Three Pillars: Ansys Structures

Robert Dickens

Lead Application Engineer

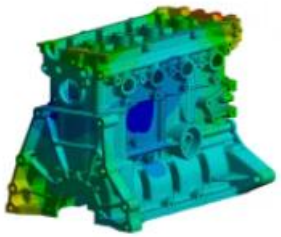
Agenda

- Highlights reel of the last few releases of Mechanical
- Highlights of 2023 R2

Structures Product Portfolio

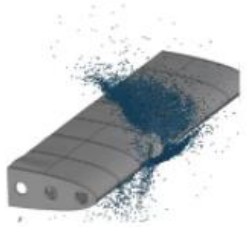
Structural analysis for every application and experience level

Ansys Mechanical



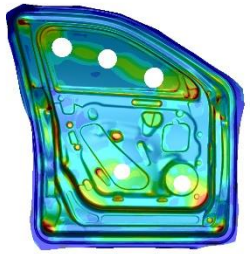
Best-in-class FEA solver for structural engineering applications

Ansys LS-DYNA



Industry-leading software for drop tests, impact and penetration, smashes and crashes, occupant safety, and more

Ansys Forming

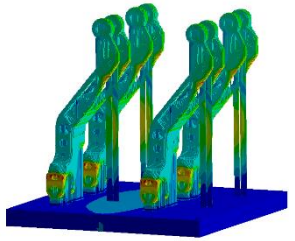


NEW sheet metal stamping tool, backed by the LS-DYNA solver

Ansys Motion

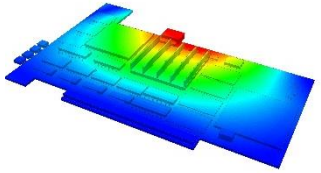
The most robust and advanced simulation solution for multibody dynamic system design

Ansys Additive



The most powerful simulation solution for metal additive manufacturing

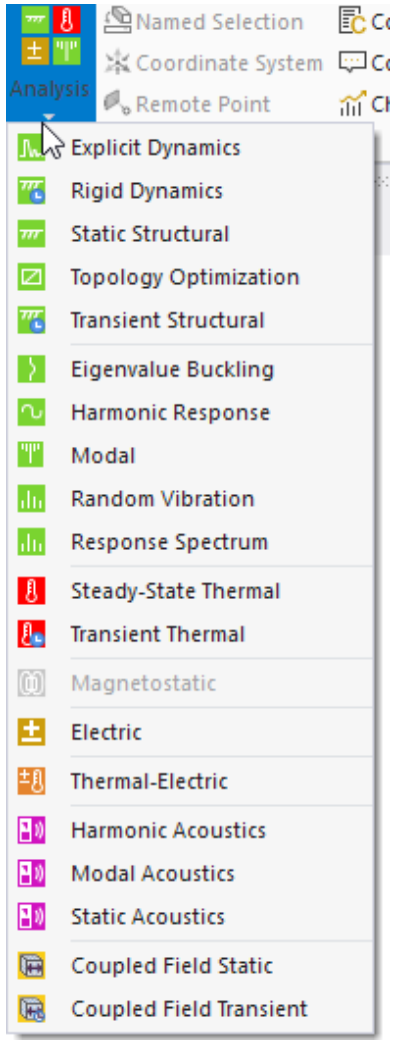
Ansys Sherlock

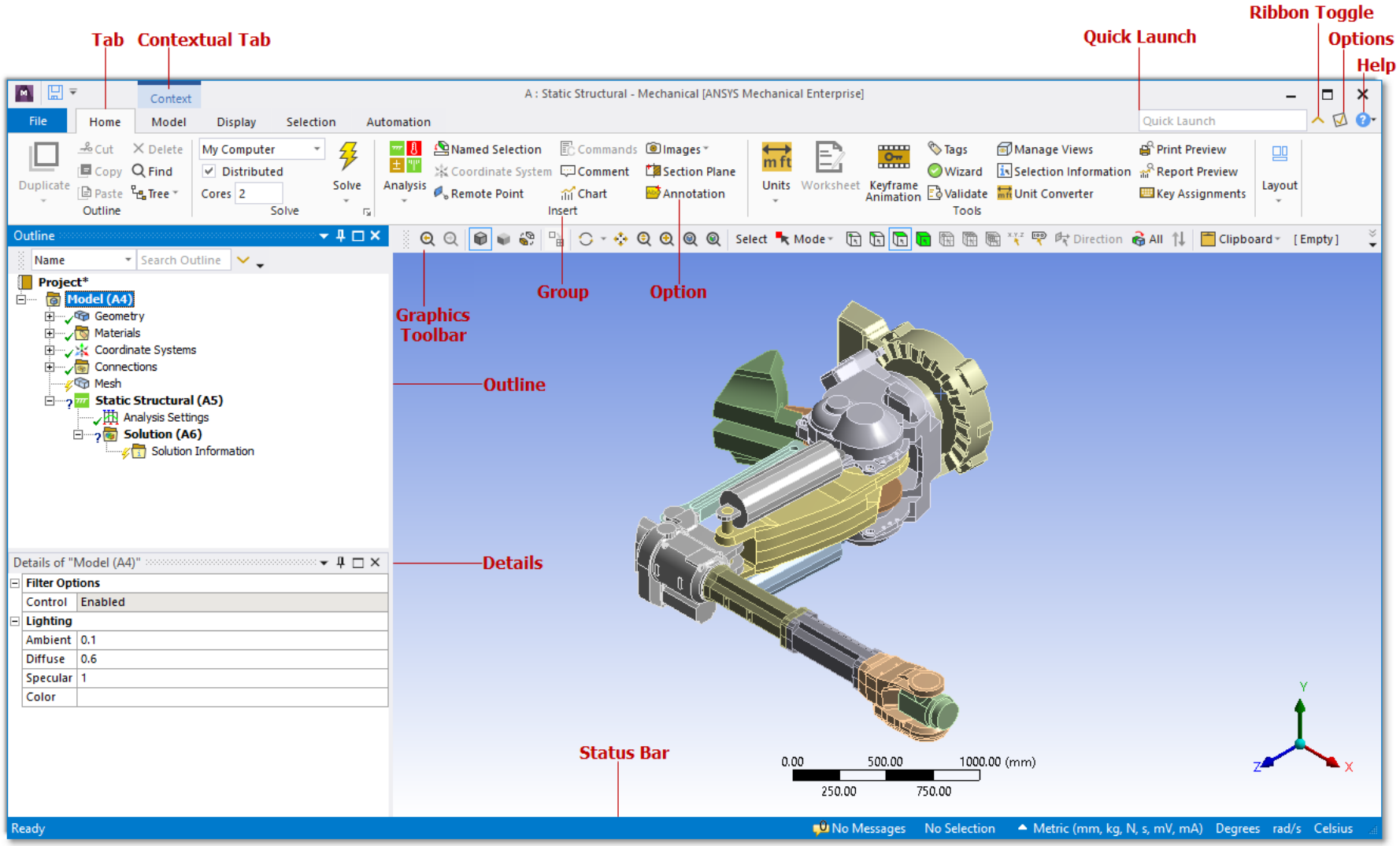


Validated life predictions for electronic hardware

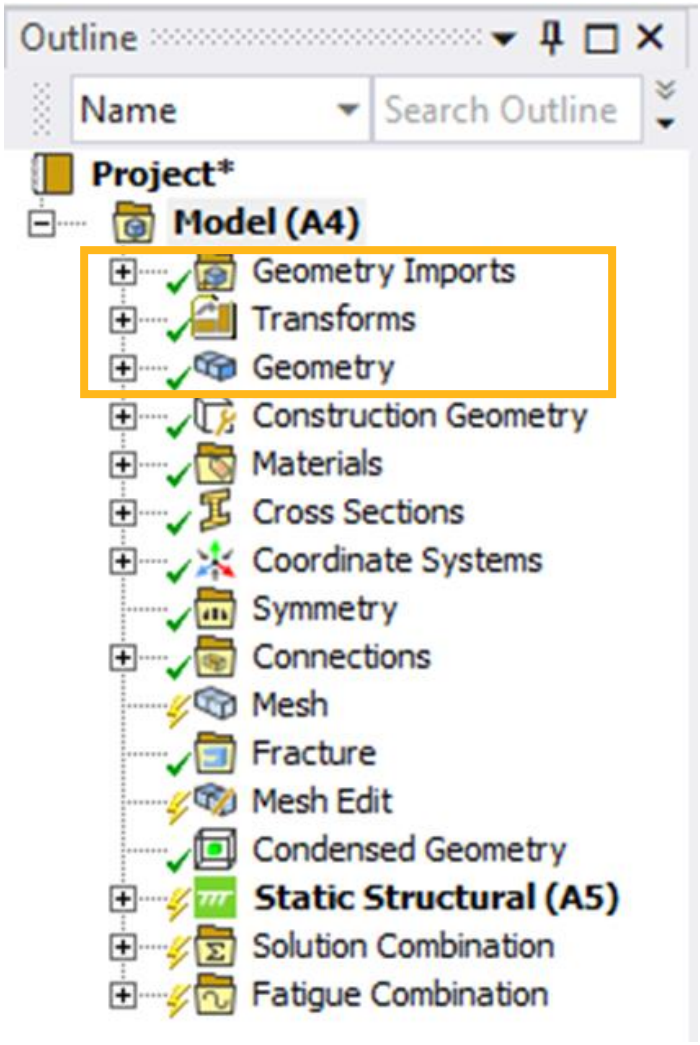
3

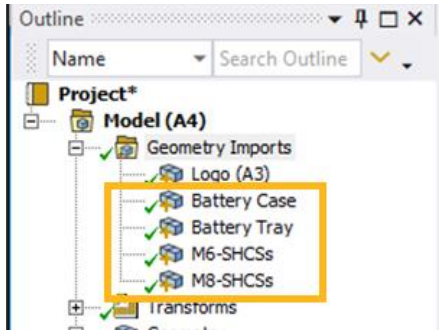
Ansys Mechanical Interface Update

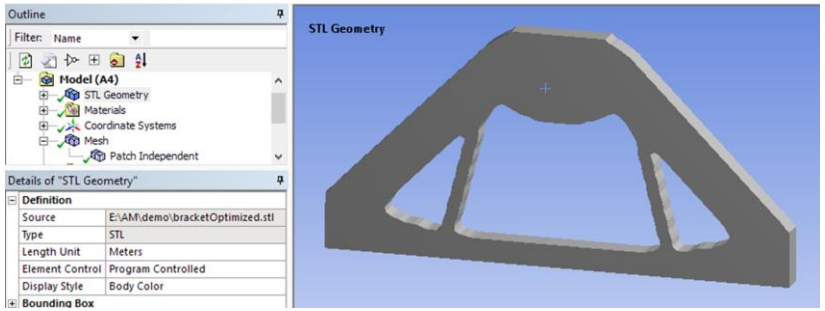


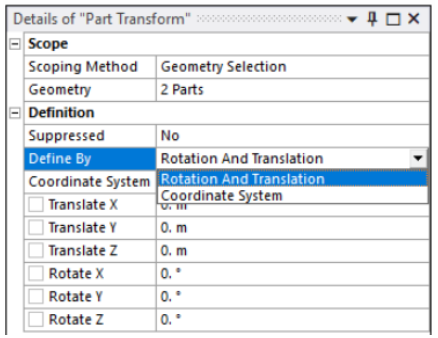


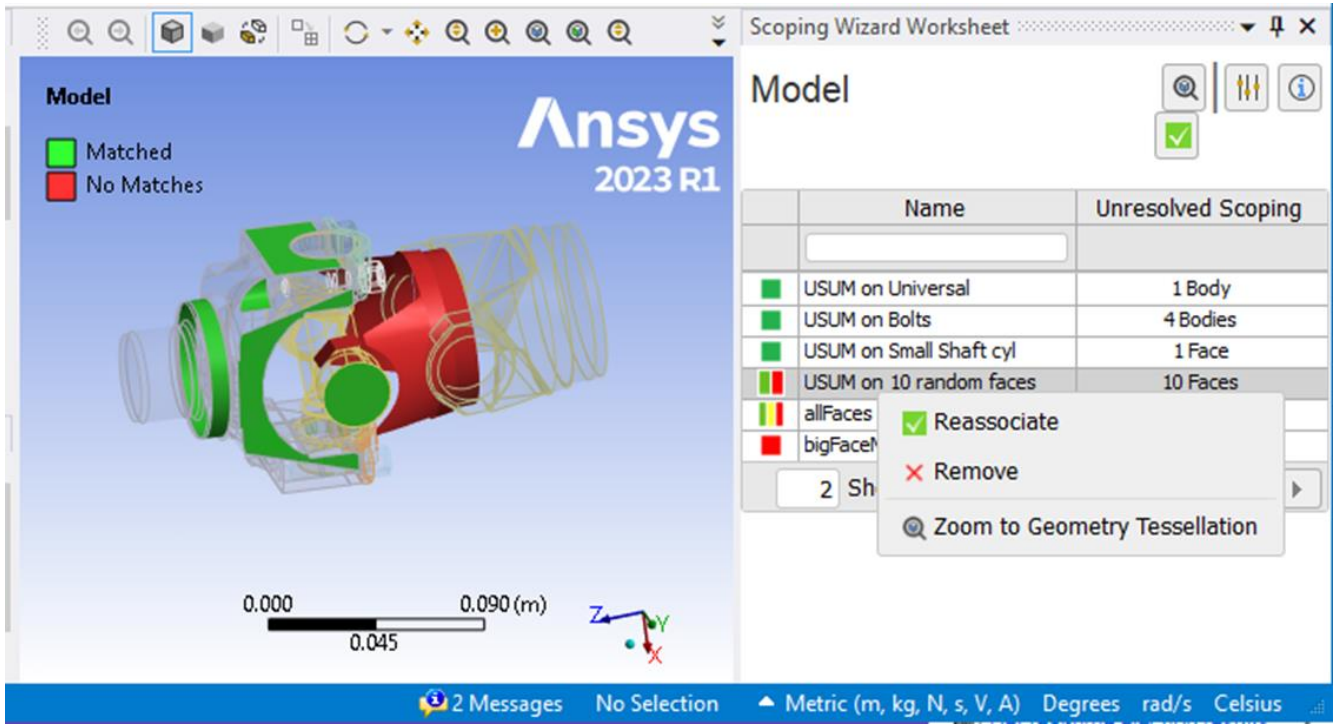
Import and Manipulate Geometry within Mechanical







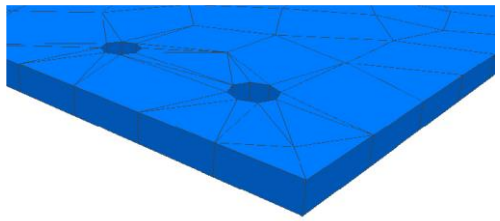




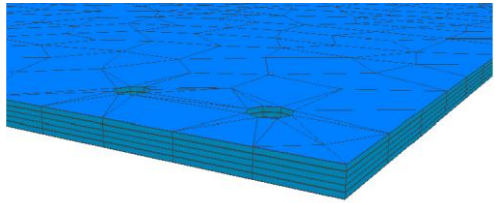
- ✓ Modify a CAD model, without losing the associativity of the model's features after setup
- ✓ Using geometry-based re-associativity with the Scoping Wizard detects and reestablishes scoping

PCB Modeling Technologies

Homogenized/Lumped



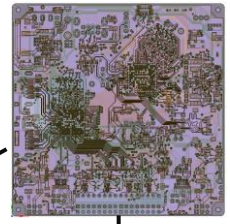
Lumped over entire board



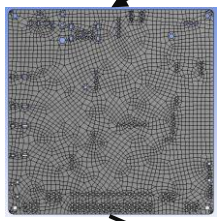
Lumped by Layer

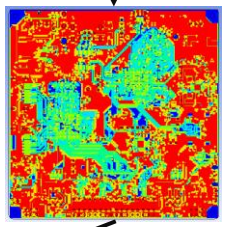
Trace Mapping

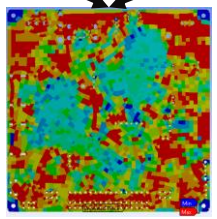
ECAD Data



Mesh



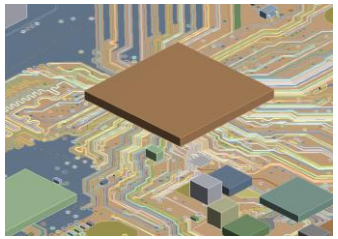




High resolution Cartesian metal map from ECAD

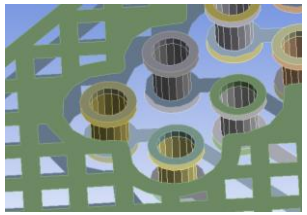
Element-wise material mapping

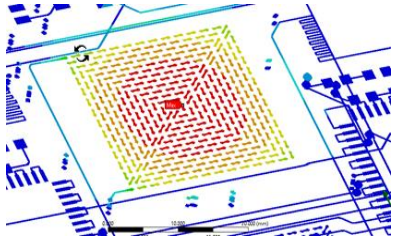
Reinforcements



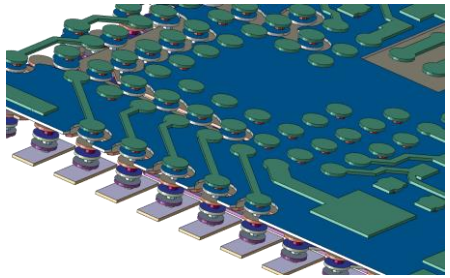
Embed surfaces (traces) and beams (vias) within a base mesh

