



# 2019 Innovation User Conference

22 May 2019, Coventry

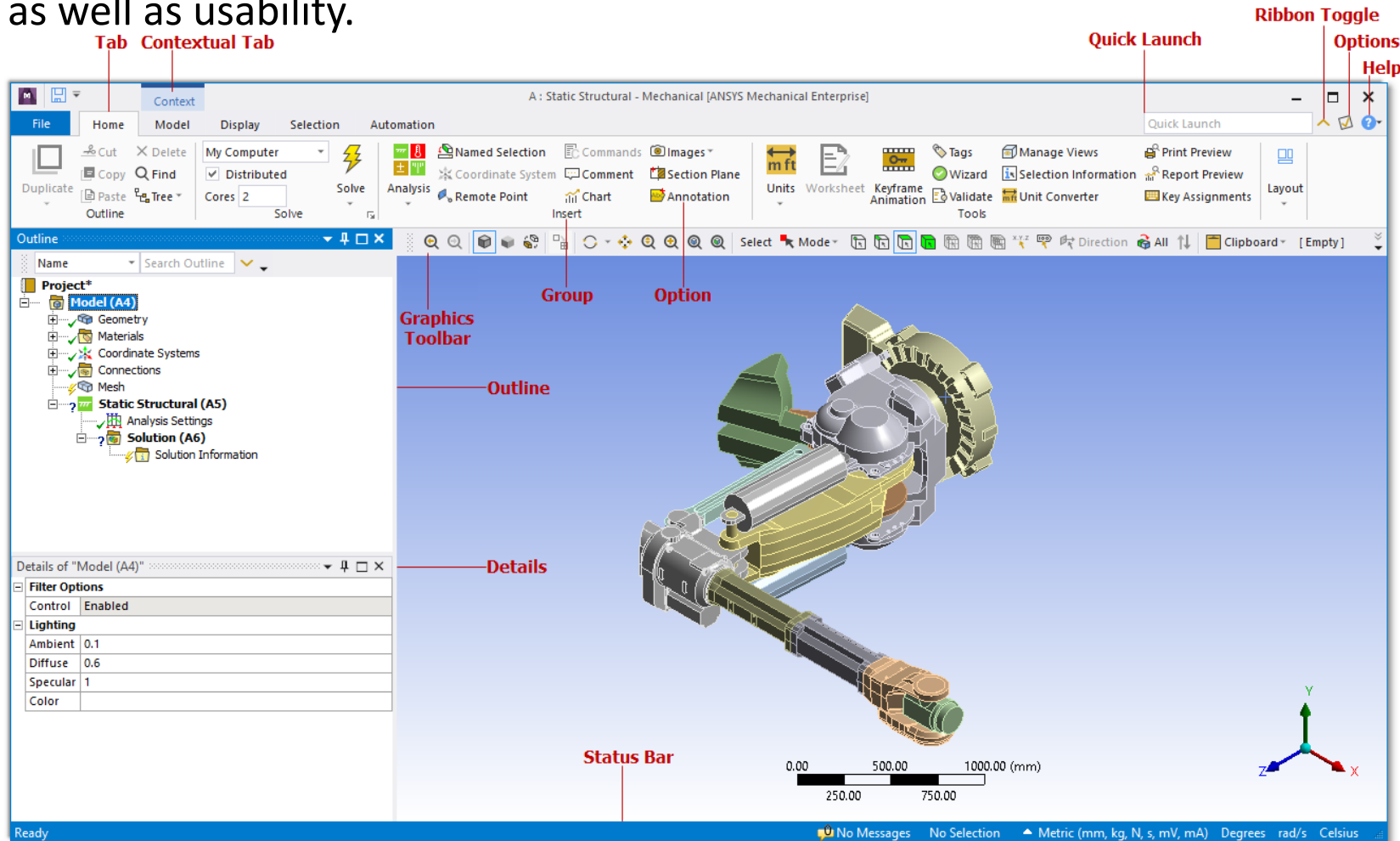


# Mechanical Interface Update

By John Lin  
Lead Engineer  
ANSYS UK

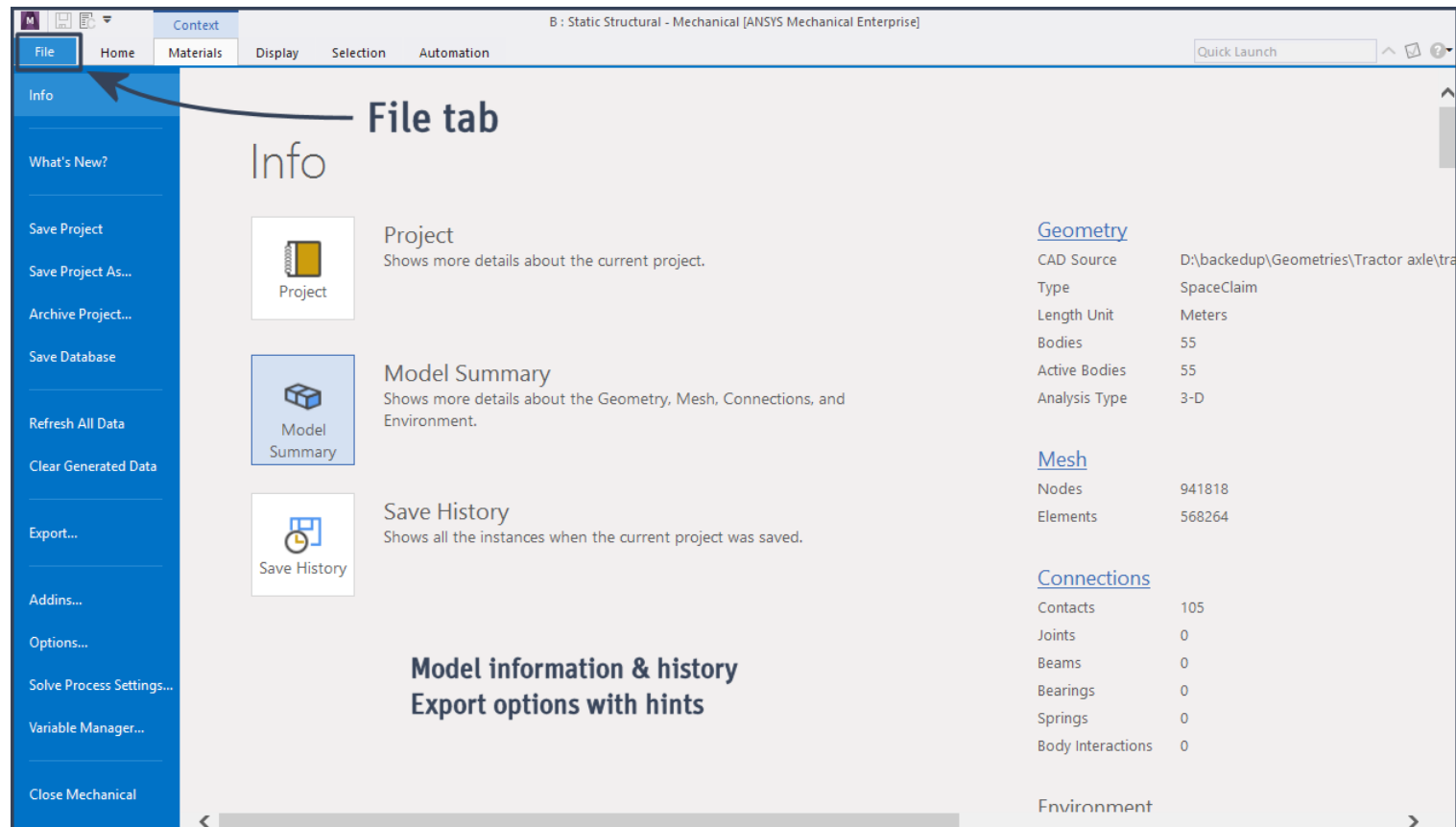
# New Ribbon Interface

- To improve usability, Mechanical now organizes its tools and commands using a **contextual ribbon**. Similar commands are organized together in Tabs, making the interface faster to use, more intuitive, and helpful.
- Additions like the Quick Launch option, expanded Tool Tips and customizable Graphics Toolbar improve discoverability as well as usability.



