

Three Pillars: Ansys Structures

Robert Dickens

Lead Application Engineer

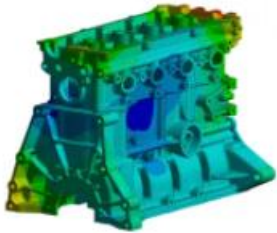
Agenda

- 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



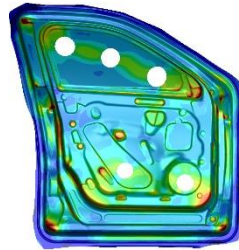
Best-in-class FEA solver for structural engineering applications

Ansys LS-DYNA



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

Ansys Forming



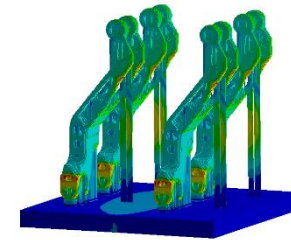
NEW sheet metal stamping tool, backed by the LS-DYNA solver

Ansys Motion



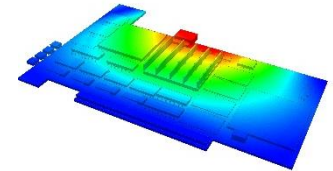
The most robust and advanced simulation solution for multibody dynamic system design

Ansys Additive



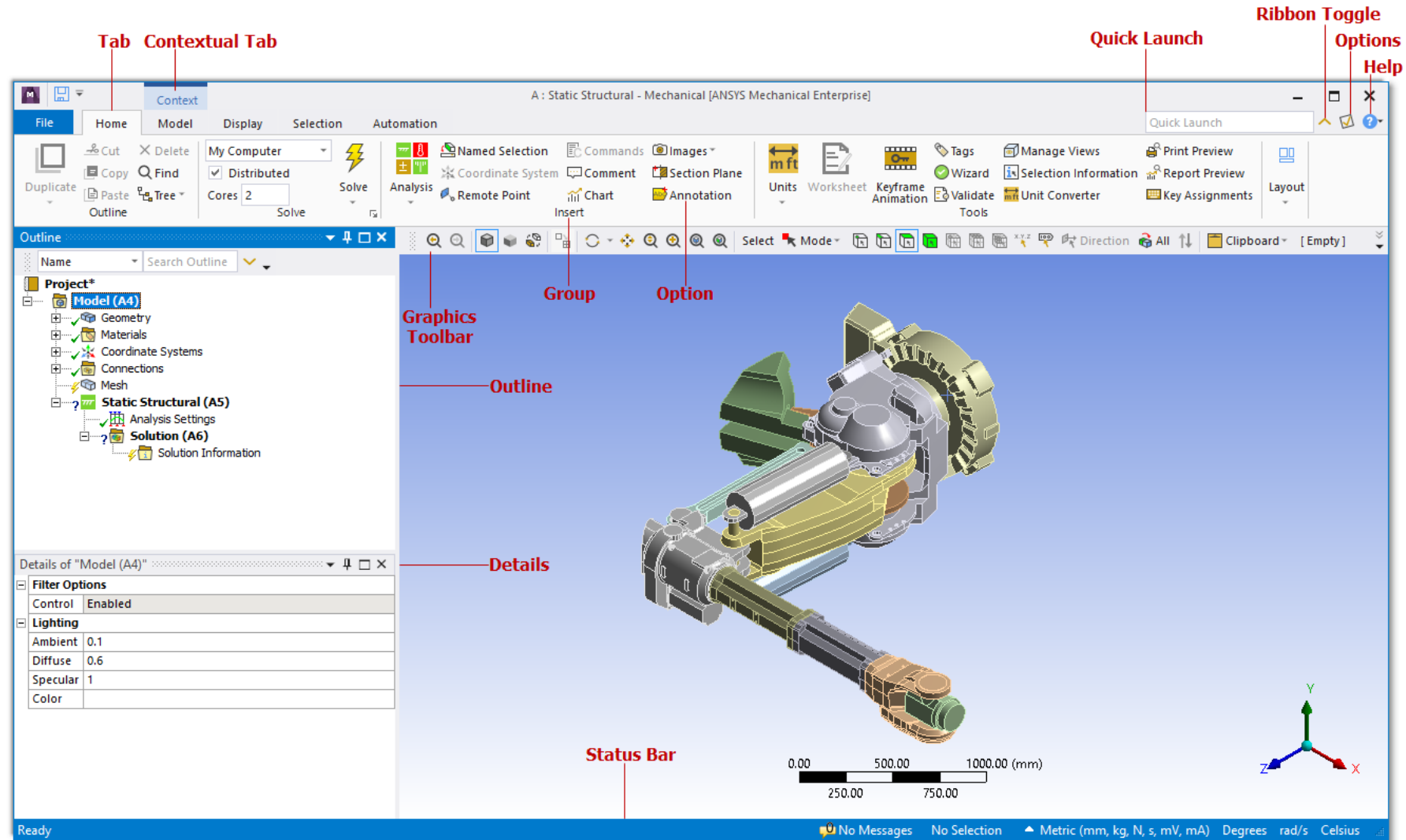
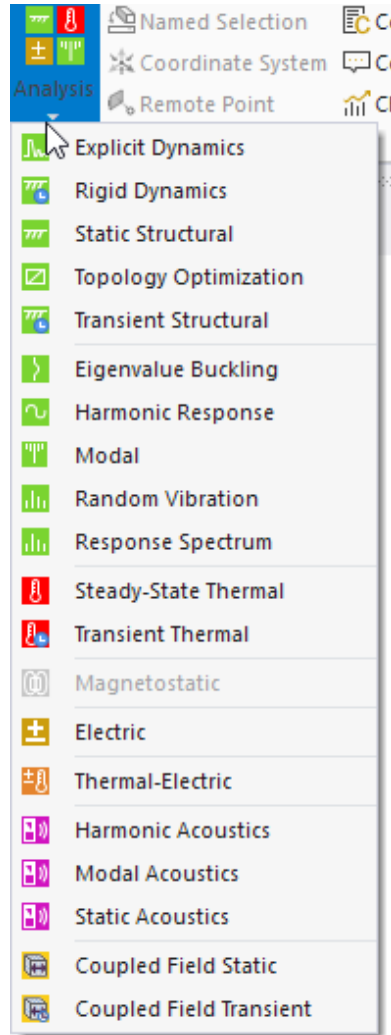
The most powerful simulation solution for metal additive manufacturing

Ansys Sherlock

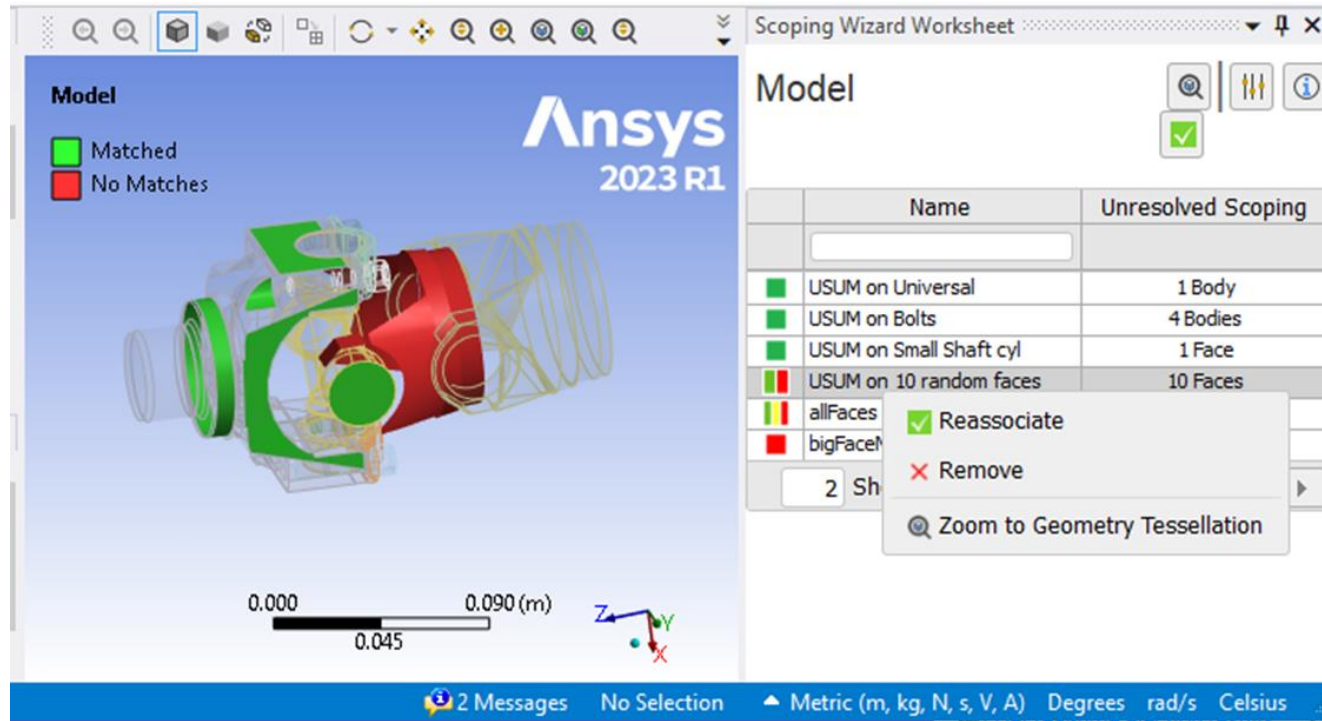
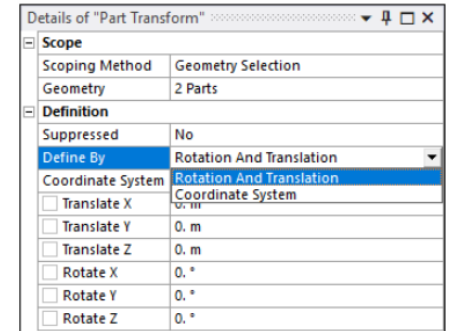
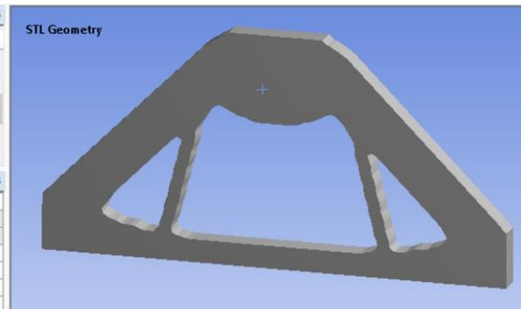
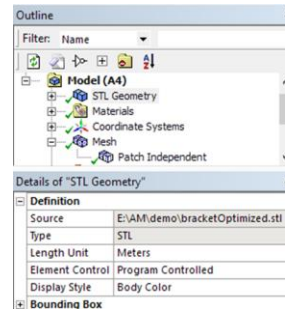
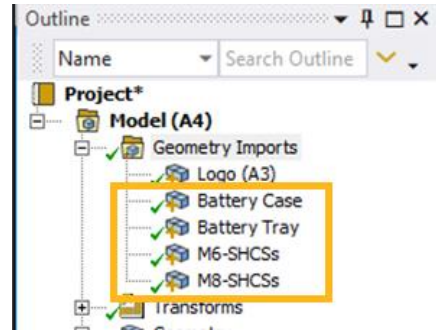
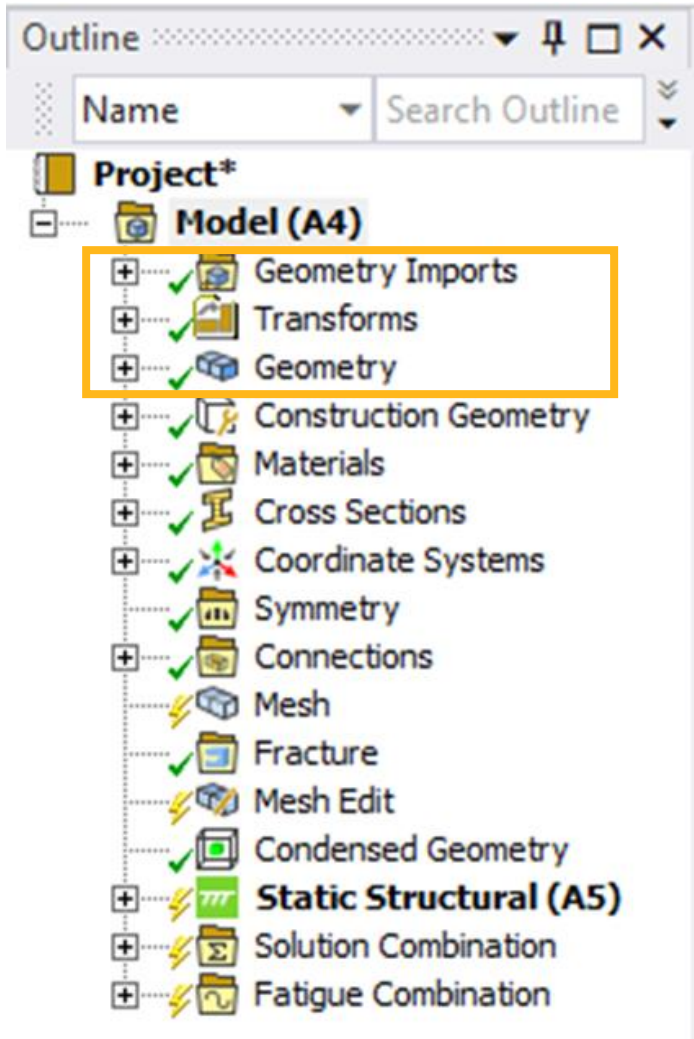


Validated life predictions for electronic hardware

ANSYS Mechanical Interface Update



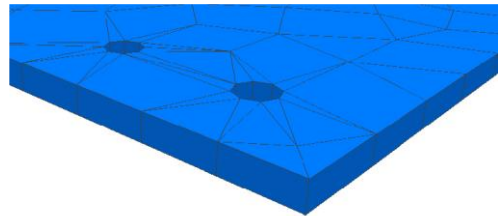
Import and Manipulate Geometry within Mechanical



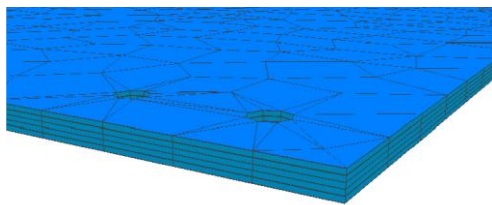
- ✓ Modify a CAD model, without losing the associativity of the model's features after setup
- ✓ Using geometry-based re-associativity with the Scoping Wizard detects and reestablishes scoping

PCB Modeling Technologies

Homogenized/Lumped

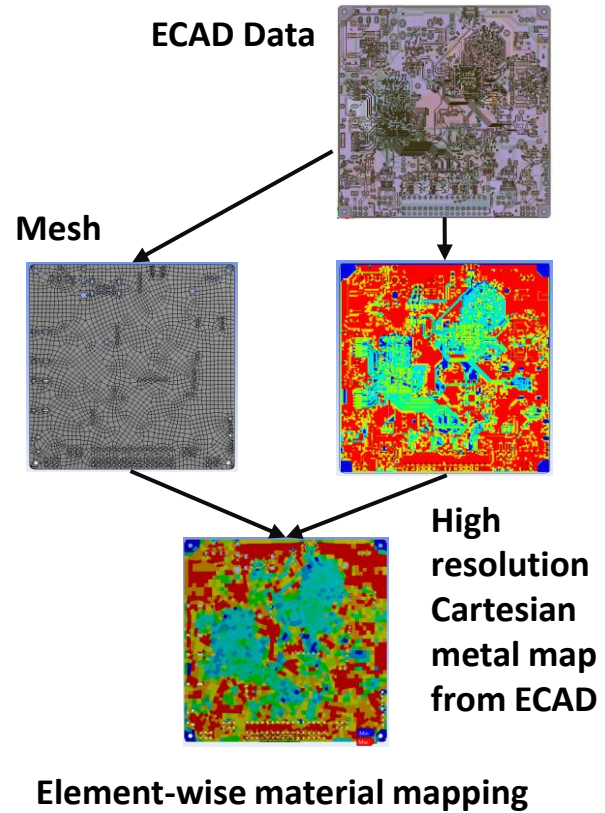


Lumped over entire board

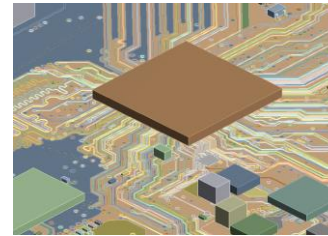


Lumped by Layer

Trace Mapping

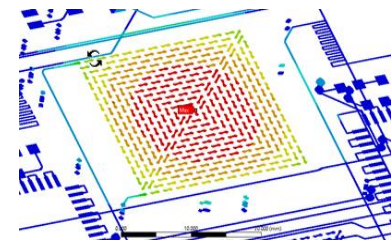
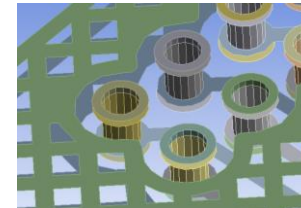


Reinforcements

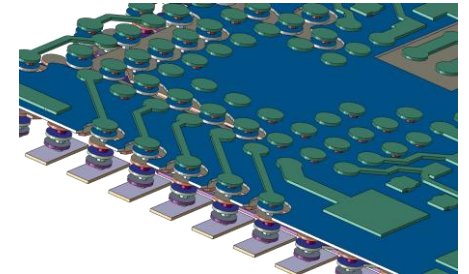


Embed surfaces (traces) and beams (vias) within a base mesh

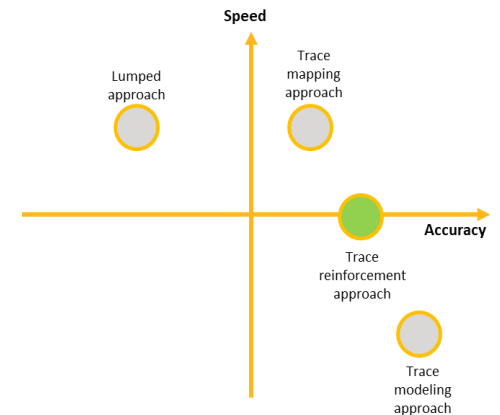
Improved accuracy without full 3D Trace Modeling