Improved accuracy without full 3D Trace Modeling

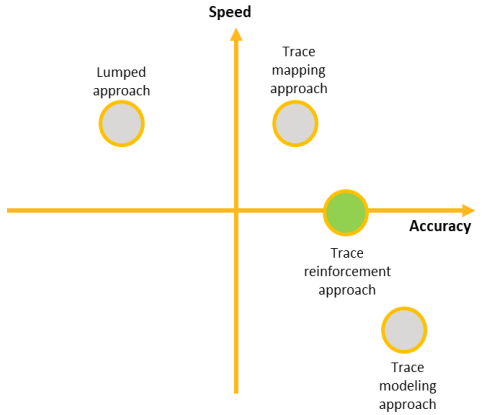




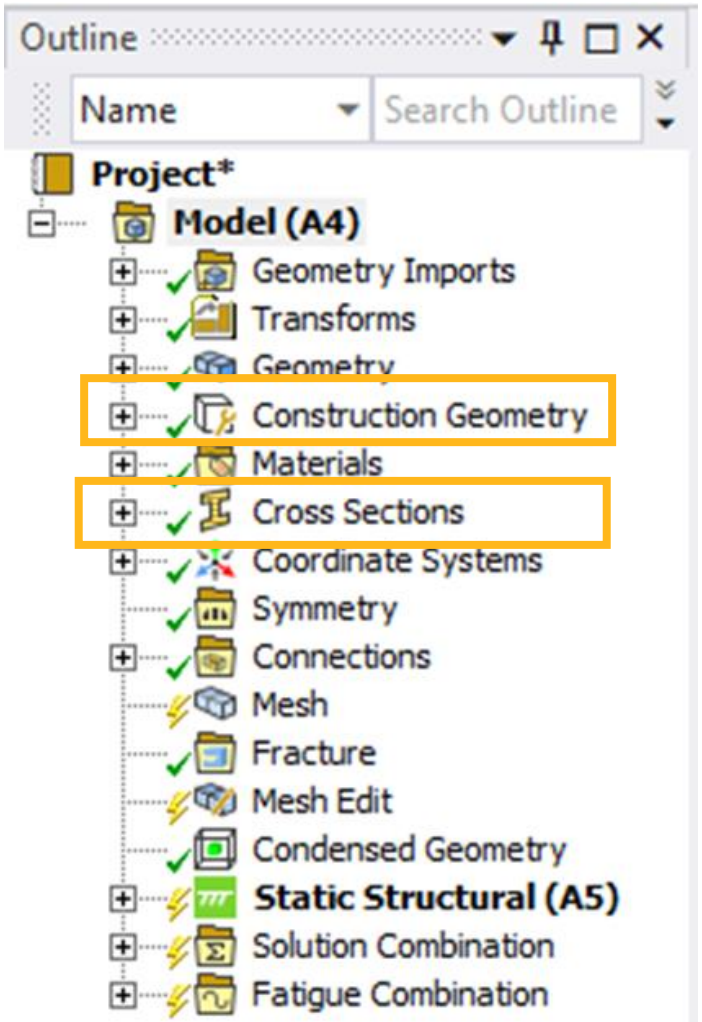
Trace Modeling

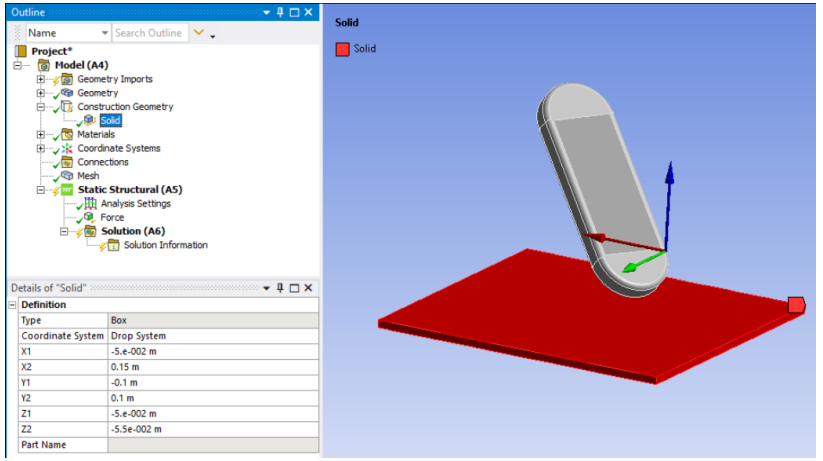


Full 3D Detail

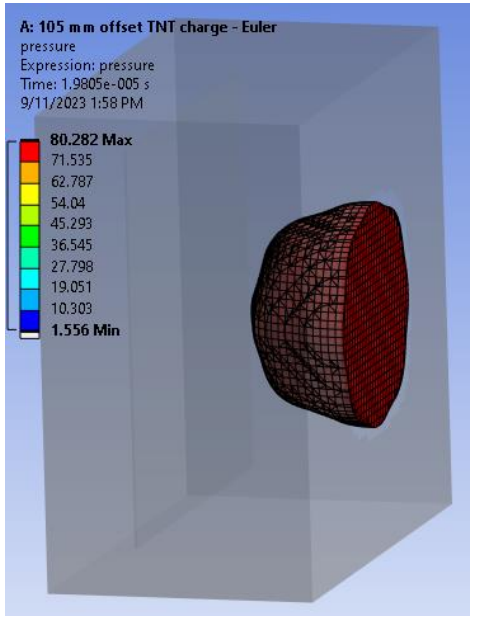


Construct geometry within Mechanical

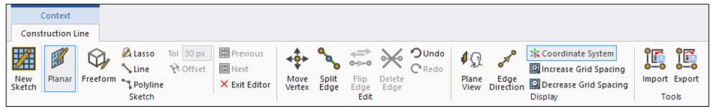


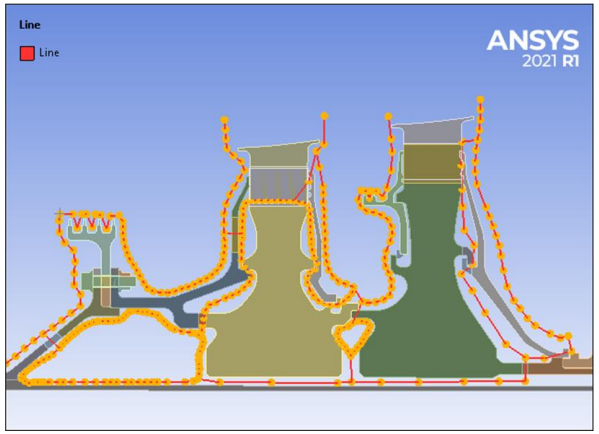


Add Ground surface for Drop

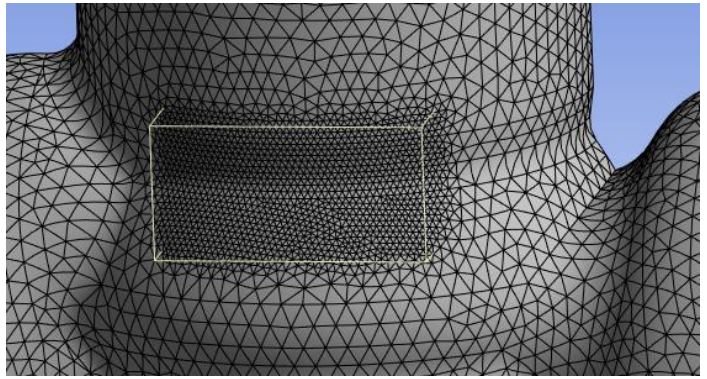


Define Euler Region



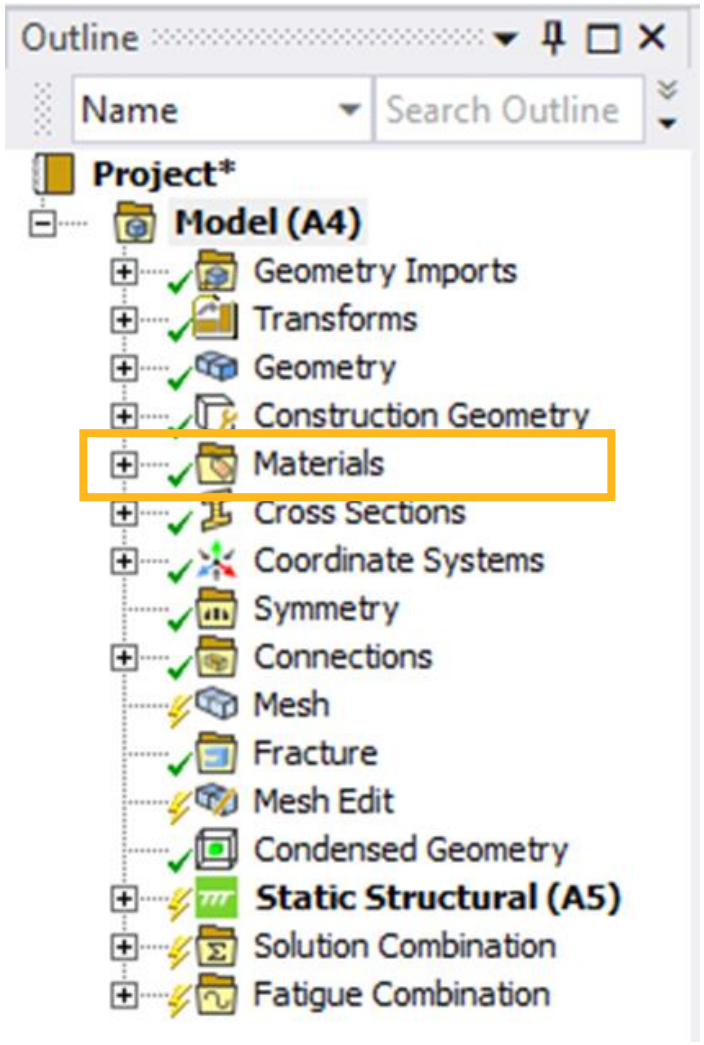


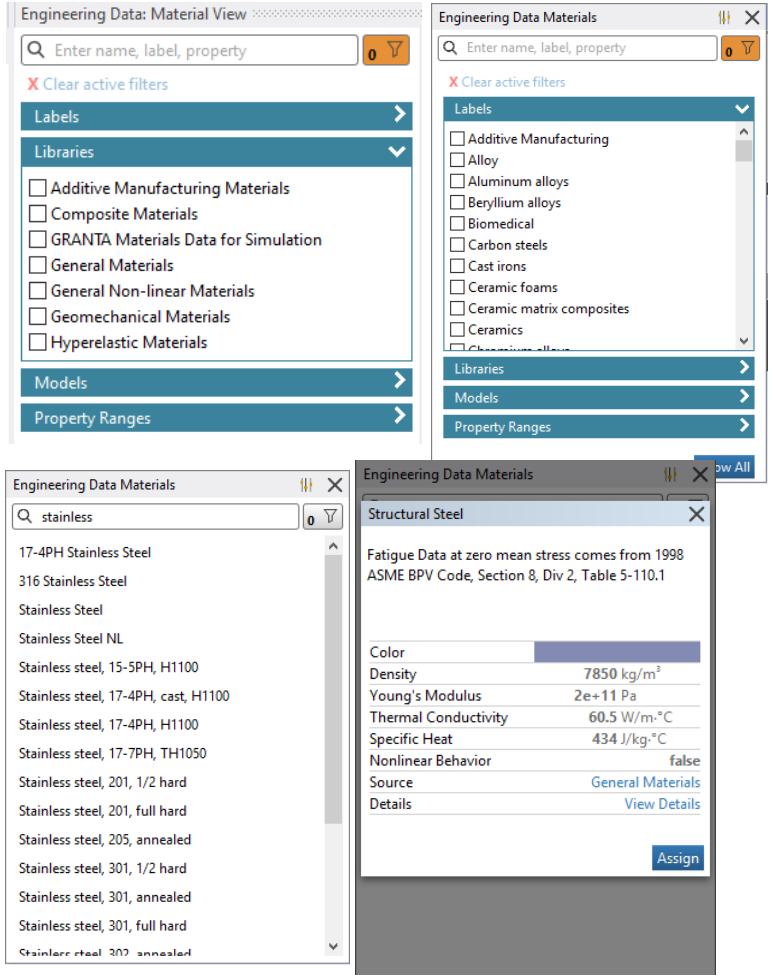
Thermal Fluid definition

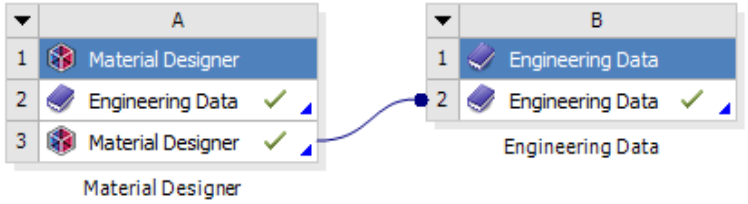


Body of Influence

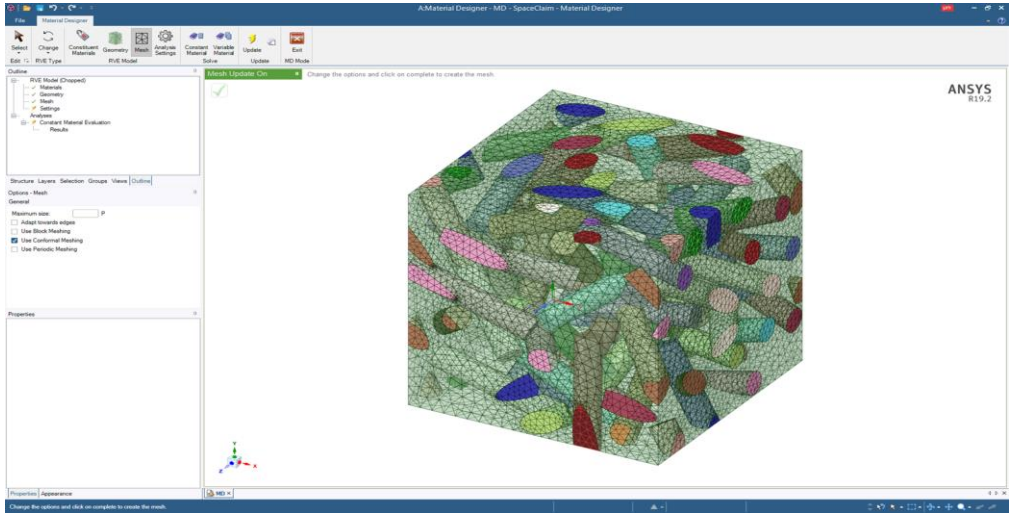
Material Browsing, Searching, Assignments, and Designer



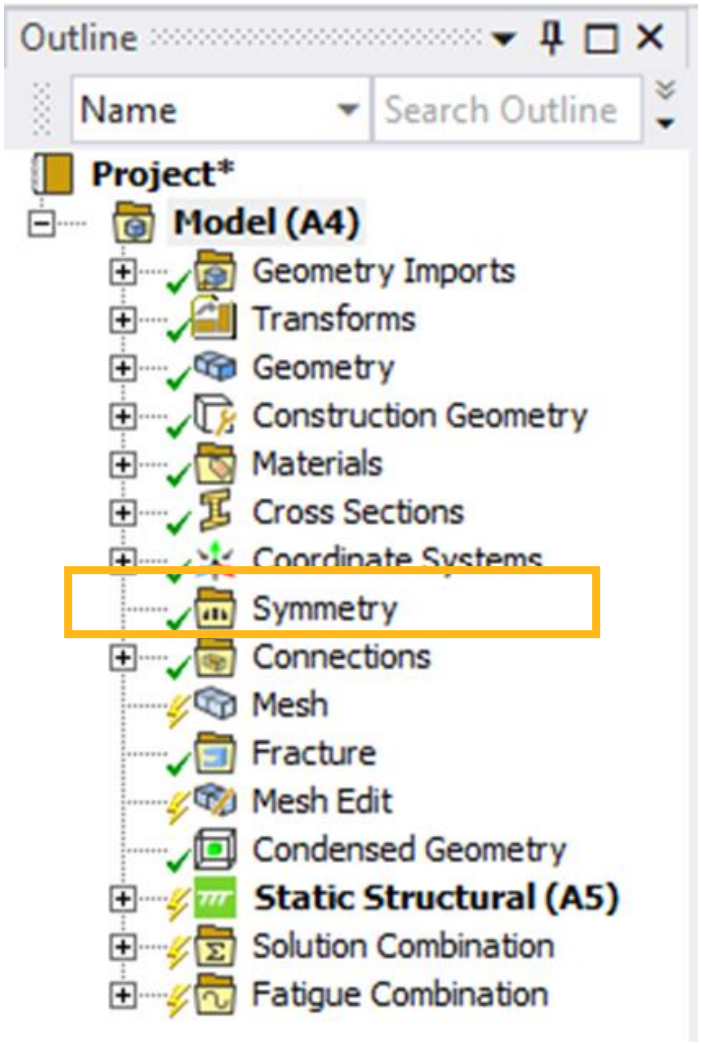


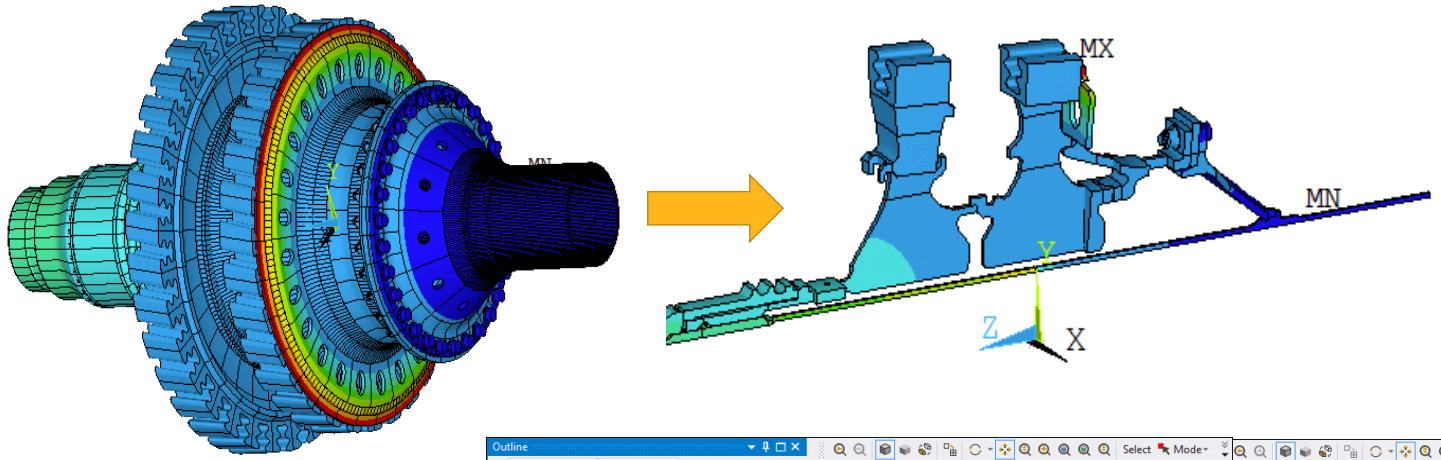


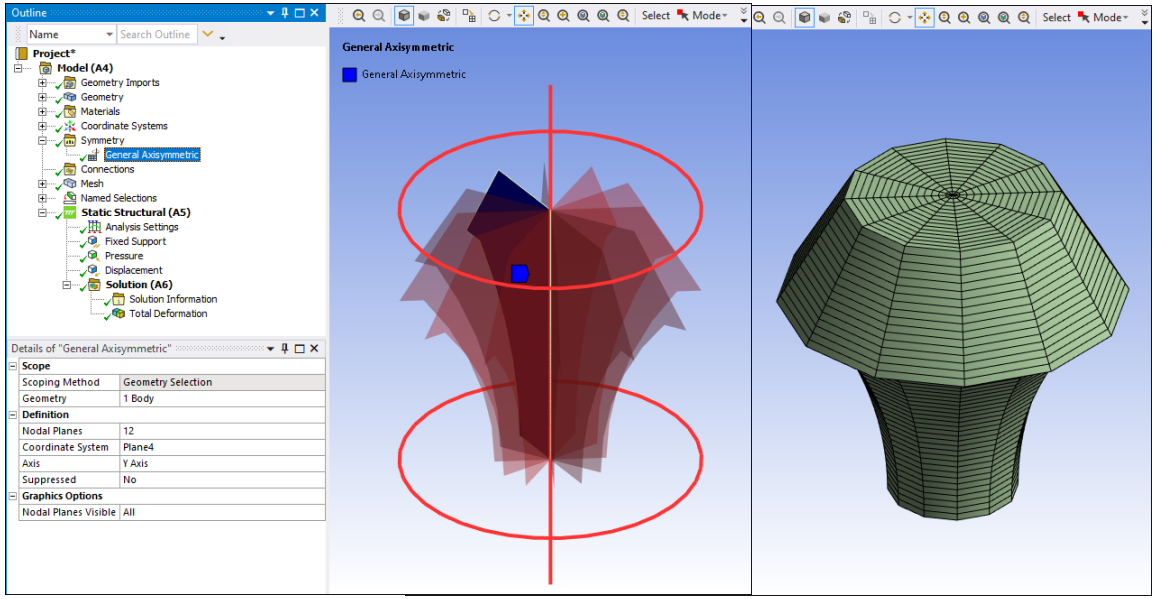
Obtain homogenized (averaged) material properties



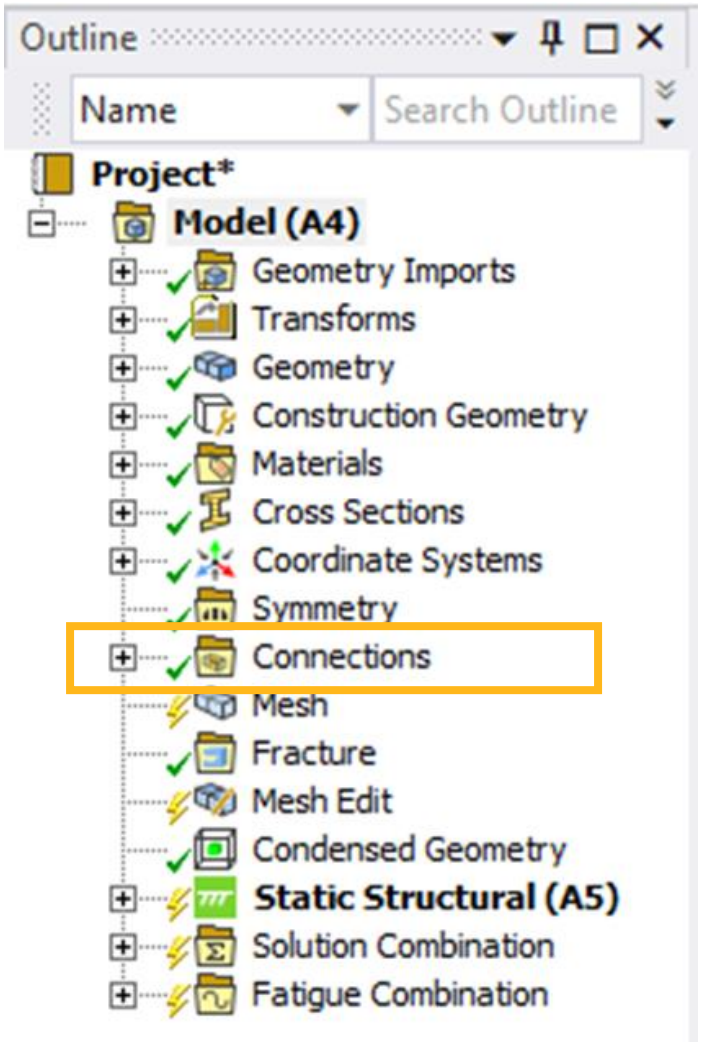
Multistage Cyclic and General Axisymmetric Symmetry

