\phi_j\}^T [M] \{u_0\} \quad \text{(15–41)}$$

If velocity initial conditions are specified, they are transferred to the modal coordinates initial velocity, $\dot{y}_j^0$ using:

$$\dot{y}_j^0 = \{\phi_j\}^T [M] \{\dot{u}_0\} \quad \text{(15–42)}$$

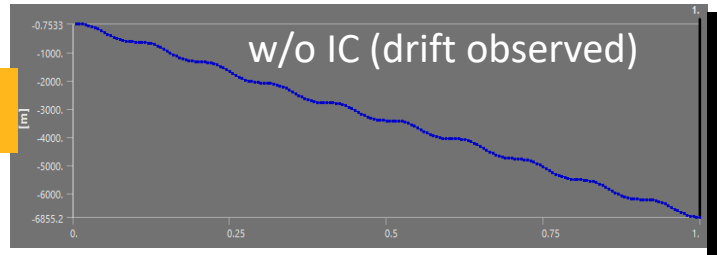
Truss

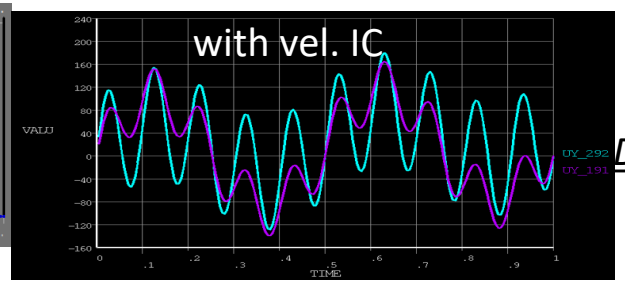
e.g.1



Truss

e.g.1

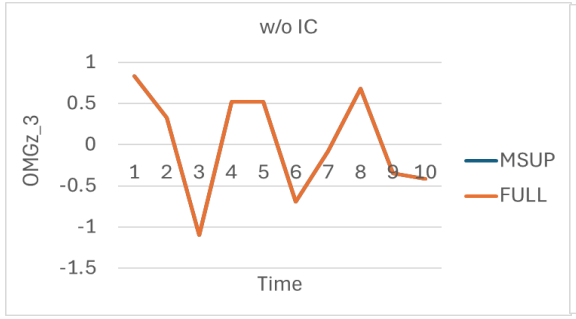


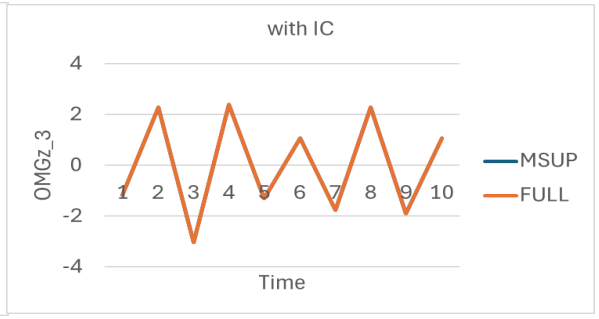


Drift eliminated with IC

Pendulum

e.g.2

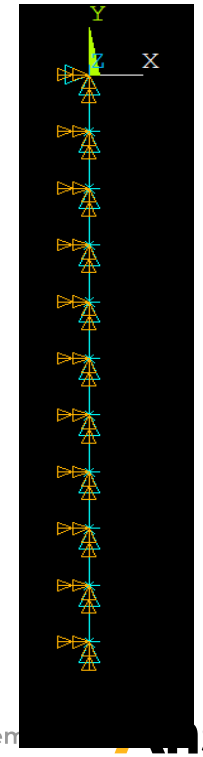




MSUP-IC matching exactly with Full-IC

Pendulum (including MASS21)

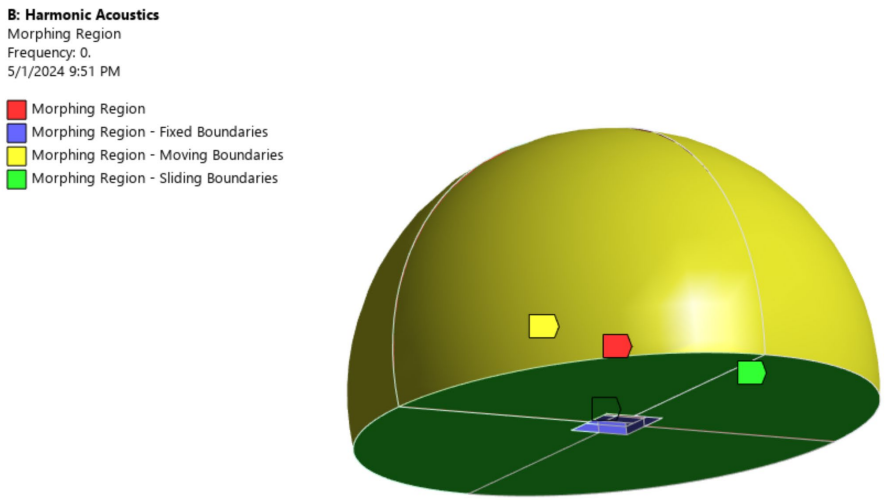
e.g.2



Acoustics

Morphing – Sliding Constraint

- Morphing Region now allows to define a sliding plane to constraint the nodes to lie on a plane in order to take into account the ground or a symmetry



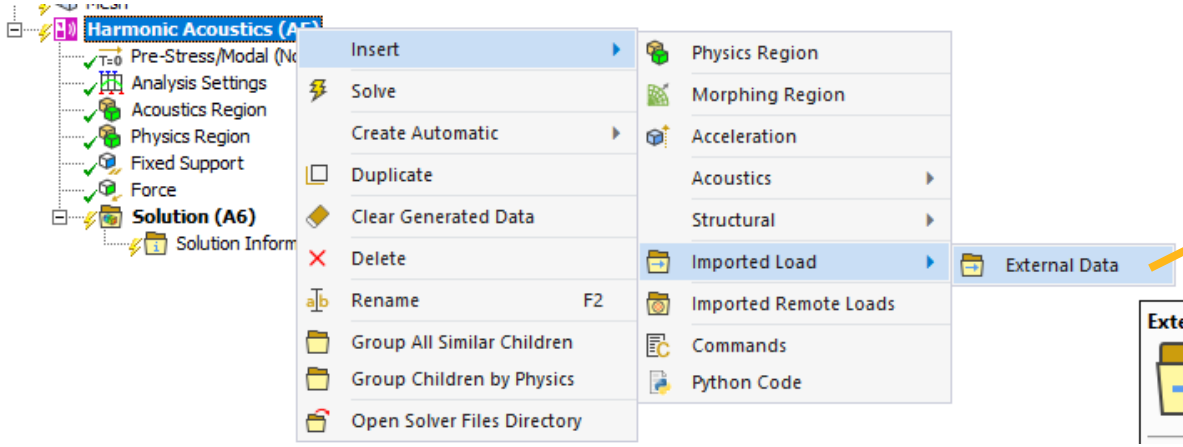
Details of "Morphing Region"

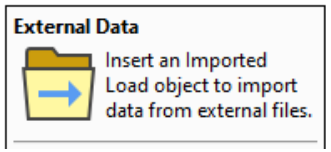
Morphing Region	
Scoping Method	Geometry Selection
Geometry	4 Bodies
Fixed Boundaries	
Scoping Method	Geometry Selection
Geometry	198 Faces
Moving Boundaries	
Scoping Method	Geometry Selection
Geometry	4 Faces
Base Mesh Parameters	
Base Frequency	500. Hz
Morphing Region Thickness	0.4 m
Morphing Parameters	
Minimum Frequency	Minimum Frequency
Maximum Frequency	Maximum Frequency
Morphing Intervals	Each Frequency
Sliding Constraints	
Sliding Constraints	Yes
Scoping Method	Geometry Selection
Geometry	4 Faces

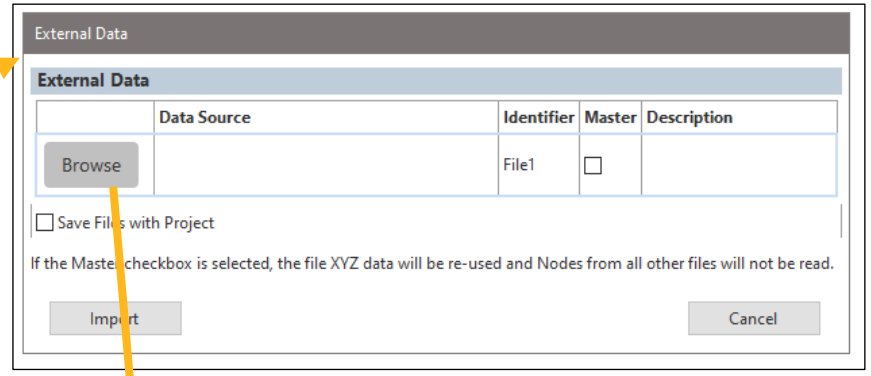
Vibro-Acoustics Mapping

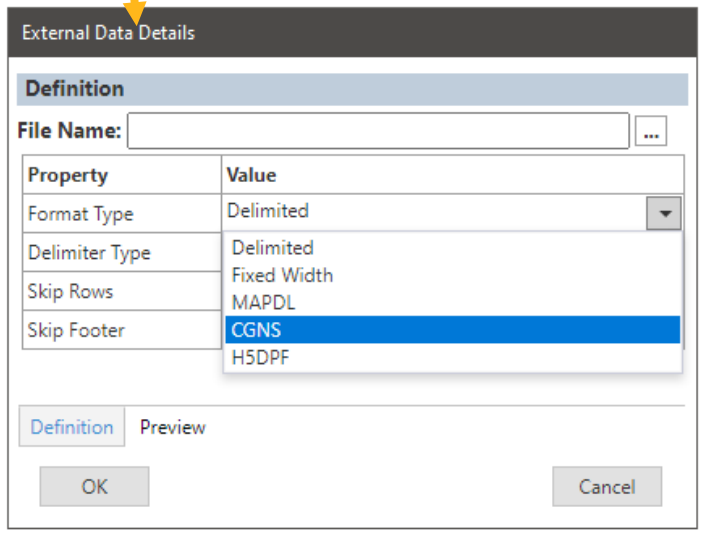
Imported CFD Pressure via External Data

- In 2025 R1, **Imported CFD Pressure** load is supported via **External Data** option under **External Load** group

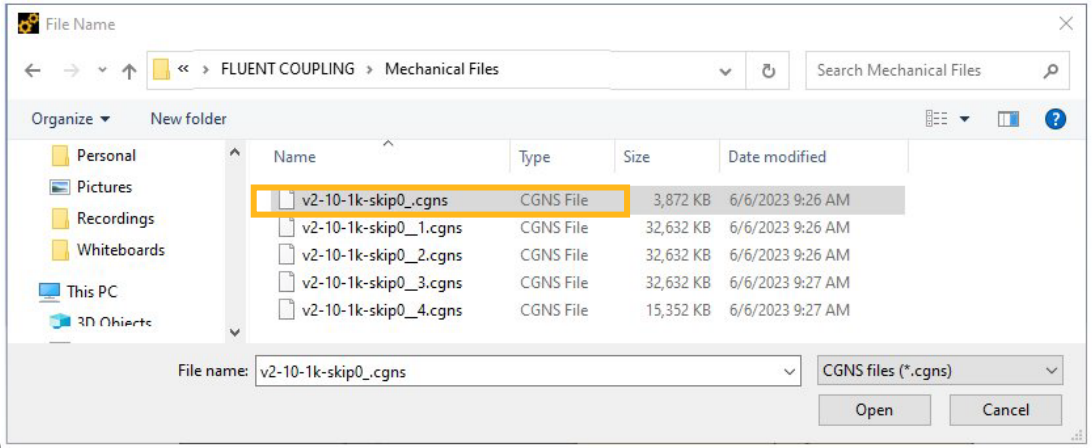








Important:
If the CFD pressure data available as multiple CGNS files, just select the master cgns file only. Internally Mechanical will identify all the relevant dependent files