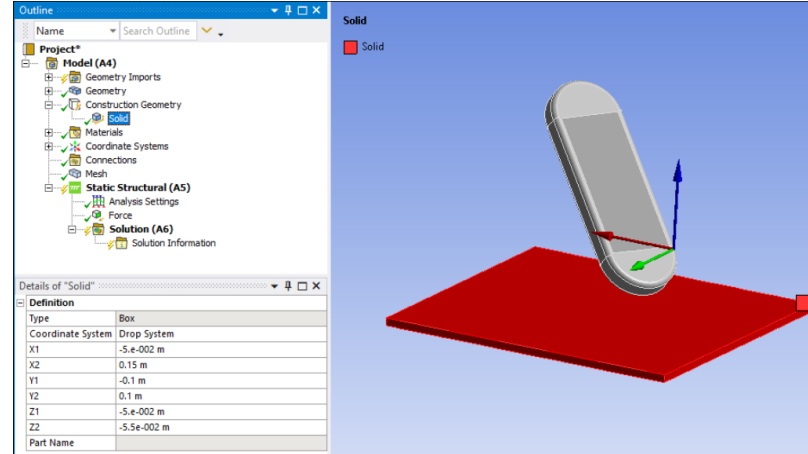
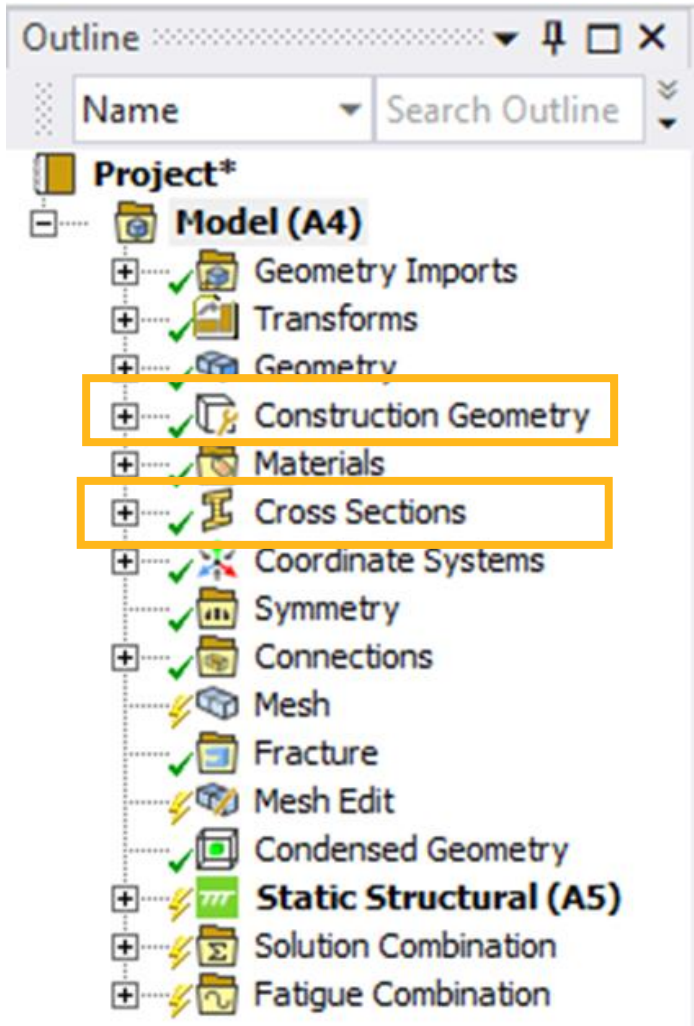
Trace Modeling



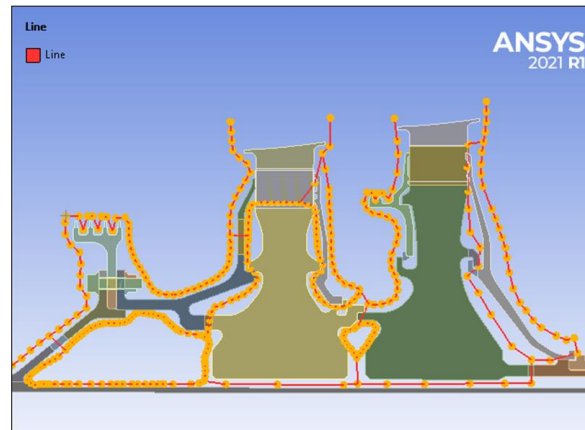
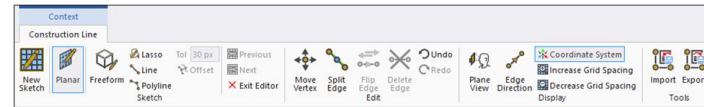
Full 3D Detail



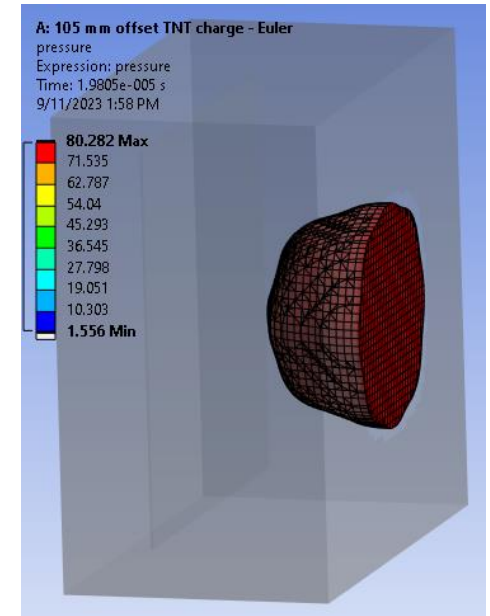
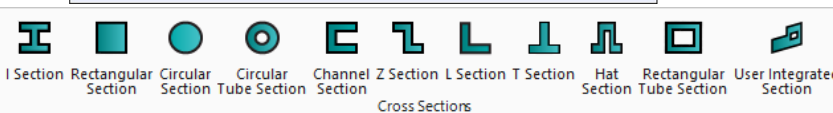
Construct geometry within Mechanical



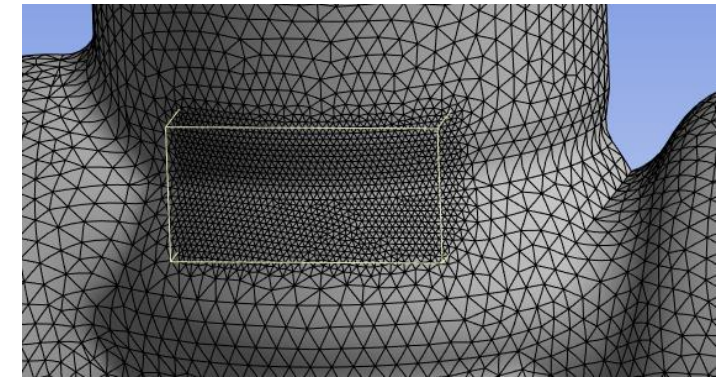
Add Ground surface for Drop



Thermal Fluid definition

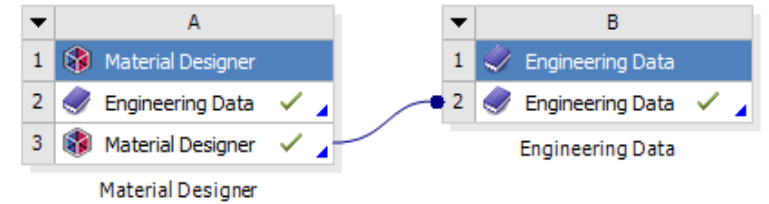
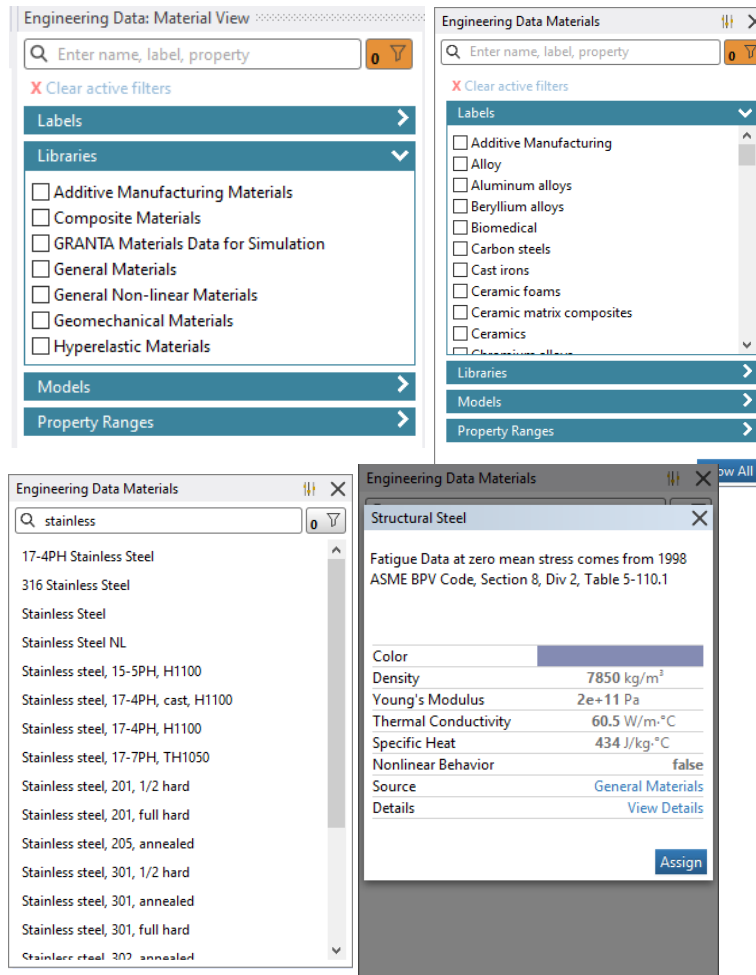
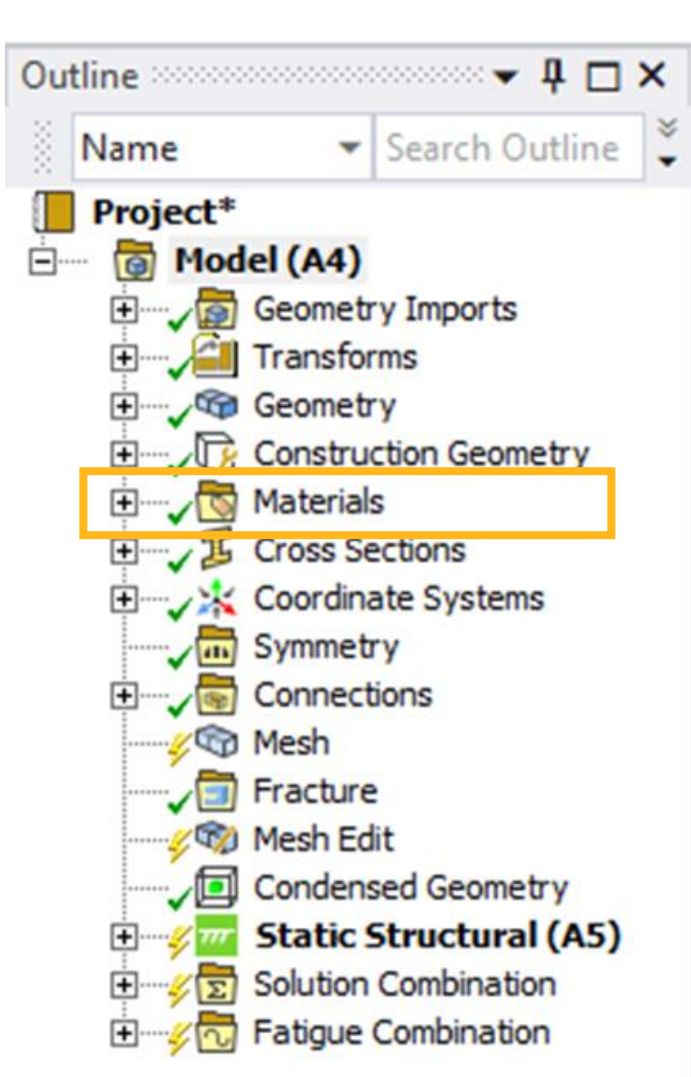


Define Euler Region

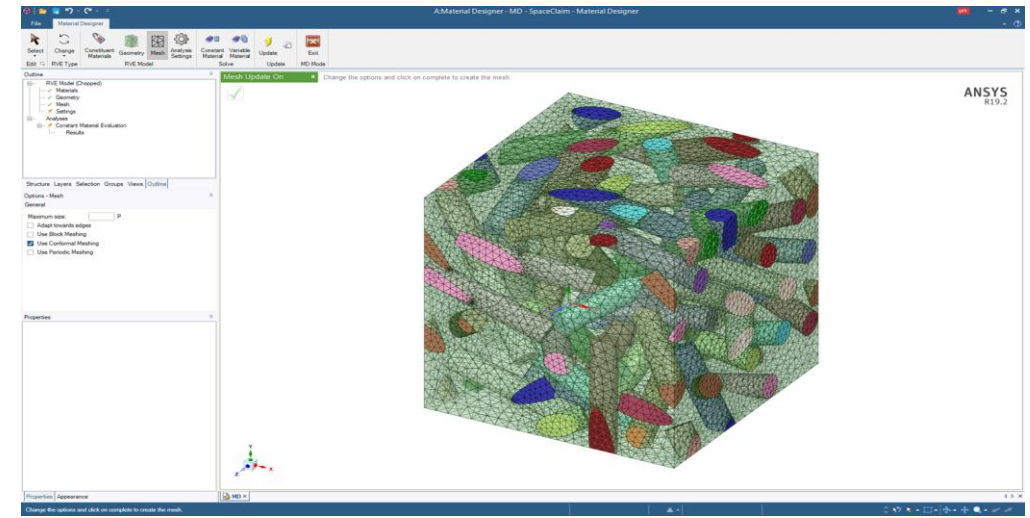


Body of Influence

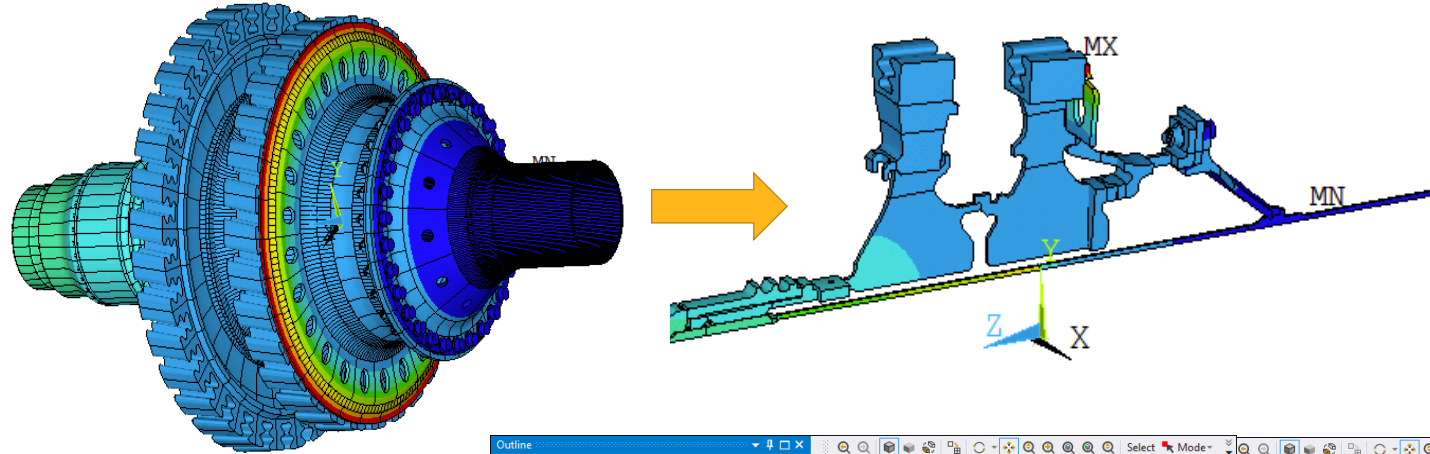
Material Browsing, Searching, Assignments, and Designer



Obtain homogenized (averaged) material properties



Multistage Cyclic and General Axisymmetric Symmetry



Outline

Name Search Outline

Project*

- Model (A4)
 - Geometry Imports
 - Transforms
 - Geometry
 - Construction Geometry
 - Materials
 - Cross Sections
 - Coordinate Systems
 - Symmetry**
 - Connections
 - Mesh
 - Fracture
 - Mesh Edit
 - Condensed Geometry
 - Static Structural (A5)
 - Solution Combination
 - Fatigue Combination