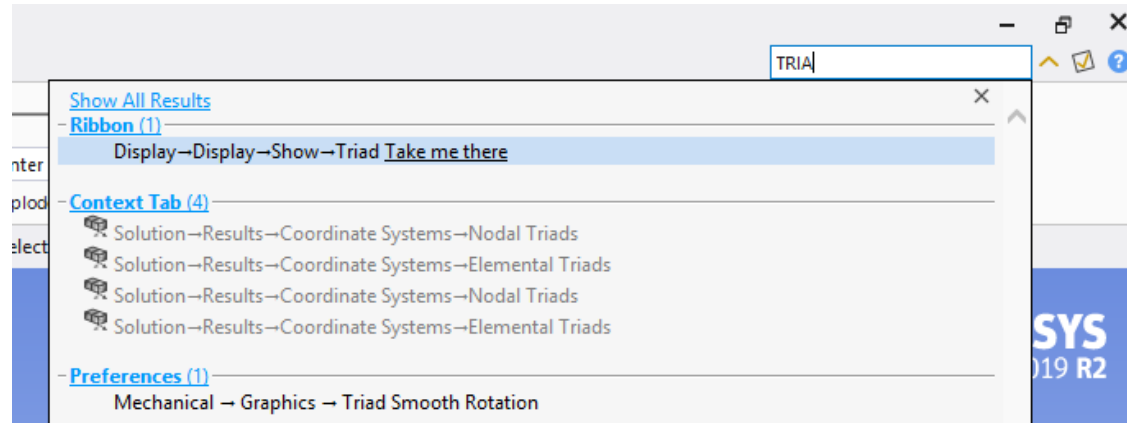
# File Tab

- The File tab contains a variety of options for managing your project, exporting data, making changes to default application settings and setting up how you want your simulation to run.

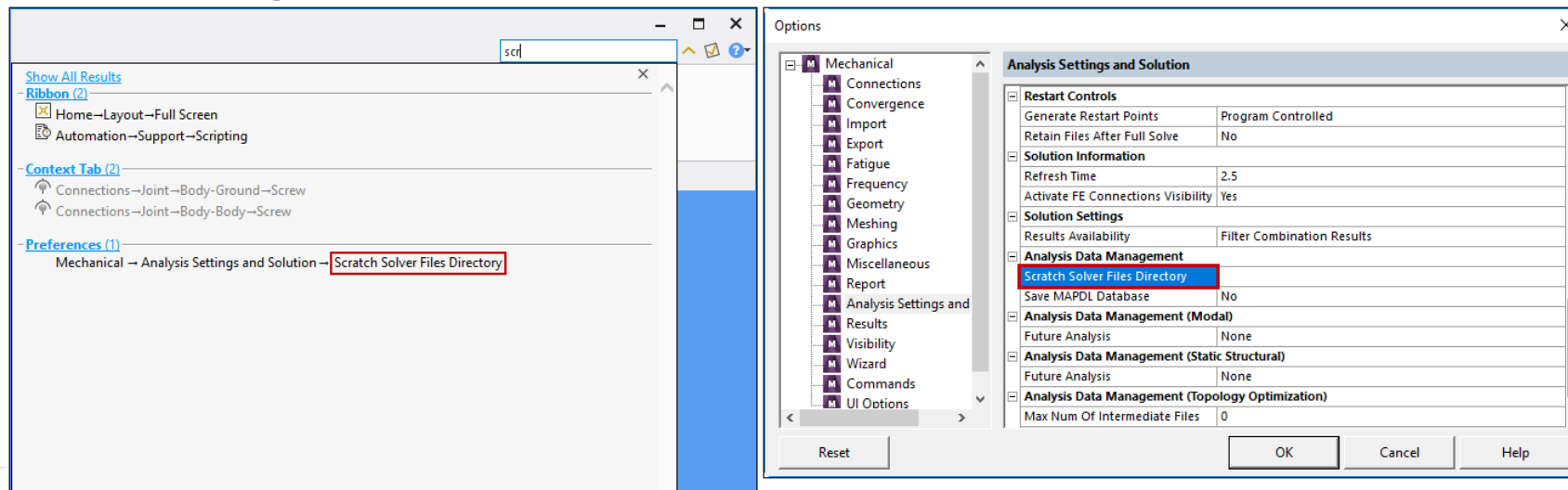


# Quick Launch

- The **Quick Launch** tool enables you to quickly search for a desired feature or interface option and automatically insert or launch the desired item or highlight the pertinent interface option (**Take me there**).