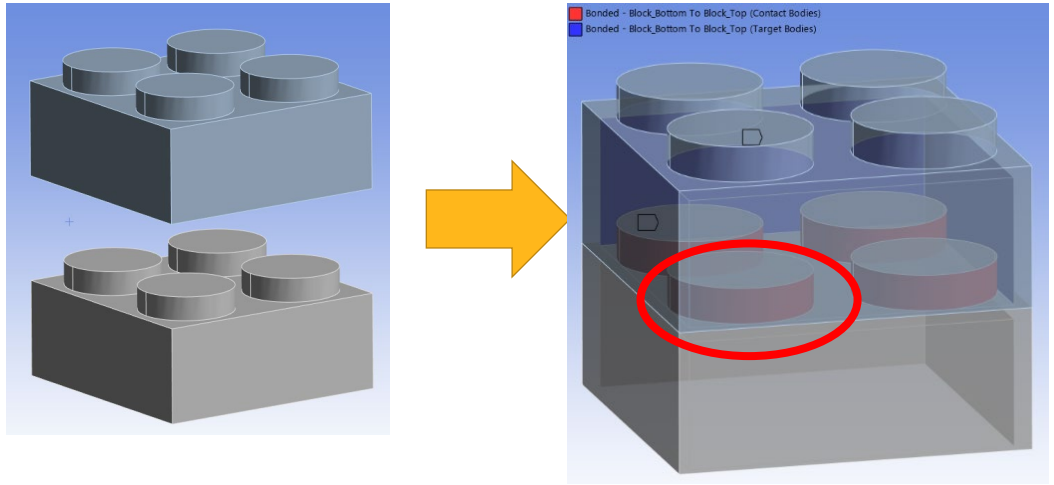
Multibody Dynamics (Motion)

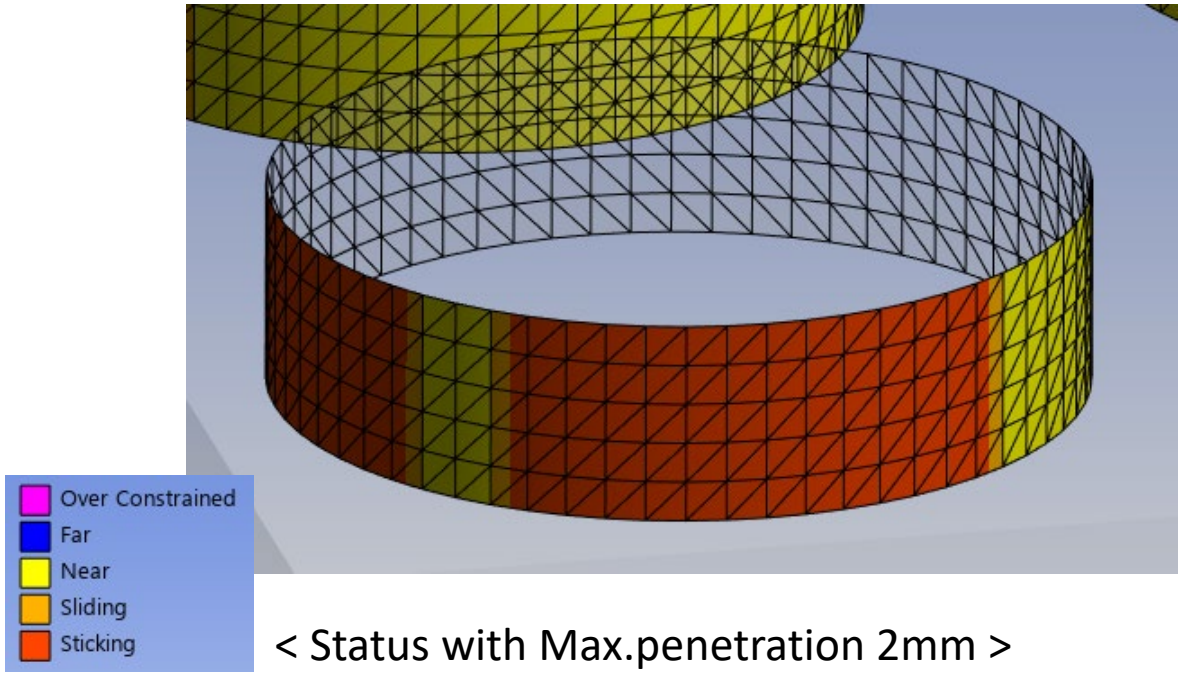
Common – Solver, Postprocessor

Pure part – Mechanical Motion / STD preprocessor

Contact output for bonded Contact

- Several output option has been added for bonded type contact. (Tie contact)
  - Contour(Both), Vector display and Plot(STD post) are available.
  - The contour pattern indicates where bonded-contacts are being applied.

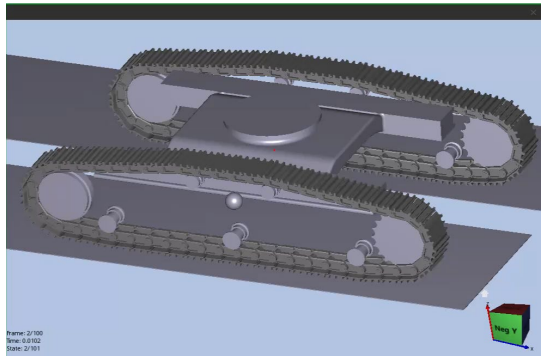




< Status with Max.penetration 2mm >

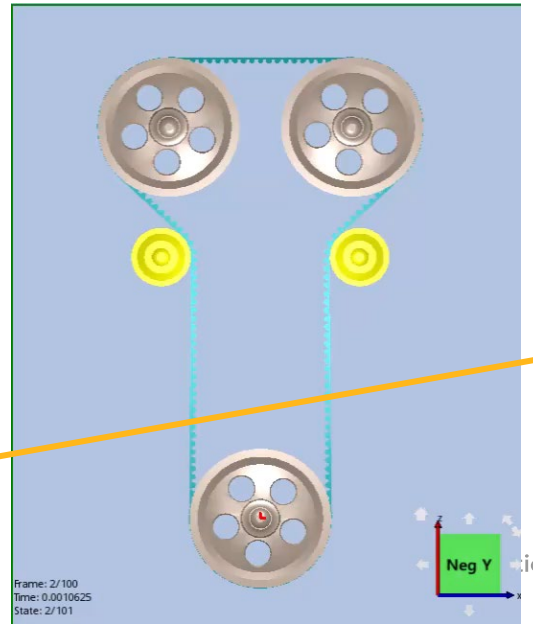
Improve performance with multiple contacts

- The contact search algorithm for entities with rare contact has been enhanced by optimizing the multi-thread strategy.
  - While 24R2 has improved memory usage, it has negatively impacted simulation performance.
  - A model, where most entities are in a non-contact condition and rarely contacted, such as the link system, has been significantly improved.



	1 Thread	8 Thread	Ratio (1th / 8th)
V241	50.41	12.60	4.00
V242	18.74	152.01	0.12
V251	17.42	7.75	2.25

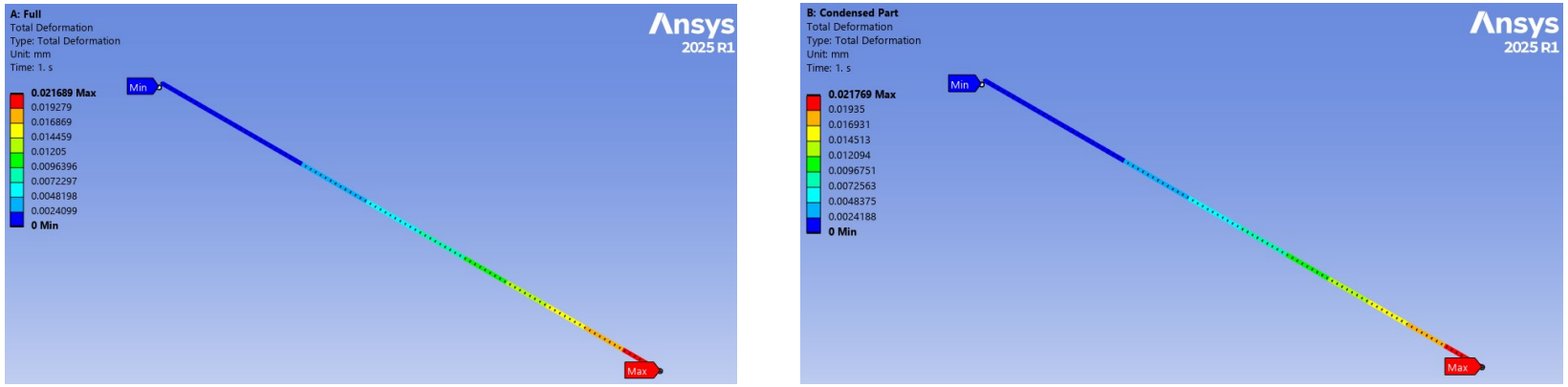
Higher is better!



	1 Thread	8 Thread	Ratio (1th / 8th)
V241	76.69	65.56	1.17
V242	43.80	134.51	0.33
V251	45.11	21.46	2.10

Support Beam bodies in Condensed Parts

- Condensed Parts and Imported Condensed Parts now support Beam bodies with a circular cross-section in Motion analysis.
- For Beam bodies used in Condensed Parts, only Deformation, Velocity, Acceleration, and Equivalent (von-Mises) Stress & Strain results are supported.

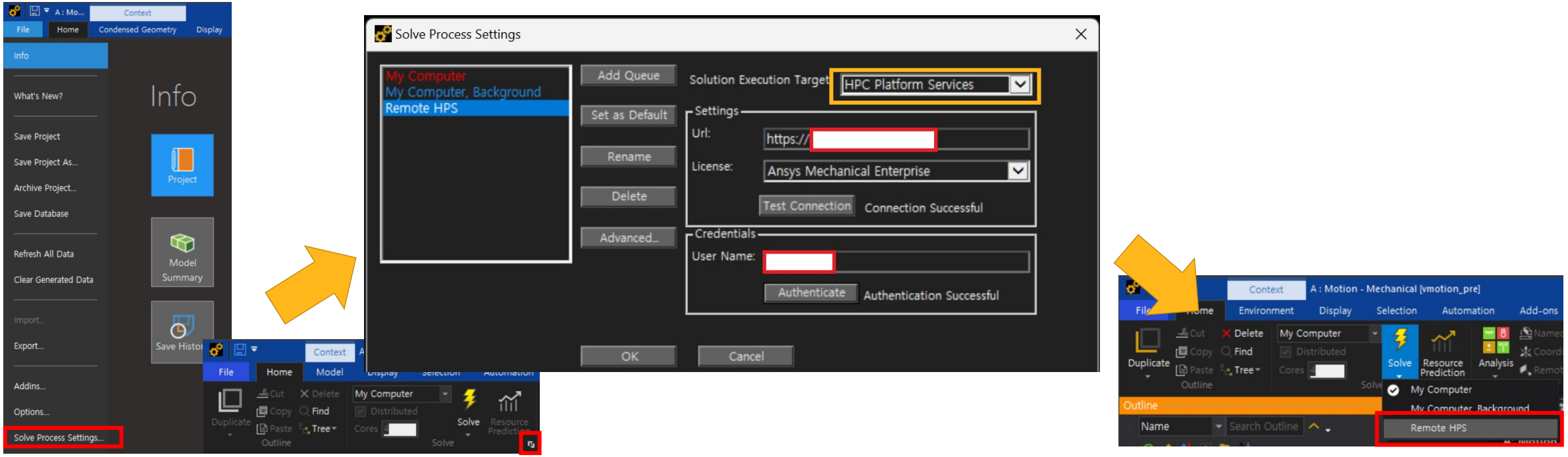


< Solution comparison between Nodal FE (left) & Condensed part (right) >

Remark : Directional stress is not available.