Outline

Name Search Outline

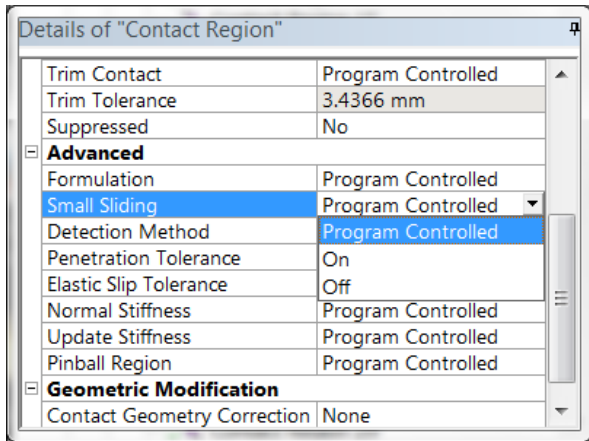
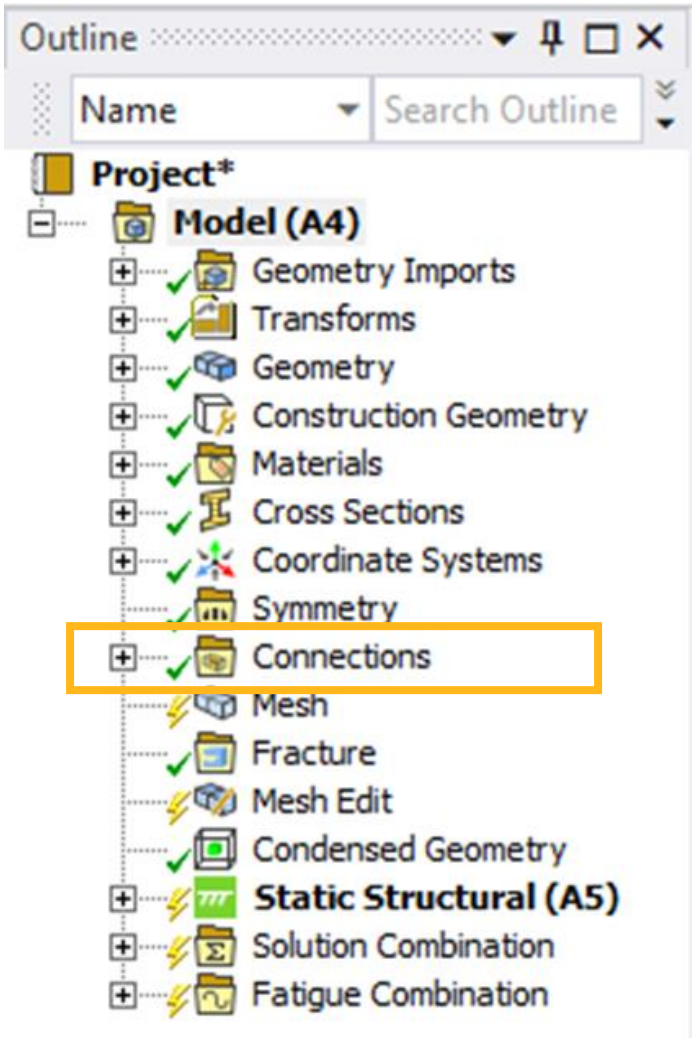
Project*

- Model (A4)
 - Geometry Imports
 - Geometry
 - Materials
 - Coordinate Systems
 - Symmetry
 - General Axisymmetric**
 - Connections
 - Named Selections
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Displacement
 - Solution (A6)
 - Solution Information
 - Total Deformation

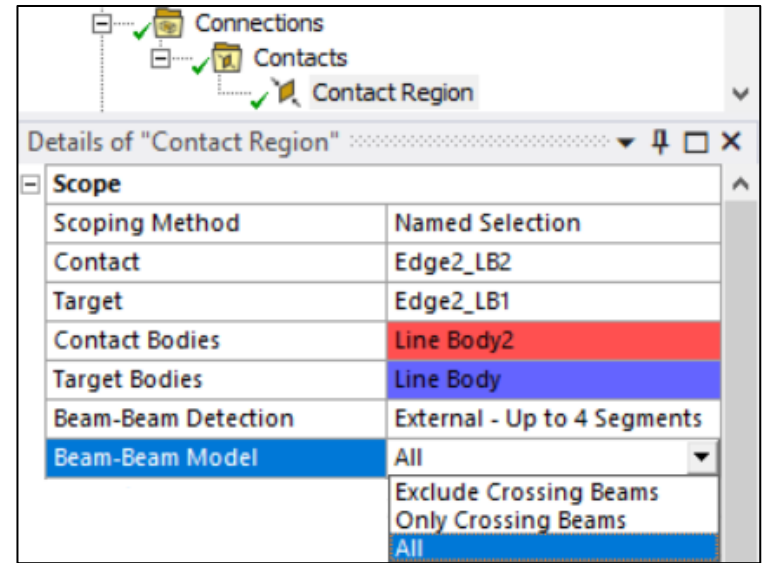
Details of "General Axisymmetric"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Nodal Planes	12
Coordinate System	Plane4
Axis	Y Axis
Suppressed	No
Graphics Options	
Nodal Planes Visible	All

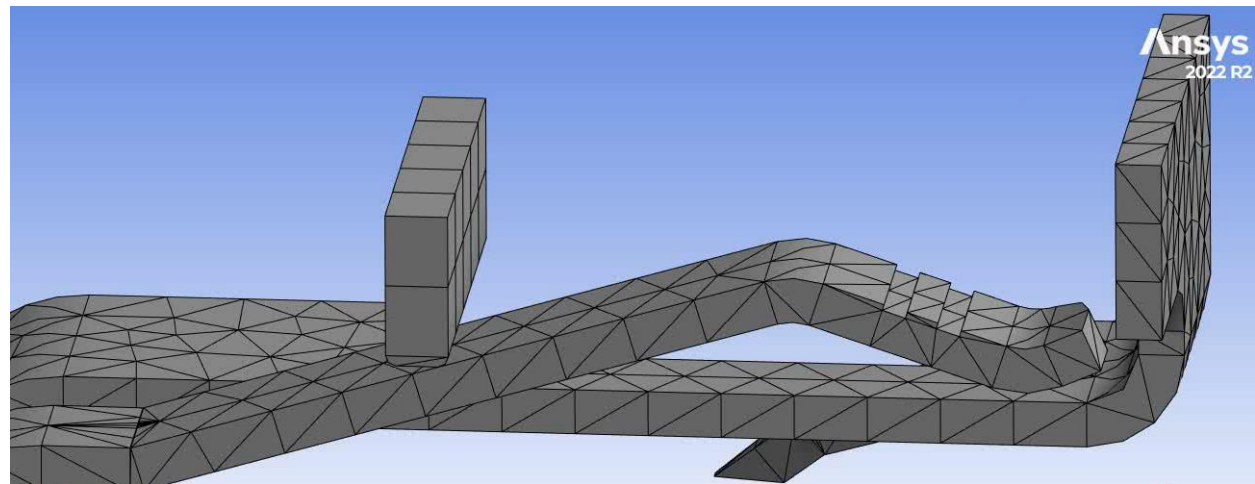
Contact



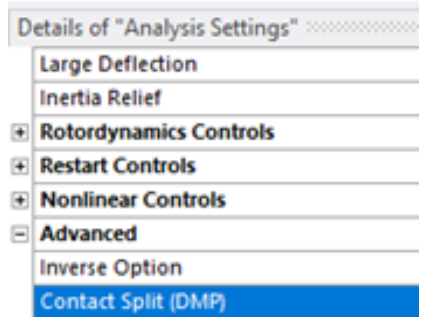
Small Sliding Contact Option



Beam to Beam Contact

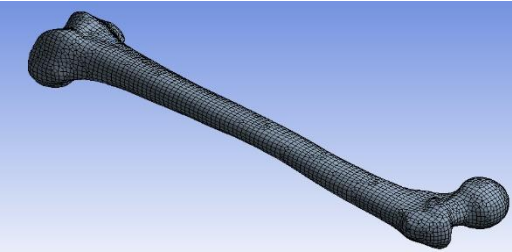
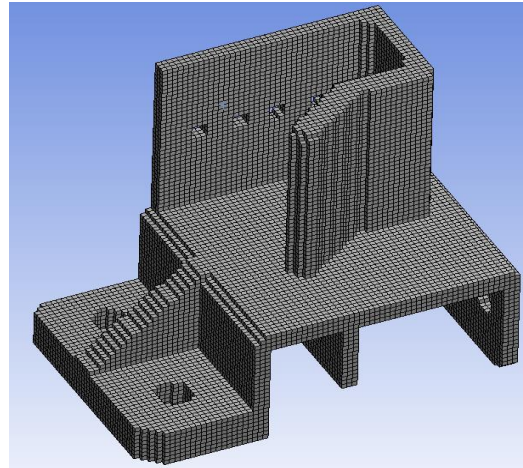


Unified Contact Detection Method

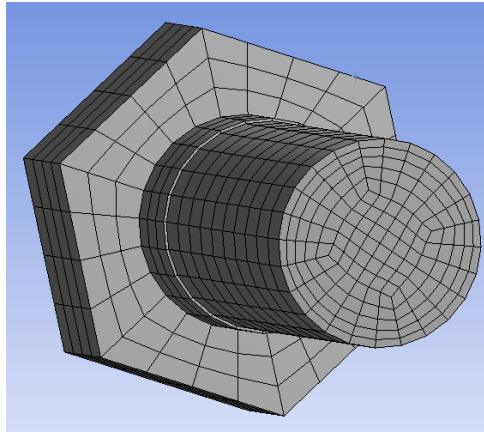
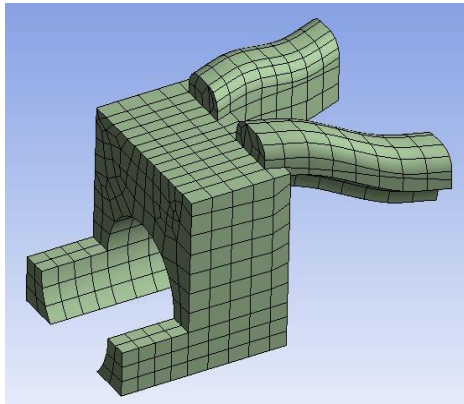
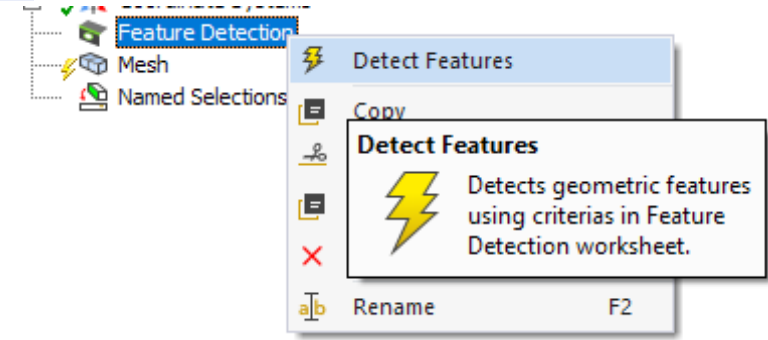


Multi-core, Distributed Contact solution

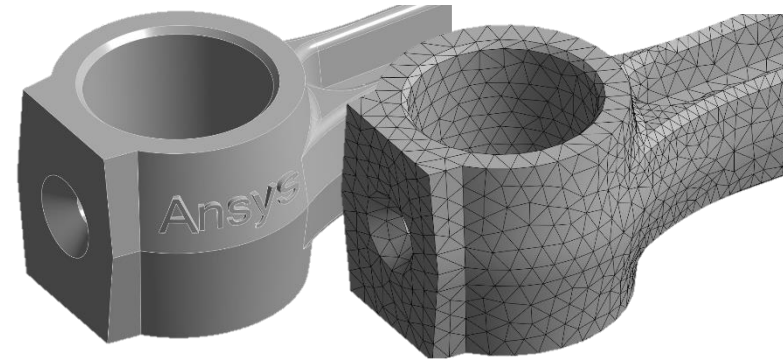
Meshing



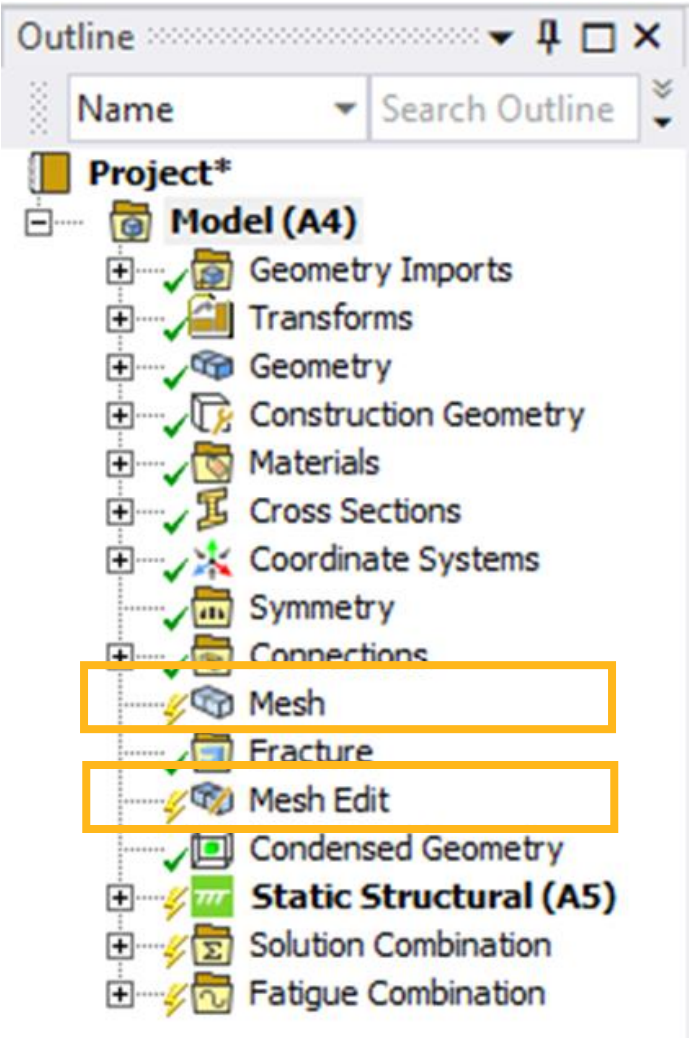
Cartesian Mesh Method



Multi-zone Mesh Method Improvements



Feature detection and suppression

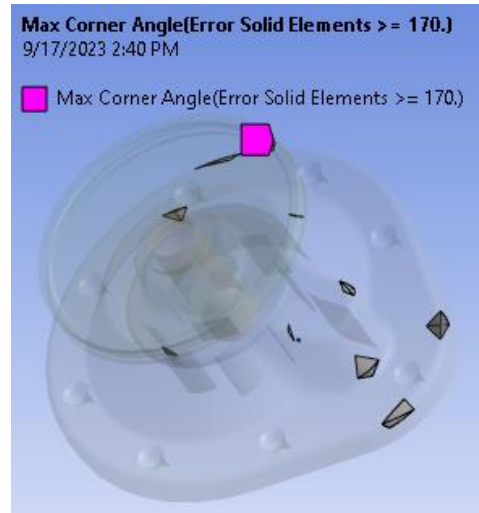


Meshing Diagnostics Tools

Named Selections

Details of "Mesh"

Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default
Sizing	
Quality	
<input checked="" type="checkbox"/> Check Mesh Quality	Yes, Errors
Error Limits	No
<input type="checkbox"/> Target Element Quality	Yes, Errors and Warnings
Smoothing	Mesh Quality Worksheet

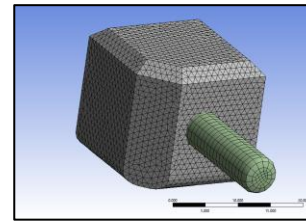


Worksheet

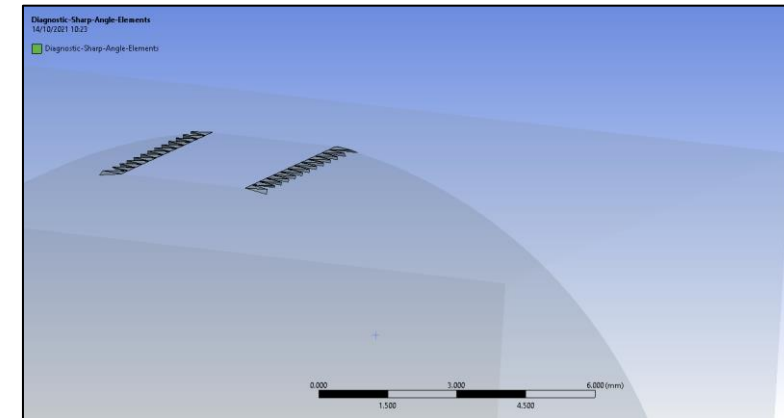
Diagnostic-Named-Selection

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

	Action	Entity Type	Criterion	Operator	Units	Value
<input checked="" type="checkbox"/>	Add	Body	Type	Equal	N/A	Surface
<input checked="" type="checkbox"/>	Diagnostics	Mesh Element	Free Mesh Edges	N/A	N/A	N/A
			Intersecting Elements			
			Body Interference Elements			
			Free Mesh Edges			
			Sharp Angle Elements			