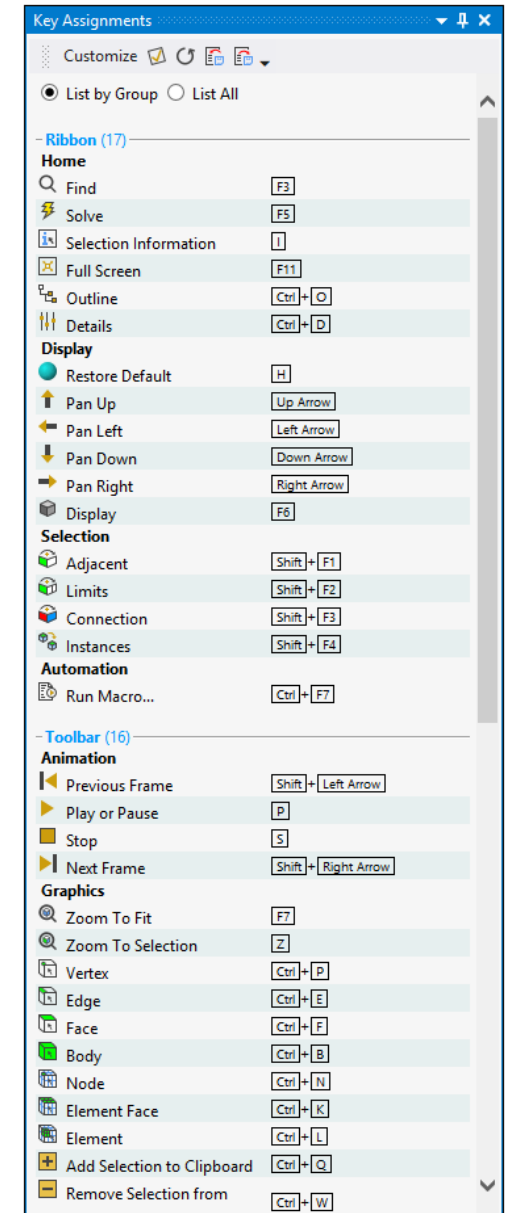
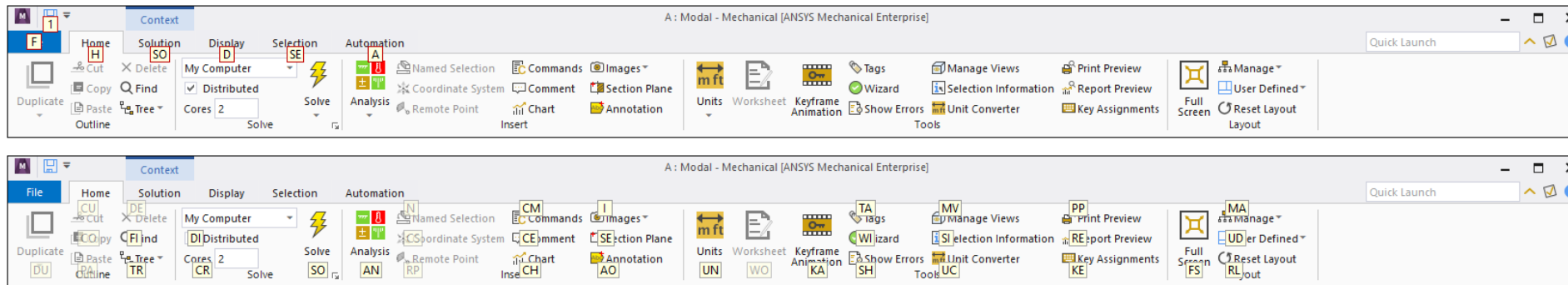


- Results display for three categories: **Ribbon**, **Context Tab**, and **Preferences**.