HPS Support for Motion

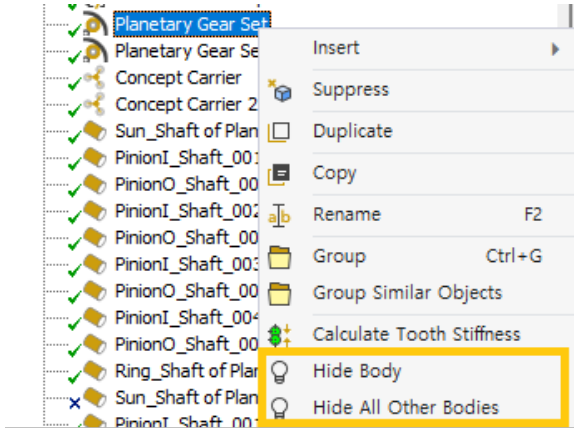
- HPC Platform Services (HPS) enables you to solve Mechanical Motion analysis. (Beta to official)
  - Solving jobs can be submitted to the remote machine once the configuration of the HPS web server is set.



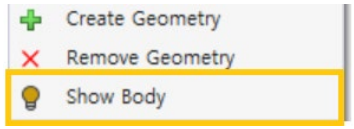
Show & Hide Option for CAD Geometry Import Objects

- Added “Show” & “Hide” option for CAD geometry at the environment level.
- Improves usability in large models by simplifying the visibility management of imported geometry.
- Applicable to specific elements like those in DT, Links, and Car(tire) toolkits.

DT – Gear/Shaft/Bearing

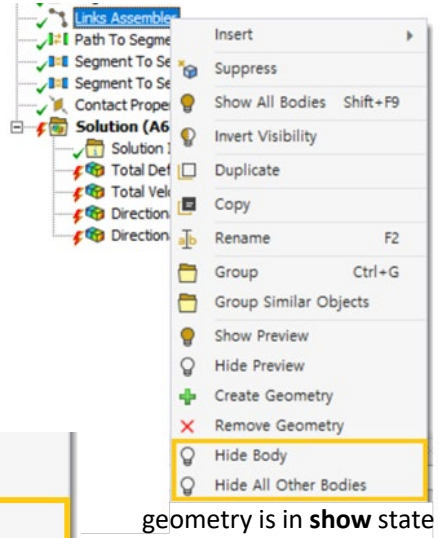


geometry is in **show** state

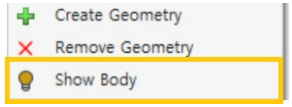


geometry is in **hide** state

Links

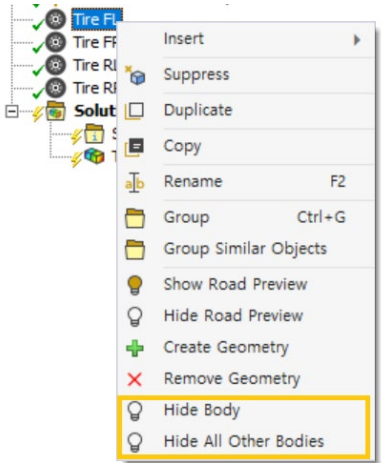


geometry is in **show** state

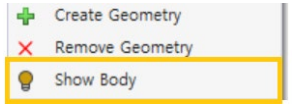


geometry is in **hide** state

Car(Tire)



geometry is in **show** state



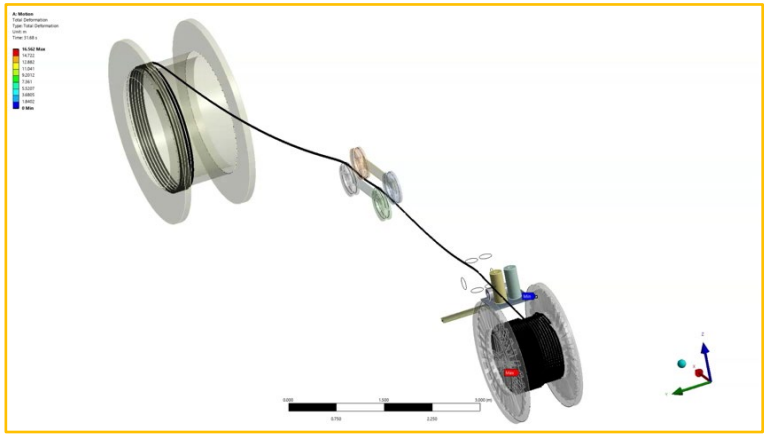
geometry is in **hide** state

Cable Elements (Beta)

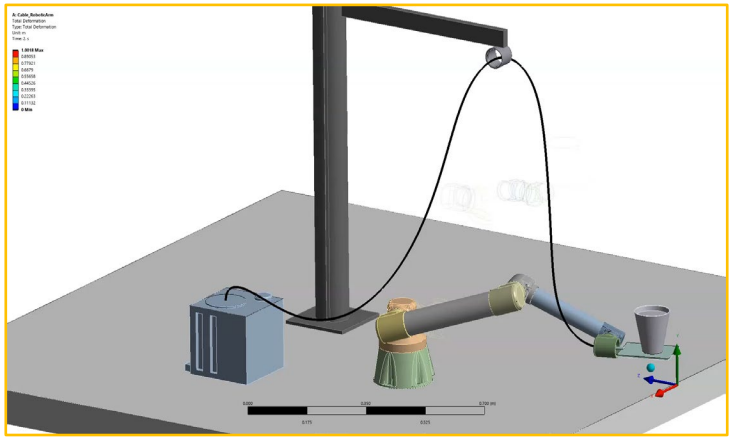
- The “Cable” Model Type for line bodies is now available.
- Key Capabilities
  - Beam type flexibility with stiffness scaling
  - High efficiency contact algorithm based on circular geometry.
  - Prestress in bending direction

Details of "SYS\Beam (Circle)"

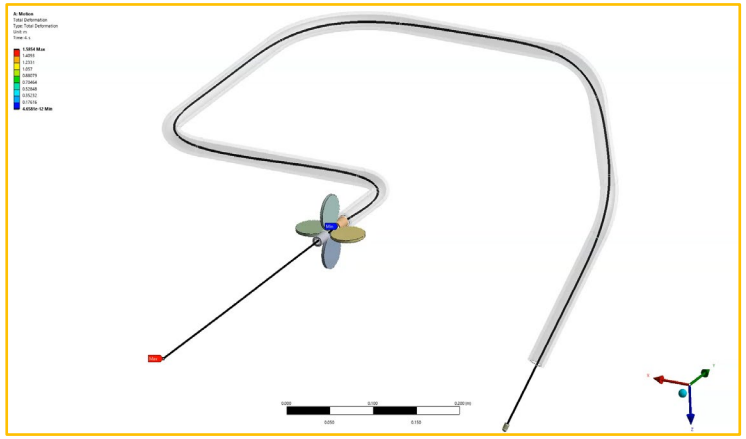
Graphics Properties	
Definition	
Suppressed	No
Model Type	Cable
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Cross Section	Circle
Treatment	None
Cable Definition (Motion) (Beta)	
Synchronize Free Length with Geometry (Beta)	No
-- Free Length (Beta)	0.16 m
Use Stiffness Scale Factor (Beta)	Yes
-- Bar Scale Factor (Beta)	1.
-- Bending Scale Factor (Beta)	1.
-- Torsional Scale Factor (Beta)	1.



< Winding>



< Robot Arm>

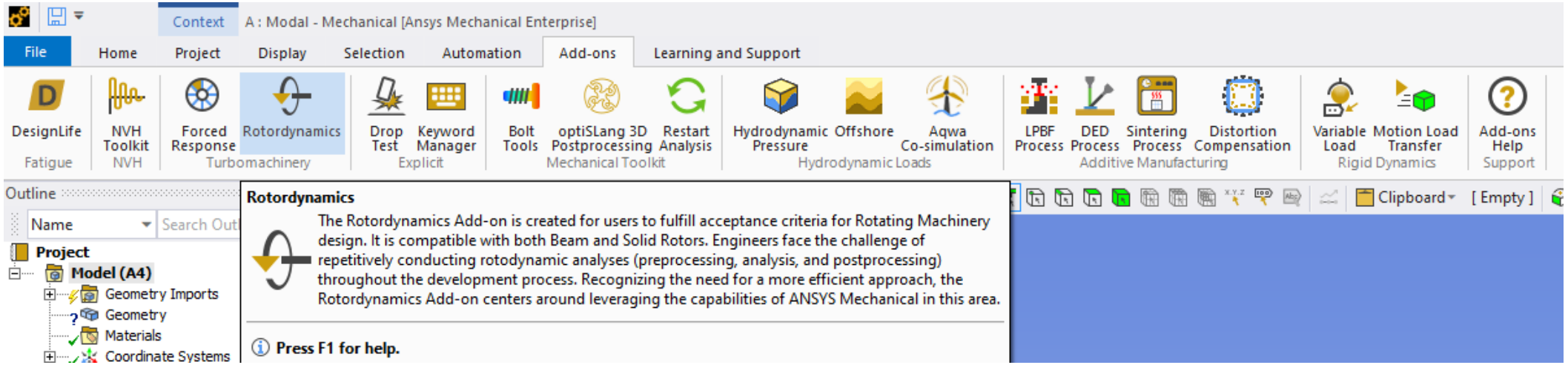


<Tube>

RotorDynamics Add-on

New : Rotordynamics Addon

The Rotordynamics Add-on is created for users to fulfill acceptance criteria for Rotating Machinery design. It is compatible with both Beam and Solid Rotors.



DesignLife Add-on

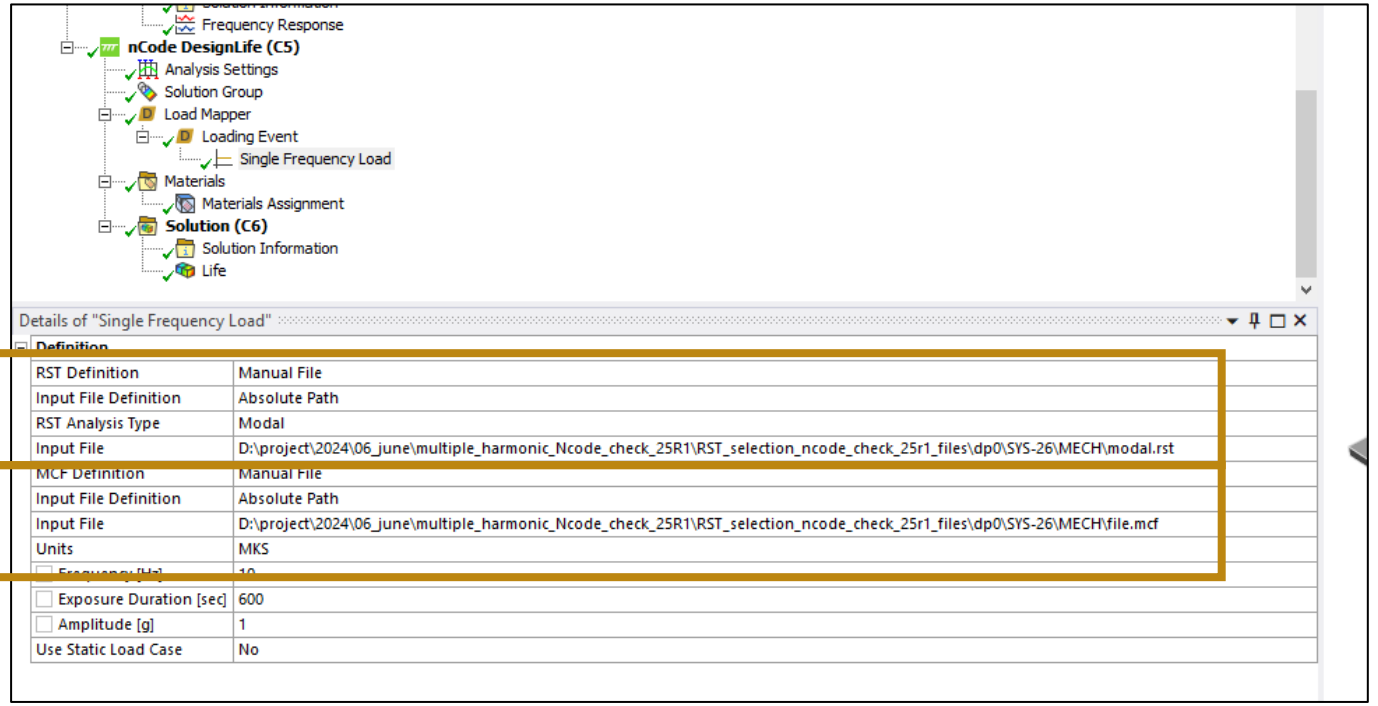
DesignLife Addon reading Element Solution Location results

- Solution Location Option in Analysis Settings is now available: Allowing LS-Dyna fatigue calculations to use Element or NodeOnElement Solution Location
  - For LSDYNA cases, NodeOnElement Solution Location is recommended
  - If users prefer to use Element Solution Location:
    - Using Multiaxial set to None is recommended (performance-wise)
    - Using Combination Method set to AbsMaxPrincipal is recommended (performance-wise)
    - Users can still use Critical Plane, but note that the performance would be highly impacted.