Body Interference

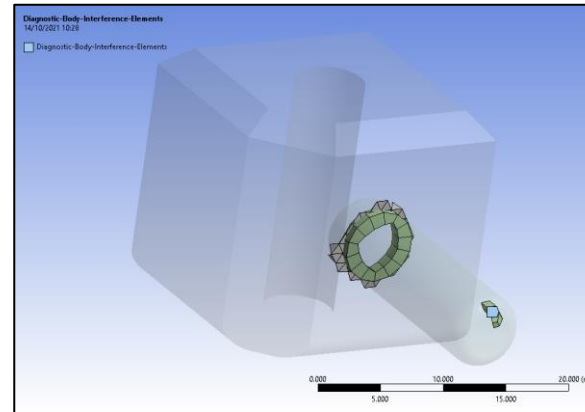


Sharp Angle Elements

Mesh Quality Worksheet

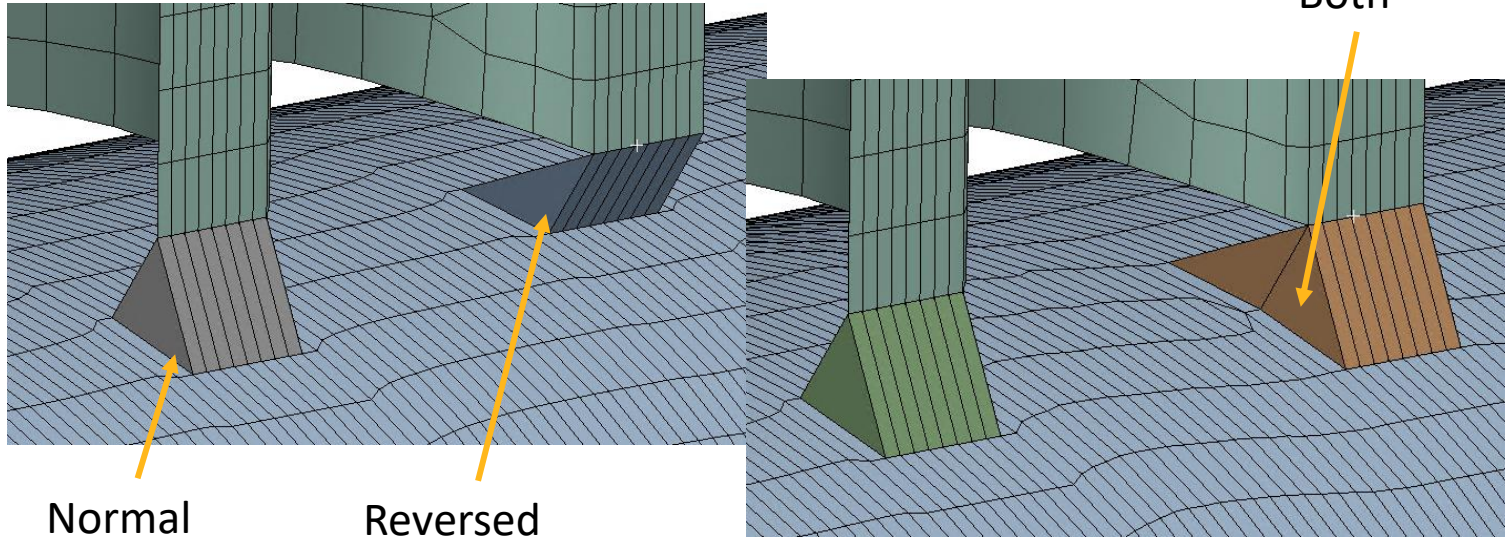
Advanced View Reset Load Save Refresh Validate Error

Error Check	Quality Criterion	Warning (Target) Limit	Error (Failure) Limit	Worst
<input checked="" type="checkbox"/>	Min Element Quality	Default (0.05)	Default (5e-04)	0.051
<input type="checkbox"/>	Max Aspect Ratio	Default (5)	Default (1000)	67.143
<input type="checkbox"/>	Min Jacobian Ratio (Corner Nodes)	Default (0.05)	Default (0.025)	0.025
<input type="checkbox"/>	Min Jacobian Ratio (Gauss Points)	Default (0.05)	Default (0.025)	0.279
<input type="checkbox"/>	Max Corner Angle	Default (150 °)	Default (170 °)	173.58 °
<input type="checkbox"/>	Max Skewness	Default (0.9)	Default (0.999)	1
<input type="checkbox"/>	Min Tet Collapse	Default (0.1)	Default (1e-03)	0.052



Weld Meshing

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

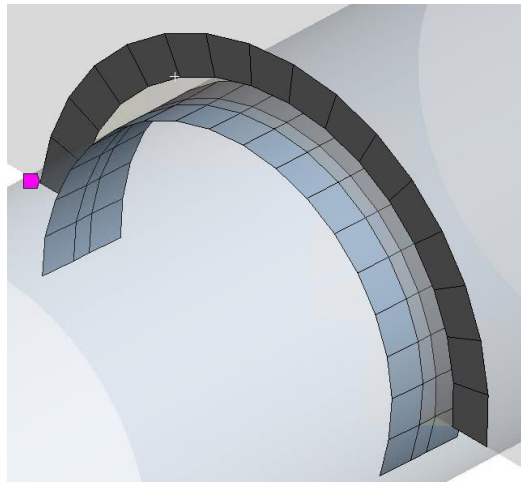


Tent, Extension options are supported for *Source Geometry*

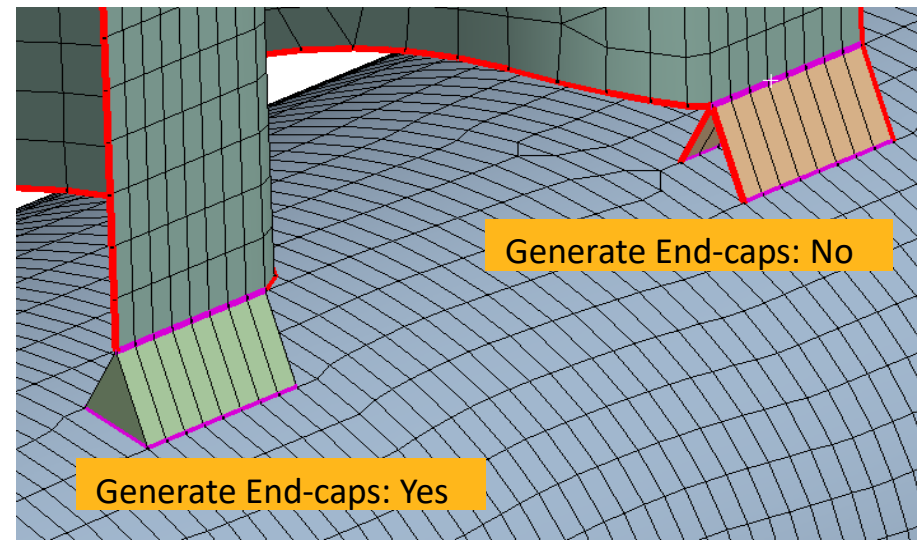
Normal Reversed

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

Option to control Tent face direction



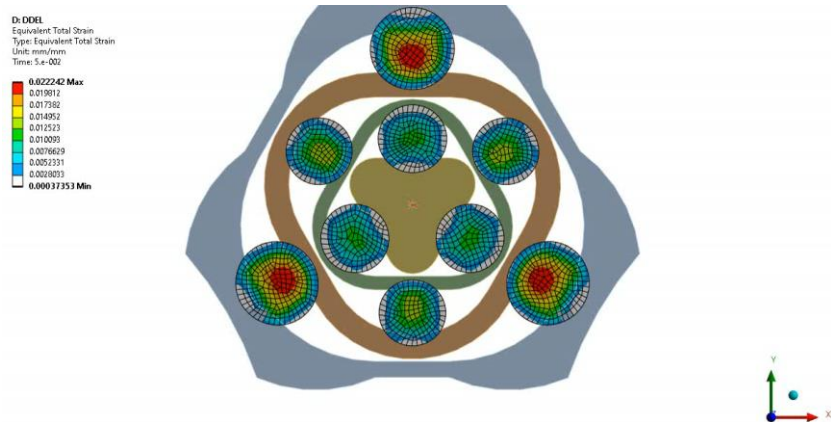
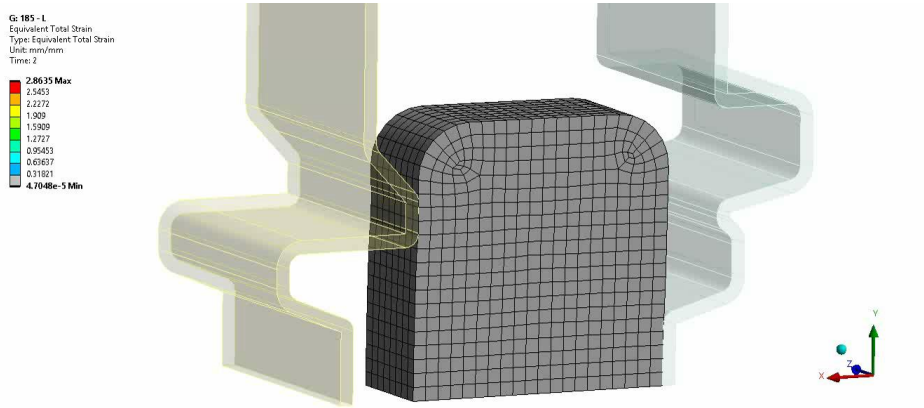
Mesh with offset layers



Generate End-caps: No

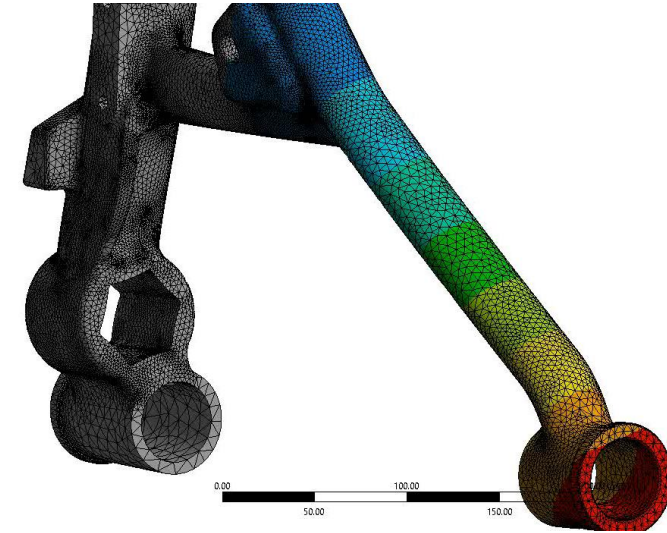
Generate End-caps: Yes

Mesh Adaptivity – NLAD, GPAD



Nonlinear Mesh Adaptivity - NLAD

Geometry Preserving Mesh Adaptivity - GPAD

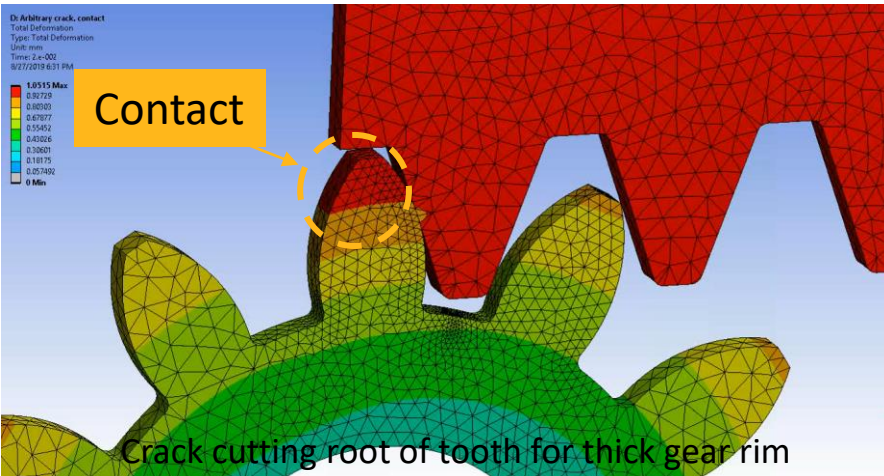
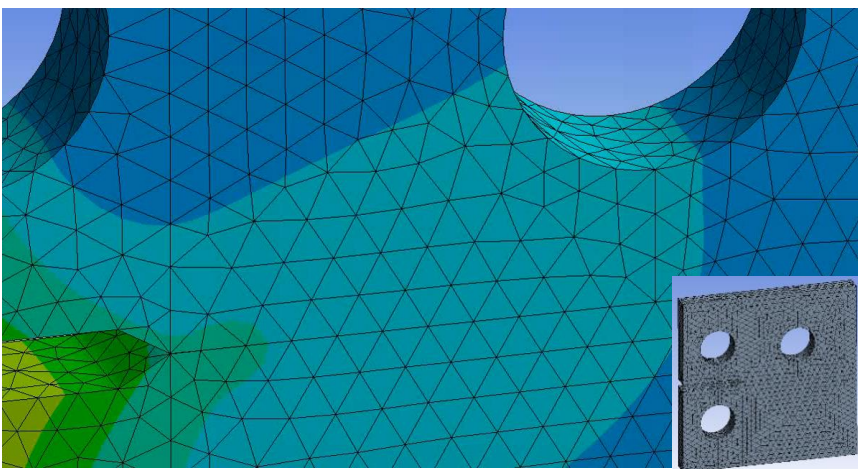
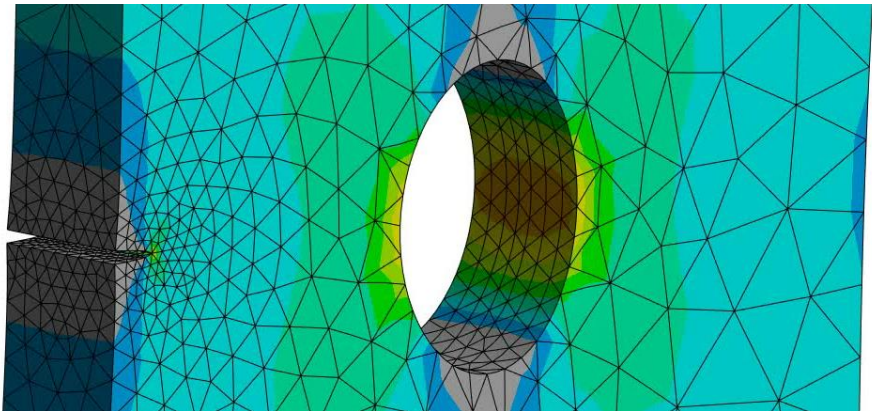
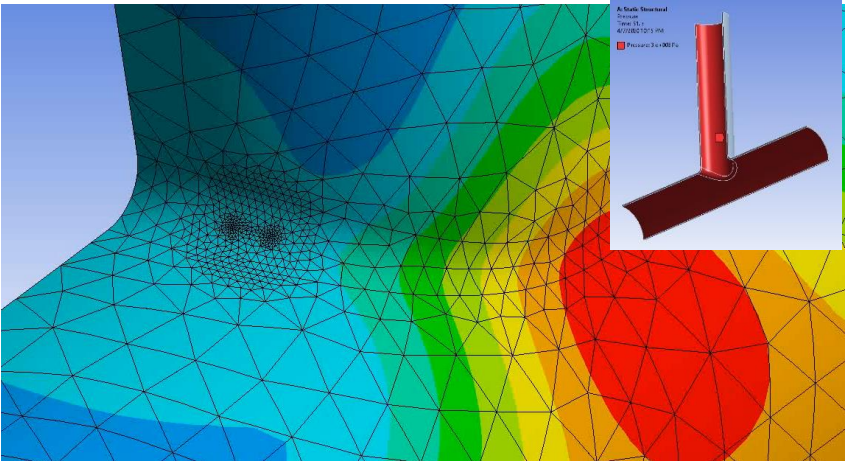


- ✓ Improved simulation accuracy on complex models and parts utilizing geometry preserving adaptivity (GPAD) that automatically refines a mesh based on the initial geometry
- ✓ Eliminates the need for an over-refined mesh or advanced user knowledge

SMART crack modeling

Support multiple pre-existing cracks and multiple crack initiations

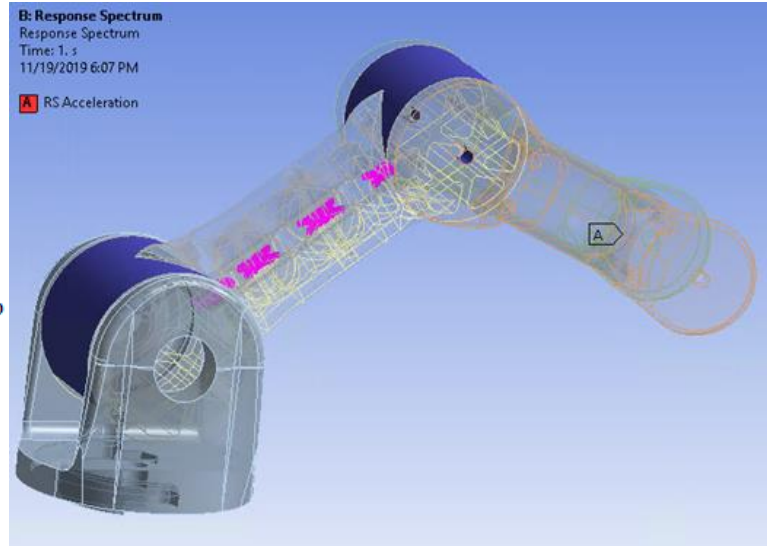
The screenshot shows the 'Outline' window in Ansys. The 'Project*' tree is expanded to show 'Model (A4)'. Under 'Model (A4)', the following items are listed: Geometry Imports, Transforms, Geometry, Construction Geometry, Materials, Cross Sections, Coordinate Systems, Symmetry, Connections, Mesh, Fracture (highlighted with a yellow box), Mesh Edit, Condensed Geometry, Static Structural (A5), Solution Combination, and Fatigue Combination.