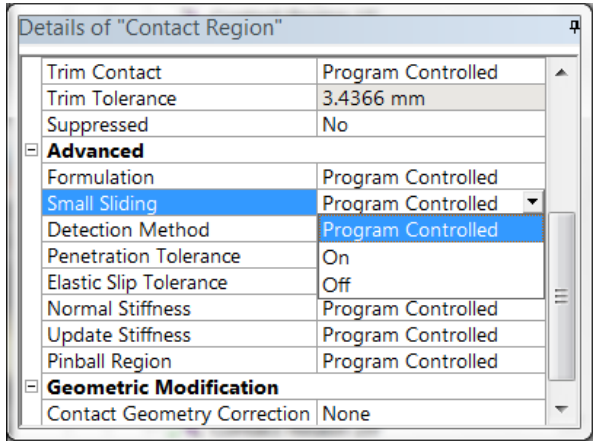




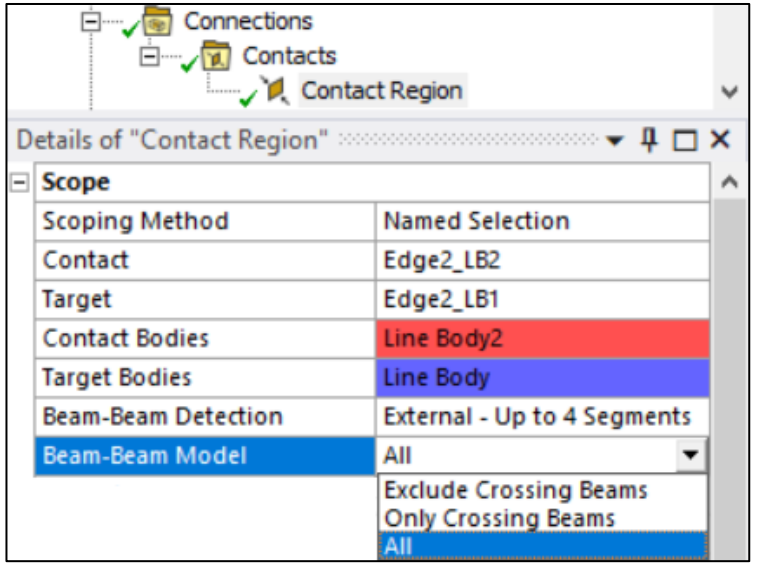


Contact

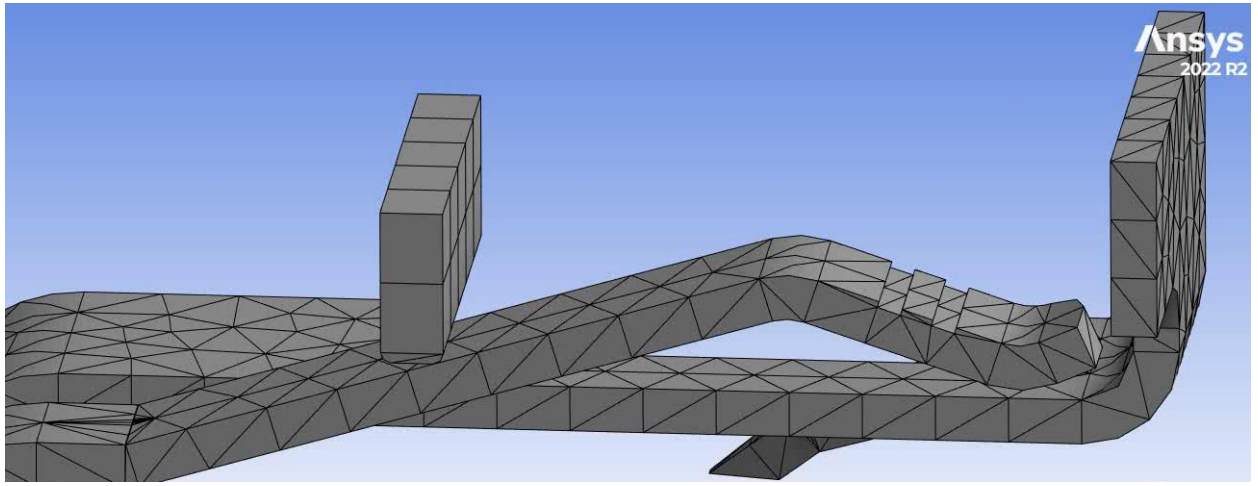




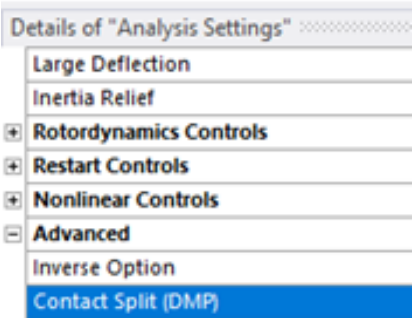
Small Sliding Contact Option



Beam to Beam Contact

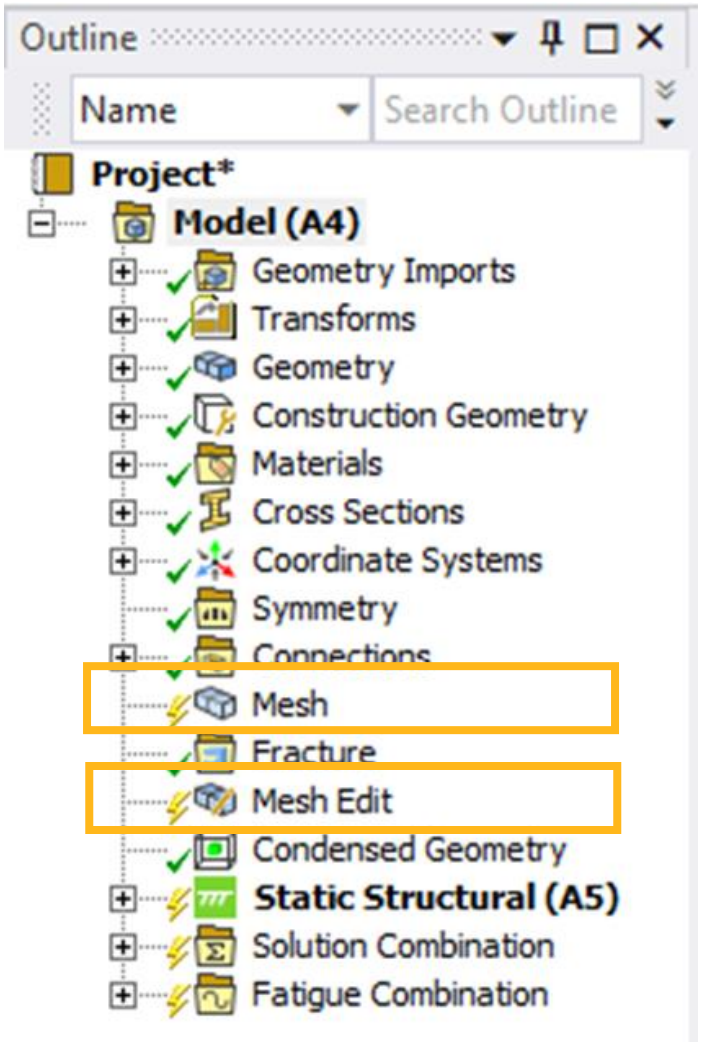


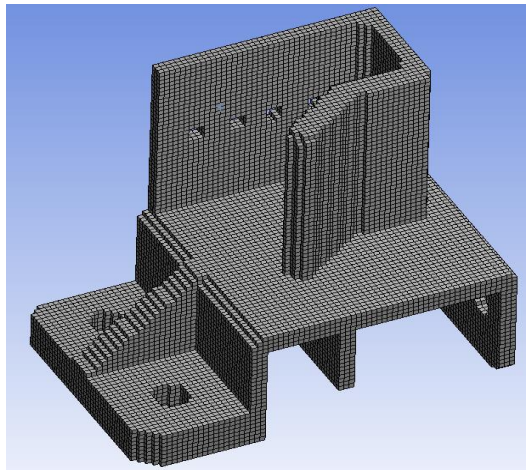
Unified Contact Detection Method

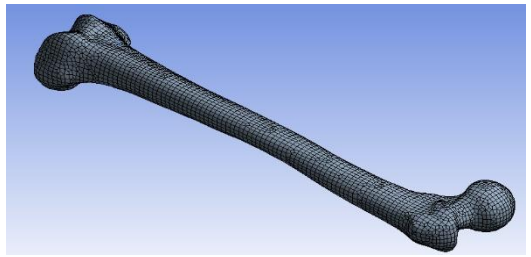


Multi-core, Distributed Contact solution

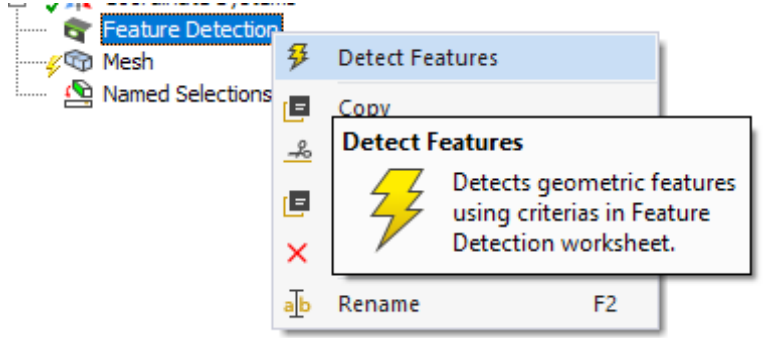
Meshing

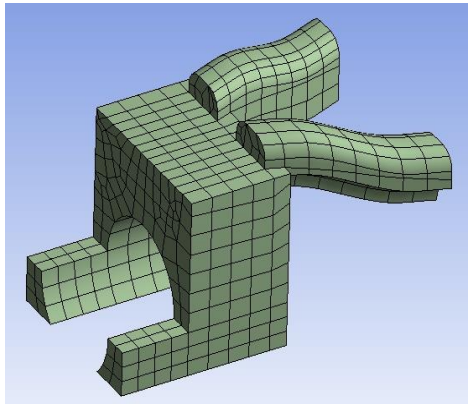


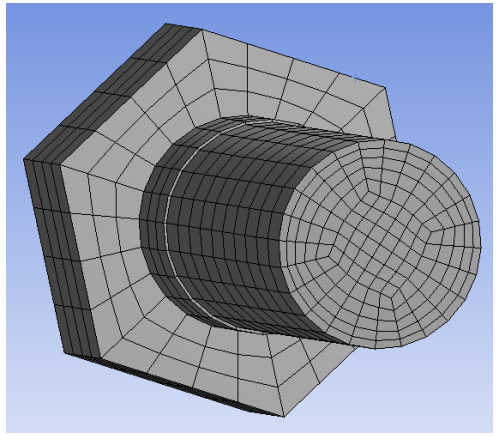




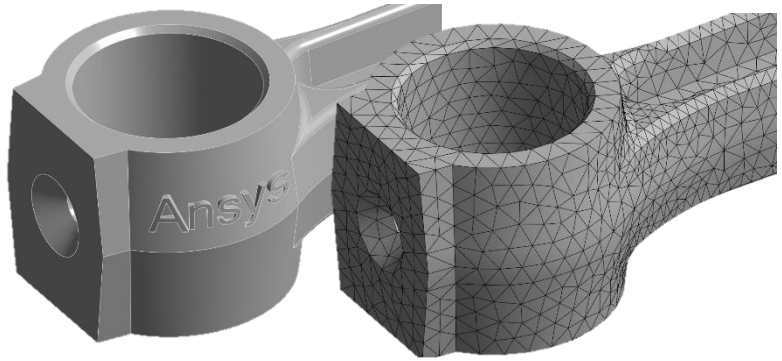
Cartesian Mesh Method





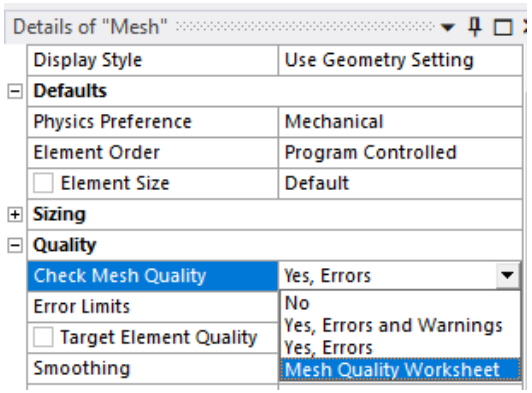


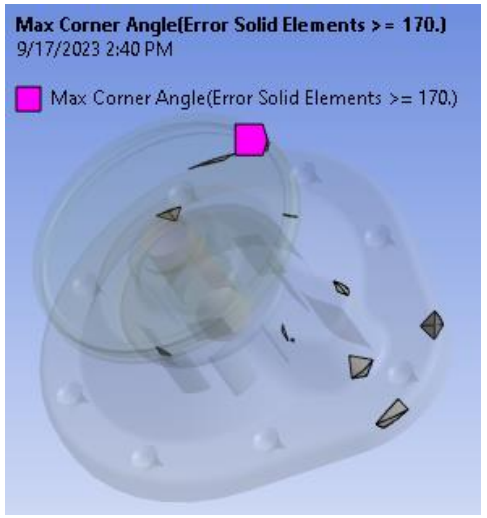
Multi-zone Mesh Method Improvements

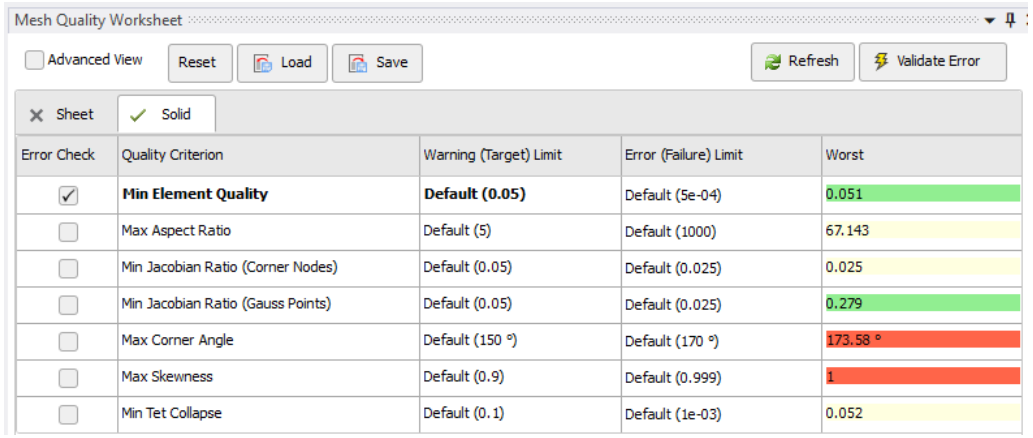


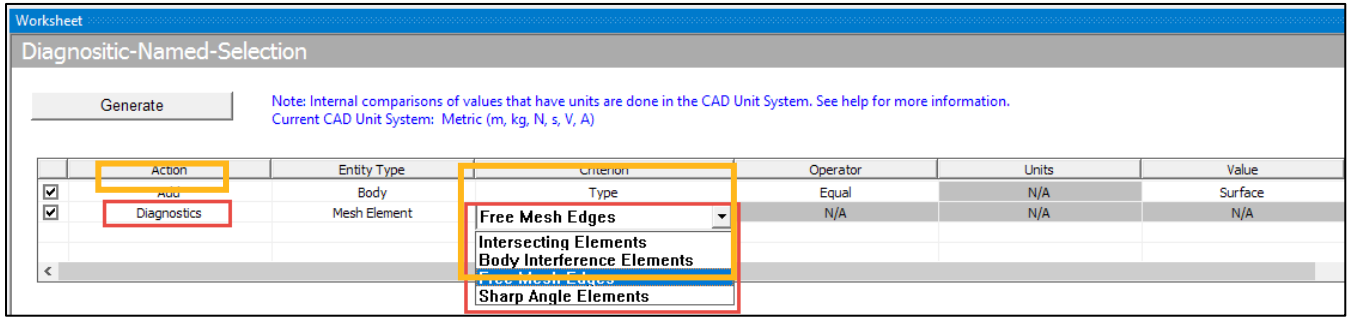
Feature detection and suppression

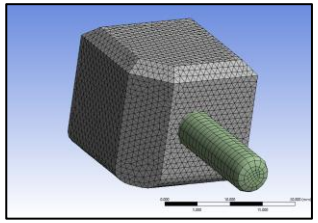
Meshing Diagnostics Tools



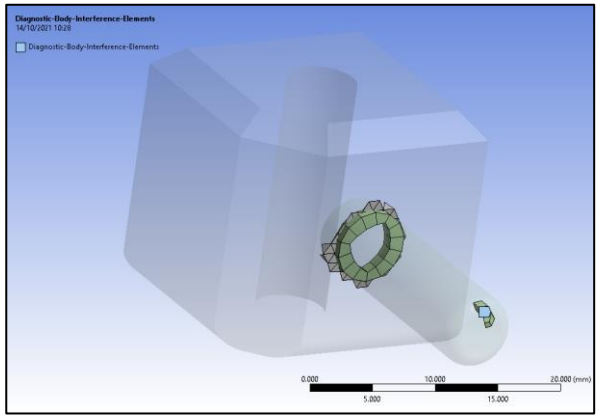


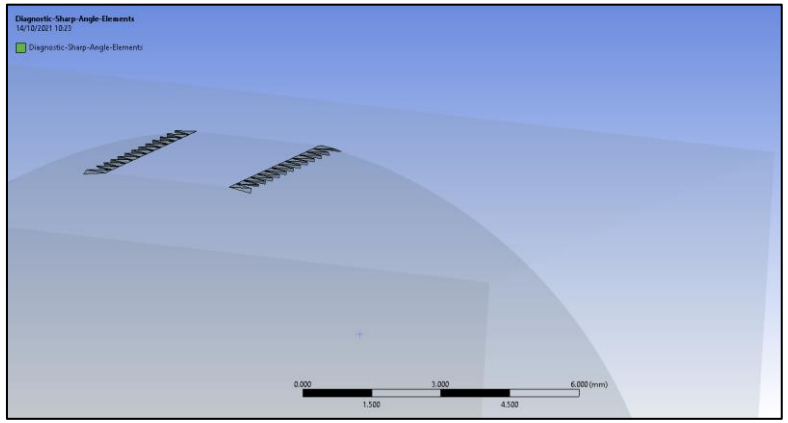






Body Interference





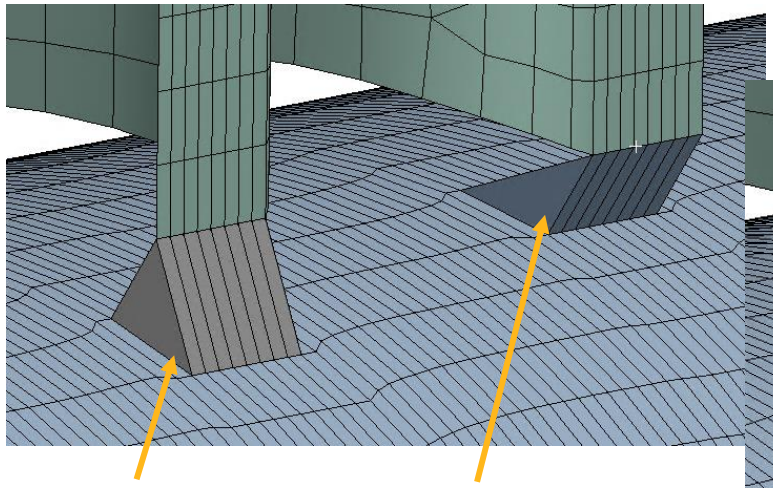
Sharp Angle Elements

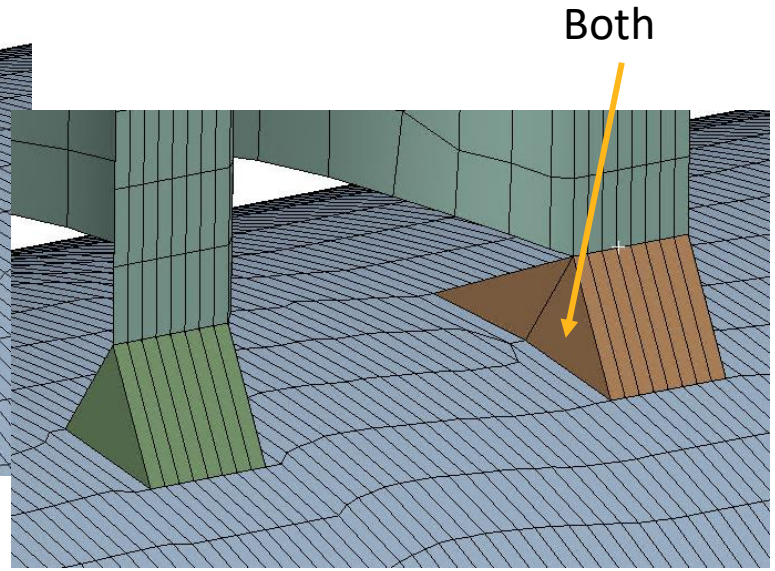
Weld Meshing

Scope	
Scoping Method	Geometry Selection
Type	Seam
Source	Geometry
Modeled As	Tent and Extension
Use Worksheet	
Extension Surfaces	
Tent Surfaces	

Tent and Extension
Tent
Extension
Parent Bodies Connection

Tent, Extension options are supported for *Source Geometry*

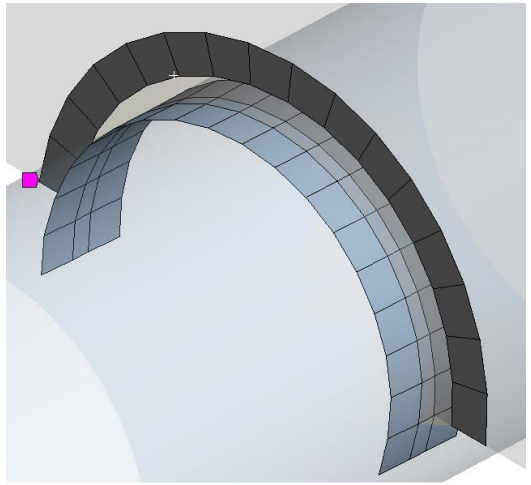




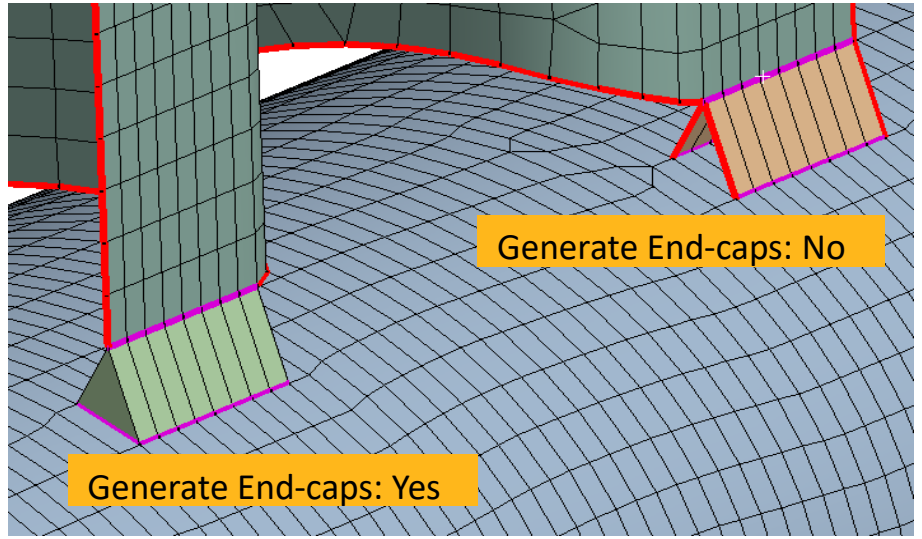
Scope	
Type	Seam
Source	Mesh
Modeled As	Tent and Extension
Create Using	Curves
Tent Direction	Normal
Use Worksheet	
Curve Scoping	
Weld Curve	1 Body