Fatigue Life estimation for MSUP Harmonic

- Manual .rst file selection does allow the use of .rst files from modal or harmonic systems.
  - The user must define whether the RST file is coming from a Modal or Harmonic system by setting the option "RST Analysis Type" accordingly. If the RST corresponds to a Modal .rst file, then a Harmonic system must be linked using the Environment or Manual File option for the MCF Environment property. If the RST corresponds to a Harmonic .rst file, then no MCF Definition is requested.

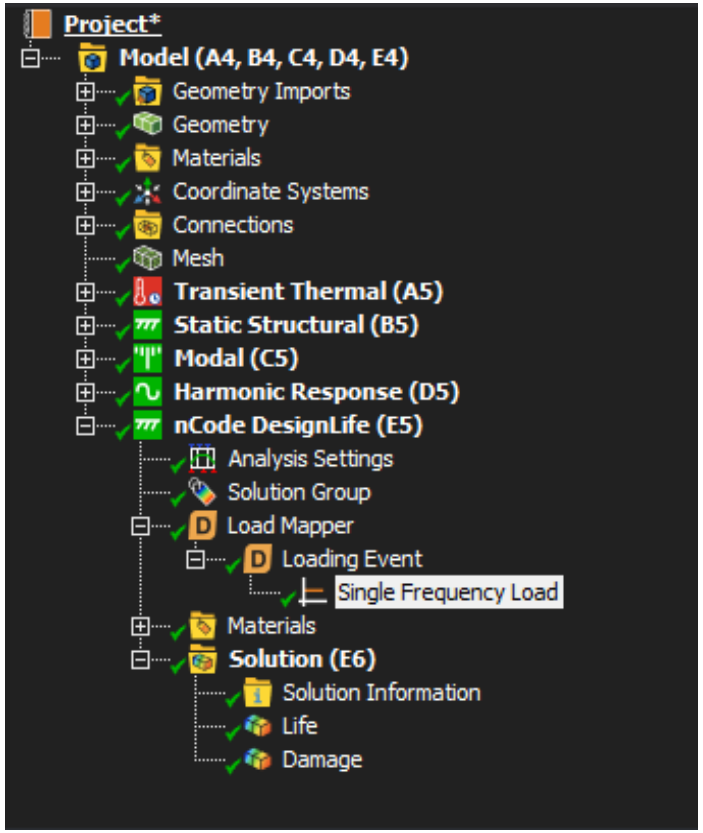
- This newly introduced Manual Selection option in 25R1 helps user to perform fatigue life estimation for Harmonic loading including the residual vector which was not supported previously.
  - Modal result file (“Modal.rst file”) saved in Harmonic directory can be picked using manual selection in NCode
  - Modal coordinate file (“file.mcf file”) saved in Harmonic directory can be picked using manual selection in NCode

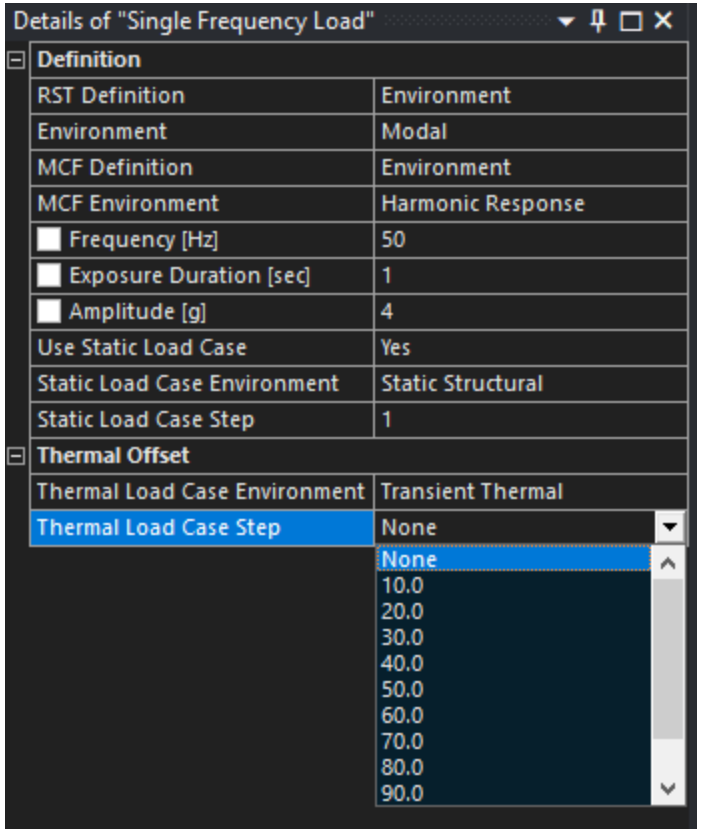


Import Temperature cases for Vibration Fatigue Offset

- Thermal Offset properties are only visible if the project contains a thermal analysis upstream system.
  - Thermal Load Case Environment: Select the thermal analysis system to create the Temperature Load Case offset. The corresponding .rth file is passed as a FE Filename to the ResultsSet to consider the Offset.

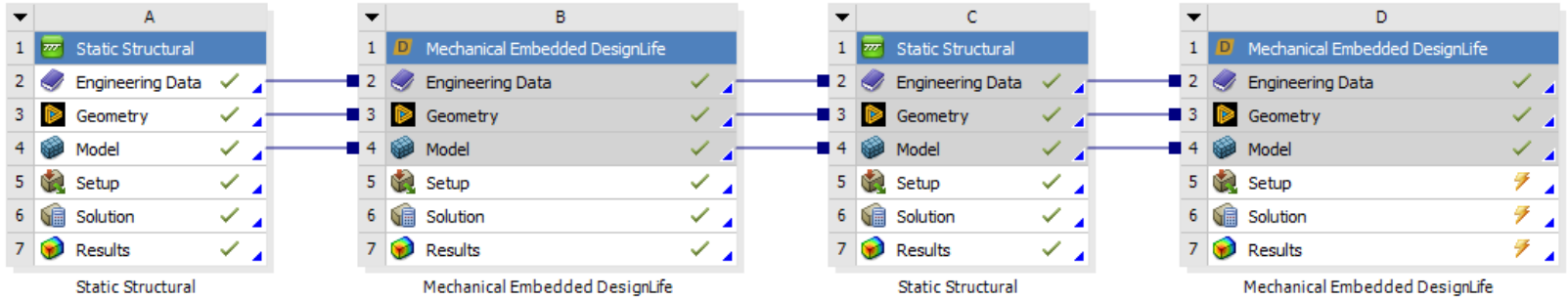
- Thermal Load Case Step:
  - Set this option to None if no thermal offset is desired.
  - Set this option to All to set the FE temperature steps to be used to all the ones in the selected thermal analysis.
  - Set this option to a particular load case of the selected thermal analysis. Sets the list of FE temperature steps to that particular load step.



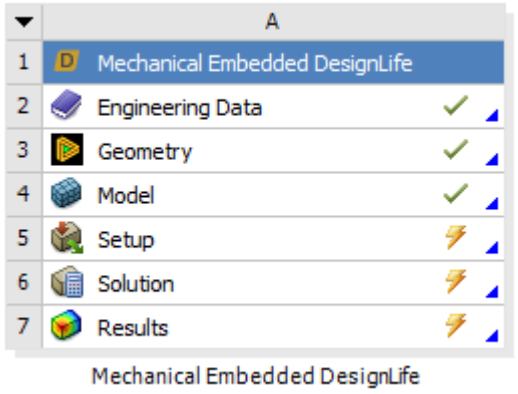


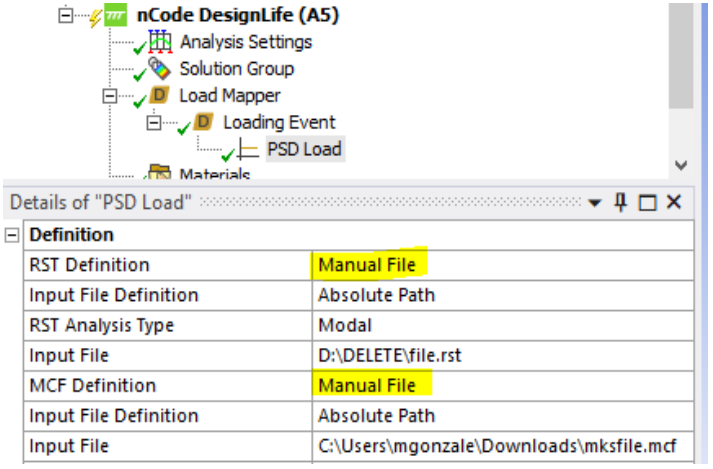
Officially support multi-DL systems

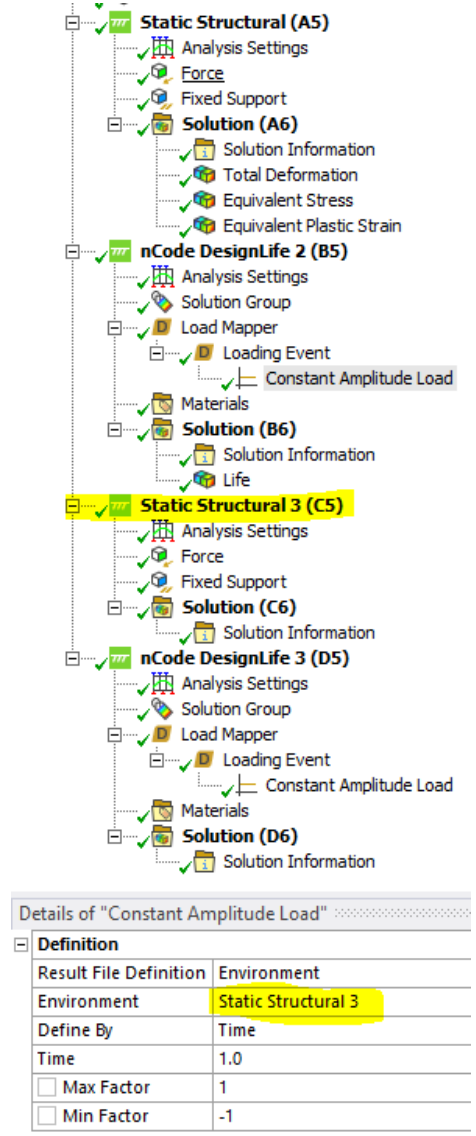
- Additional workflows flexibility
  - Multiple DesignLife Add-on systems allowed in Mechanical
  - DesignLife Add-on systems can be intercalated with other systems in between.