Substructuring in Mechanical (Top Down/Bottom Up)

Model (A4, B4)

- Geometry
- Materials
- Coordinate Systems
- Connections
 - Contacts
 - Joints
 - Revolute - 2e onderdeel\Shoulder To BTop
 - Revolute - Multiple To BTop
- Mesh
 - Condensed Geometry
 - Condensed Part
 - Condensed Part 2
 - Condensed Part 3
 - Solution Information
- Named Selections
- Modal (A5)**
 - Pre-Stress (None)
 - Analysis Settings
 - Fixed Support
 - Fixed Support 2
- Solution (A6)**
- Response Spectrum (B5)**
 - Modal (Modal)
 - Analysis Settings
 - RS Acceleration
- Solution (B6)**
 - Solution Information
 - Total Deformation



Substructure Generation (C5)

- Pre-Stress
- Analysis Settings
- Substructure Definition
- Solution (C6)**
 - Solution Information

Project*

- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Mesh
 - Condensed Geometry
 - Condensed Part
 - Named Selections
 - Modal (A5)
 - Pre-Stress (None)
 - Analysis Setting
 - Fixed Support
 - Fixed Support 2

Details of "Condensed Part"

Scope	
Scoping Method	Geor
Geometry	1 Boi

Exported Part Context Menu:

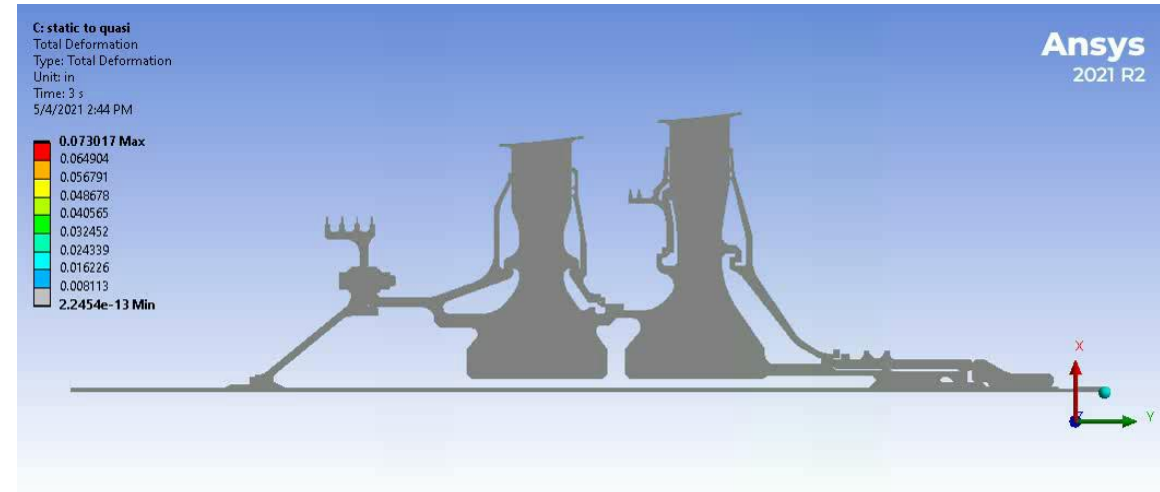
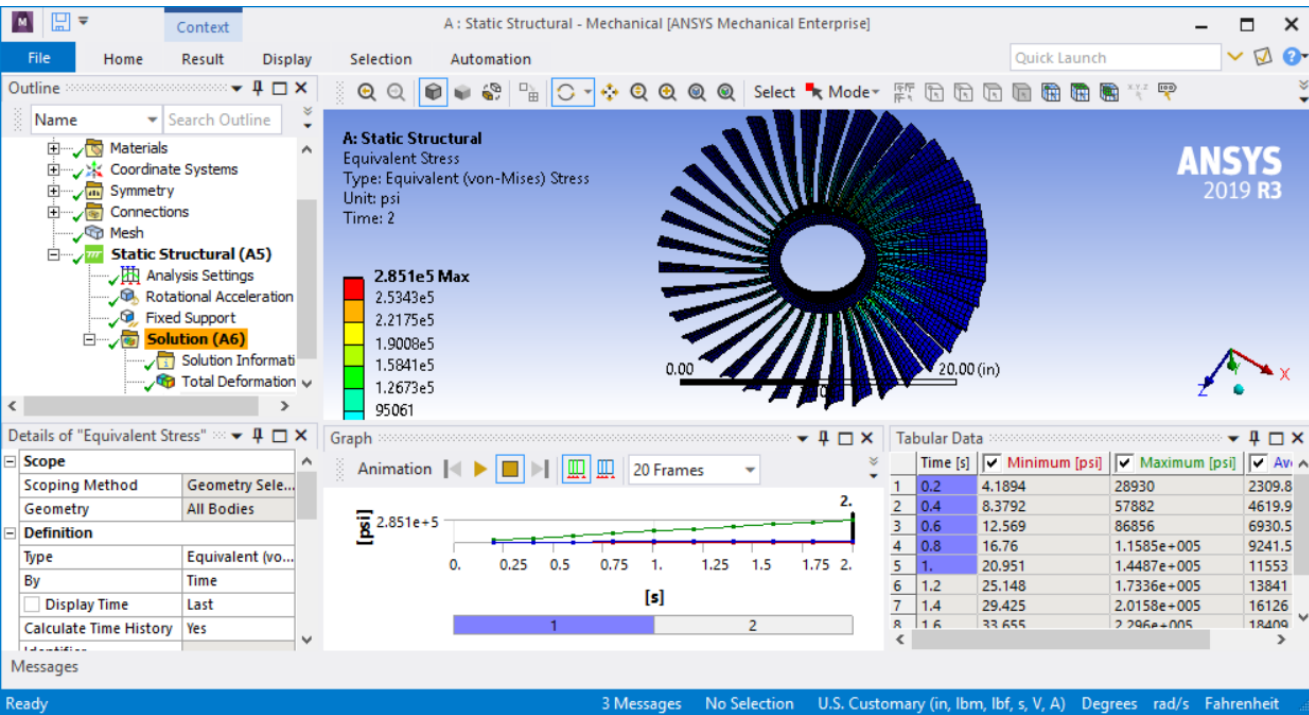
- Insert
- Detect Condensed Part Interface
- Export Condensed Part
- Suppress
- Duplicate
- Copy
- Copy To Clipboard (Beta)
- Clear Generated Data

Export Condensed Part

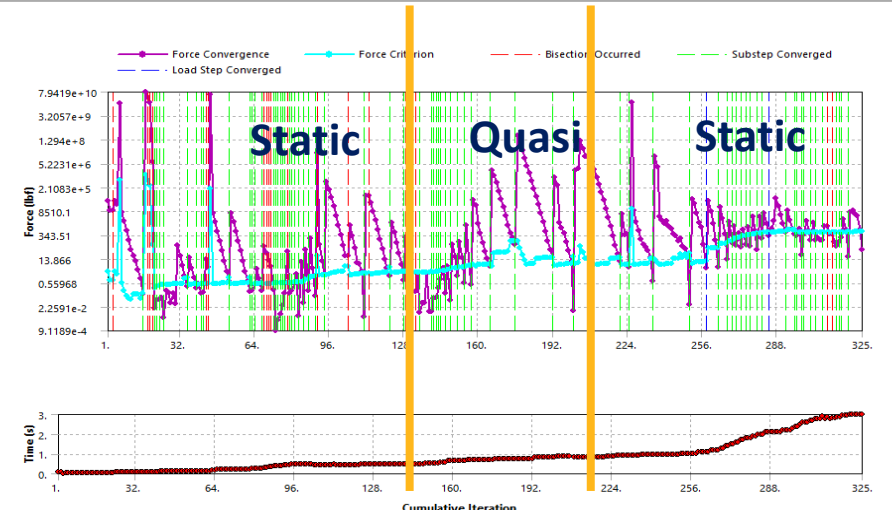
Export the Condensed Part Solution as which can be imported in any Mechanical model.

Press F1 for help.

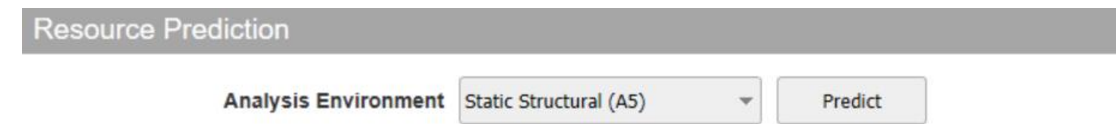
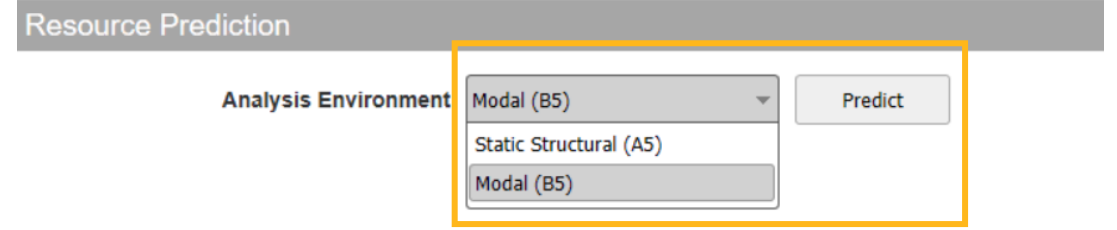
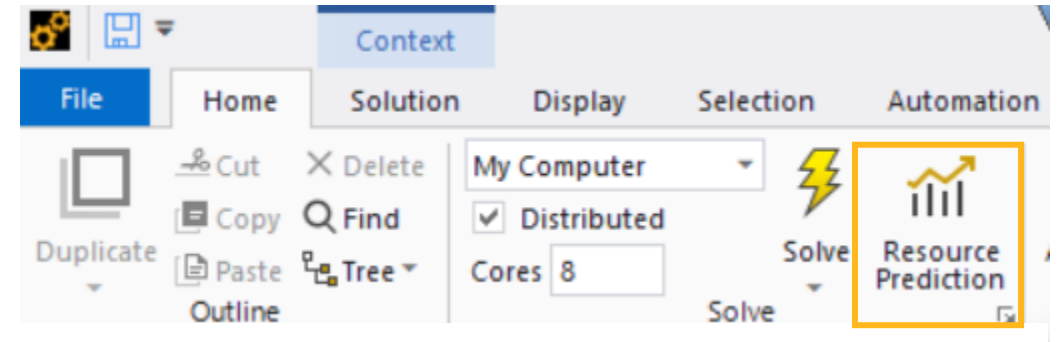
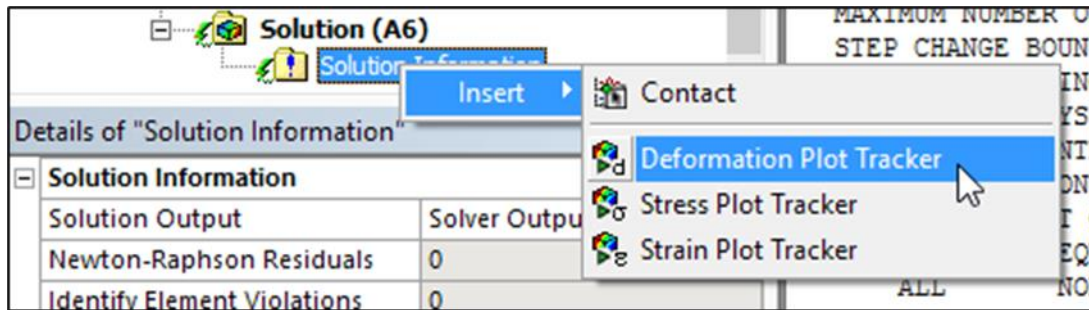
Solution - Inverse Analysis, Automatic "Static to Transient"



Force Convergence



Solution - Resource Prediction, On the Fly Solution Tracking



Predicted Memory Usage

Direct : 32 - 64 GB (Solver Type chosen by Mechanical)

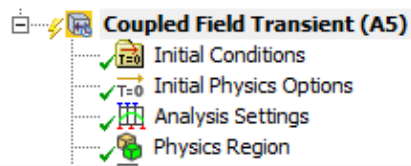
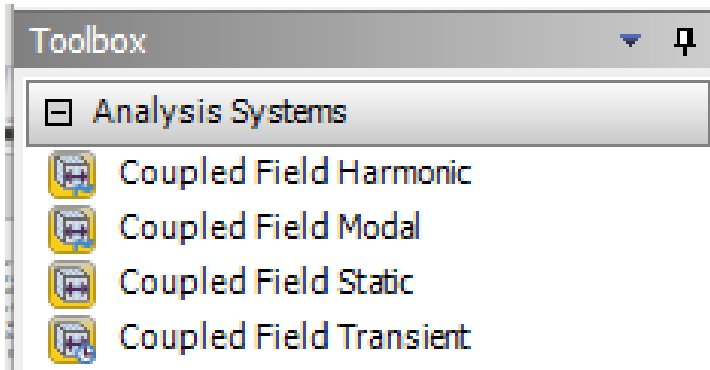
Iterative : 8 - 16 GB

For the selected analysis system the model will require 32 - 64 GB of RAM when run on 4 cores.

This data is based on simulations with similar characteristics.

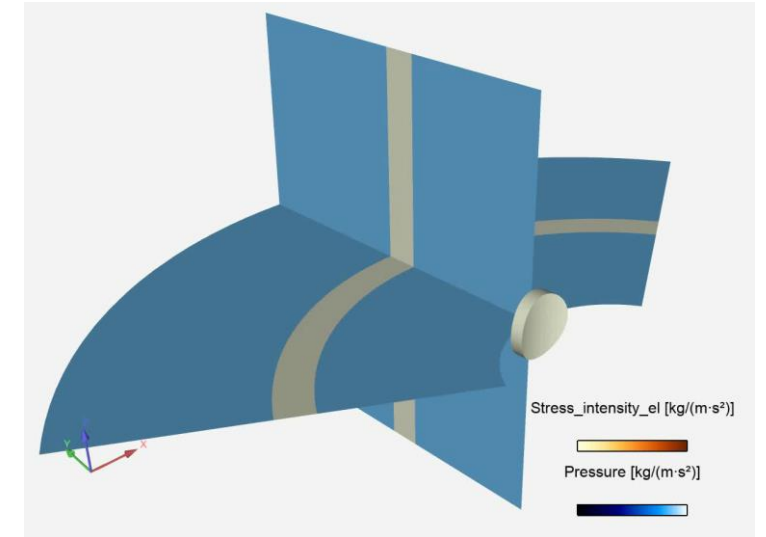
To be used as a guidance only, your actual model may perform differently.

Solution – Coupled Field Problems



Details of "Coupled Field Transient (A5)"

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

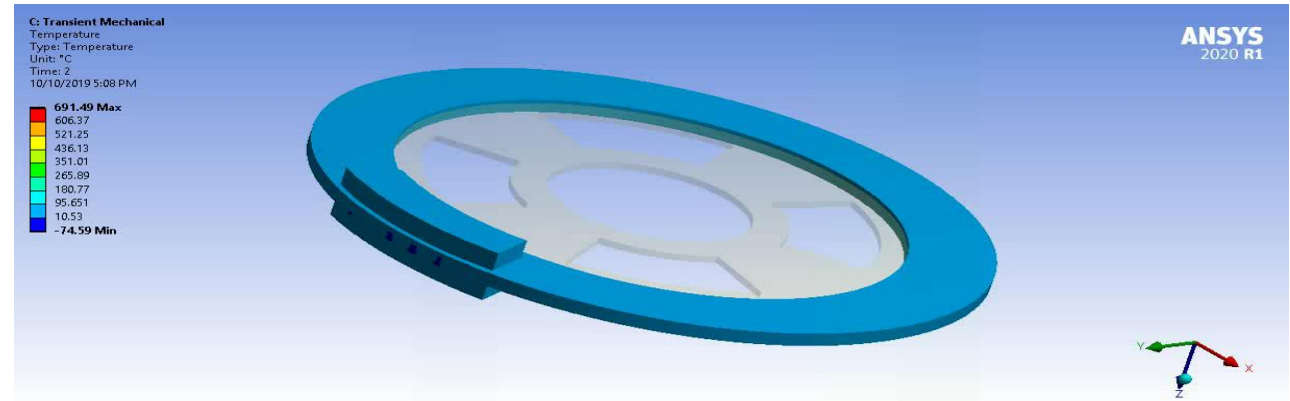


Piezoelectric Acoustics



Details of "Coupled Field Static (A5)"

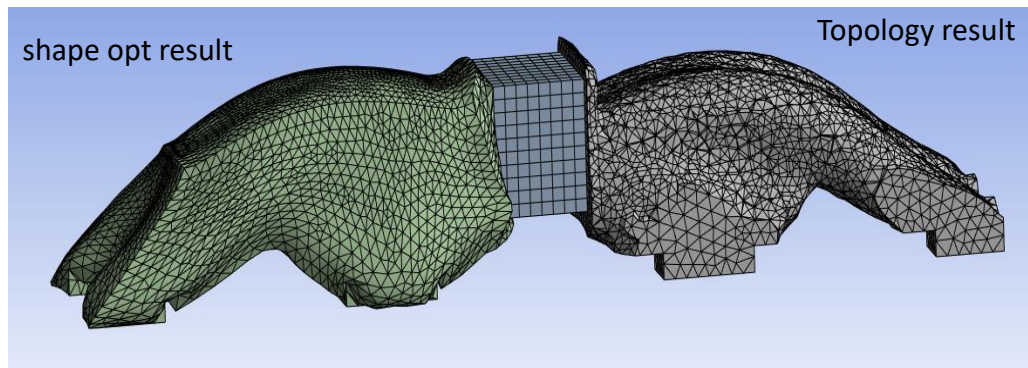
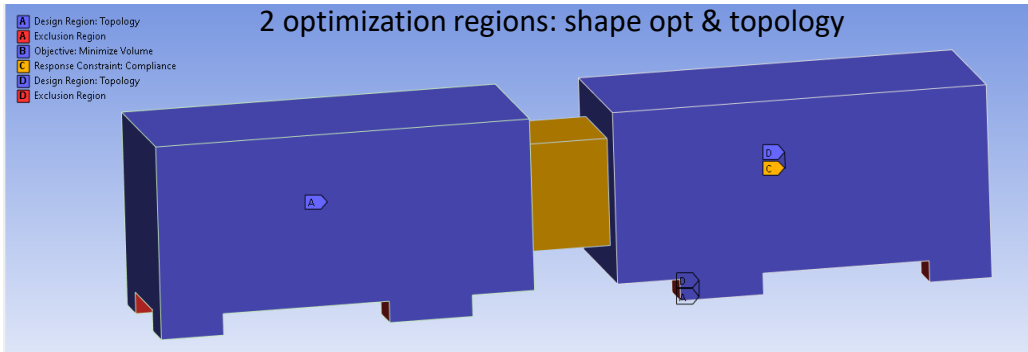
Physics Definition	
Structural	Yes
Acoustics	No
Thermal	No
Electric	Yes
Definition	
Analysis Type	Static
Solver Target	Mechanical APDL
Options	
Generate Input Only	No