Normal
Reversed
Both

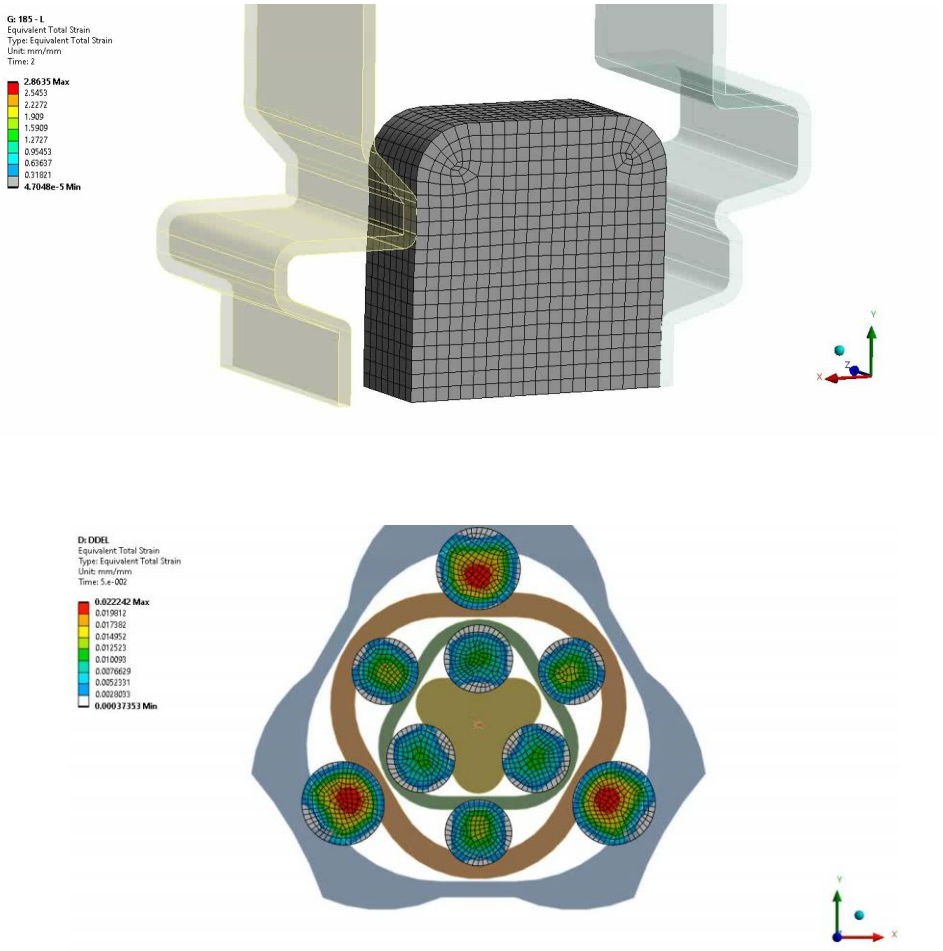
Option to control Tent face direction



Mesh with offset layers

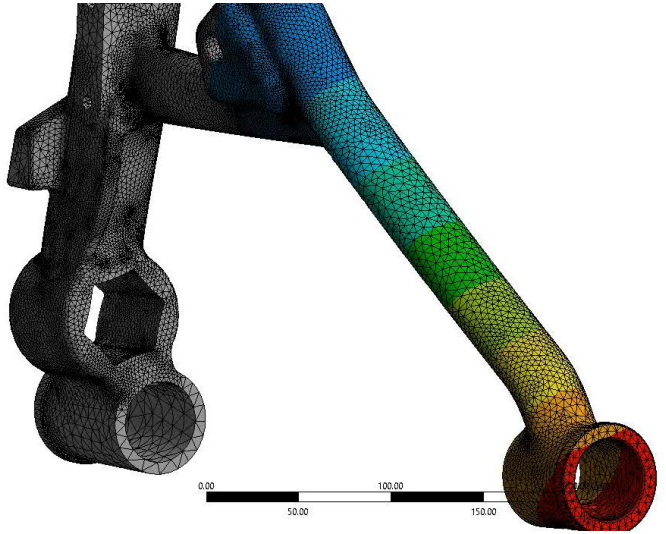


Mesh Adaptivity – NLAD, GPAD



Nonlinear Mesh Adaptivity - NLAD

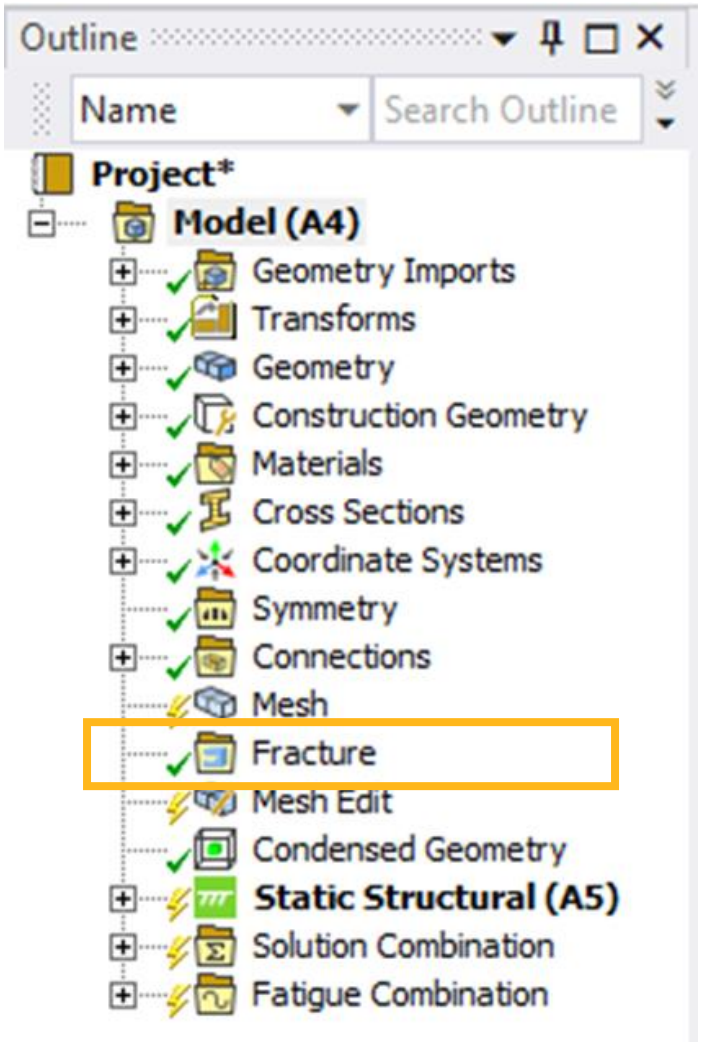
Geometry Preserving Mesh Adaptivity - GPAD

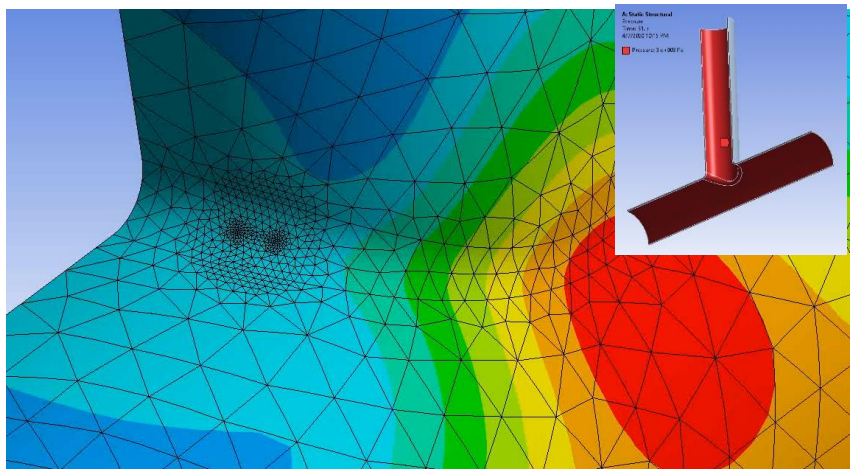


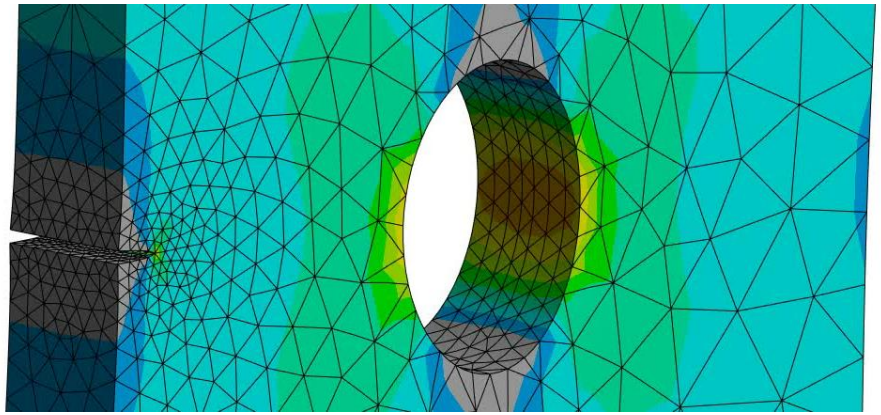
- ✓ Improved simulation accuracy on complex models and parts utilizing geometry preserving adaptivity (GPAD) that automatically refines a mesh based on the initial geometry
- ✓ Eliminates the need for an over-refined mesh or advanced user knowledge

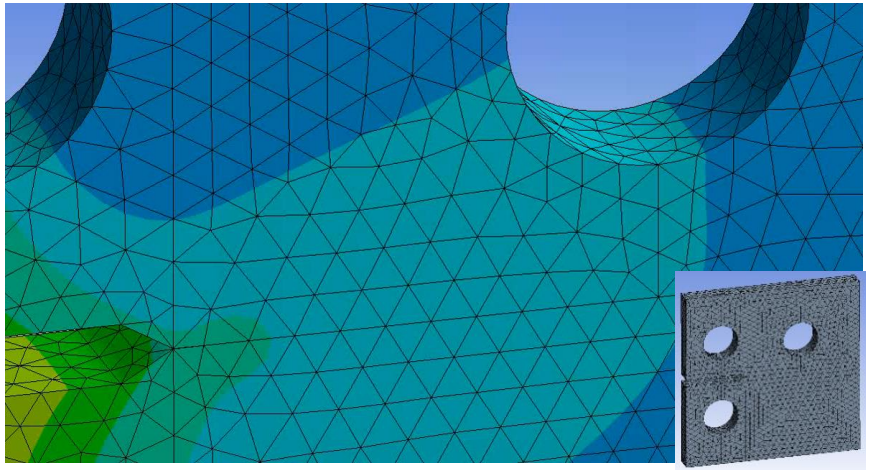
SMART crack modeling

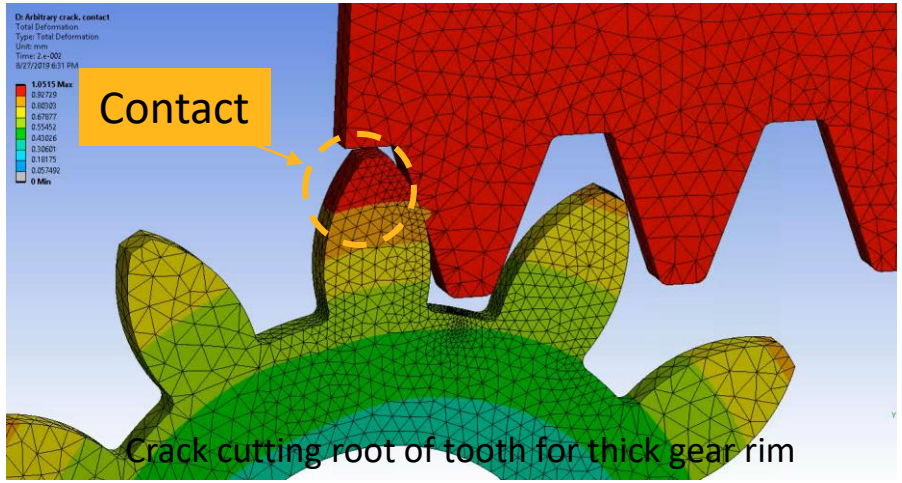
Support multiple pre-existing cracks and multiple crack initiations





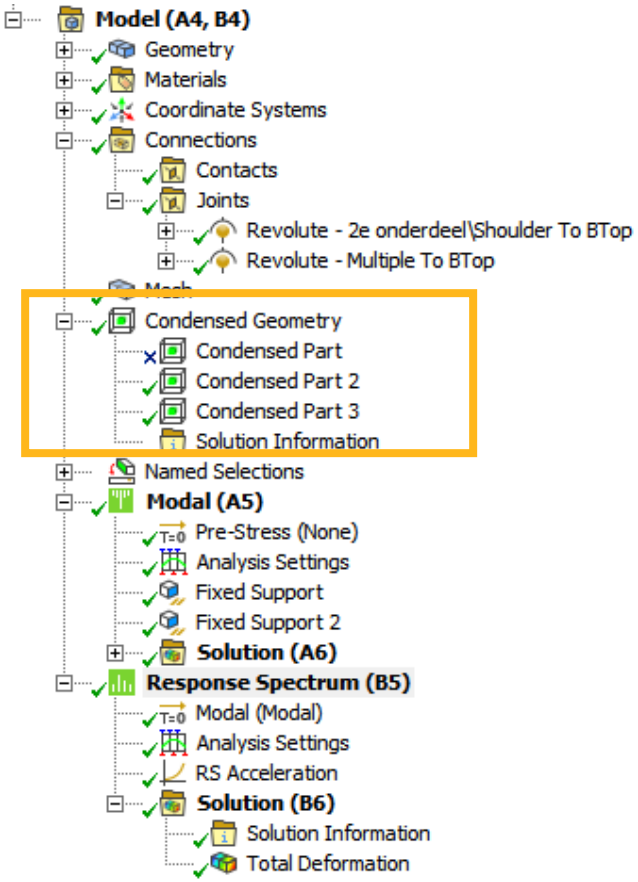


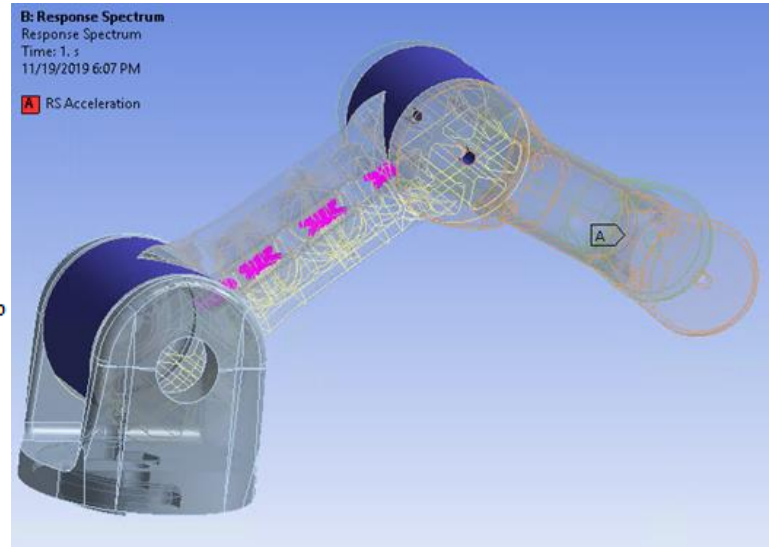


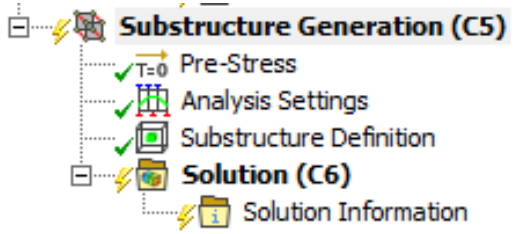


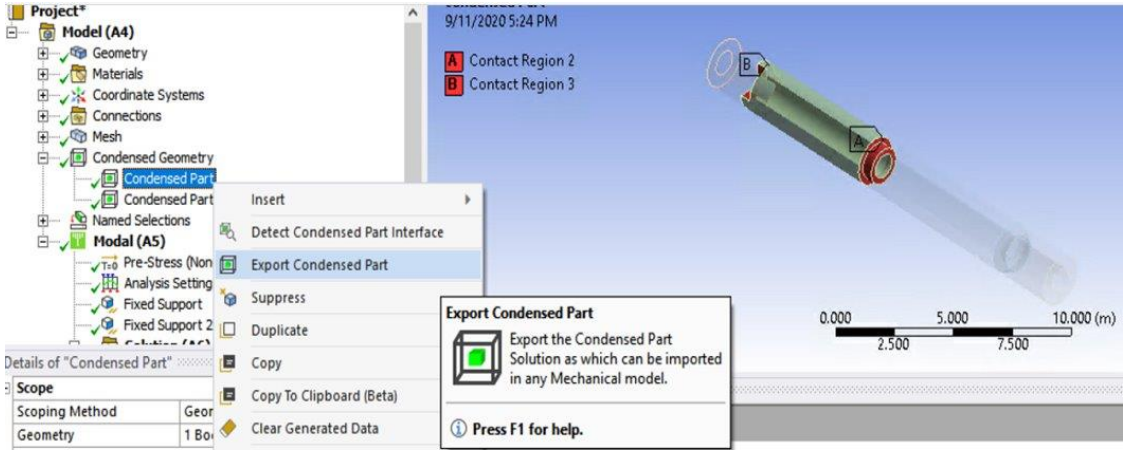
Crack cutting root of tooth for thick gear rim

Substructuring in Mechanical (Top Down/Bottom Up)