  - DesignLife Add-on system can load environments from upstream or downstream systems
  - DesignLife Add-on systems can be independent if only manual .rst files are loaded



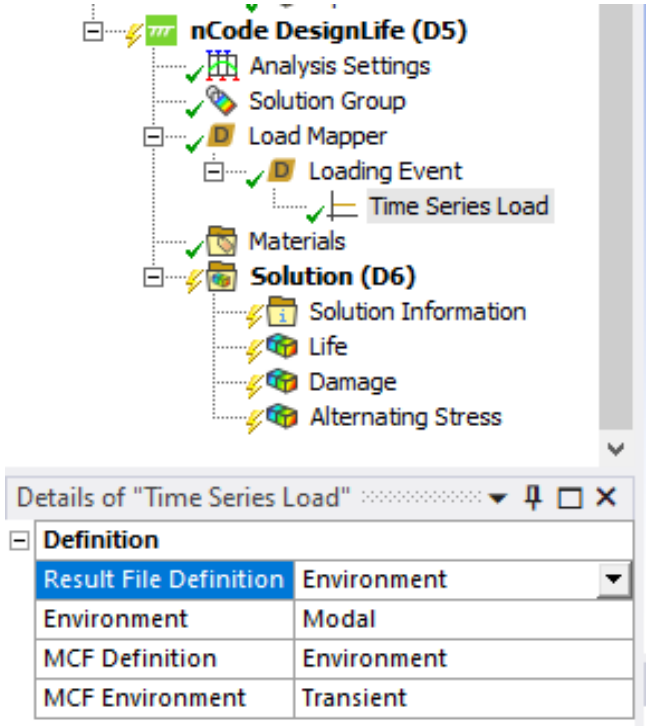




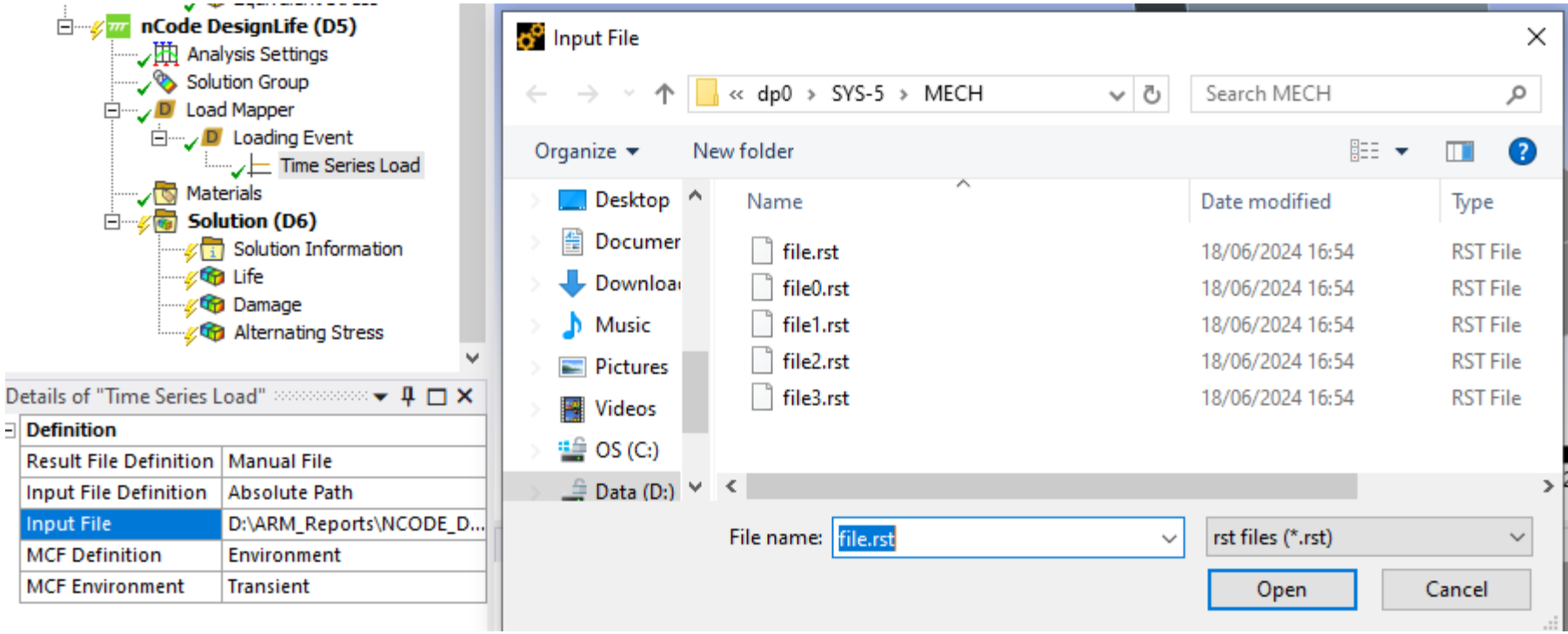
Modal .rst manual selection for Modal + Transient Case

Ability to manually pick a Modal .rst file for Time Domain analysis Modal + Transient case.

- Environment Modal .rst selection



- Manual Modal .rst selection

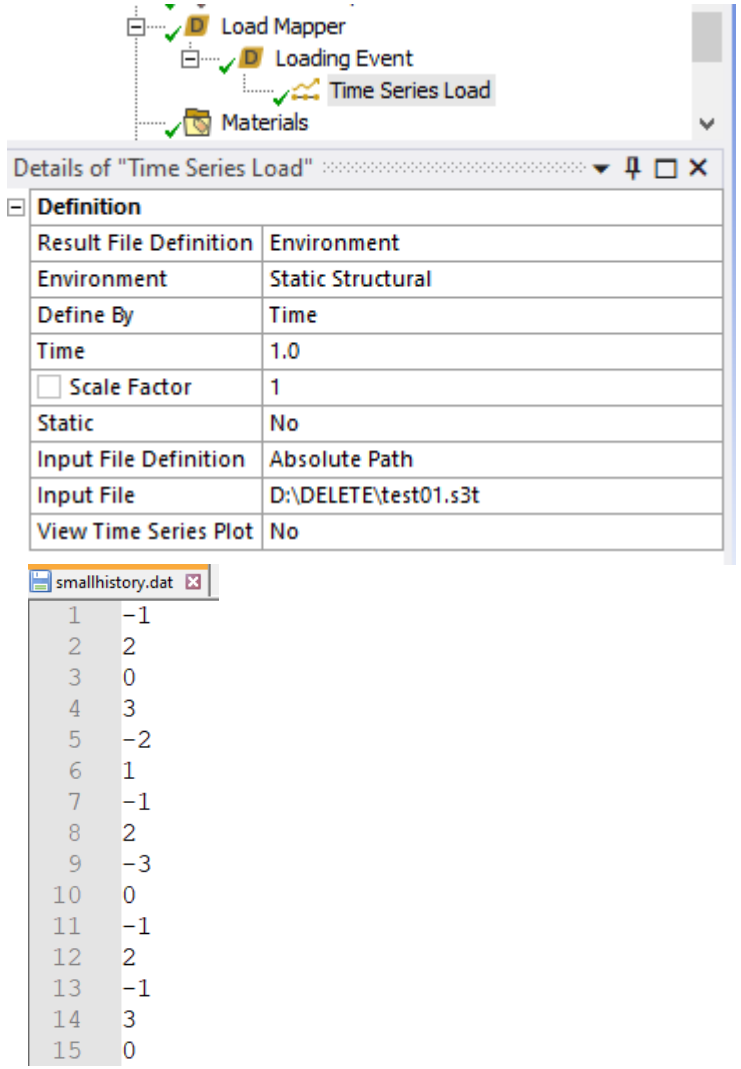


Ability to read .s3t Time Series File

The Time Series **Input File** must be in either **`*.s3t`** or **`*.dat`** text file format. Note that for the same loading event, loads with different file formats cannot be mixed. However, for different loading events, you can use either **`*.dat`** or **`*.s3t`** files.

- **s3t** is an nCode-specific multi-channel binary time series format. It is a specific implementation of the s3 (structured storage system) for time series type data. The time series data is stored as real floating-point single-precision values. Note that when using **`*.s3t`** format, the time series file cannot be visualized as GlyphWorks would be needed.

- **dat** files must be text files containing the factors of the time series, as shown in the example.



HPC for PCG Solver

Distributed Memory Parallel Enhancements

- MPI library support
  - Upgraded to Intel MPI 2021 Update 13 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 now default when running on AMD processors

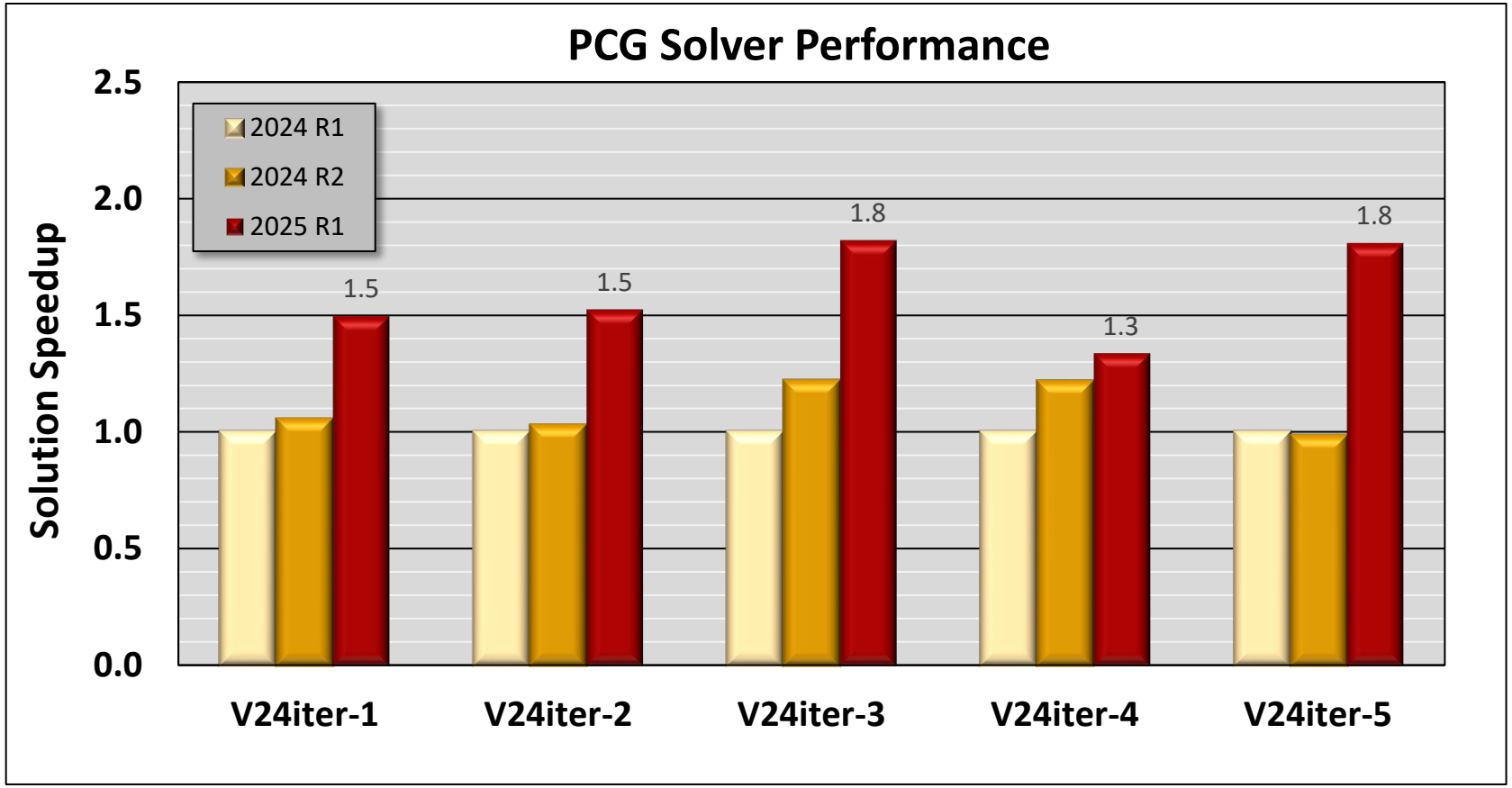
AMD Enhancements

- Open MPI v4.0.5 is now default when running AMD processors
  - Better stability and improved performance on AMD-based clusters
- Support added for AOCL 4.2.1
  - Optimized library for certain “LAPACK” functions
  - Optimized for performance on the latest AMD processor architectures