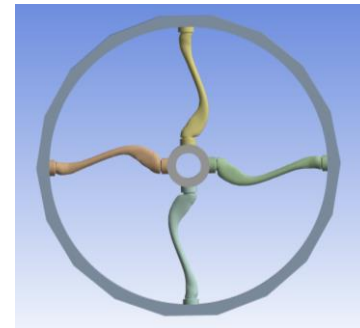
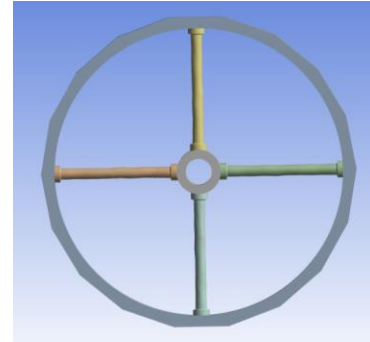
Structural Thermal

Structural Optimization – Shape, Topology, Topography, Lattice

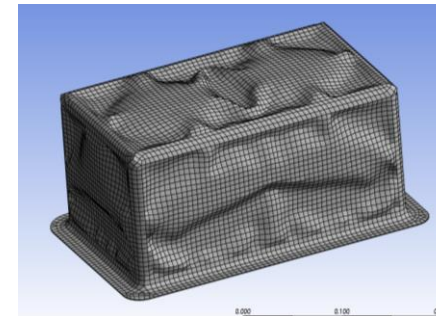
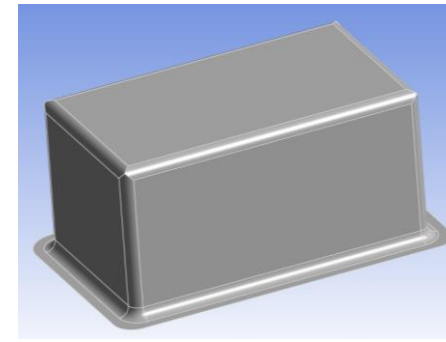
Mixed Shape and Topology



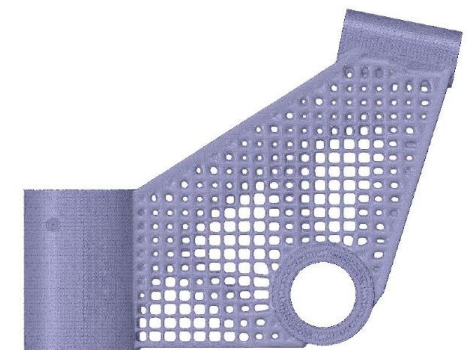
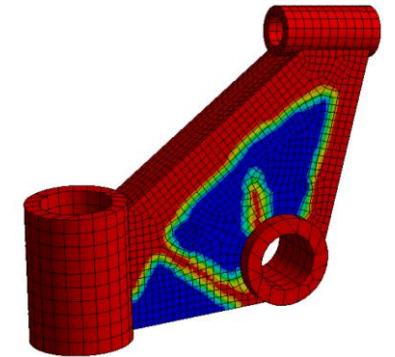
Shape



Topography

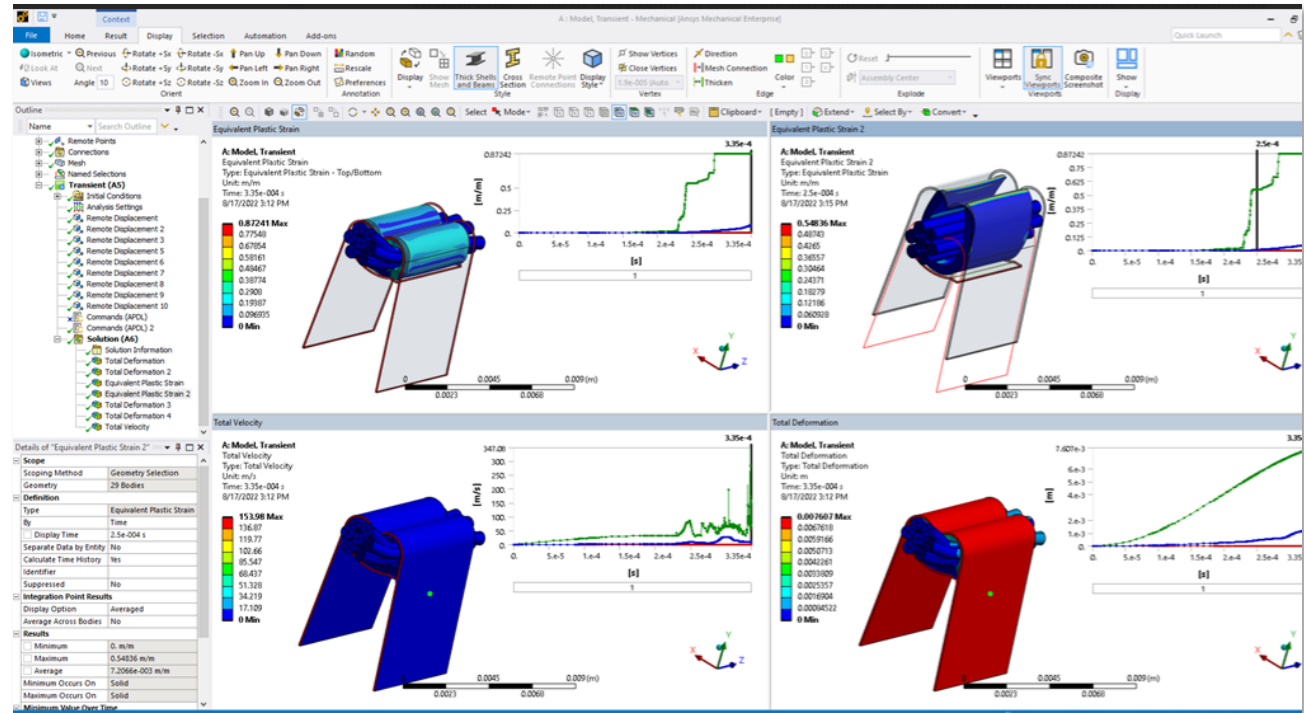
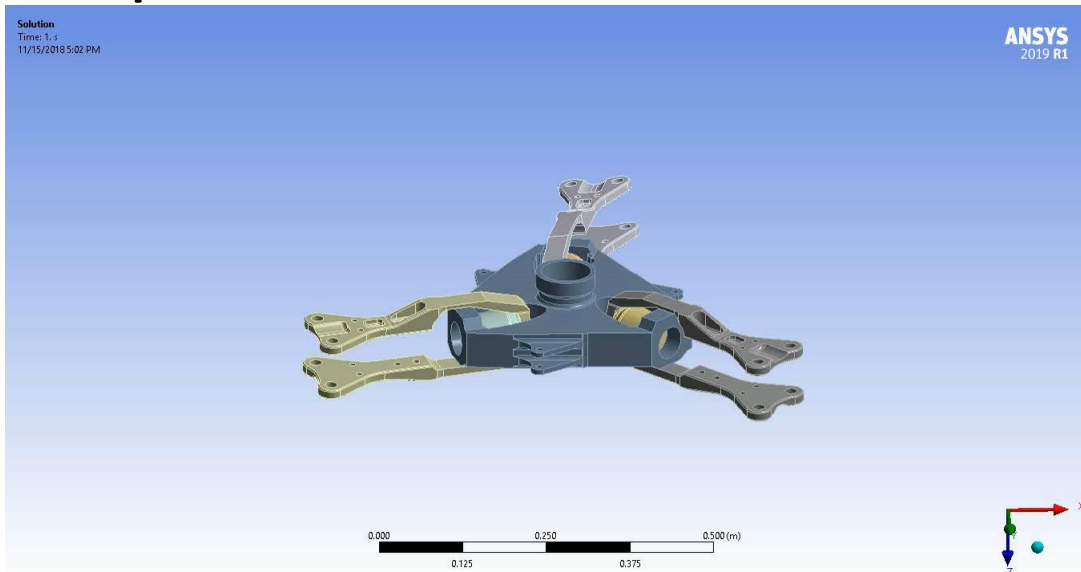


Lattice



Source:
CADFEM

Post Processing - Keyframe Animation, Embedded Graphs, Separated Data

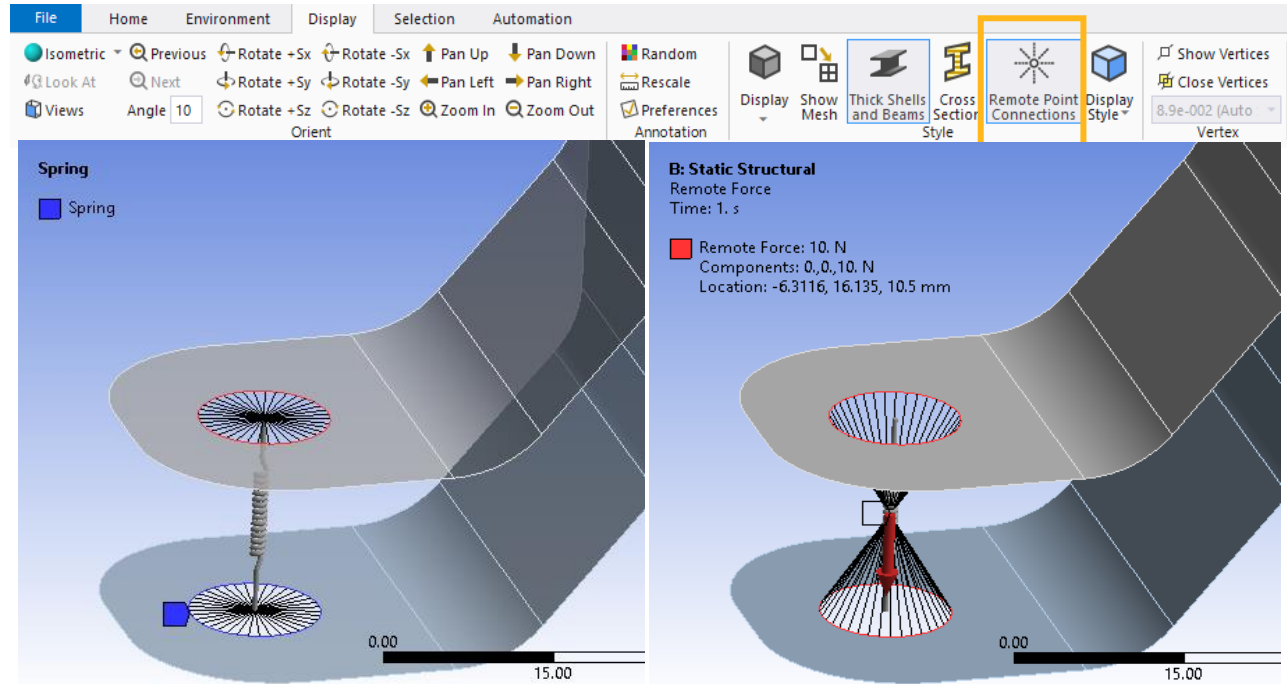
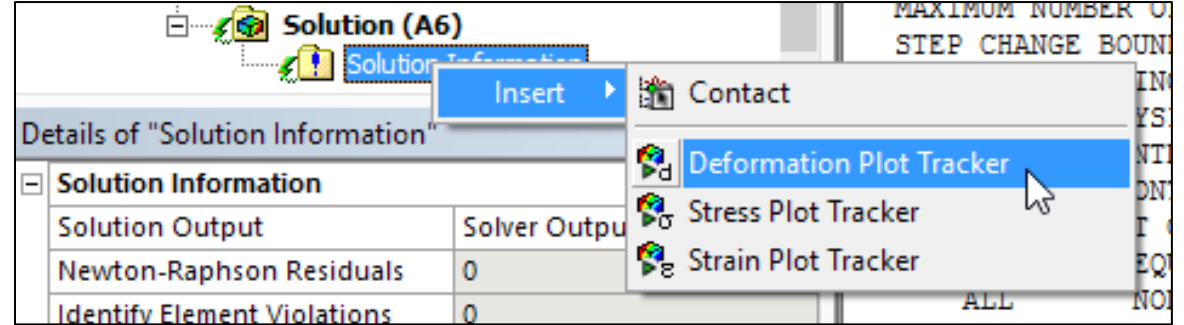
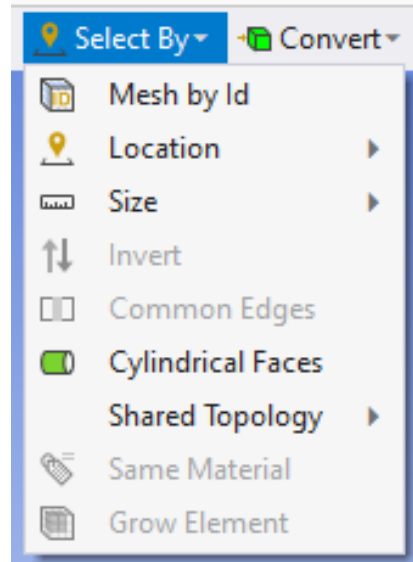
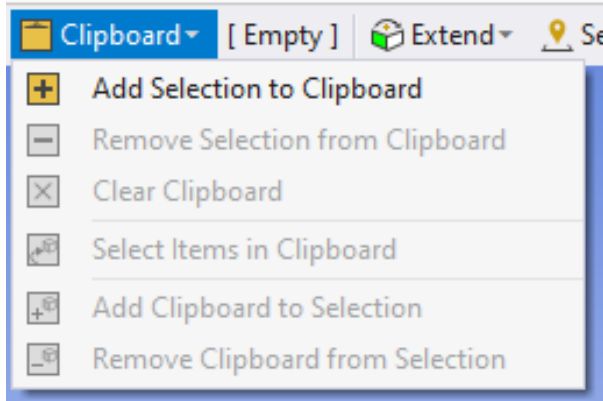
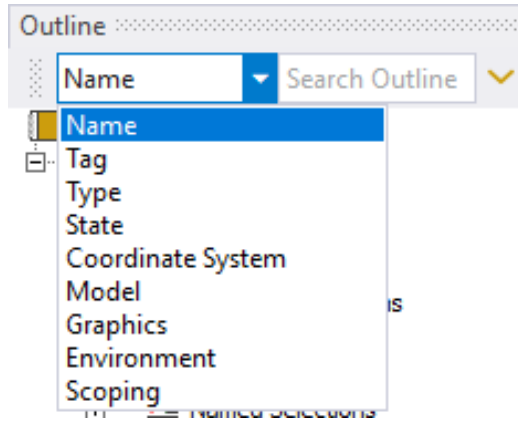


Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
By	Time
<input type="checkbox"/> Display Time	Last
Separate Data by Entity	Yes
Calculate Time History	Yes
Identifier	

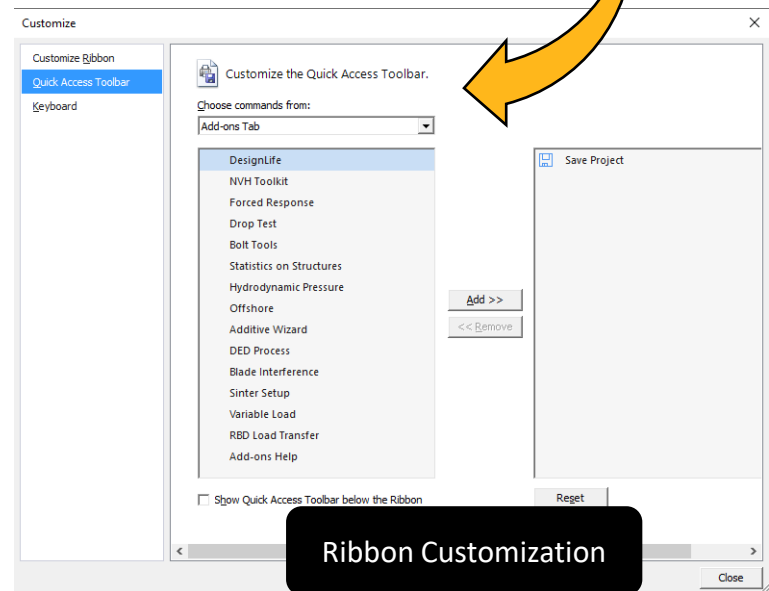
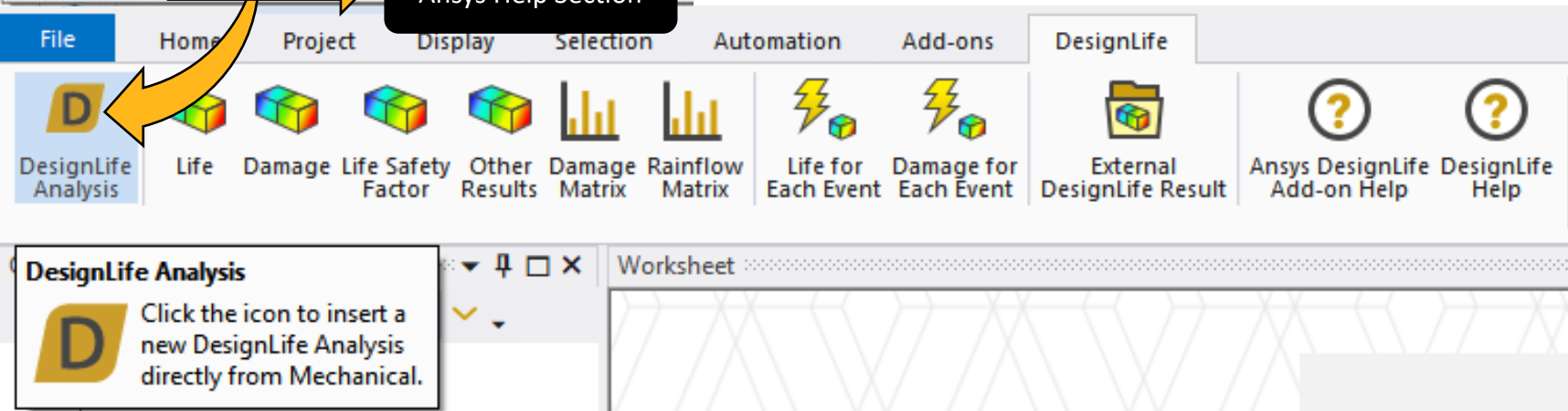
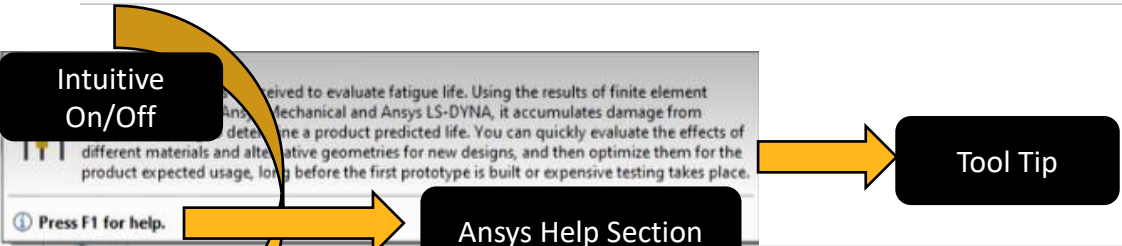
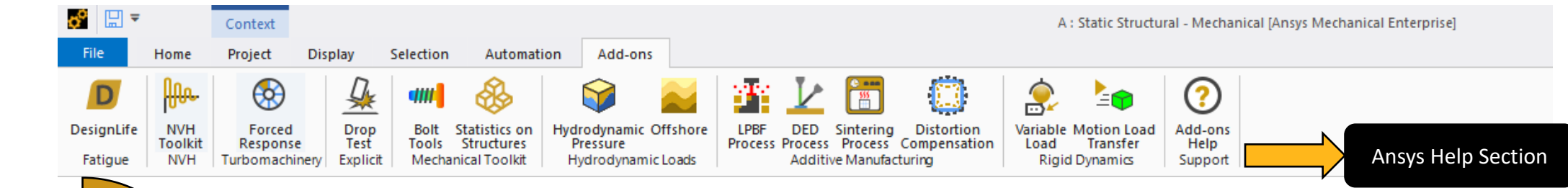
Tabular Data		Min 1 - Reference 6 [m]	Max 1 - Reference 6 [m]	Min 2 - Reference 33 [m]	Max 2 - Reference 33 [m]	Min 3 - Reference 60 [m]	Max 3 - Reference 60 [m]
1	1.	7.3201e-006	9.5788e-006	9.7814e-007	7.4912e-006	0.	3.5401e-006
2	2.	1.464e-005	1.9158e-005	1.9563e-006	1.4982e-005	0.	7.0803e-006
3	3.	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