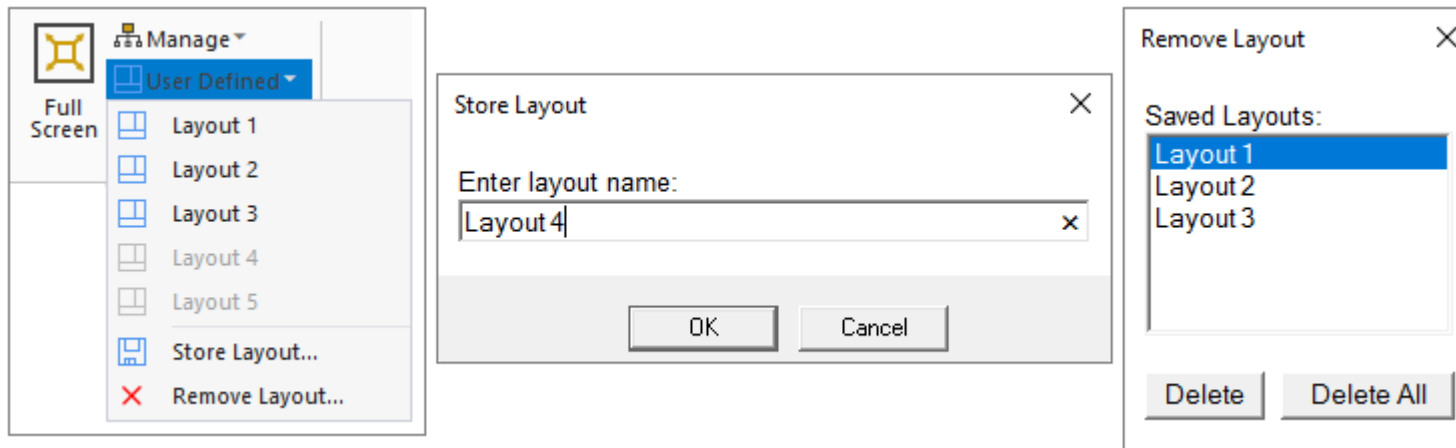
# Key Assignments

- Mechanical Hotkeys are now contained in the **Key Assignments** window.
- Accessible from the **Tools** group on the Home tab.
- Most keyboard key and key combination shortcuts available in the application can now be customized.
- **Import** and **Export** options are available.
- In addition, selecting the **[Alt]** key displays additional keyboard selection options.



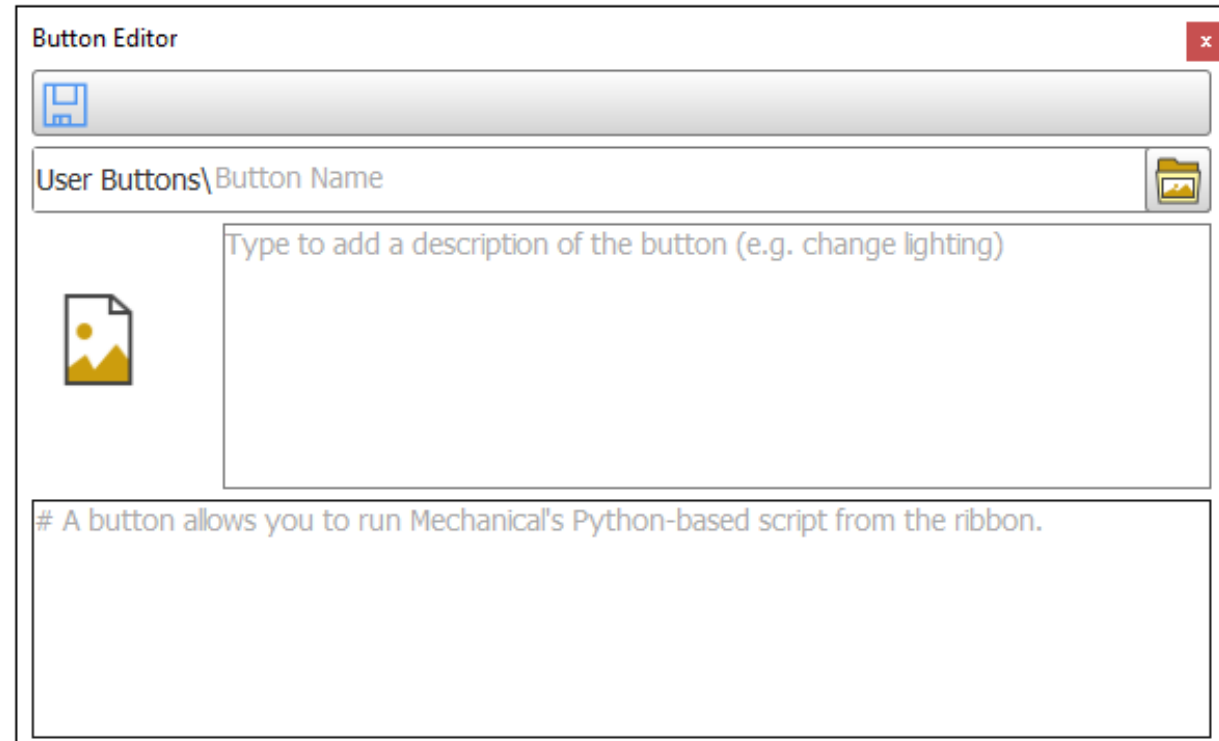
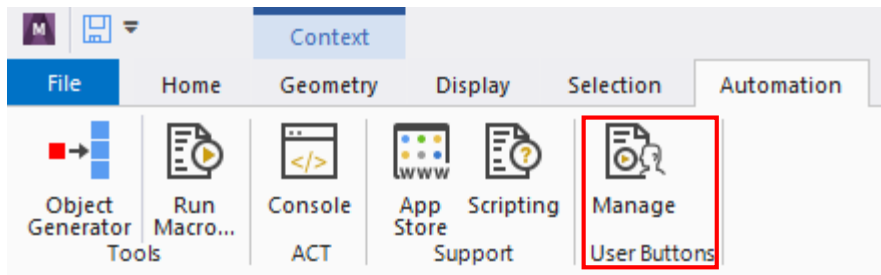
# Manage Interface Layouts