GPU Acceleration Enhancements

- PCG solver has more computations offloaded onto GPUs in 2025 R1

Measuring entire PCG iterative solver elapsed time



• R24 Benchmark set run on Linux server with 2 Intel Xeon Gold 6548Y+ processors, 1024 GB RAM, SSD, 1 NVIDIA H100 NVL card, RHEL 8.10

GPU Acceleration Enhancements

- PCG solver improvements for GPU acceleration
  - More calculations are offloaded to the GPU for acceleration
    - Some preconditioner operations are now offloaded
    - Coupling/Constraint equation operations are now offloaded
  - CUDA/ROCm upgrades also improve performance

- With more calculations offloaded → larger speedups when using a GPU

Ansys LS-DYNA

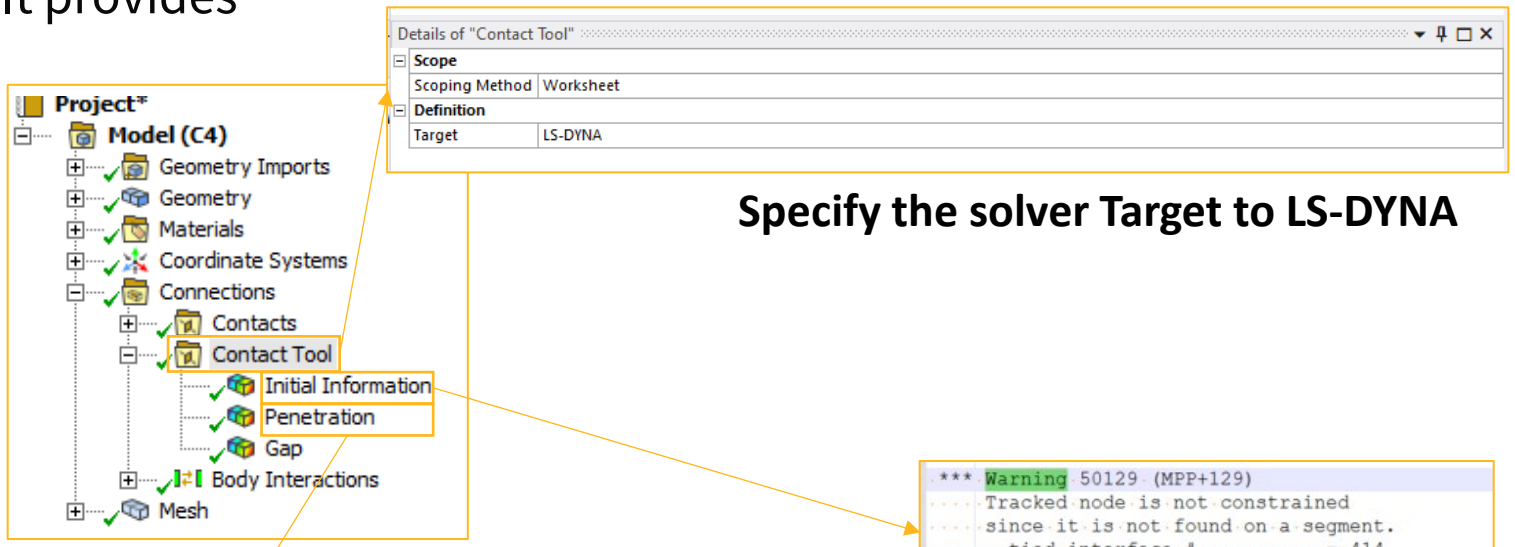
DRD

Contact Tool

The **Contact Tool** is an object you can insert under a **Connections** object for examining initial contact conditions. It is now supported for LS-DYNA based on the extraction of information from LS-DYNA output files (d3hsp, messag). It provides

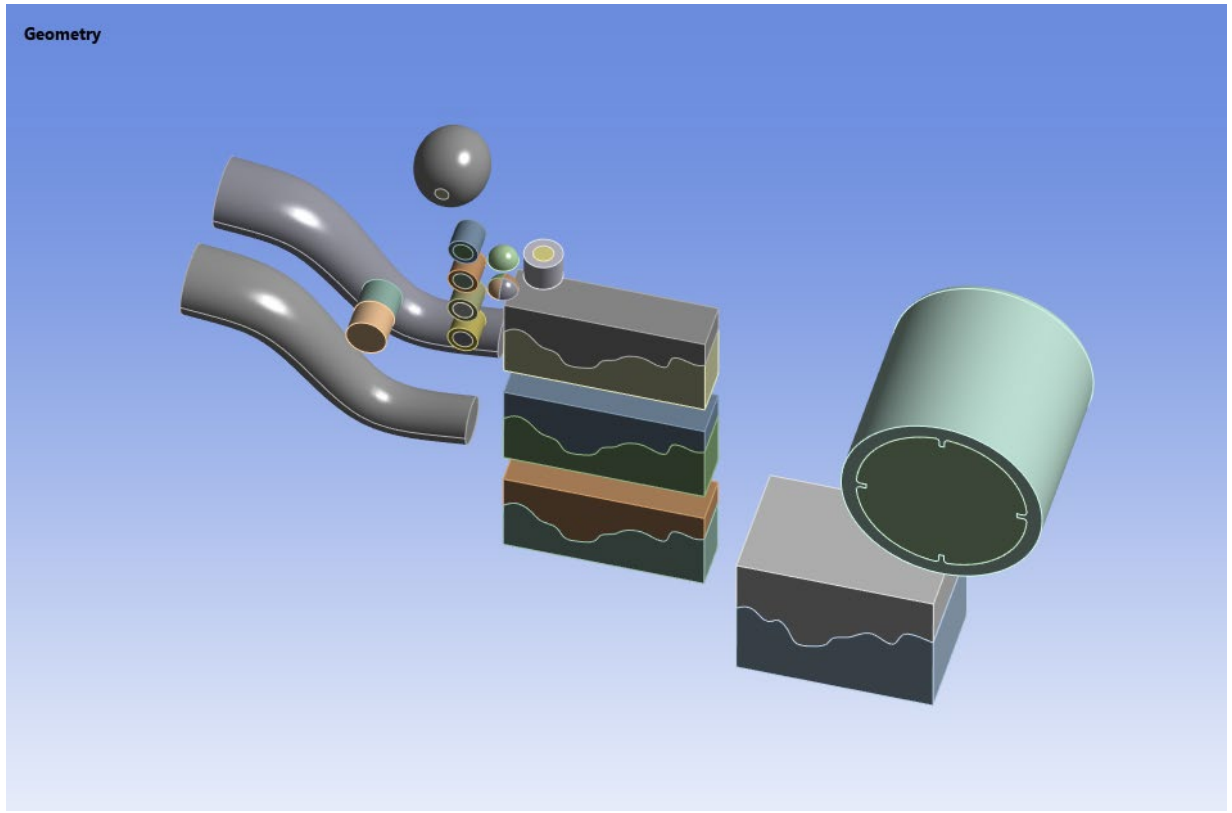
- Maximum penetration,
- Maximum gap and
- Contact status

before the application of the Shooting Node logic inside LS-DYNA, what may fix some of the warnings.



Contact Tool

Penetration found in frictional contacts

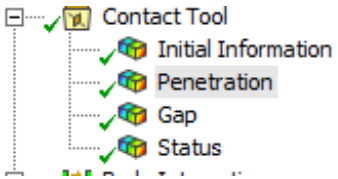


Name	Contact Side	Type	Status	Number Contacting	Penetration (m)	Gap (m)	Geometric Penetration (m)	Geometric Gap (m)	Resulting Pinball (m)	Real Constant
Contact Region	Contact	Bonded	Closed	18.	0.	0.	0.	0.	0.	-95.
Contact Region	Target	Bonded	Closed	18.	0.	0.	0.	0.	0.	95.
Contact Region 2	Contact	Bonded	Closed	18.	0.	0.	0.	0.	0.	-98.
Contact Region 2	Target	Bonded	Closed	18.	0.	0.	0.	0.	0.	98.
Contact Region 3	Contact	Bonded	Closed	18.	0.	0.	0.	0.	0.	-101.
Contact Region 3	Target	Bonded	Closed	18.	0.	0.	0.	0.	0.	101.
Contact Region 4	Contact	Bonded	Closed	18.	0.	0.	0.	0.	0.	-104.
Contact Region 4	Target	Bonded	Closed	18.	0.	0.	0.	0.	0.	104.
Contact Region 5	Contact	Bonded	Closed	18.	0.	0.	0.	0.	0.	-107.
Contact Region 5	Target	Bonded	Closed	18.	0.	0.	0.	0.	0.	107.
Contact Region 7	Contact	Bonded	Closed	42.	0.	0.	0.	0.	0.	-113.
Contact Region 7	Target	Bonded	Closed	42.	0.	0.	0.	0.	0.	113.
Contact Region 8	Contact	Bonded	Closed	115.	0.	0.	0.	0.	0.	-116.
Contact Region 8	Target	Bonded	Closed	115.	0.	0.	0.	0.	0.	116.
Contact Region 9	Contact	Bonded	Closed	115.	0.	0.	0.	0.	0.	-119.
Contact Region 9	Target	Bonded	Closed	115.	0.	0.	0.	0.	0.	119.
Contact Region 10	Contact	Bonded	Closed	115.	0.	0.	0.	0.	0.	-122.
Contact Region 10	Target	Bonded	Closed	115.	0.	0.	0.	0.	0.	122.
Frictional - Solid To Solid 1	Contact	Frictional	Closed	336.	9.29e-004	0.	0.	0.	0.	-195.
Frictional - Solid To Solid 1	Target	Frictional	Closed	336.	9.29e-004	0.	0.	0.	0.	195.
Contact Region 16	Contact	Bonded	Closed	1094.	0.	0.	0.	0.	0.	-252.
Contact Region 16	Target	Bonded	Closed	1094.	0.	0.	0.	0.	0.	252.
Contact Region 12	Contact	Bonded	Closed	476.	0.	0.	0.	0.	0.	-304.
Contact Region 12	Target	Bonded	Closed	476.	0.	0.	0.	0.	0.	304.
Contact Region 13	Contact	Bonded	Closed	73.	0.	0.	0.	0.	0.	-307.
Contact Region 13	Target	Bonded	Closed	73.	0.	0.	0.	0.	0.	307.
Contact Region 14	Contact	Bonded	Closed	4944.	0.	0.	0.	0.	0.	-310.
Contact Region 14	Target	Bonded	Closed	4944.	0.	0.	0.	0.	0.	310.
Contact Region 15	Contact	Bonded	Closed	498.	0.	0.	0.	0.	0.	-313.
Contact Region 15	Target	Bonded	Closed	498.	0.	0.	0.	0.	0.	313.
Contact Region 18	Contact	Bonded	Closed	3.	0.	0.	0.	0.	0.	-326.
Contact Region 18	Target	Bonded	Closed	3.	0.	0.	0.	0.	0.	326.
Contact Region 19	Contact	Bonded	Closed	7.	0.	0.	0.	0.	0.	-329.
Contact Region 19	Target	Bonded	Closed	7.	0.	0.	0.	0.	0.	329.
Contact Region 20	Contact	Bonded	Closed	12.	0.	0.	0.	0.	0.	-332.
Contact Region 20	Target	Bonded	Closed	12.	0.	0.	0.	0.	0.	332.
Contact Region 21	Contact	Bonded	Closed	7.	0.	0.	0.	0.	0.	-335.
Contact Region 21	Target	Bonded	Closed	7.	0.	0.	0.	0.	0.	335.
Bonded - Solid To Solid	Contact	Bonded	Far Open	0.	0.	0.	0.	0.	0.	-414.
Bonded - Solid To Solid	Target	Bonded	Far Open	0.	0.	0.	0.	0.	0.	414.