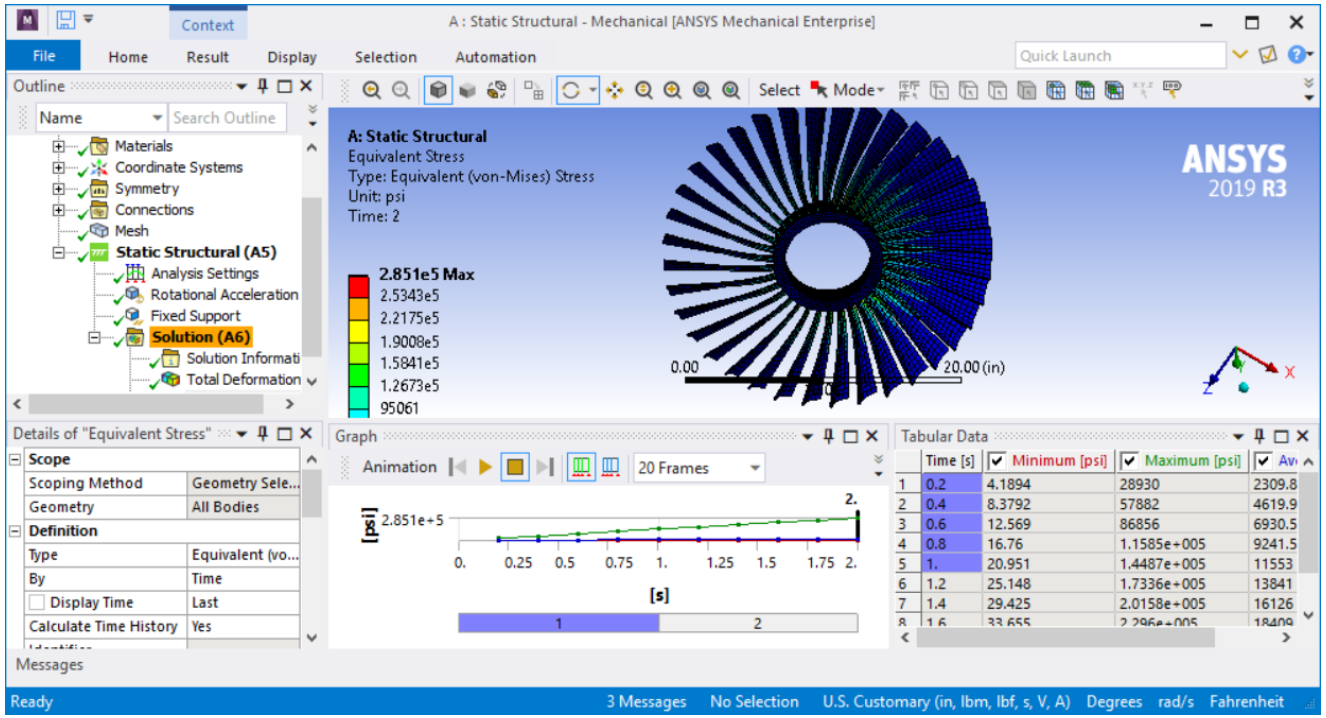


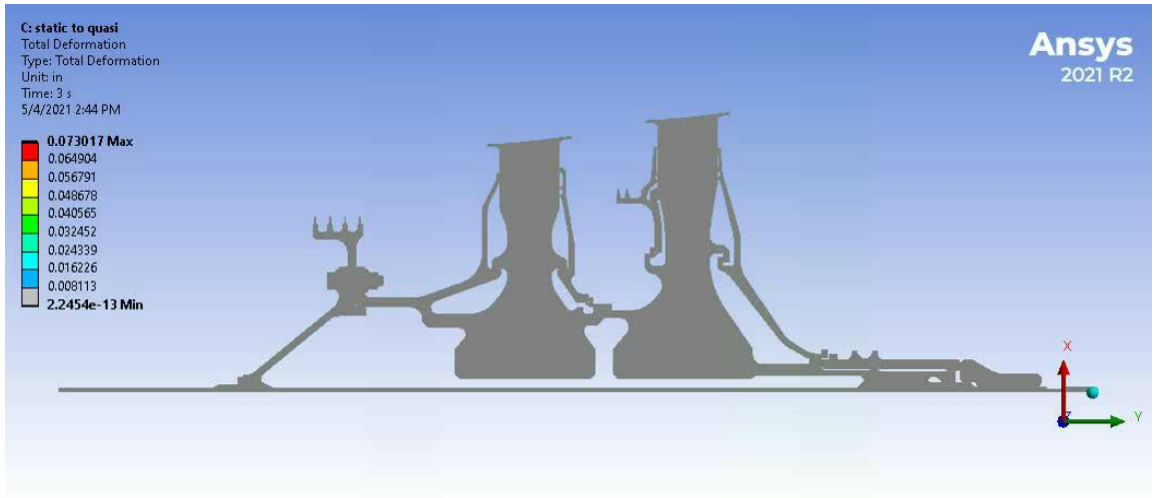


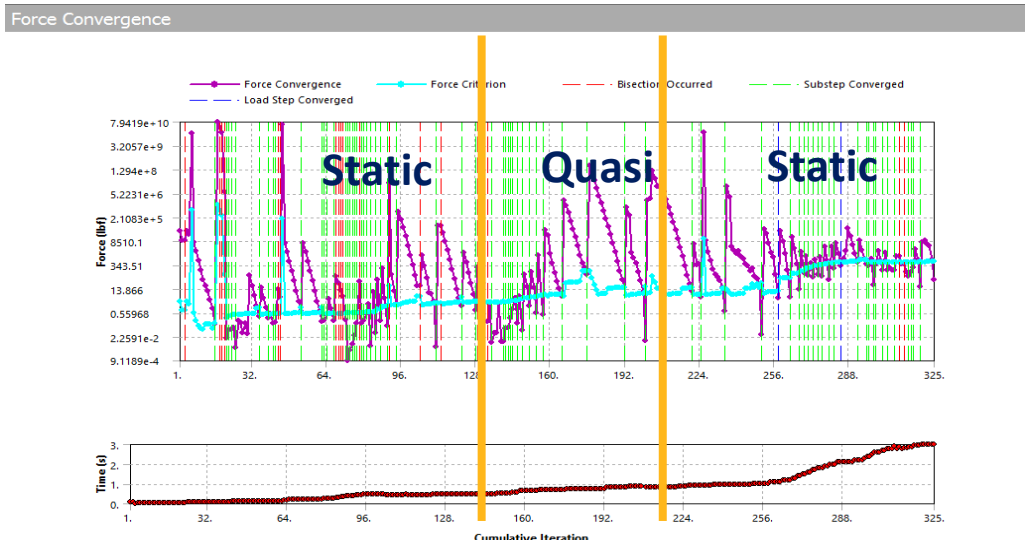




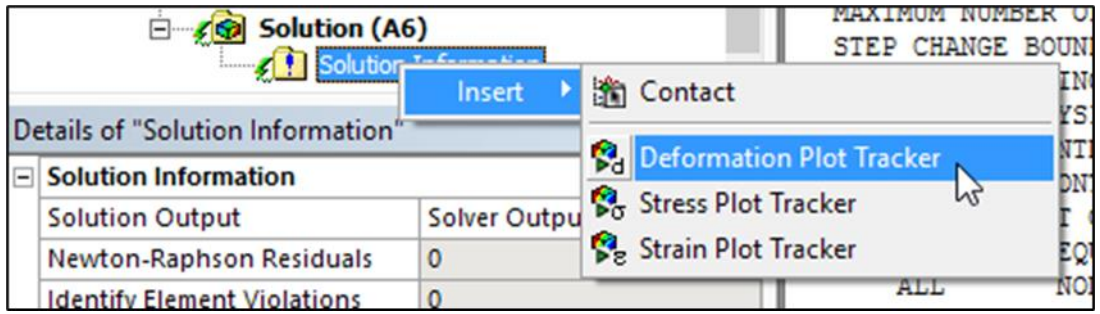
Solution - Inverse Analysis, Automatic “Static to Transient”

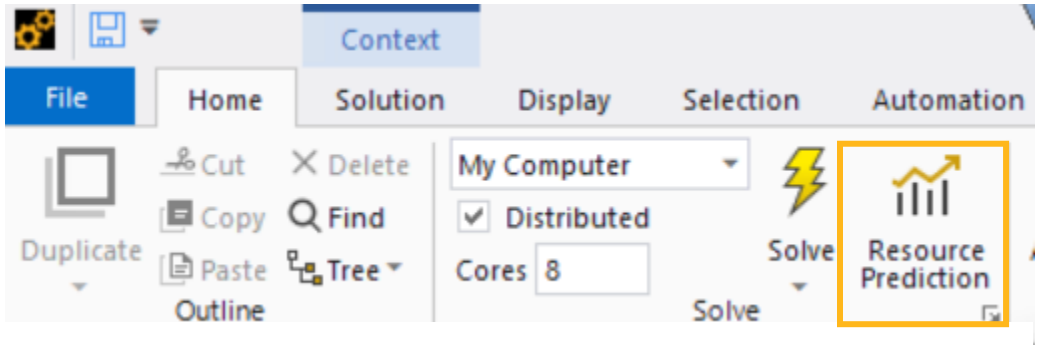


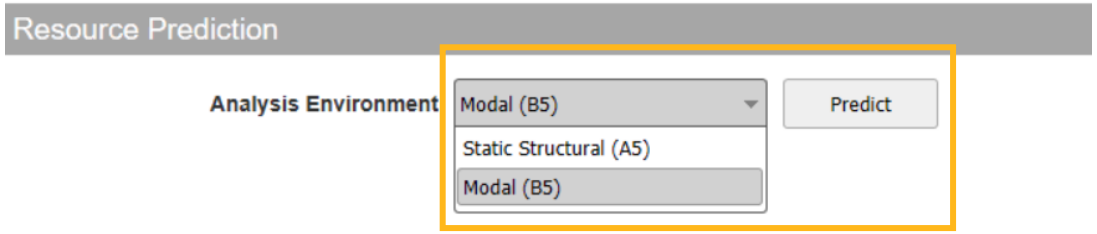


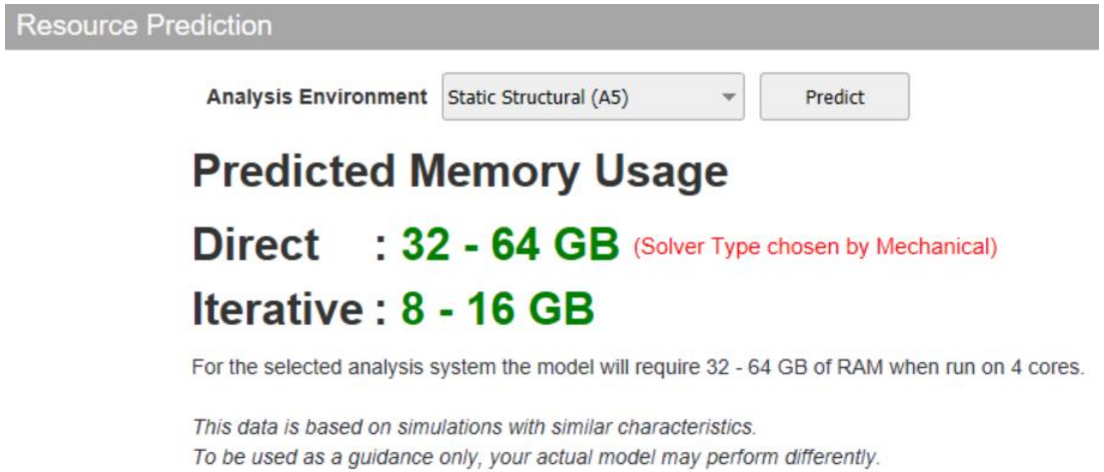


Solution - Resource Prediction, On the Fly Solution Tracking

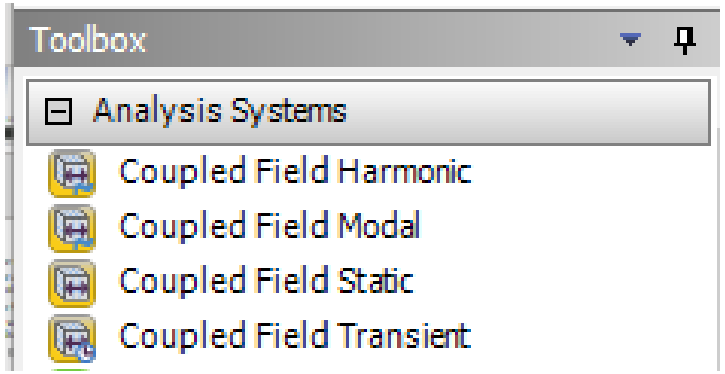


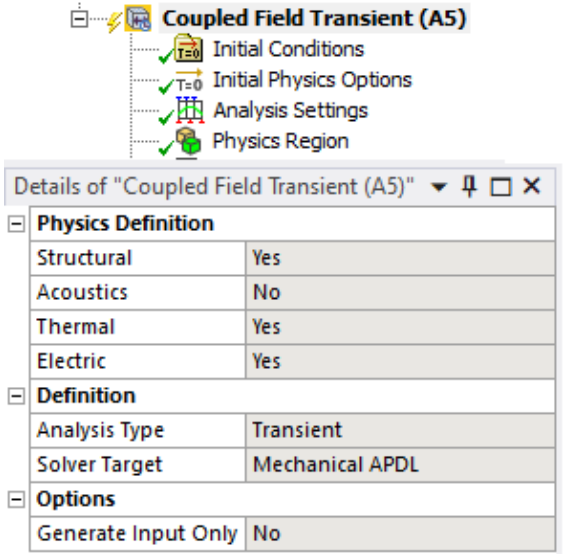




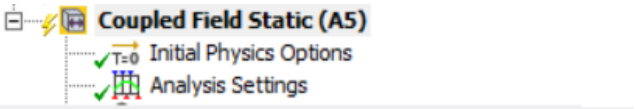


Solution – Coupled Field Problems



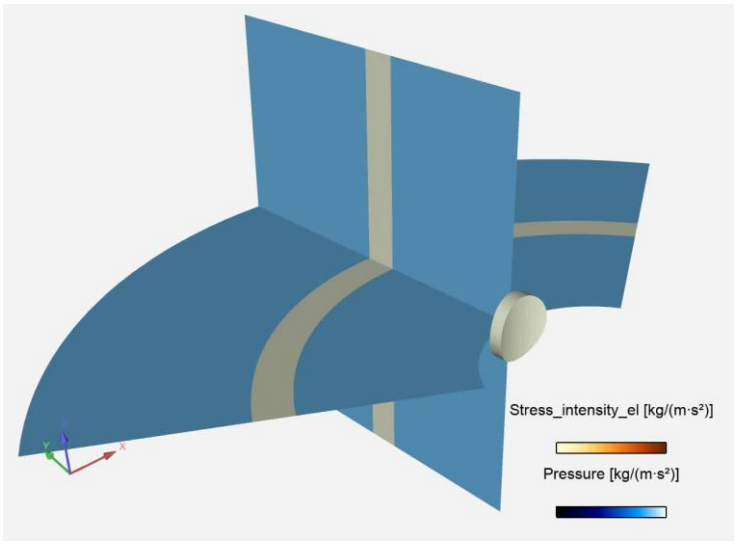


Physics Definition	
Structural	Yes
Acoustics	No
Thermal	Yes
Electric	Yes
Definition	
Analysis Type	Transient
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

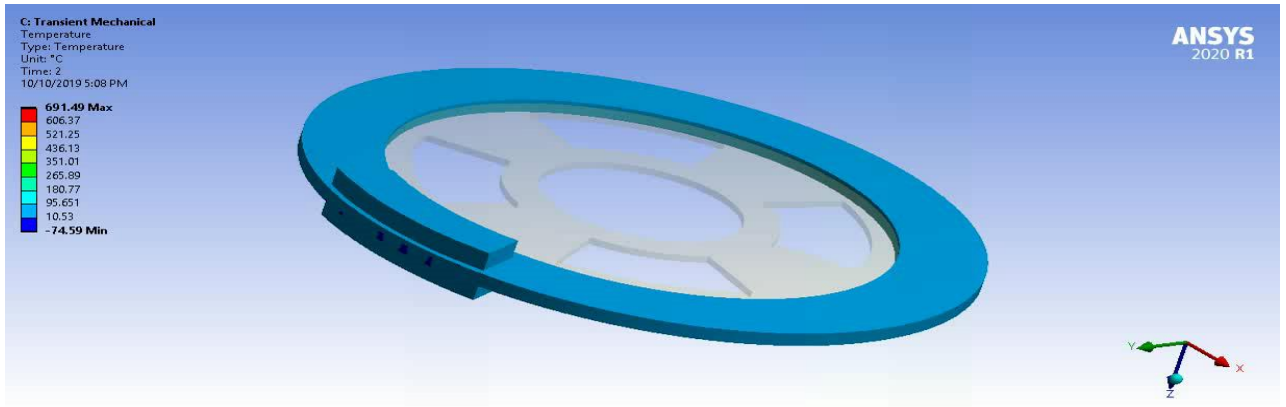


Details of "Coupled Field Static (A5)"

Physics Definition	
Structural	Yes
Acoustics	No
Thermal	No
Electric	Yes
Definition	
Analysis Type	Static
Solver Target	Mechanical APDL
Options	
Generate Input Only	No



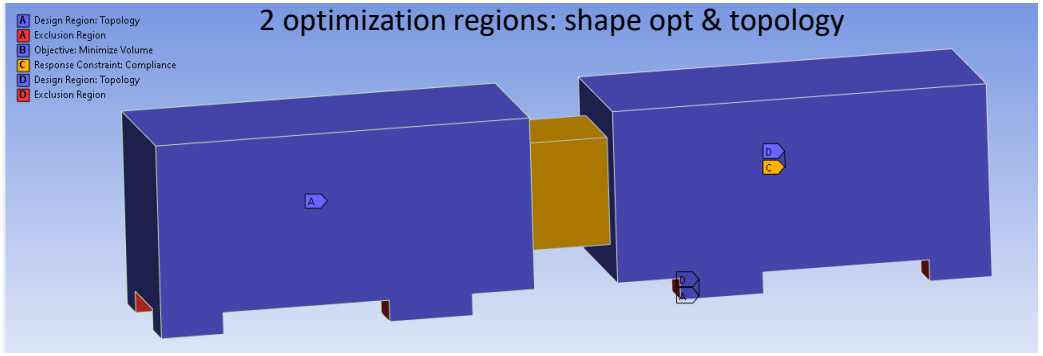
Piezoelectric Acoustics

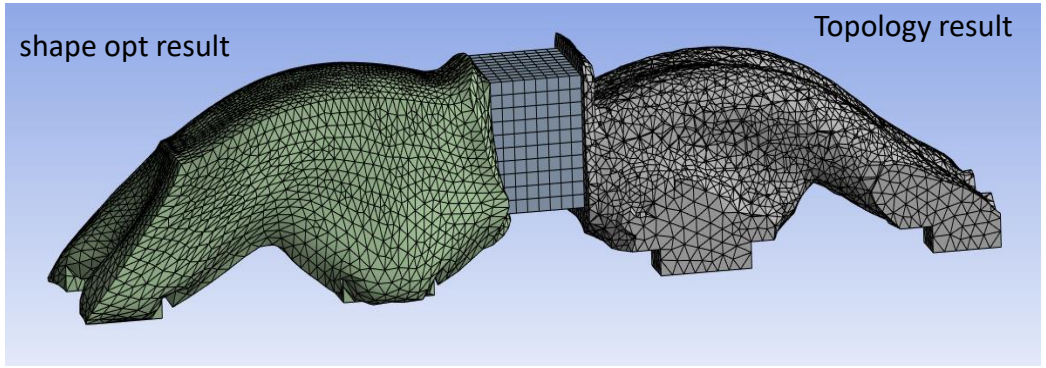


Structural Thermal

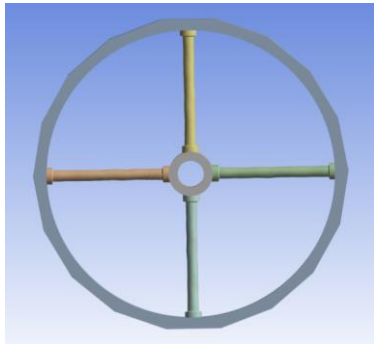
Structural Optimization – Shape, Topology, Topography, Lattice