Body ID

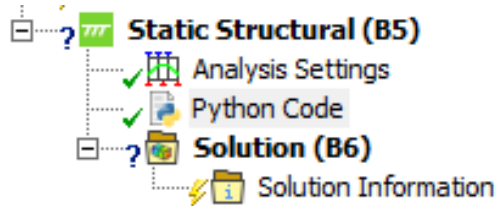
Some Quality of Life Improvements ...



ANSYS Mechanical Add-ons



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

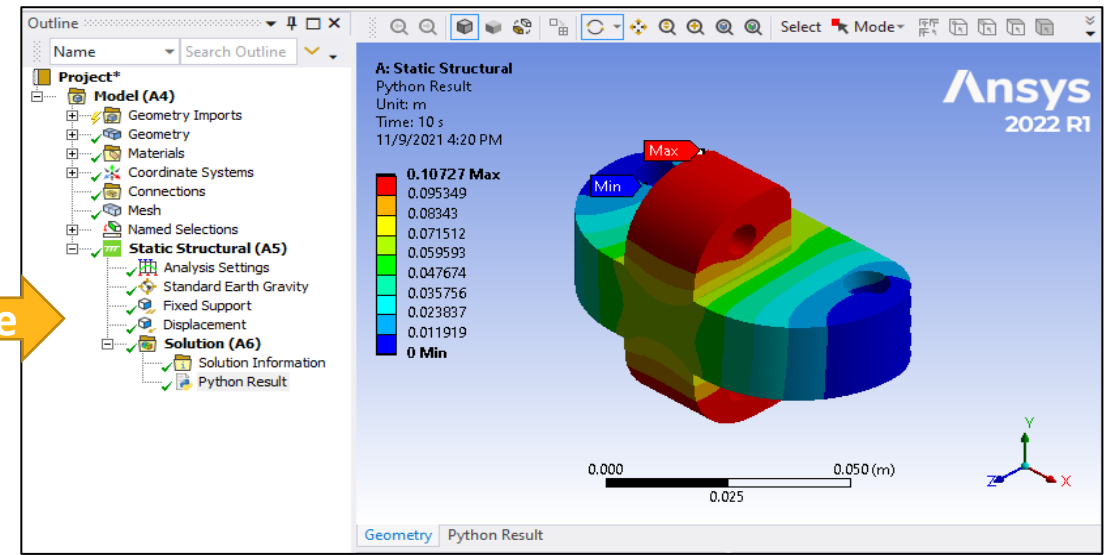
```

Python Code
1 def before_solve(this, analysis):# Do not edit this line
2     """
3     ... Called before solving the parent analysis.
4     ... Keyword Arguments:
5     ... this -- the datamodel object instance of the python code object you have selected in the tree
6     ... analysis -- Static Structural
7     ... """
8
9
10    ... # To access properties created using the Property Provider, please use the following command.
11    ... # this.GetCustomPropertyByPath("your_property_group_name/your_property_name")
12
13    ... # To access scoping properties use the following to access geometry scoping and named selection respectively:
14    ... # this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Geometry.Selection")
15    ... # this.GetCustomPropertyByPath("your_property_group_name/your_property_name/Named.Selection")
16
17    ... pass
    
```

Default Script

```

1 def post_started(sender, analysis):# Do not edit this line
2     define_dp_workflow(analysis)
3
4     # Uncomment this function to enable retrieving results from the table/chart
5     # def table_retrieve_result(value):# Do not edit this line
6     #     import mech_dp
7     #     import Ans.DataProcessing as dpf
8     #     wf = dpf.Workflow(this.WorkflowId)
9     #     wf.Connect('contour_selector', value)
10    #     this.Evaluate()
11
12    def define_dp_workflow(analysis):
13        import mech_dp
14        import Ans.DataProcessing as dpf
15        mech_dp.setExtAPI(ExtAPI)
16        dataSource = dpf.DataSources(analysis.ResultFileName)
17        u = dpf.operators.result.displacement()
18        nrm = dpf.operators.math.norm_fc()
19        # timeScop = dpf.Scoping()
20        # timeScop.Ids = [1]
21        # u.inputs.time_scoping.Connect(timeScop)
22        u.inputs.data_sources.Connect(dataSource)
23        nrm.Connect(u)
24        dpf_workflow = dpf.Workflow()
25        dpf_workflow.Add(u)
26        dpf_workflow.Add(nrm)
27        # dpf_workflow.SetInputName(u, 0, 'time')
28        # dpf_workflow.Connect('time', timeScop)
29        dpf_workflow.SetOutputContour(nrm)
30        dpf_workflow.Record('wf_id', False)
31        this.WorkflowId = dpf_workflow.GetRecordedId()
    
```



Mechanical Scripting, Recording, Debugging

Scripting
The Mechanical Scripting View exposes tools that you can use to develop scripts and interactively test out commands.

```

1 sum = 0
2
3 for geoid in ExtAPI.SelectionManager.CurrentSelection.Ids:
4     geoEntity = DataModel.GeoData.GeoEntityById(geoid)
5     if geoEntity.Type == GeoCellTypeEnum.GeoBody:
6         sum += geoEntity.Volume
7         type = "volume"
8     if geoEntity.Type == GeoCellTypeEnum.GeoFace:
9         sum += geoEntity.Area
10        type = "area"
11    if geoEntity.Type == GeoCellTypeEnum.GeoEdge:
12        sum += geoEntity.Length
13        type = "length"
14
15 # values are reported in the CAD unit system so get that...
16 unit = Model.Geometry.LengthUnit
17 print("Total selected " + type + " is: " + str(sum) + " " + str(unit))
    
```

```

1
2 #region Context.Menu.Action
3 id_list = [57,62,151]
4 for solve_object in DataModelObjectList(id_list).DataModelObjects:
5     solve_object.Solve(True)
6 #endregion
7
    
```

```

1 #Insertion of Mohr-Coulumb Stress Tool
2 solution_2 = DataModel.GetObjectById(51)
3 stress_tool_2 = solution_2.AddStressTool()
4
5 stress_safety_factor_2 = stress_tool_2.AddSafetyFactor()
6
7 stress_tool_2.Theory = SafetyTheoryType.MohrCoulumbStress
8 stress_tool_2.Activate()
9
10 #Insertion of Deformation Plot Tracker with scoping
11 solution_information_1 = DataModel.GetObjectById(52)
12 total_deformation_2 = solution_information_1.AddDeformationPlotTracker()
13
14 selection = ExtAPI.SelectionManager.CreateSelectionInfo(SelectionTypeEnum.GeometryEntities)
15 selection.Ids = [5]
16 total_deformation_2.Location = selection
17
18 #Insertion of Eroded Internal Energy with scoping
19 solution_information_2 = DataModel.GetObjectById(64)
20 eroded_internal_energy_tracker_1 = solution_information_2.AddErodedInternalEnergy()
21
22 selection = ExtAPI.SelectionManager.CreateSelectionInfo(SelectionTypeEnum.GeometryEntities)
23 selection.Ids = [5]
24 eroded_internal_energy_tracker_1.Location = selection
25
    
```

Start Debugger (F5)

```

1 ...
2 Change the contact type to Frictionless and set the Pinball Region
3 to use a radius
4 ...
5
6 with Transaction():
7     connections = DataModel.GetObjectsByType(DataModelObjectCategory.Connections)
8     contacts = connections[0].GetChildren(DataModelObjectCategory.ContactRegion, True)
9     for contact in contacts:
10        contact.ContactType = ContactType.Frictionless
11        contact.PinballRadius = 0.005
12
13 print("Done with script, changed %s contact regions" % (len(contacts)))
15
    
```

Watch Expression	Value
contact.PinballRadius	'0.005 [m]'

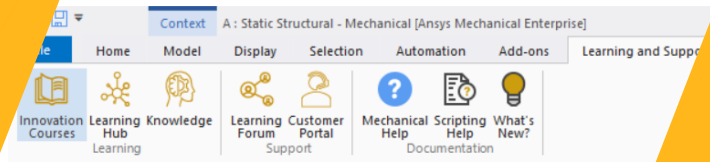


2023 R2

Highlights



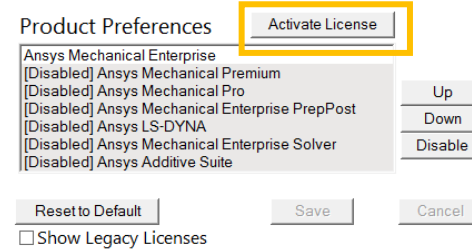
Mechanical



Learning and Support Tab

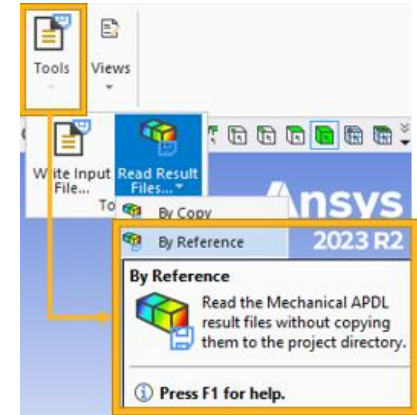
- ✓ Give user access to Innovation Courses, Learning Hub, Knowledge, Learning Forum, Customer Portal by adding Learning and Support Tab in Mechanical.
- ✓ Add access to Mechanical Help, Scripting Help, and What's New to this tab.

License Options



Activate License within Mechanical

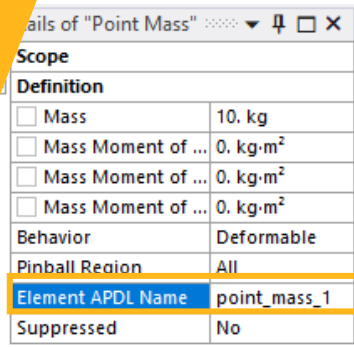
- ✓ If a license is not available/checked out when you open Mechanical, the application automatically opens in Read-Only Configuration mode
- ✓ From the License Options pane select a license and click the activate button. This removes the application from read-only mode and activates Mechanical
- ✓ This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



Read Result Files by Reference

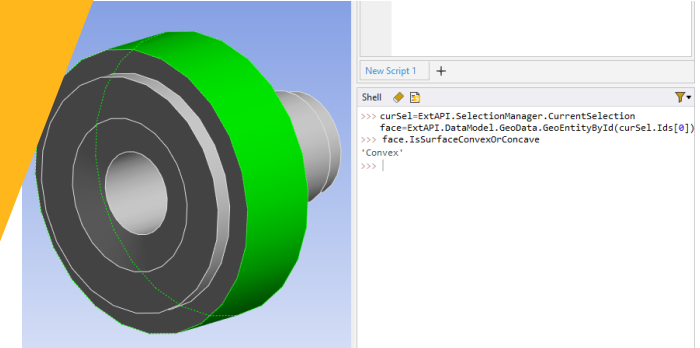
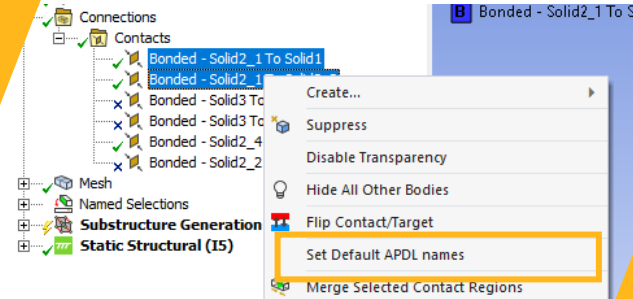
- ✓ Read Result Files... button is replaced by a menu button with two options:
 - By Copy: this is the old method which copies the files to the solver directory.
 - By Reference: using this new option, one can specify the path to Mechanical APDL result files, and the application reads the files without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

Contact & Target APDL Name

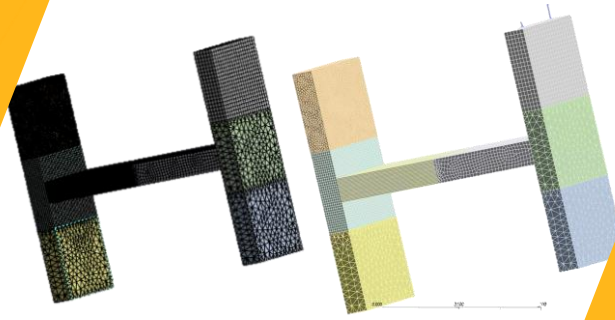
New APIs

- ✓ 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

- ✓ 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>`

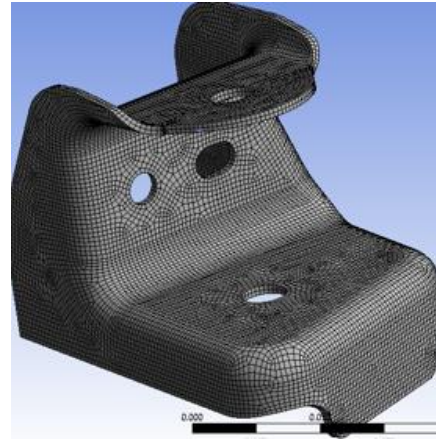
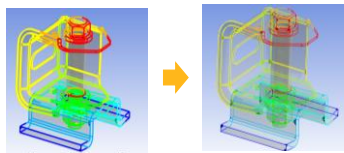
- ✓ New API to query whether the surface is Concave or Convex for Cone and Cylinder.
 - `IsSurfaceConvexOrConcave`
- ✓ New API to get radius of Cylindrical Face and Spherical Face
- ✓ New API to get radius of Circular Edge

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

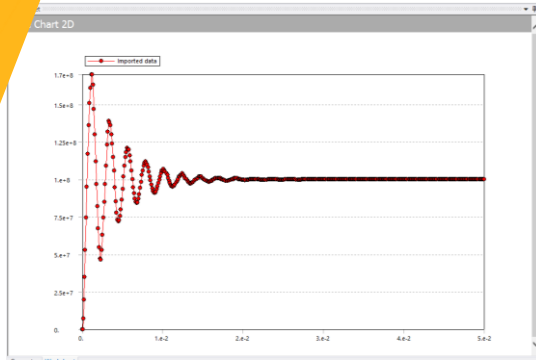
- ✓ Panning, zooming, and rotating is significantly more responsive for results with large FE scoping (1 million+)
- ✓ For models using shared mesh beam sections expect 4-5x improvements in mesh display speed
- ✓ For models using many shared non-mesh beam sections (10k+) expect a more modest 1.5x improvement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

	A	B	C	D
1	# Result File Name:	C:\Users\		
2	# Environment Name:	Static Structural		
3	# Result Name:	Total Deformation 2		
4	# Result Type:	Total Deformation		
5	# Unit:	m		
6	# Coordinate System:	Global Coordinate System		
7	# Time:	10. s		
8				
9	Node Number	Total Deformation (m)		
10	1011	0		
11	1033	2.05E-11		
12	1044	0		
13	1135	0		

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



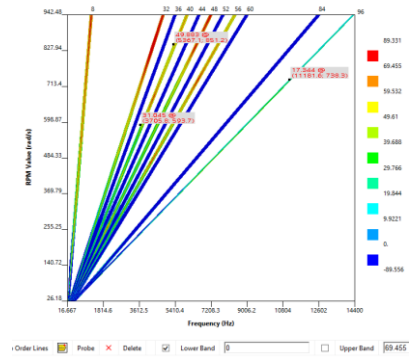
Line Chart object (LS-Dyna)