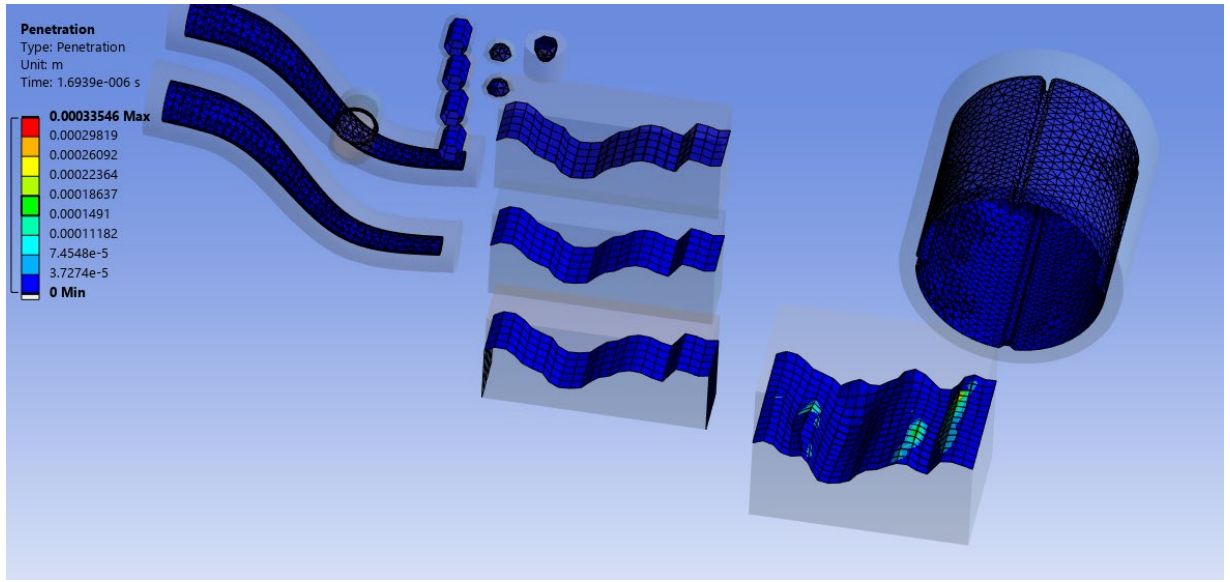
Contact defined as bonded but with no nodes in contact

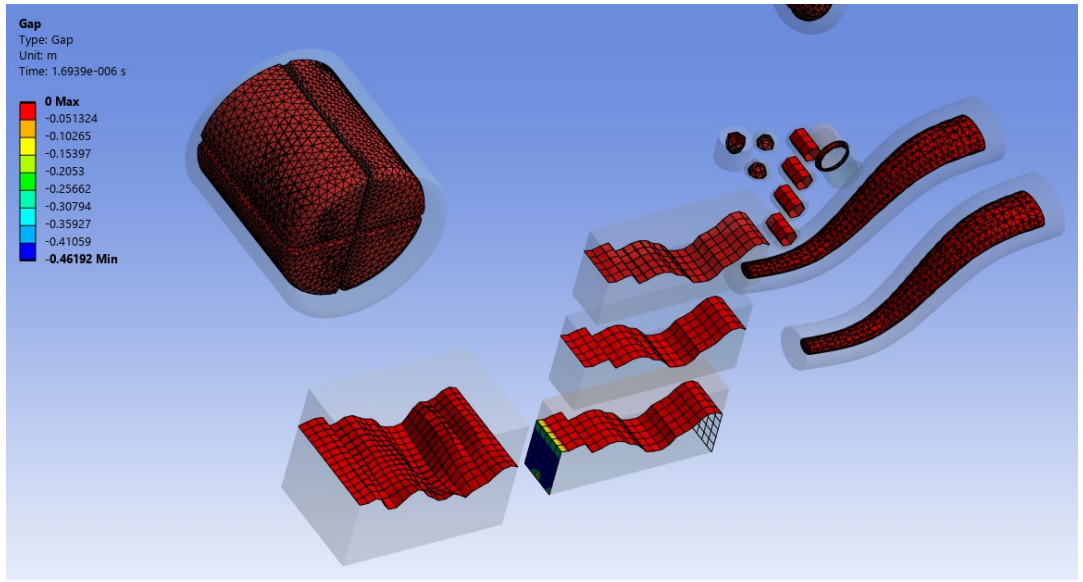
Contact Tool

The Contact Tool is using the solver **Model Check** feature, which is running a couple of cycles to assess if there is any major issue in the model. The Penetration, Gap and Status at the end of this phase can be visualized through the Contact Tool.



Penetration and Gap at the end of the Model Check



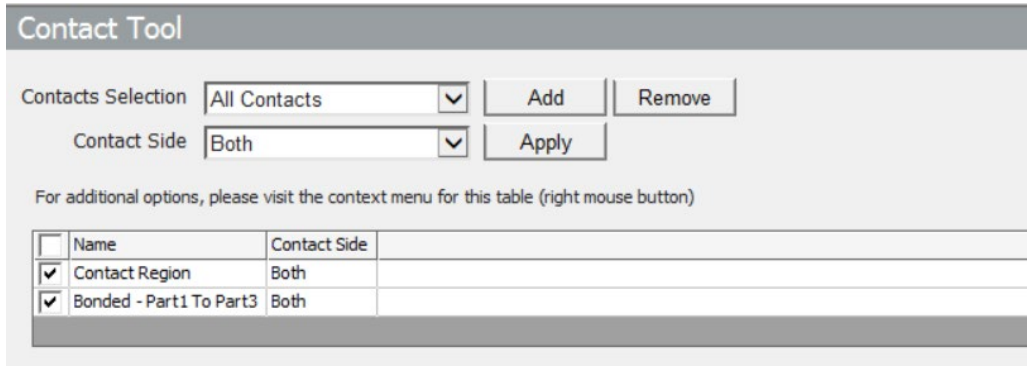


Contact Tool (Post-Processing)

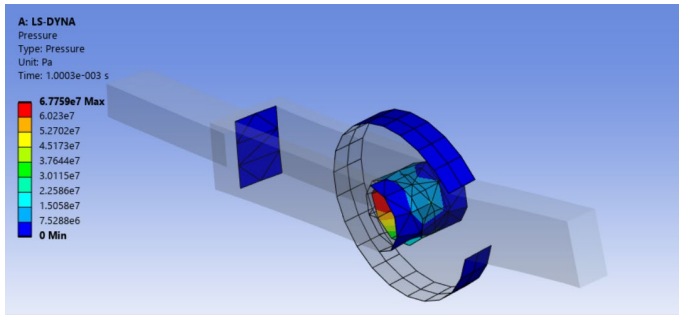
The same **Contact Tool** is also enabled during Post-Processing.
It enables display of

- **Pressure**,
- **Penetration**,
- **Frictional Stress and others**

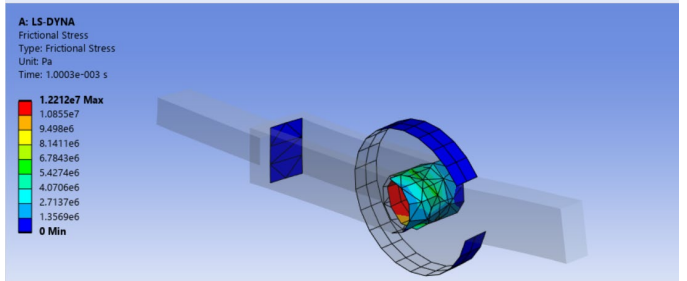
for multiple contact regions



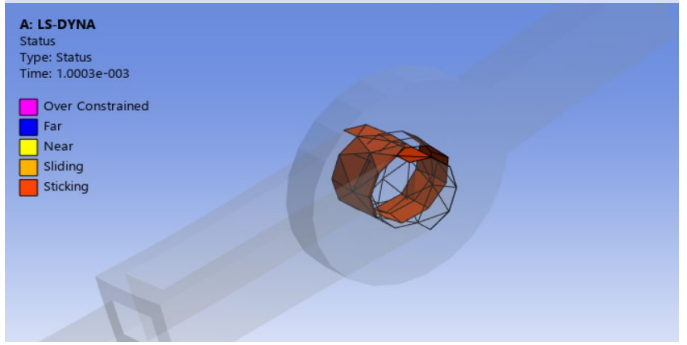
Pressure



Frictional Stress



Contact State



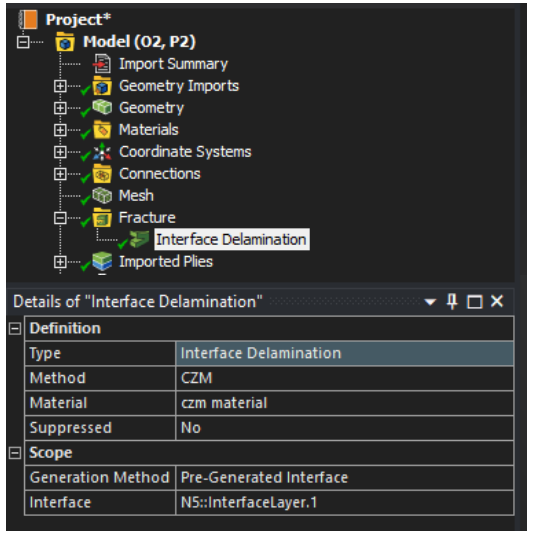
Improvements towards LS-DYNA

The LS-DYNA Analysis System now support cohesive elements and tiebreak contacts.

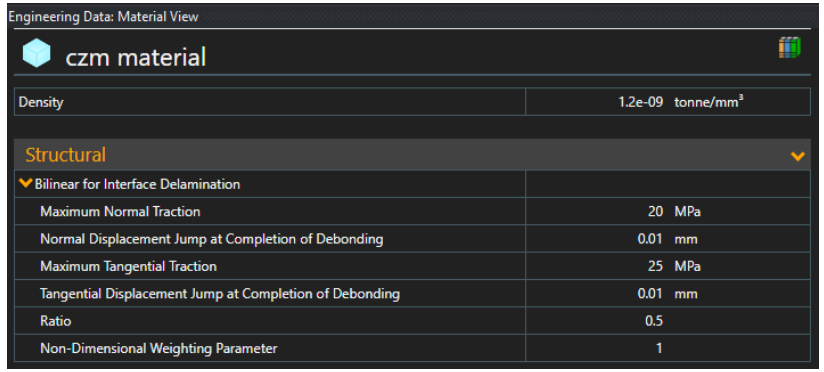
- The LS-DYNA Analysis System module now supports Interface Layers of ACP Solid Models to simulate delamination of plies. Interface Layers are converted into zero volume LS-DYNA cohesive elements (ELFORM=19) with a material model *MAT_COHESIVE_MIXED_MODE. In Engineering Data it is defined by material model "Bilinear for Interface Delamination".
- Contact debonding between bodies is supported now as well. Bonded contacts are converted into LS-DYNA tiebreak contacts with a fracture model based on *MAT_COHESIVE_MIXED_MODE. “Fracture-Energies based Debonding” from Engineering Data is the supported Cohesive Zone material model.

See Ansys LS-DYNA help for more details.