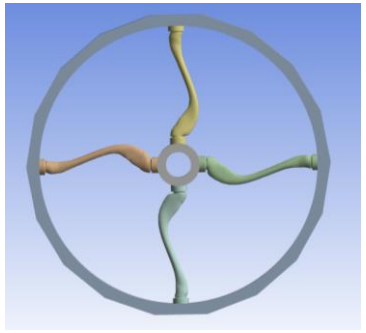
Mixed Shape and Topology



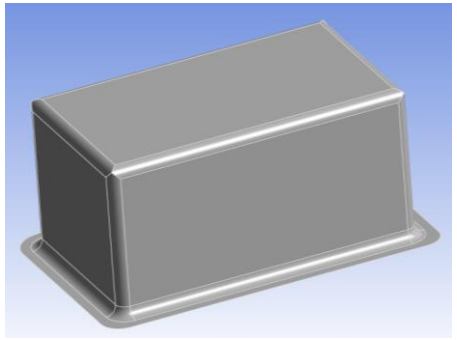


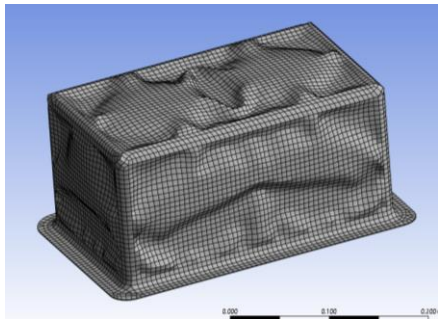
Shape



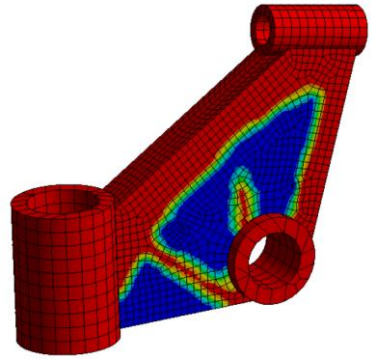


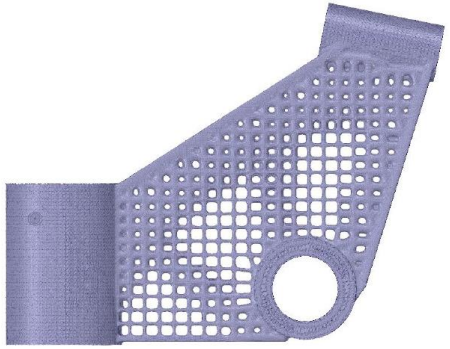
Topography





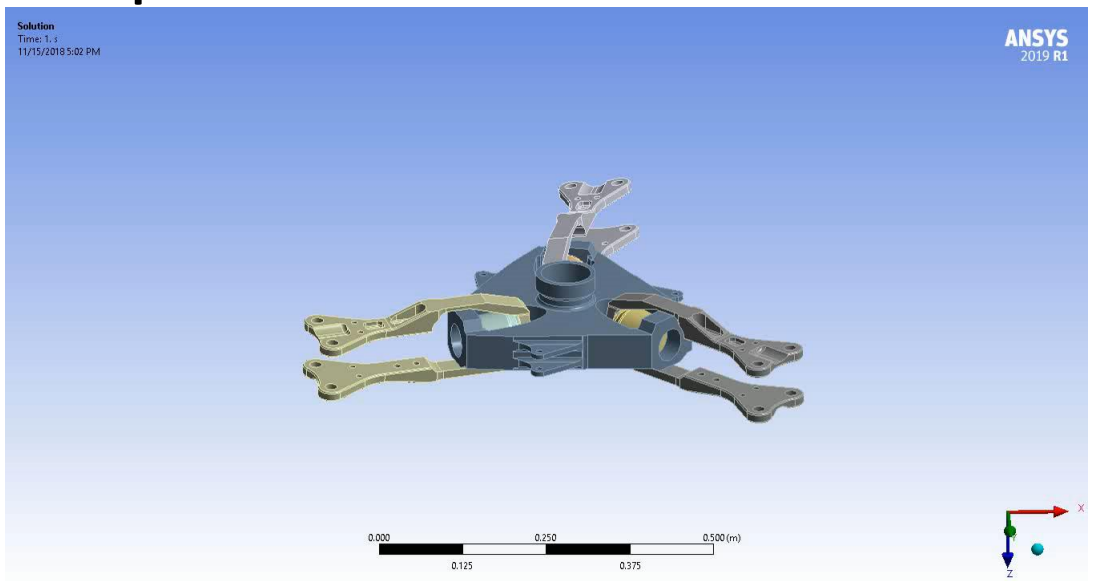
Lattice

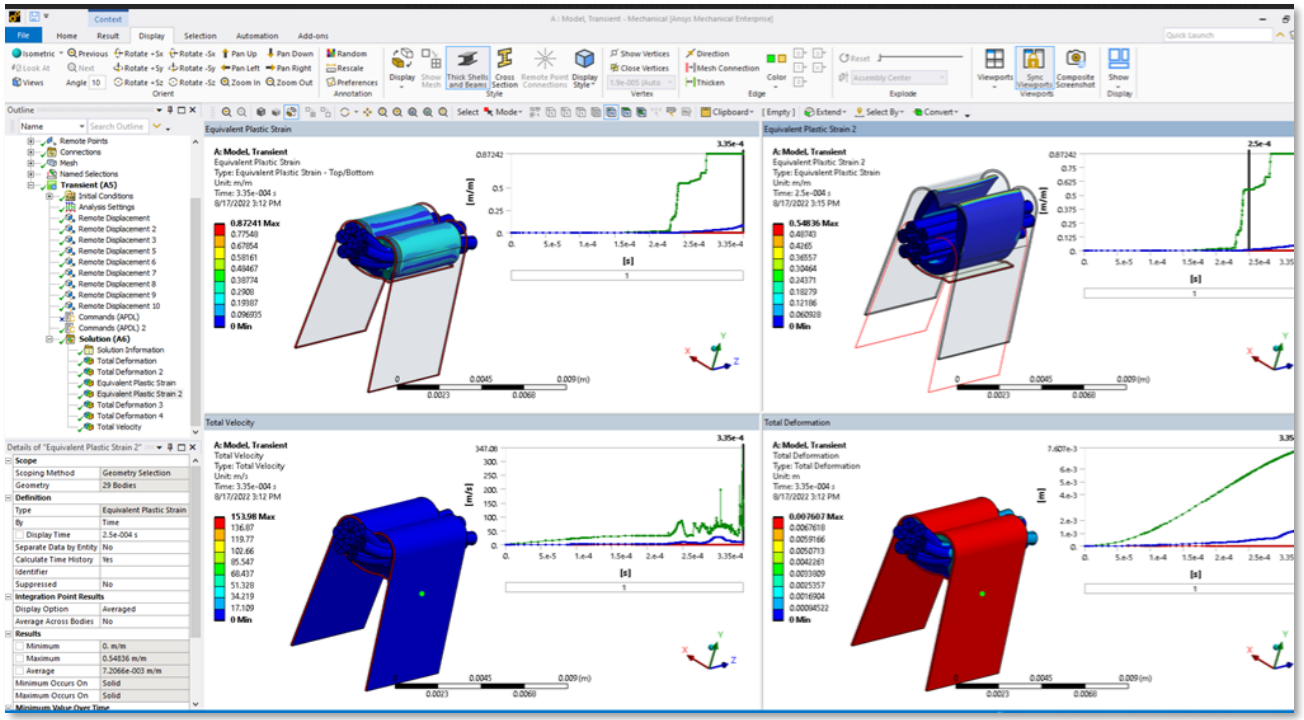




Source:
CADFEM

Post Processing - Keyframe Animation, Embedded Graphs, Separated Data





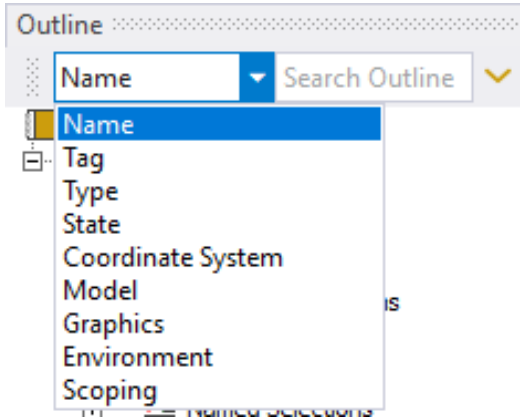
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
By	Time
Display Time	Last
Separate Data by Entity	Yes
Calculate Time History	Yes
Identifier	

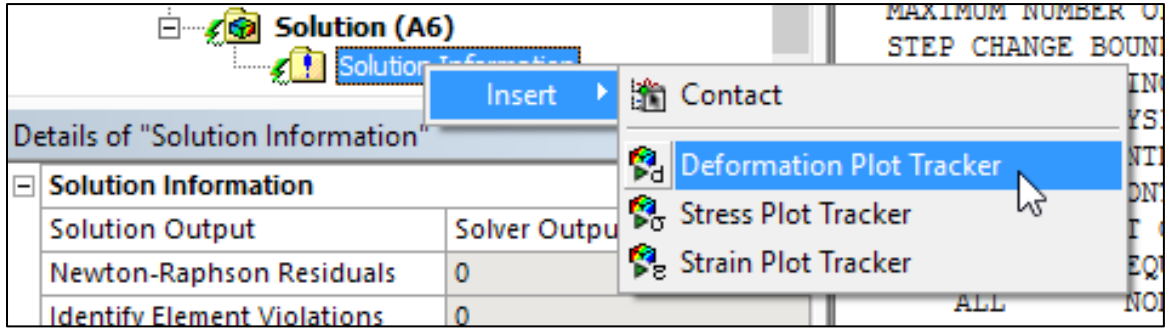
Body ID

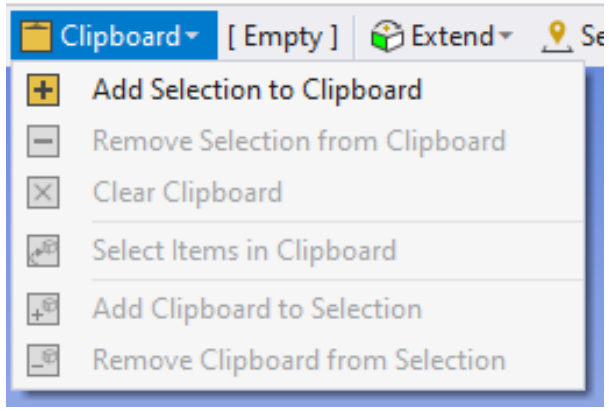
Tabular Data

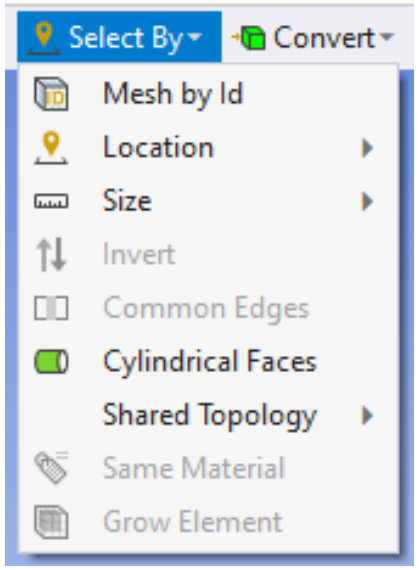
	Time [s]	Min 1 - Reference 6 [m]	Max 1 - Reference 6 [m]	Min 2 - Reference 33 [m]	Max 2 - Reference 33 [m]	Min 3 - Reference 60 [m]	Max 3 - Reference 60 [m]
1	1.	7.3201e-006	9.5788e-006	9.7814e-007	7.4912e-006	0.	3.5401e-006
2	2.	1.464e-005	1.9158e-005	1.9563e-006	1.4982e-005	0.	7.0803e-006
3	3.	2.196e-005	2.8736e-005	2.9344e-006	2.2474e-005	0.	1.062e-005
4	4.	2.928e-005	3.8315e-005	3.9126e-006	2.9965e-005	0.	1.4161e-005
5	5.	3.6601e-005	4.7894e-005	4.8907e-006	3.7456e-005	0.	1.7701e-005
6	6.	4.3921e-005	5.7473e-005	5.8688e-006	4.4947e-005	0.	2.1241e-005
7	7.	5.1241e-005	6.7052e-005	6.847e-006	5.2438e-005	0.	2.4781e-005
8	8.	5.8561e-005	7.6631e-005	7.8251e-006	5.9929e-005	0.	2.8321e-005
9	9.	6.5881e-005	8.6209e-005	8.8033e-006	6.7421e-005	0.	3.1861e-005
10	10.	7.3201e-005	9.5788e-005	9.7814e-006	7.4912e-005	0.	3.5401e-005

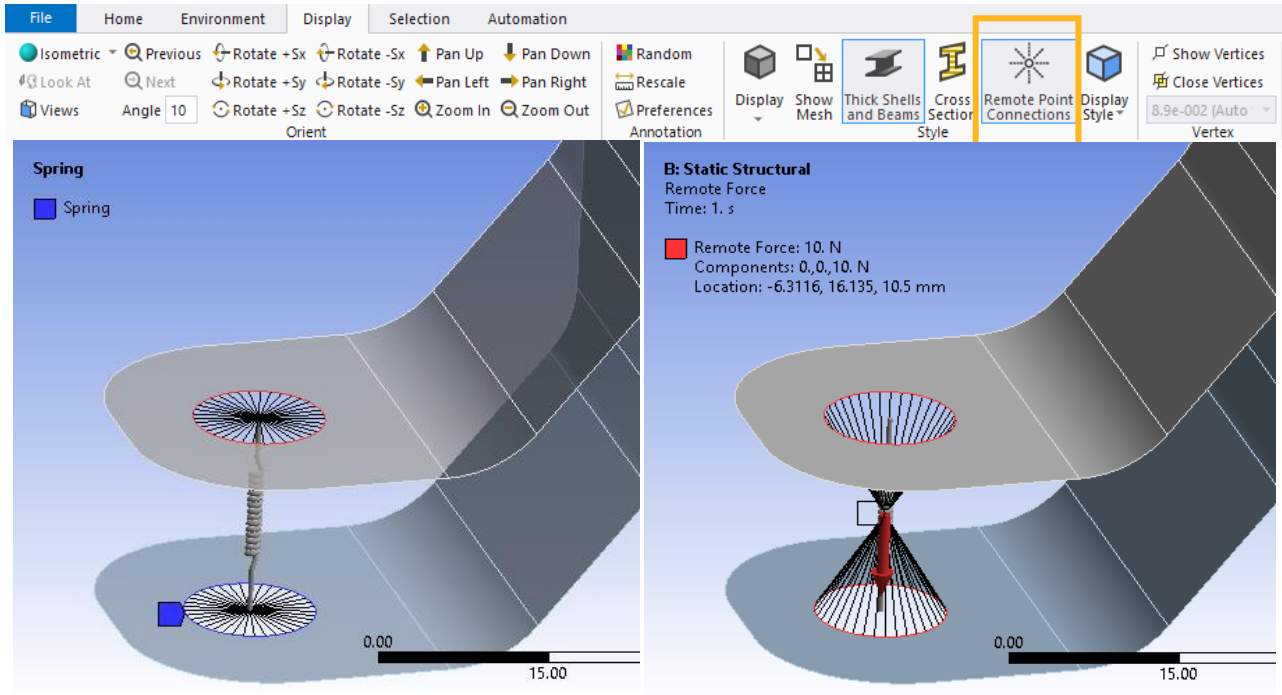
Some Quality of Life Improvements …



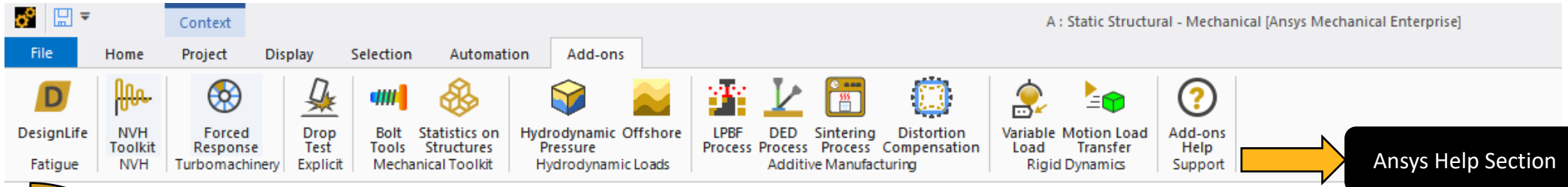






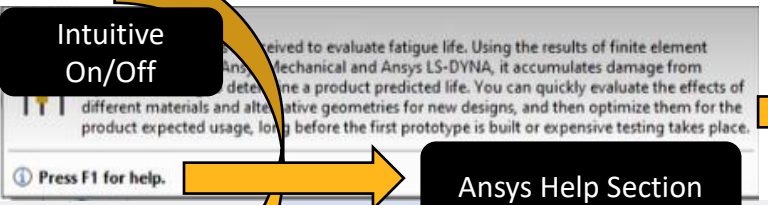


Ansys Mechanical Add-ons



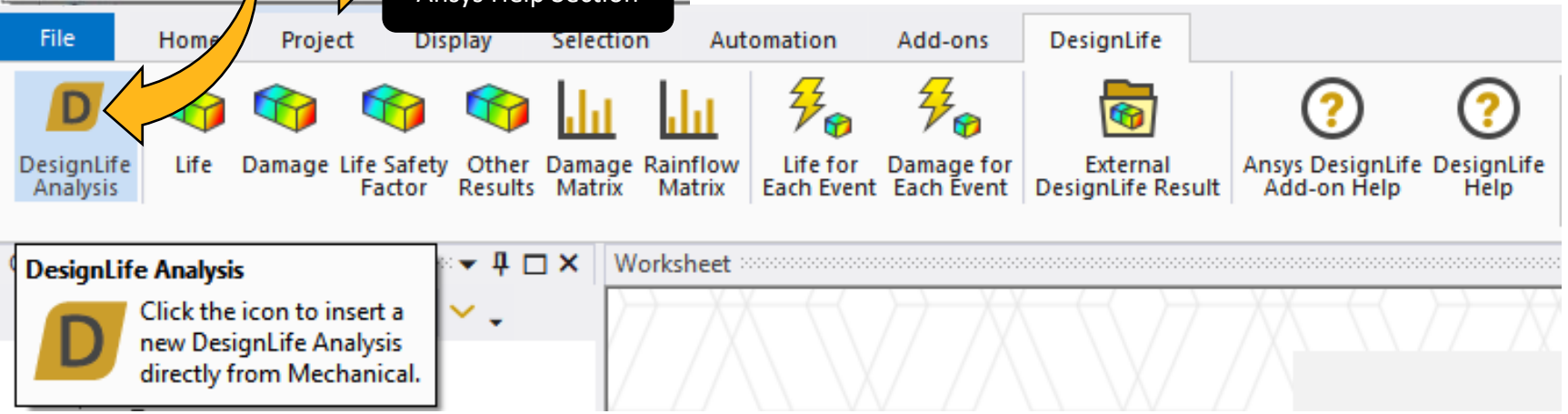
Ansys Help Section

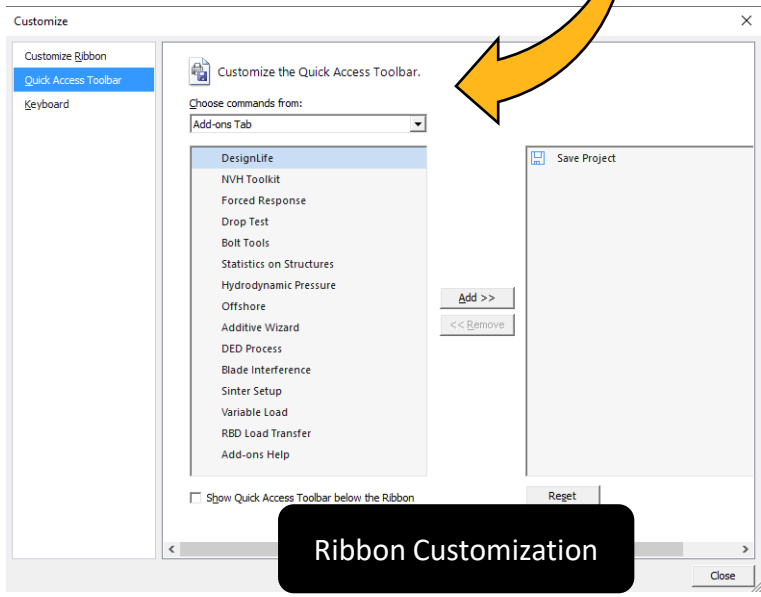
Intuitive On/Off



Tool Tip

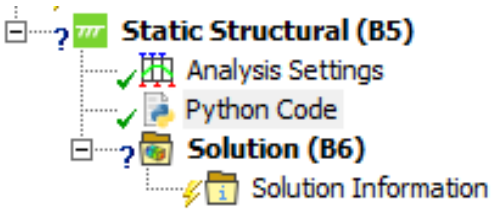
Ansys Help Section





Ribbon Customization

Python Code for Control and Custom Result Processing

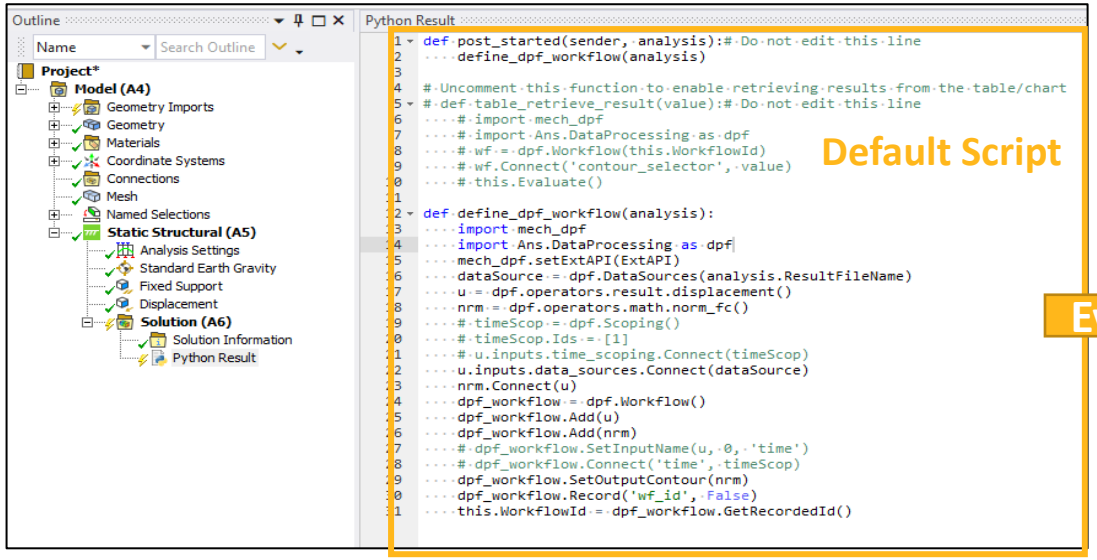


Details of "Python Code"

Definition	
Target Callback	Before Solve
Suppressed	No
Connected	True
Advanced	
Group 1	
Double Property 1	3
Group 2	
Define By	Geometry Selection
Geometry Selection	3 Faces
Options Property	Value 1

Python Code

```
def before_solve(this, analysis):# Do not edit this line
    """
    Called before solving the parent analysis.
    Keyword Arguments :
        this -- the datamodel object instance of the python code object you have selected in the tree
        analysis -- Static Structural
    """


    #To access properties created using the Property Provider, please use the following command.
    #this.GetCustomPropertyByPath("your_property_group_name/your_property_name")

    #To access scoping properties use the following to access geometry scoping and named selection respectively:
    #this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Geometry Selection")
    #this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Named Selection")

    pass
```