- Users often take the time to position the various interface windows and panes of their simulation in a specific manner.
- You can now save your window layout configurations using the **Store Layout** option of the **User Defined** drop-down menu on the **Home** tab.
- Stored layouts can be applied to the interface at any point.



# User Buttons

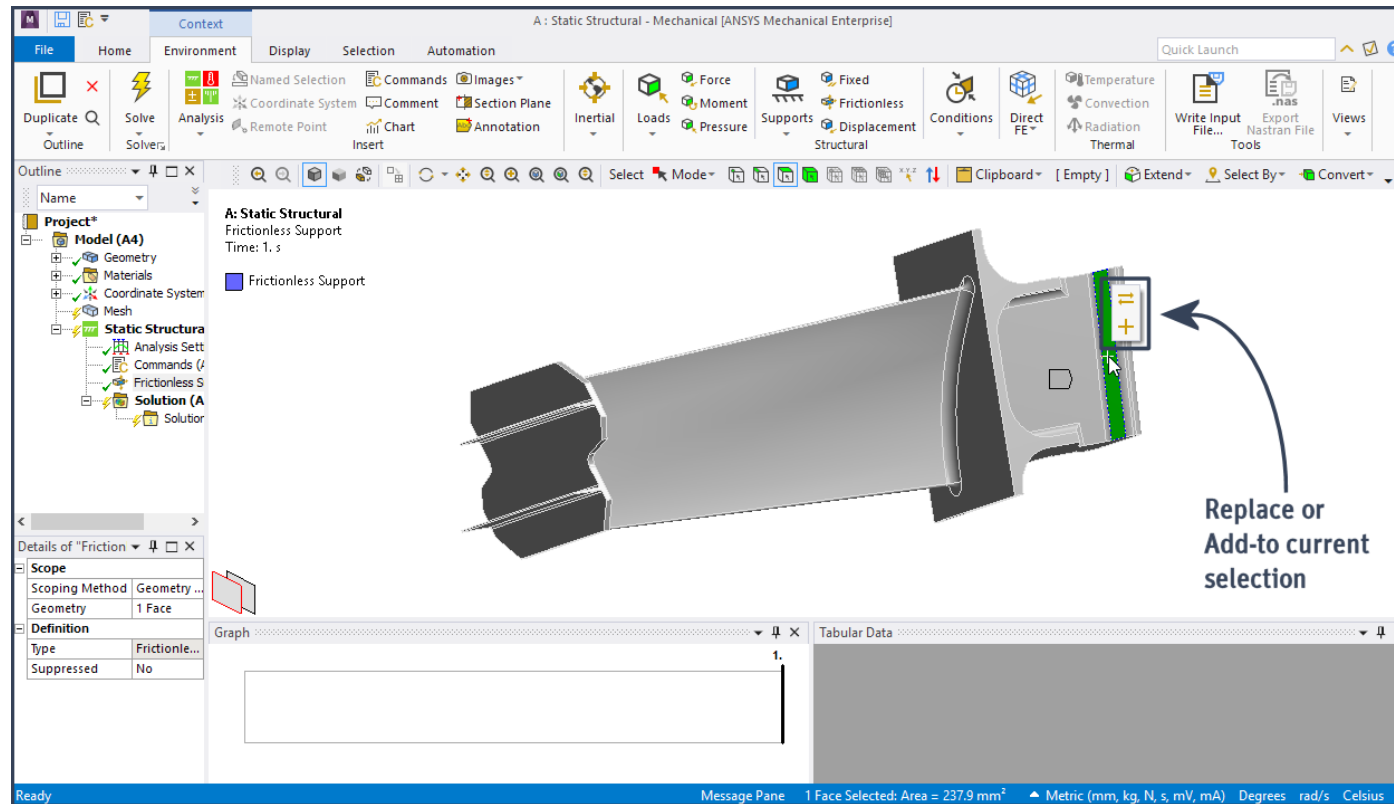
- Mechanical now enables you to **add** buttons to the **Automation** tab that can perform specific actions based on Mechanical's scripting API.
- Using the **Manage** option of the **User Buttons** group of the **Automation** tab, you can create, edit, and manage **User Defined** buttons.
- An icon can be assigned to each button.