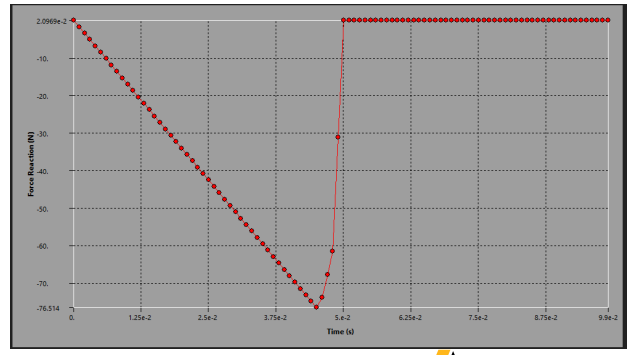
Fracture object defined by an Interface Layer



Material properties



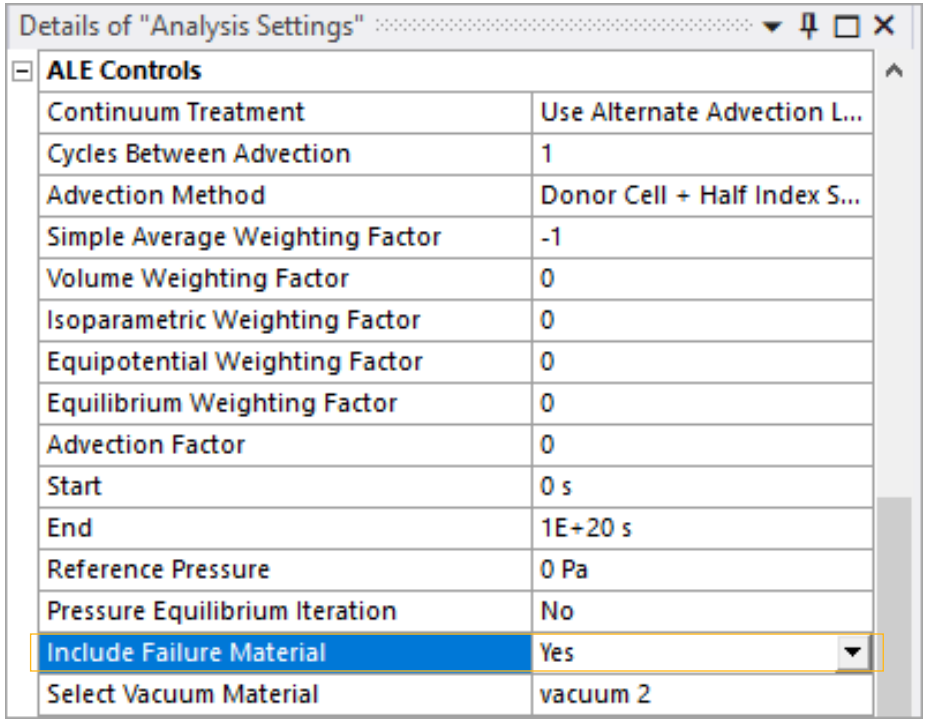
Reaction forces



ALE Material Failure

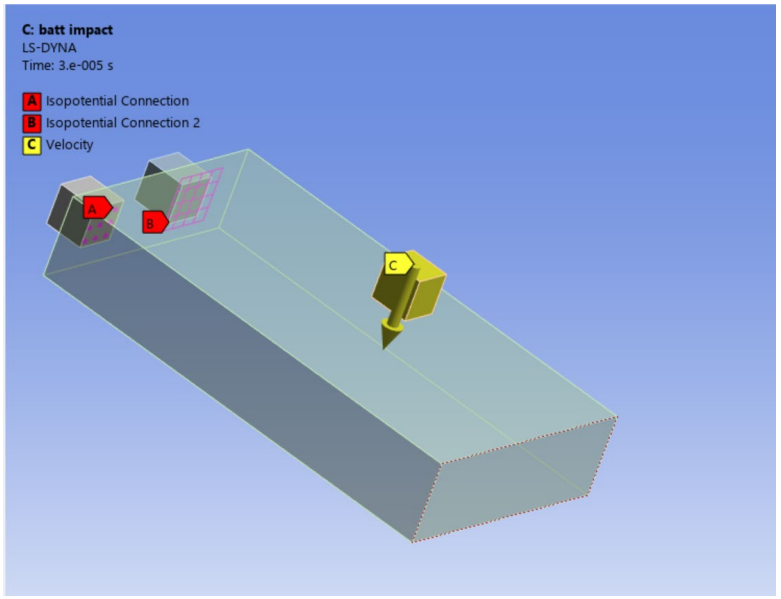
Material failure is available in the LS-DYNA Analysis System for ALE and S-ALE simulations. This approach replaces the failed material with a dummy vacuum material by creating a dummy part with its own ID and associating a *MAT_VACUUM keyword with the part.

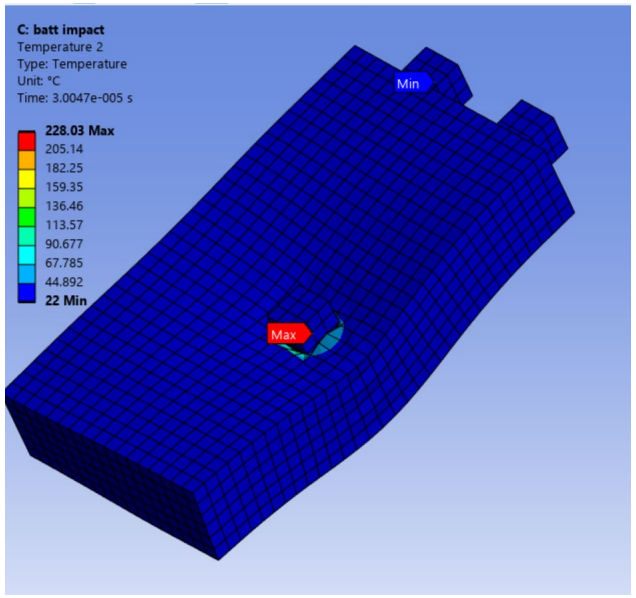
To enable material failure in your simulation, select *Yes* for the *Include Failure Material* option under the ALE Controls category of the LS-DYNA system Analysis Settings.

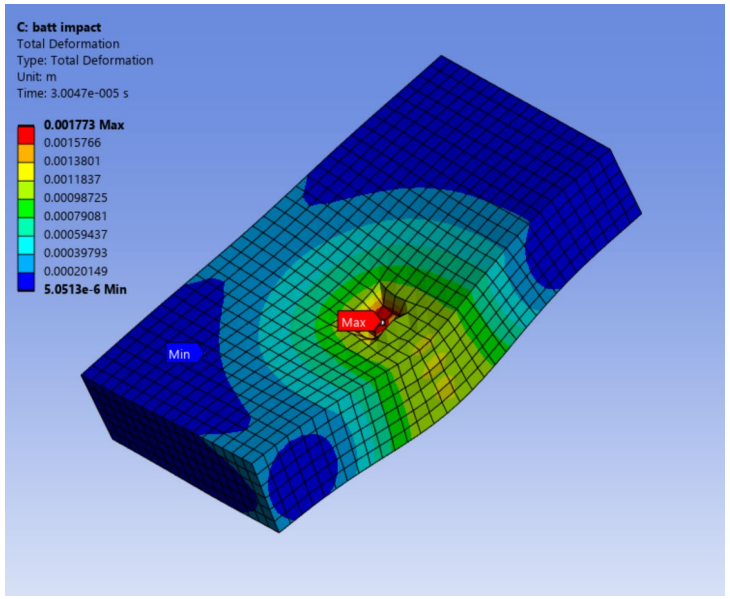


Battery Modeling

Battery modeling has been introduced in Ansys LS-DYNA to enable the simulation of battery cells under normal use conditions as well as during abusive scenarios during which the cells undergo a damaging and an internal short may occur.







Impact on Battery Cell

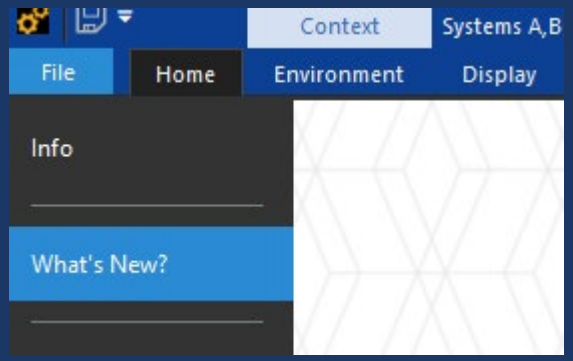
Autodyn Eulerian Technology In LS-DYNA

(Beta Release)

Autodyn Finite Volume Euler Solvers Added to LS-DYNA

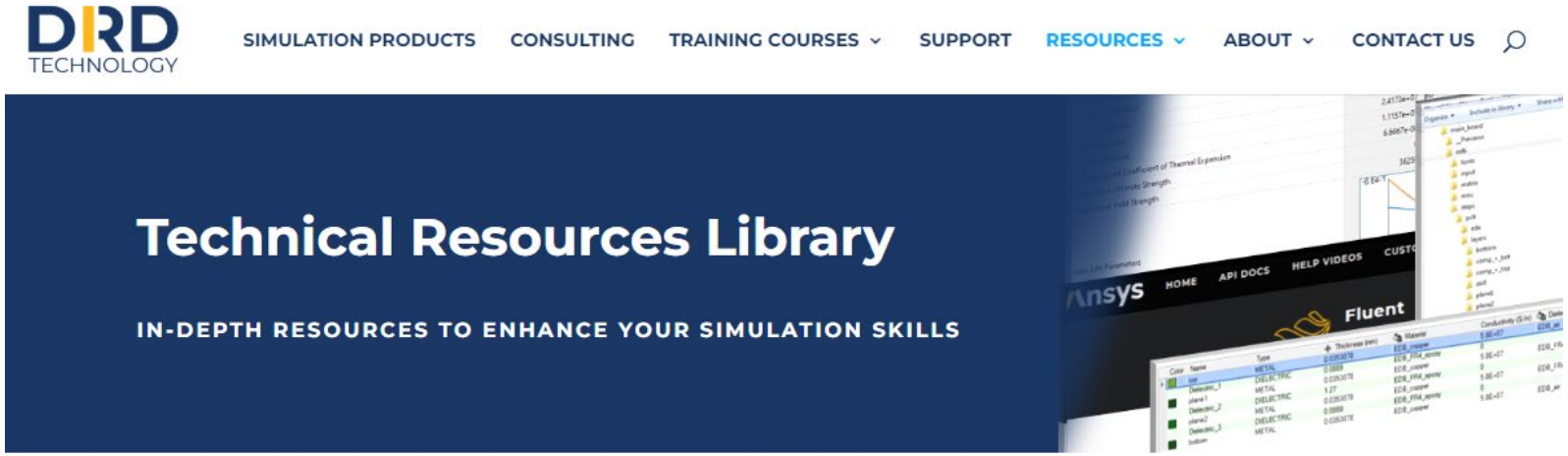
- Migration of the Autodyn Eulerian solvers into the LS-DYNA solver (as a beta feature):
  - Multi-Material Euler solver (2d and 3d) for modelling high deformation solids, liquids and fluids
  - Ideal Gas solver (2d and 3d) for high fidelity blast simulations
- Fluid Structure Interaction (FSI) included between both Eulerian solvers and finite element structures (solids and shells) based on the Autodyn coupling algorithm
- Introduction of mapping algorithms to initialise an Eulerian or ALE simulation from the results of another Eulerian or ALE simulation.

What's New! There's more...

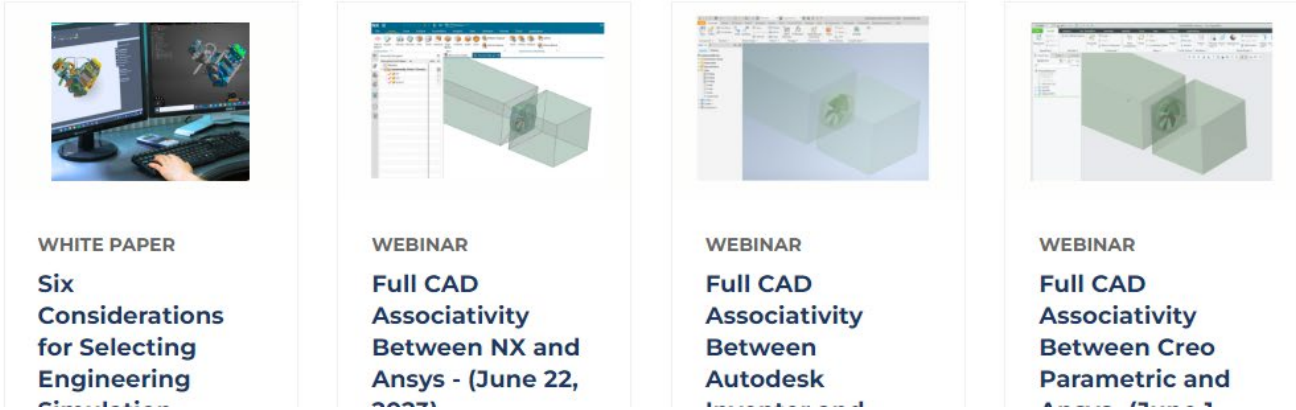


Wrap Up

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?