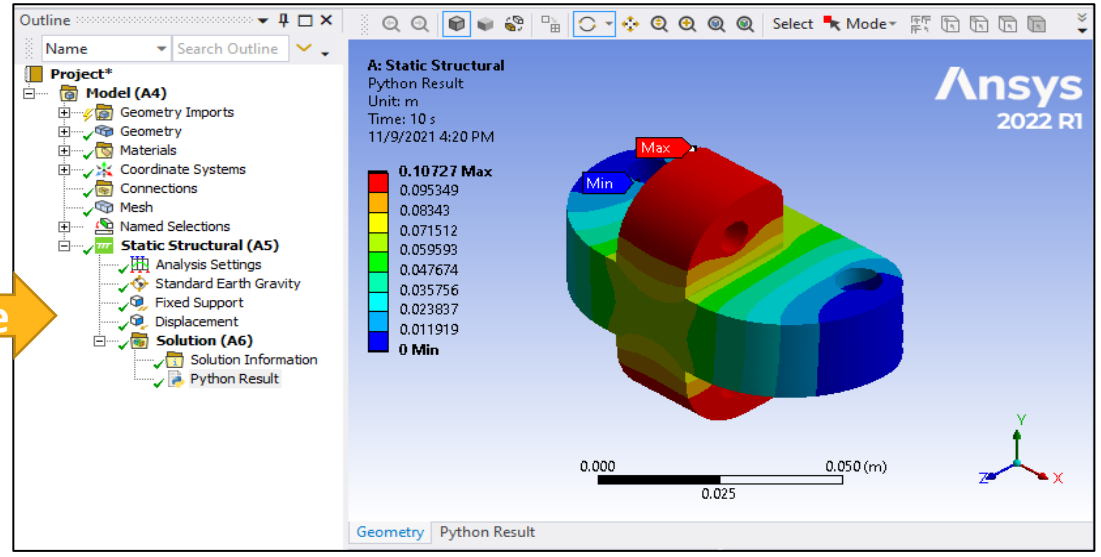


Python Result

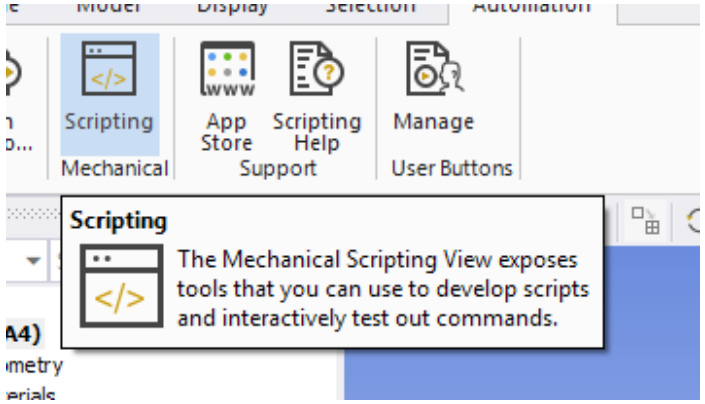
```
def post_started(sender, analysis):# Do not edit this line
    define_dpf_workflow(analysis)

# Uncomment this function to enable retrieving results from the table/chart
# def table_retrieve_result(value):# Do not edit this line
    # import mech_dpf
    # import Ans.DataProcessing as dpf
    # wf = dpf.Workflow(this.WorkflowId)
    # wf.Connect('contour_selector', value)
    # this.Evaluate()

def define_dpf_workflow(analysis):
    import mech_dpf
    import Ans.DataProcessing as dpf
    mech_dpf.setExtAPI(ExtAPI)
    dataSource = dpf.DataSources(analysis.ResultFileName)
    u = dpf.operators.result.displacement()
    nrm = dpf.operators.math.norm_fc()
    # timeScop = dpf.Scoping()
    # timeScop.Ids = [1]
    # u.inputs.time_scoping.Connect(timeScop)
    u.inputs.data_sources.Connect(dataSource)
    nrm.Connect(u)
    dpf_workflow = dpf.Workflow()
    dpf_workflow.Add(u)
    dpf_workflow.Add(nrm)
    # dpf_workflow.SetInputName(u, 0, 'time')
    # dpf_workflow.Connect('time', timeScop)
    dpf_workflow.SetOutputContour(nrm)
    dpf_workflow.Record('wf_id', False)
    this.WorkflowId = dpf_workflow.GetRecordedId()
```

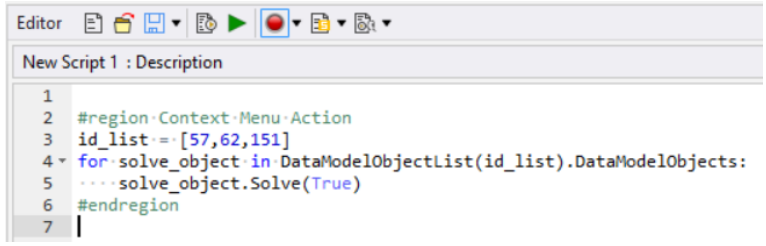
Default Script

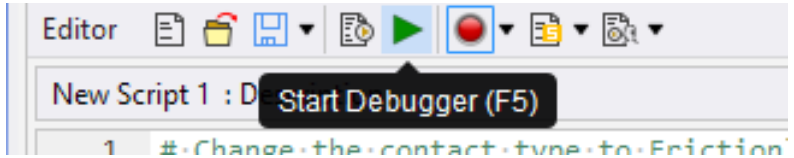
Evaluate

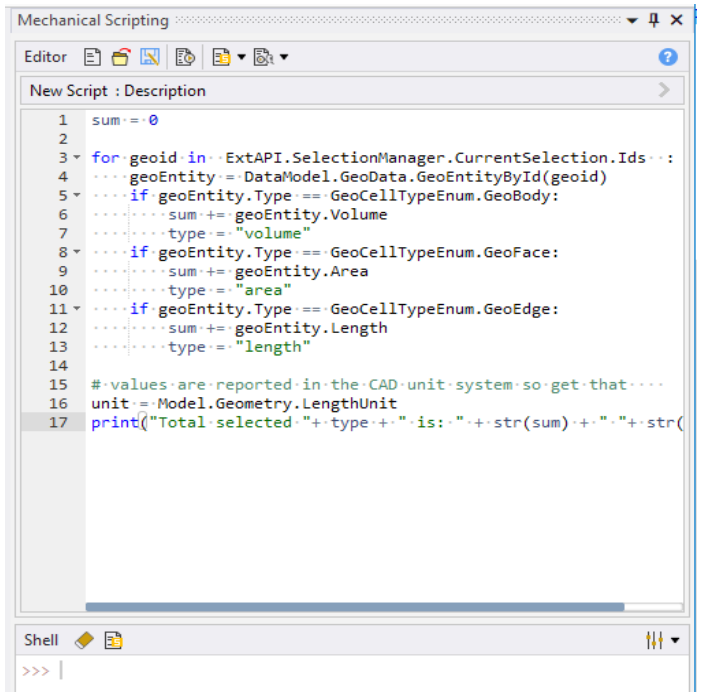


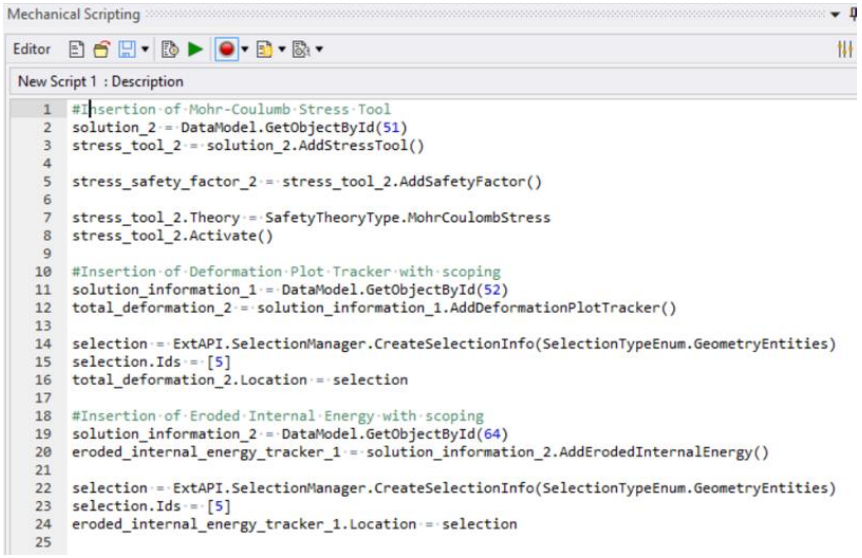
Mechanical Scripting, Recording, Debugging

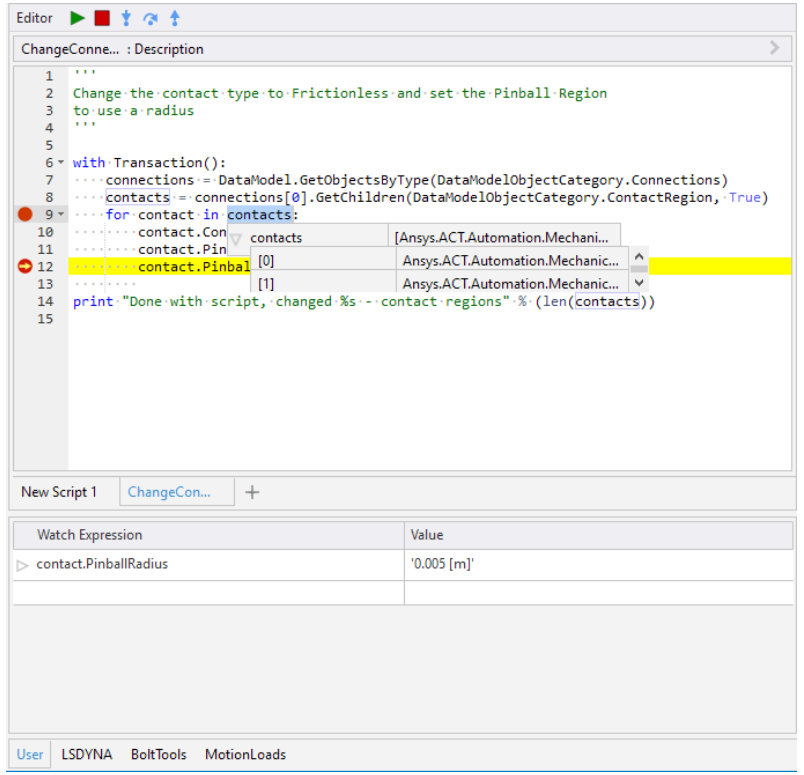










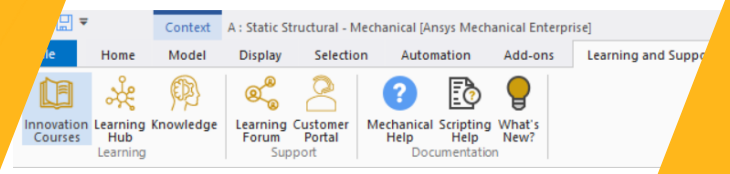


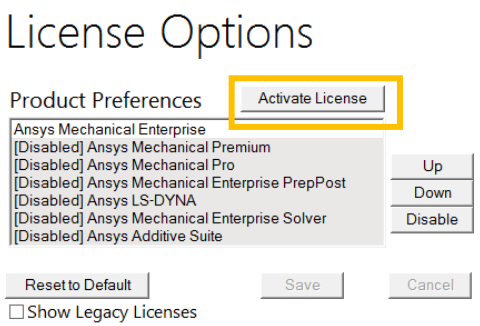
2023 R2

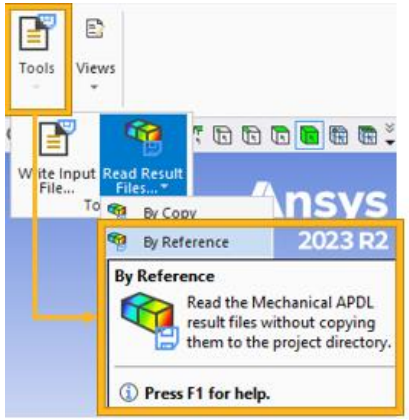
Highlights

Ansys

Mechanical







Learning and Support Tab

- ✓ Give user access to Innovation Courses, Learning Hub, Knowledge, Learning Forum, Customer Portal by adding Learning and Support Tab in Mechanical.
- ✓ Add access to Mechanical Help, Scription Help, and What's New to this tab.

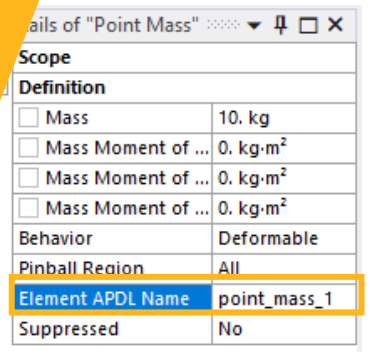
Activate License within Mechanical

- ✓ If a license is not available/checked out when you open Mechanical, the application automatically opens in Read-Only Configuration mode
- ✓ From the License Options pane select a license and click the activate button. This removes the application from read-only mode and activates Mechanical
- ✓ This option is only available when no license is checked out and the application is in Read-Only Configuration mode

Read Result Files by Reference

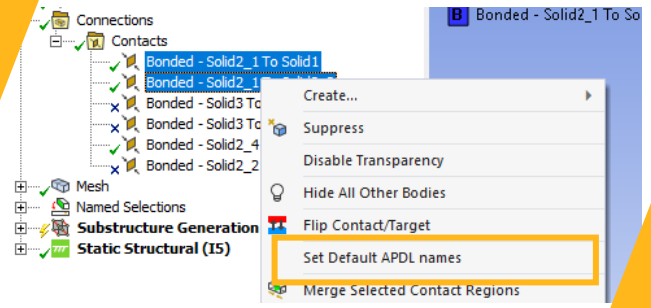
- ✓ Read Result Files... button is replaced by a menu button with two options:
  - By Copy: this is the old method which copies the files to the solver directory.
  - By Reference: using this new option, one can specify the path to Mechanical APDL result files, and the application reads the files without copying them to the solver directory.

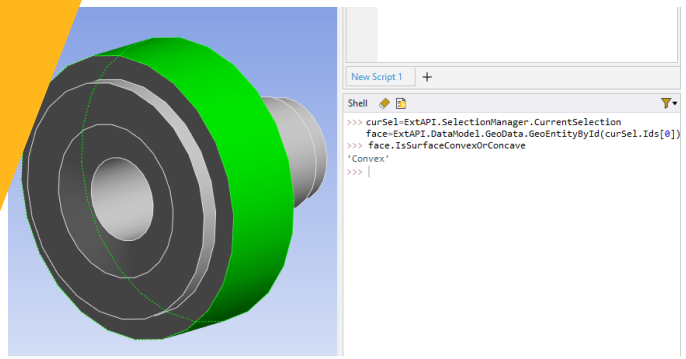
Mechanical Usability and Scripting enhancements



ds.dat:

***set,_tid,13**
POINT_MASS_1 = 13
et,_tid, 21





Element APDL Name

- ✓ This feature shows the Element APDL Name property for Point Mass, Distributed Mass and Contact Region objects.
- ✓ It sets Element type number to a user defined name.
- ✓ Later this name can be used in places where the element type number is required
- ✓ There is also a new ACT API for Element APDL Name

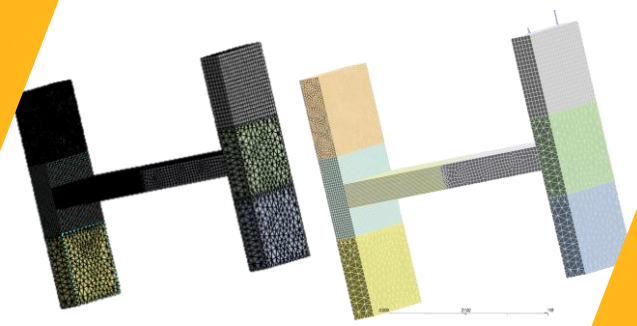
Contact & Target APDL Name

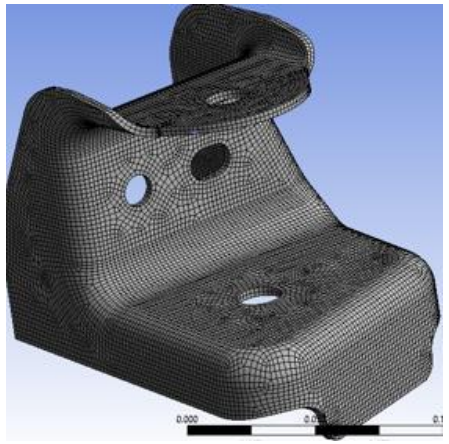
- ✓ This new feature will set default APDL names to all selected contacts
- ✓ This action is available on the right mouse click on the following objects:
  - Contact Regions
  - Connection Groups: Type = Contacts
- ✓ sets APDL name property to:
  - contact: _con<objectId>
  - target: _tgt<objectId>

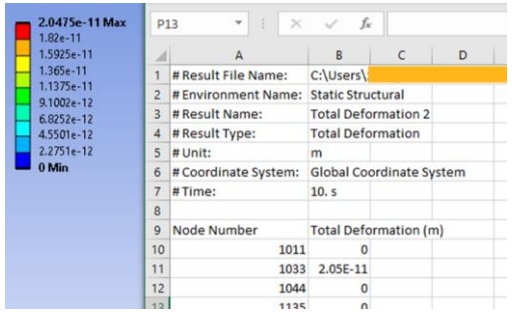
New APIs

- ✓ New API to query whether the surface is Concave or Convex for Cone and Cylinder.
  - IsSurfaceConvexOrConcave
- ✓ New API to get radius of Cylindrical Face and Spherical Face
- ✓ New API to get radius of Circular Edge

General Post and Graphics

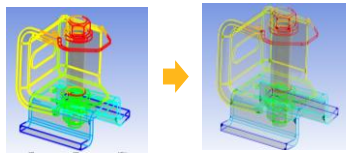






Mesh and Edge display

- ✓ new option was introduced to replace the global color of edge meshes with the color of each body.
  - See "Color mesh edge using body color" preference in Options->Graphics
- ✓ Modifying the Contour Edge Line Weight now possible



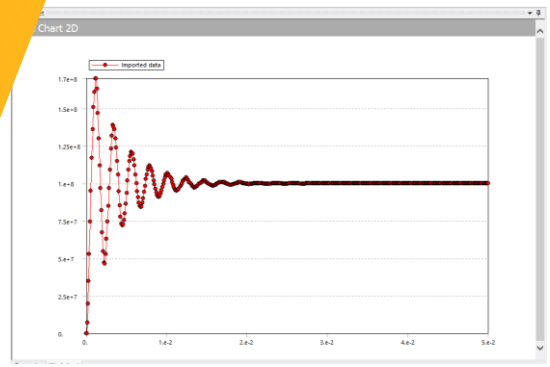
Visualization Performance

- ✓ Panning, zooming, and rotating is significantly more responsive for results with large FE scoping (1 million+)
- ✓ For models using shared mesh beam sections expect 4-5x improvements in mesh display speed
- ✓ For models using many shared non-mesh beam sections (10k+) expect a more modest 1.5x improvement in mesh display speed

File import and export

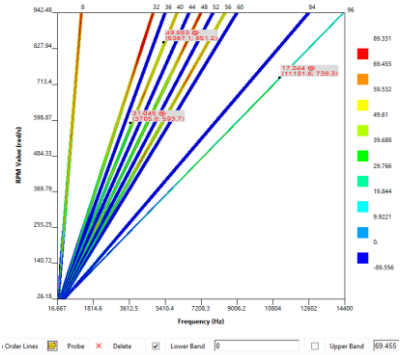
- ✓ Users can now opt to include additional header information to files Exported from Mechanical
  - contextual information will be added to the exported result
- ✓ Users can now also create a User Defined Result from a previously exported result file

New Line Chart Object in Mechanical for LS-Dyna



Details of "Line Chart 2D"

Definition	
Source Type	Imported File
Source File	Z:\test\small_curve.txt
Operation Type	None
Filter Type	Butterworth
Cutoff Frequency [Hz]	60
Cutoff Frequency Limit [Hz]	2228.64019253911
Display Original	Yes



Line Chart object (LS-Dyna)

- ✓ The new 2D Line Chart object allows users to plot a two-dimensional graph
- ✓ Currently, input data for Line Chart objects can come from Result Trackers, other Line Chart objects, imported text files (of supported formats) and also any user-defined datasets created using scripting APIs.
- ✓ Once you specify your data, you can apply the available Operation and Filter features to your graph

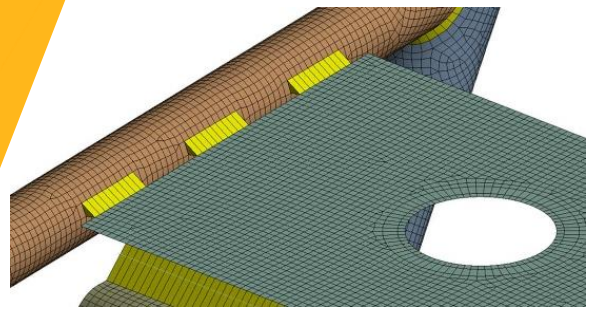
APIs

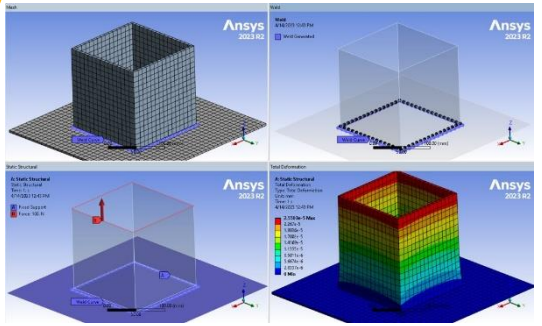
- ✓ The new Line Chart object also has a rich scripting API available for users to define datasets and to choose various display options for the datasets.
- ✓ There are also APIs available to apply Filters, perform Operations and to Import/Export chart data

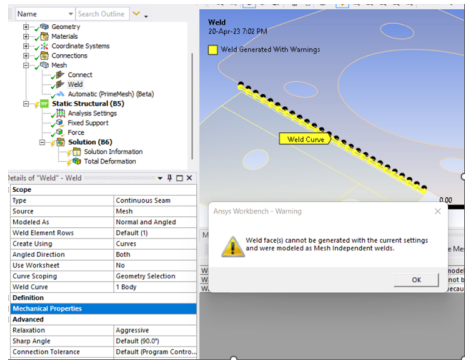
Operations and filters

- ✓ Filters currently are:
  - Differentiate and Integrate
- ✓ Operations currently are:
  - Butterworth, SAE, Cutoff frequency + limit

Weld Meshing







Weld Meshing

- ✓ Higher robustness for automatic weld attainment
- ✓ Mesh independent welds are used where connectivity is difficult to attain
- ✓ Parameter support for weld parameters
  - Support for DoE studies in Workbench