# Selection options

- New options are available in the Graphics window to quick **apply** the current selection as the scoping for the active object
- In addition, when applicable, options also allow you to **add** to or **remove** from the current scoping.



# Engineering Data in Mechanical

- A new **Assignment** panel in Mechanical lets you assign a material directly from Engineering Data libraries without accessing the Engineering Data Workspace.
- You can quickly choose a favorite, recent, or project material, or you can **search** for a material in the active libraries. Search by Label, Library, material Model and/or Properties, or use filters.

The screenshot displays the ANSYS Mechanical interface with the Engineering Data Materials panel open. The panel shows a search bar with the text "Granta bilinear stainless" and a list of materials including "Stainless steel, 15-5PH, H1100", "Stainless steel, 17-4PH, cast, H1100", "Stainless steel, 17-7PH, TH1050", "Stainless steel, 201, 1/2 hard", "Stainless steel, 201, full hard", "Stainless steel, 205, annealed", "Stainless steel, 301, 1/2 hard", "Stainless steel, 301, annealed", "Stainless steel, 301, full hard", "Stainless steel, 302, annealed", "Stainless steel, 302, grade D", "Stainless steel, 304, 1/2 hard", "Stainless steel, 304, 1/4 hard", and "Stainless steel 304 NI annealed".

The Assignment panel is also visible, showing the material "Structural Steel" assigned to "Part 1". The panel includes sections for Graphics Properties, Definition, Material, Bounding Box, Properties, and Statistics.

The Engineering Data Materials panel also shows a search bar with the text "Granta bilinear stainless" and a list of materials including "Stainless steel, 15-5PH, H1100", "Stainless steel, 17-4PH, cast, H1100", "Stainless steel, 17-7PH, TH1050", "Stainless steel, 201, 1/2 hard", "Stainless steel, 201, full hard", "Stainless steel, 205, annealed", "Stainless steel, 301, 1/2 hard", "Stainless steel, 301, annealed", "Stainless steel, 301, full hard", "Stainless steel, 302, annealed", "Stainless steel, 302, grade D", "Stainless steel, 304, 1/2 hard", "Stainless steel, 304, 1/4 hard", and "Stainless steel 304 NI annealed".

The Engineering Data Materials panel also shows a search bar with the text "Granta bilinear stainless" and a list of materials including "Stainless steel, 15-5PH, H1100", "Stainless steel, 17-4PH, cast, H1100", "Stainless steel, 17-7PH, TH1050", "Stainless steel, 201, 1/2 hard", "Stainless steel, 201, full hard", "Stainless steel, 205, annealed", "Stainless steel, 301, 1/2 hard", "Stainless steel, 301, annealed", "Stainless steel, 301, full hard", "Stainless steel, 302, annealed", "Stainless steel, 302, grade D", "Stainless steel, 304, 1/2 hard", "Stainless steel, 304, 1/4 hard", and "Stainless steel 304 NI annealed".

The Engineering Data Materials panel also shows a search bar with the text "Granta bilinear stainless" and a list of materials including "Stainless steel, 15-5PH, H1100", "Stainless steel, 17-4PH, cast, H1100", "Stainless steel, 17-7PH, TH1050", "Stainless steel, 201, 1/2 hard", "Stainless steel, 201, full hard", "Stainless steel, 205, annealed", "Stainless steel, 301, 1/2 hard", "Stainless steel, 301, annealed", "Stainless steel, 301, full hard", "Stainless steel, 302, annealed", "Stainless steel, 302, grade D", "Stainless steel, 304, 1/2 hard", "Stainless steel, 304, 1/4 hard", and "Stainless steel 304 NI annealed".