- ✓ 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

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

APIs

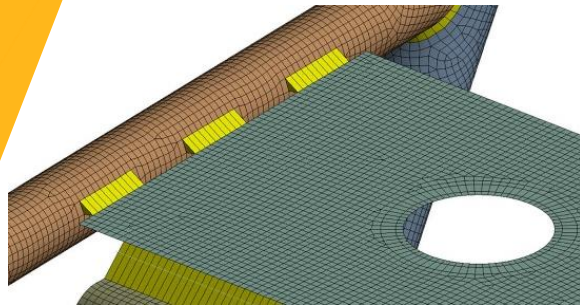
- ✓ The new Line Chart object also has a rich scripting API available for users to define datasets and to choose various display options for the datasets.
- ✓ There are also APIs available to apply Filters, perform Operations and to Import/Export chart data



Operations and filters

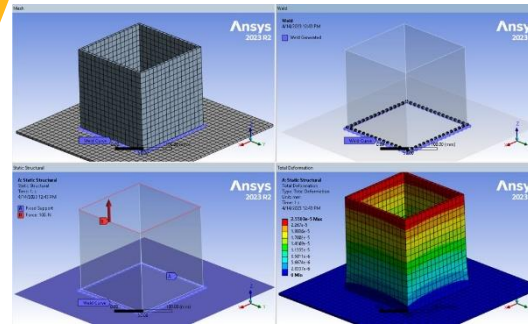
- ✓ Filters currently are:
 - Differentiate and Integrate
- ✓ Operations currently are:
 - Butterworth, SAE, Cutoff frequency + limit

Weld Meshing



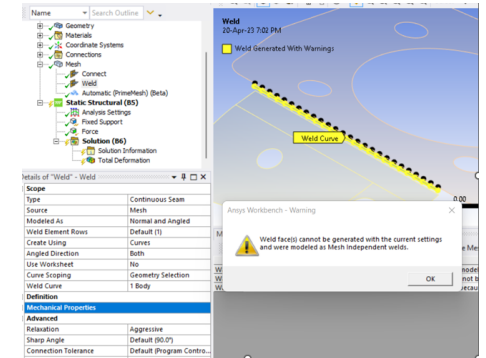
Weld Meshing

- ✓ 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

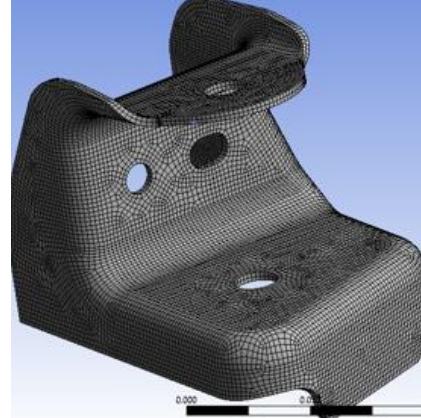
- ✓ When Weld is Modeled As “Normal” or “Angled” or “1D” or “Normal and Angled”, and the Relaxation property is set to Aggressive, sometimes Weld Face(s) fails to generate.
- ✓ In such scenarios, Mechanical internally creates Mesh Independent welds through bonded MPC contact.
- ✓ This applies for both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

Hex Meshing



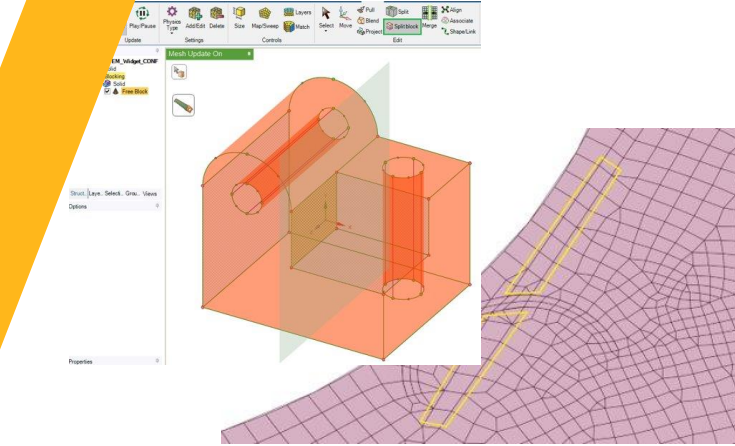
Improved Handling of Coiled Geometry for MultiZone

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



Solsh for Pull and MZ Thin Sweep

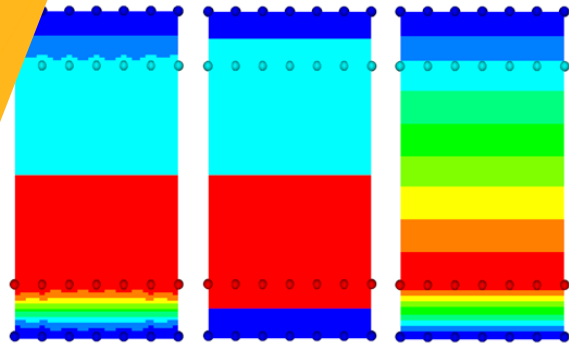
- ✓ Solid-Shell option is now available for
 - Mesh Edit Pull (Extrude) with a given vector direction
 - Multizone Decomposition Type "Thin Sweep"
- ✓ Solid-Shell element types are passed correctly to the solvers to produce Solsh190 (MAPDL) or T-Shell (LS-Dyna) element types



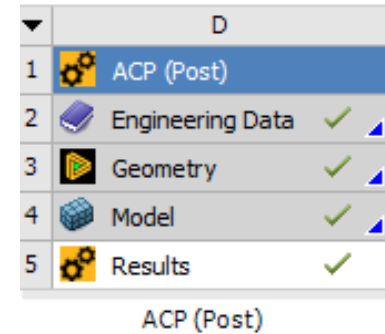
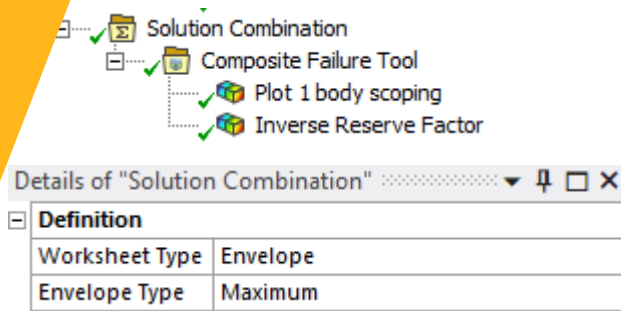
SpaceClaim Meshing

- ✓ 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).



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

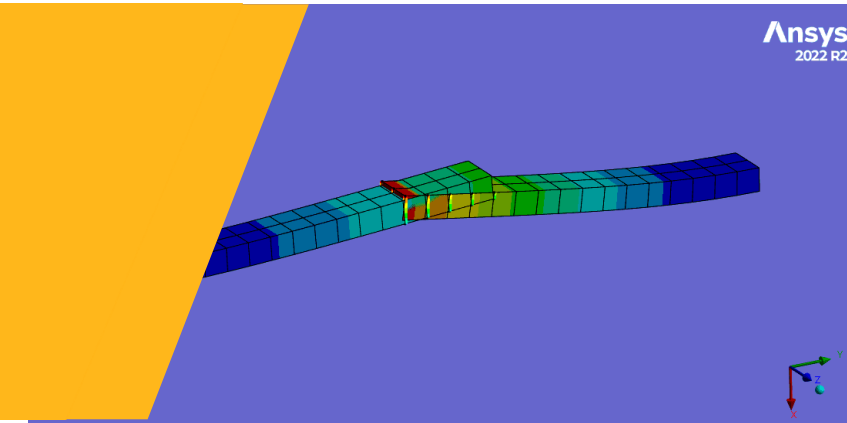
Envelope Solution in Mechanical

- ✓ 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

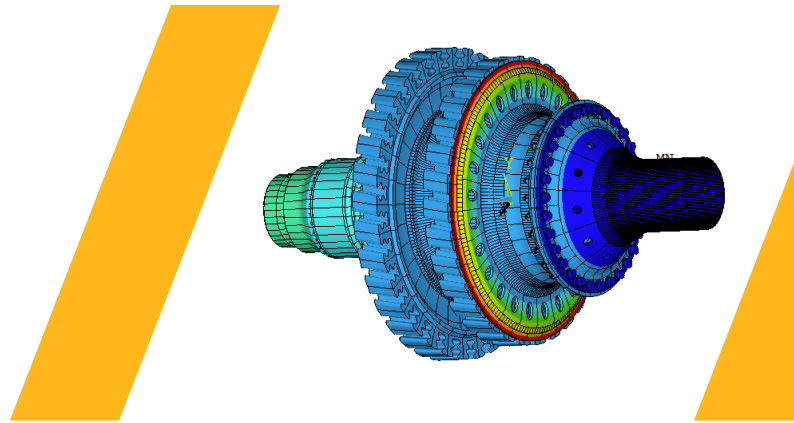
- ✓ ACP Post is in maintenance mode and is planned to be removed entirely in version 2025 R1. You may start using Ansys Mechanical to post-process composite simulations. The Composite Failure Tool, Sampling Point Tool and Envelope Solution are already available in Ansys Mechanical.
- ✓ Regarding scripting,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

Linear & Non-Linear Dynamics, Acoustics



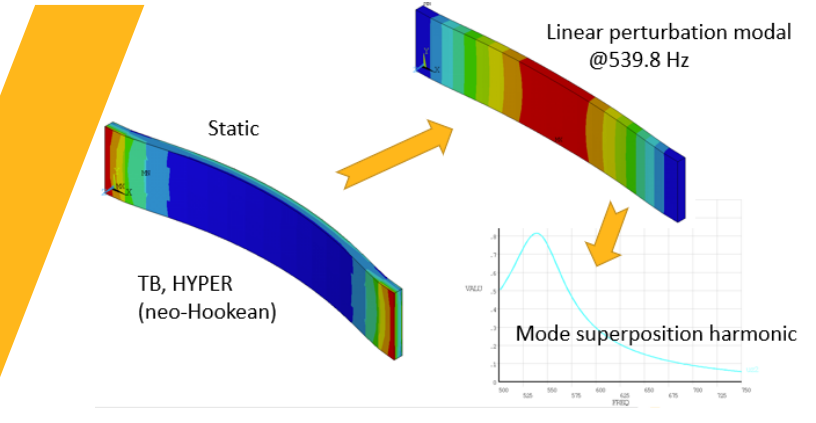
Harmonic Balance Method (HBM)

- ✓ 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



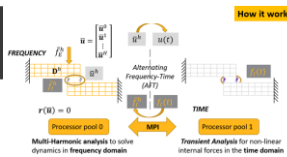
Multistage

- ✓ 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.

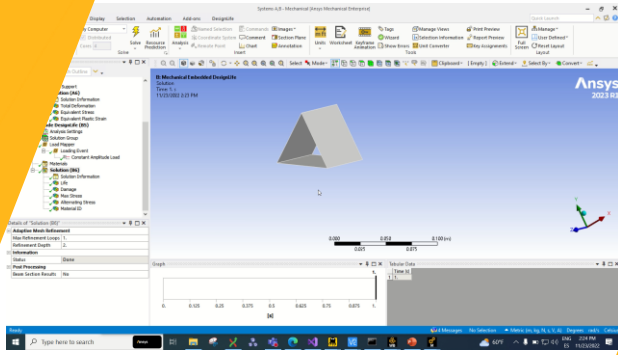


Other Enhancements

- ✓ 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

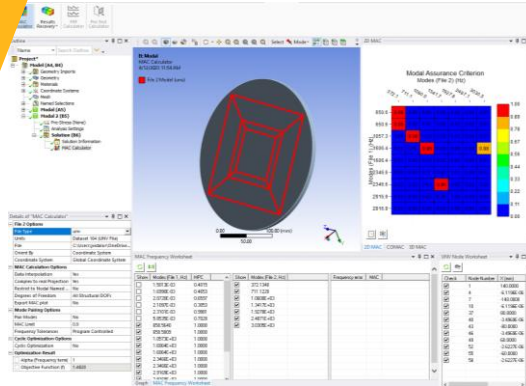


Add-ons



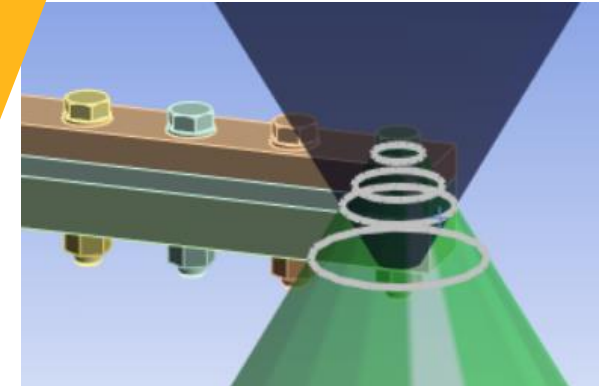
DesignLife Add-on

- ✓ 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 Analysis



NVH Add-on

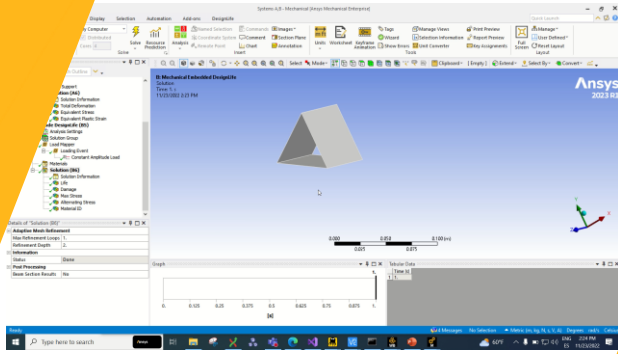
- ✓ Pre-Test Calculator is a new feature developed that introduces a result in the Tree. It identifies the optimum sensor and exciter locations for vibration tests
- ✓ Enhancements to MAC Calculator
 - Removes dependency from MAPDL commands, Preview Matching Nodes
 - Data Interpolation, Works with On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



Bolt Tools Add-on

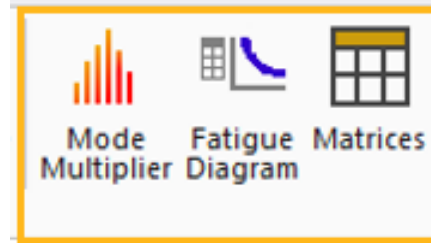
- ✓ Bolt pretension load objects can now be added for LS-DYNA analysis system using Bolt Tools (supporting both solid and beam type bolts)
- ✓ Cone of Compression Imprint Wizard - automatically creates imprints on selected geometries and defines contact between them.
- ✓ Mesh sizing: Separate mesh sizing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

Forced Response Add-on



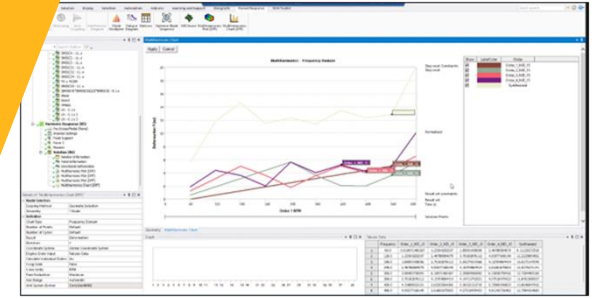
Mistuning and Aero coupling loads

- ✓ 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

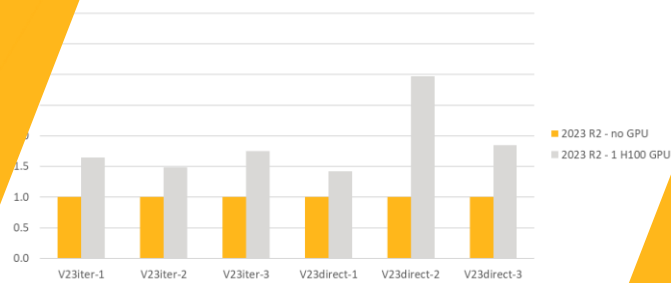
- ✓ Mode Multiplier, Matrices and Fatigue Diagram results under Harmonic System
- ✓ Multi-Stage choice is a new Add-On capability available as part of the Forced Response Add-on.
- ✓ This allows users to calculate a set of harmonics for each stage in a Multi-harmonic, multi-stage analysis



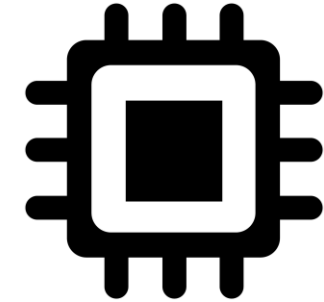
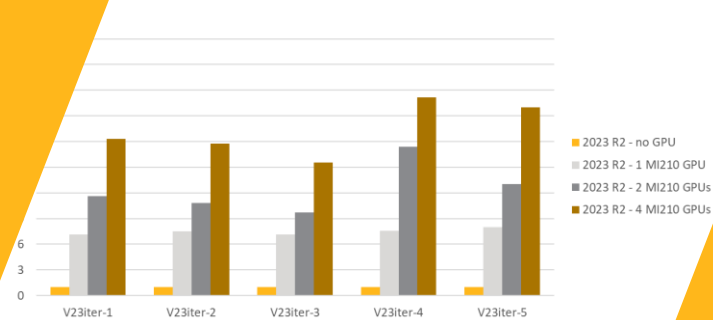
Multiharmonic Chart and Plot

- ✓ 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

Benefits of using NVIDIA H100 with 4 CPU cores



Benefits of using AMD MI210 with 4 CPU cores



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 ¹ 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 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 ³ (older) Intel MPI 2018 is automatically chosen

The Ansys logo consists of a yellow slanted bar followed by the word "Ansys" in a bold, black, sans-serif font.