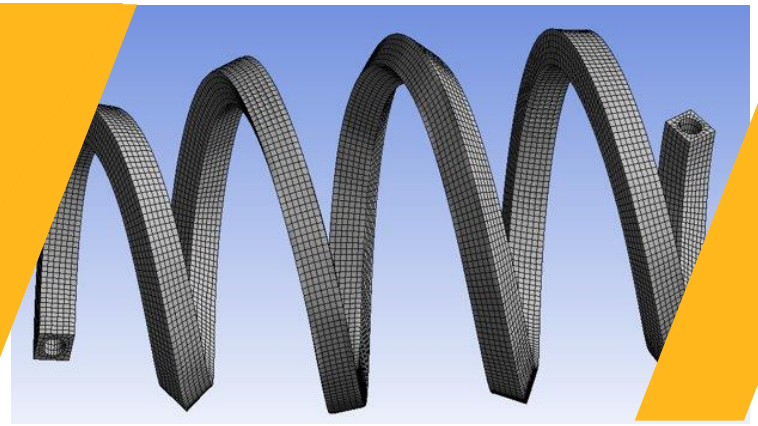
Mesh Independent Weld

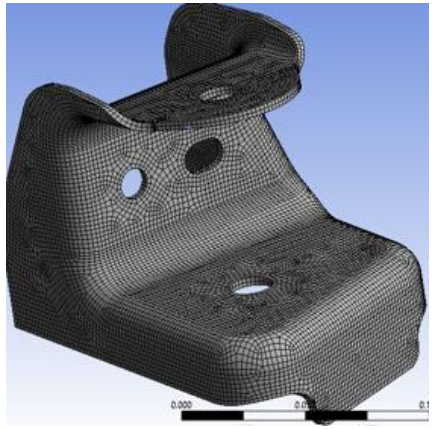
- ✓ Mesh Independent welds are now supported
- ✓ Creates mesh without weld faces or HAZ layers
- ✓ When Weld is Modeled As Mesh Independent, Mechanical creates bonded MPC contact and writes it to the input file
- ✓ Nodes on the top faces are attached to the elements on the bottom face which act as contact points

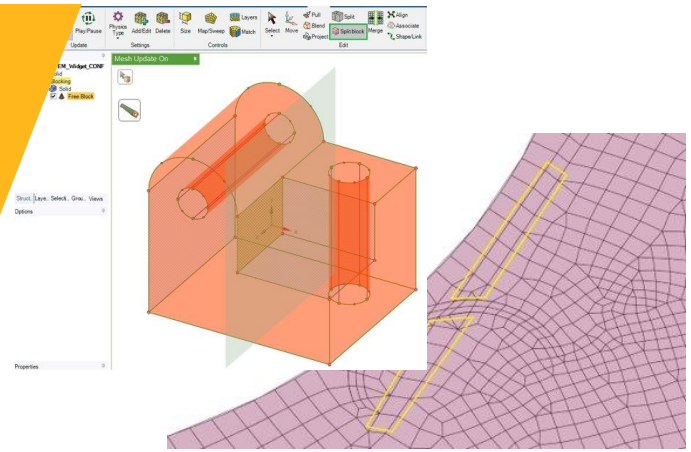
Weld with Aggressive Relaxation

- ✓ When Weld is Modeled As “Normal” or “Angled” or “1D” or “Normal and Angled”, and the Relaxation property is set to Aggressive, sometimes Weld Face(s) fails to generate.
- ✓ In such scenarios, Mechanical internally creates Mesh Independent welds through bonded MPC contact.
- ✓ This applies for both options:
  - Worksheet
  - Non-worksheet (Geometry Scoping & Body Scoping).

Hex Meshing







Improved Handling of Coiled Geometry for MultiZone

- ✓ Improved robustness and performance
- ✓ Speed improvements when using Curvature/Proximity sizing
- ✓ Auto-meshing of coiled geometries

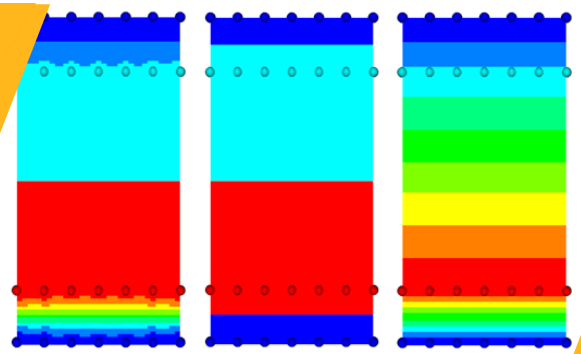
Solsh for Pull and MZ Thin Sweep

- ✓ Solid-Shell option is now available for
  - Mesh Edit ⍰ Pull (Extrude) with a given vector direction
  - Multizone ⍰ Decomposition Type "Thin Sweep"
- ✓ Solid-Shell element types are passed correctly to the solvers to produce Solsh190 (MAPDL) or T-Shell (LS-Dyna) element types

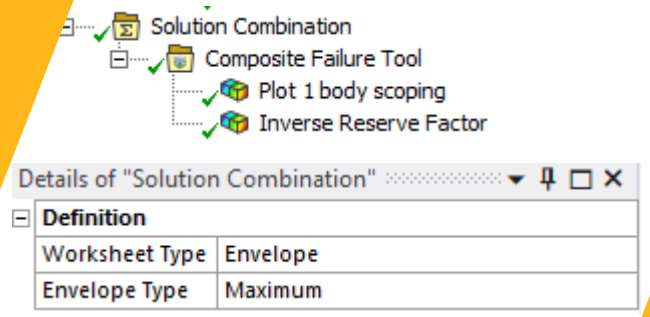
SpaceClaim Meshing

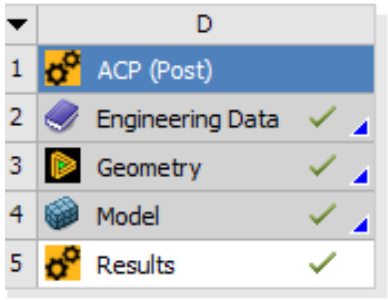
- ✓ Robustness improvements
- ✓ Quad meshing and size field speed up
- ✓ Ease of use improvements
  - Select faces based on NS Groups
  - Split blocks by plane
- ✓ Improved mesh transitions without needing to insert blocking/layers
  - O-, C- , L-Grid without blocking
- ✓ Merge Blocking...

Composites



Interpolated Values with Weighted Nearest Neighbor, Nearest Neighbor, and Linear Multivariate (from left to right).





ACP (Post)

Additional Interpolation Algorithms for the Look-Up Table

- ✓ Look-up Tables can be used to define for example variable ply thicknesses or material directions.
- ✓ Two additional interpolation algorithms - Nearest Neighbor and Linear Triangulation - are available for 3D Look-Up Tables.
- ✓ These algorithms offer more flexibility when mapping table data onto a composite model. Depending on the data and the mesh, this leads to improved accuracy

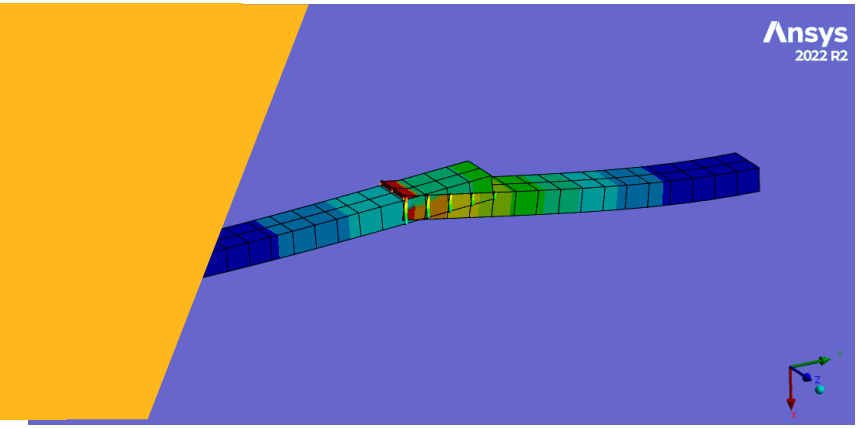
Envelope Solution in Mechanical

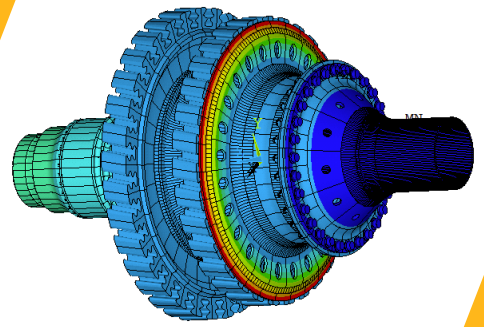
- ✓ The Envelope Solution feature of ACP Post is now also available in Mechanical. It can be used in combination with the Composite Failure Tool.
- ✓ This enables the computation of the most critical failure value from a list of selected load cases and composite failure criteria.
- ✓ The Envelope method is part of the Solution Combination feature in Mechanical.

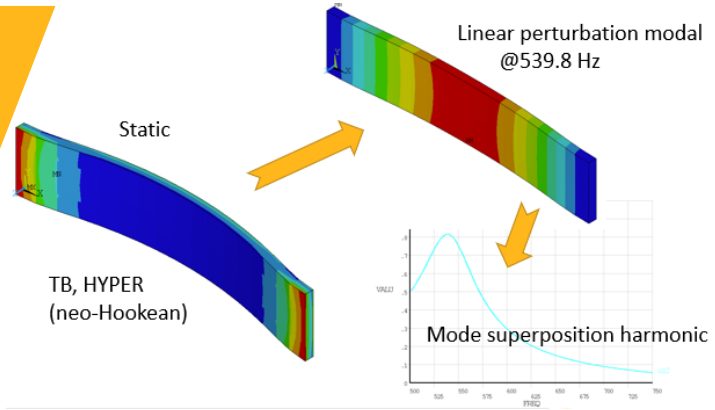
Retirement of ACP Post

- ✓ ACP Post is in maintenance mode and is planned to be removed entirely in version 2025 R1. You may start using Ansys Mechanical to post-process composite simulations. The Composite Failure Tool, Sampling Point Tool and Envelope Solution are already available in Ansys Mechanical.
- ✓ Regarding scripting, please start to use PyDPF Composites.
- ✓ For questions, contact Ansys at composites@ansys.com.

Linear & Non-Linear Dynamics, Acoustics







Harmonic Balance Method (HBM)

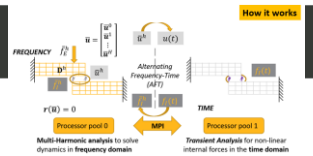
- ✓ Released new solver
- ✓ Example applications: Gaps (clearances), Friction dampers, Bolted joints, Hydrodynamics bearings, squeeze film dampers, Joints based on experimental curves
- ✓ Nonlinear Contacts: Stick-slip behavior, Unilateral contact
- ✓ Complete documentation available: HBM main equations, elements supported, tips on convergence & recommended strategy, example problems with verification
- ✓ Multi-Harmonic App for Mechanical

Multistage

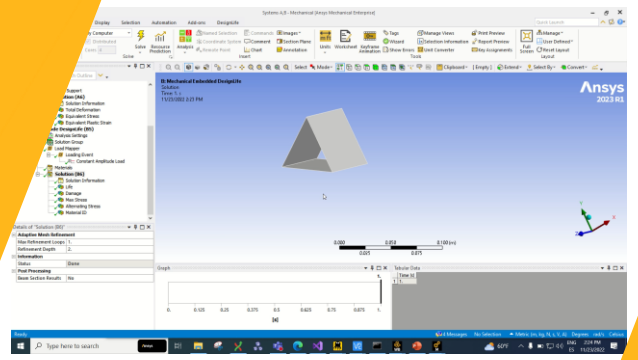
- ✓ Imported loads per stage:
  - ✓ Steady-state Thermal:
    - ✓ Temperature
    - ✓ Film coefficient
  - ✓ Static Structural:
    - ✓ Body temperature
    - ✓ Pressure
- ✓ Joint Elements (MPC184) are supported for cyclic symmetry and multistage cyclic symmetry analyses in static and modal analyses
- ✓ Harmonic index-based loads in multiharmonic static analysis. Primary variable is MSHI.

Other Enhancements

- ✓ Hyperelastic materials are now supported in a harmonic mode-superposition analysis when a linear perturbation modal analysis is performed first.
- ✓ Prestressed Harmonic, Harmonic Acoustics & Harmonic Coupled Field analysis support Multiple Steps with commands
- ✓ Displacement, Velocity and Acceleration Frequency Response charts scoped to Remote Point supported with On Demand Expansion

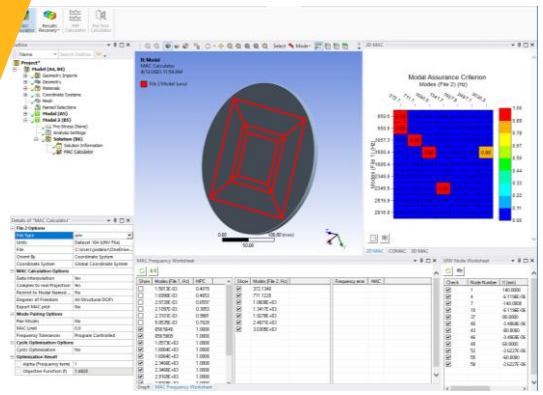


Add-ons



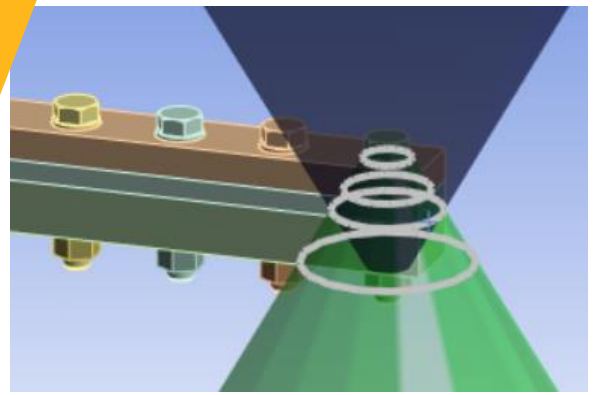
DesignLife Add-on

- ✓ MPI solving
- ✓ RSM send to Windows or Linux machine
- ✓ DCS/REP (see demo video)
- ✓ Elastic license
- ✓ LS-DYNA fatigue calculation
- ✓ PCB fatigue analysis
  calculations, Sherlock data can also be connected as an upstream system
- ✓ SpotWeld Analisys



NVH Add-on

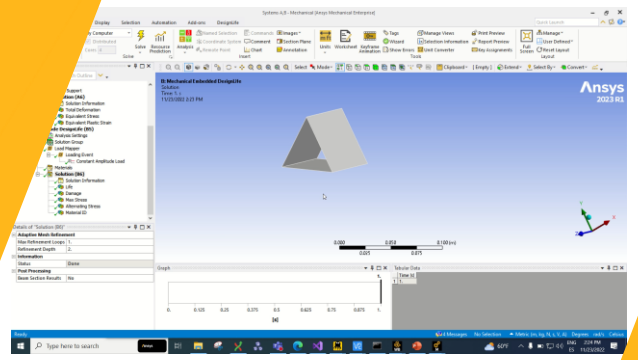
- ✓ Pre-Test Calculator is a new feature developed that introduces a result in the Tree. It identifies the optimum sensor and exciter locations for vibration tests
- ✓ Enhancements to MAC Calculator
  - Removes dependency from MAPDL commands, Preview Matching Nodes
  - Data Interpolation, Works with On Demand Expansion
- ✓ Enhancements to FRF Calculator
  - Export FRF Data in the UNV file format

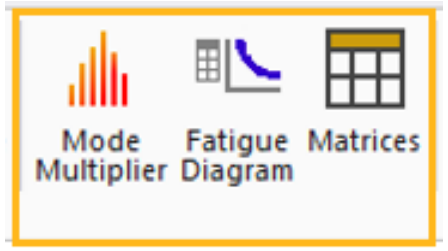


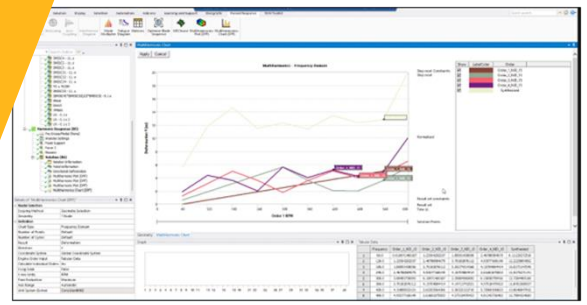
Bolt Tools Add-on

- ✓ Bolt pretension load objects can now be added for LS-DYNA analysis system using Bolt Tools (supporting both solid and beam type bolts)
- ✓ Cone of Compression Imprint Wizard - automatically creates imprints on selected geometries and defines contact between them.
- ✓ Mesh sizing: Separate mesh sizing options from shank and thread faces of the bolt.
- ✓ Contact Extraction Type option exposed for contact reaction probe.

Forced Response Add-on







Mistuning and Aero coupling loads

- ✓ Mistuning and Aero coupling loads inserted to Cyclic Mode Superposition Harmonic system
- ✓ Small mistuning effects (on the order of a few percent) may be included in the analysis by introducing blade-to-blade variations in the stiffness (frequency) of each blade.
- ✓ Aerodynamic coupling effects can be included in the Forced Response analysis. Aerodynamic coefficients account for vibration-induced pressure fluctuations on the blade surface, and contribute to the stiffness and damping of the system

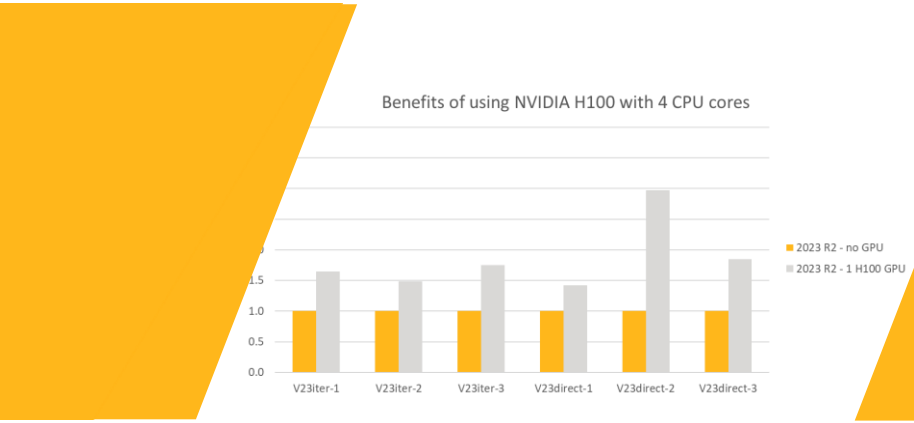
Mode Multiplier, Matrices and Fatigue Diagram results under Harmonic System

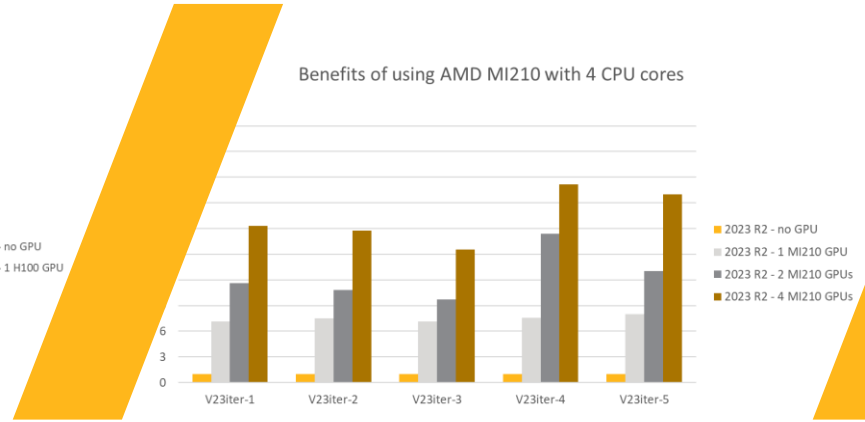
- ✓ Mode Multiplier, Matrices and Fatigue Diagram results under Harmonic System
- ✓ Multi-Stage choice is a new Add-On capability available as part of the Forced Response Add-on.
- ✓ This allows users to calculate a set of harmonics for each stage in a Multi-harmonic, multi-stage analysis

Multiharmonic Chart and Plot

- ✓ Multiharmonic Chart is a new capability available as part of Forced Response Add-On
- ✓ Graphs the contribution of each order. This facilitates the identification of order/frequencies contributing the most
- ✓ Multiharmonic Plot is a new capability available as part of Forced Response Add-On
- ✓ Contour plot of results obtained as composition of harmonic responses excited at different frequencies

HPC





GPU NVIDIA Enhancements

- ✓ Tuned and optimized code for “Hopper” GPUs from NVIDIA
  - New generation (Hopper) cards significantly faster than previous generation (Ampere)
  - Upgraded to CUDA 12 → requires GPU driver update

GPU AMD Enhancements

- ✓ Improved support for Instinct GPUs from AMD
- ✓ PCG iterative solver now supported
- ✓ Upgraded to HIP/ROCm 5.4.2 → requires GPU driver update

Distributed Memory Parallel Enhancements

- ✓ Improved processing of constraint and/or coupling equations (CP/CE)
- ✓ Faster simulations with better scaling to higher core counts
- ✓ MPI library support
  - Upgraded to Intel MPI 2021 Update 8 on Windows and Linux
  - Improves performance, scalability and robustness
  - Linux clusters using (older) Mellanox Infiniband 4.x → (older) Intel MPI 2018 is automatically chosen


Ansys