The Engineering Data Materials panel also shows a search bar with the text "Granta bilinear stainless" and a list of materials including "Stainless steel, 15-5PH, H1100", "Stainless steel, 17-4PH, cast, H1100", "Stainless steel, 17-7PH, TH1050", "Stainless steel, 201, 1/2 hard", "Stainless steel, 201, full hard", "Stainless steel, 205, annealed", "Stainless steel, 301, 1/2 hard", "Stainless steel, 301, annealed", "Stainless steel, 301, full hard", "Stainless steel, 302, annealed", "Stainless steel, 302, grade D", "Stainless steel, 304, 1/2 hard", "Stainless steel, 304, 1/4 hard", and "Stainless steel 304 NI annealed".

# Engineering Data in Mechanical

- When you select a material from the **Materials** folder in the **Outline**, you can view the data being used by Mechanical and also access edit capabilities in the Engineering Data Workspace.

The screenshot shows the 'Engineering Data: Material View' window for 'Structural Steel'. The left pane displays the 'Outline' with 'Structural Steel' selected under the 'Materials' folder. The main pane shows the material's properties, including density, structural properties, and common material properties. A 'Strain-Life Parameters' graph is also visible.

**Engineering Data: Material View**

**Structural Steel**

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

Density: 7850 kg/m<sup>3</sup>

**Structural**

Derive from: Young's Modulus and Poisson's Ratio

Property	Value
Young's Modulus	2e+11 Pa
Poisson's Ratio	0.3
Bulk Modulus	1.6667e+11 Pa
Shear Modulus	7.6923e+10 Pa
Isotropic Secant Coefficient of Thermal Expansion	1.2e-05 1/°C
Compressive Ultimate Strength	0 Pa
Compressive Yield Strength	2.5e+08 Pa

**Strain-Life Parameters**

Graph showing Strain Amplitude (log10) vs. Reversals to Failure, 2N (log10). The y-axis ranges from -5.37 to -0.662, and the x-axis ranges from 0.00 to 10.0. Two curves are plotted: a blue line for the Baseline and an orange line for the Fatigue Life.

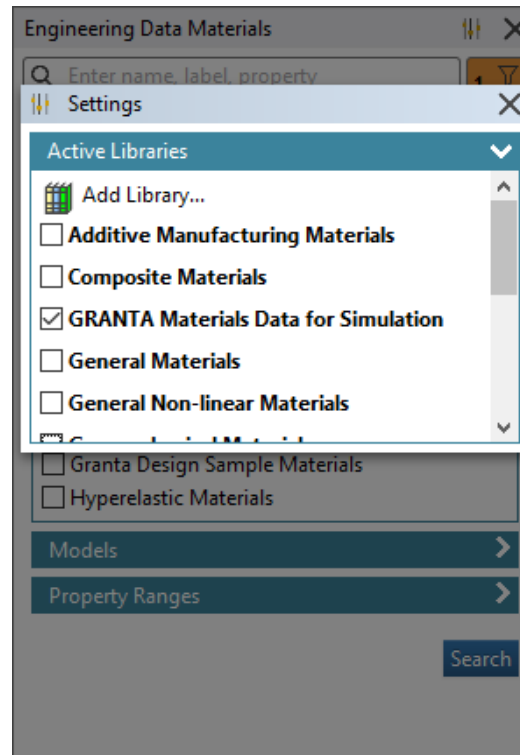
The screenshot shows the 'Engineering Data: Material View' window for 'Structural Steel', specifically the 'Strain-Life Parameters' graph. The graph plots Strain Amplitude (log10) on the y-axis (ranging from -5.37 to -0.662) against Reversals to Failure, 2N (log10) on the x-axis (ranging from 0.00 to 10.0). Two curves are shown: a blue line for the Baseline and an orange line for the Fatigue Life. Below the graph, a table provides the parameters for the Baseline and Fatigue Life curves.

**Structural Steel: Strain-Life Parameters**

Strength Coefficient [Pa]	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient [Pa]	Cyclic Strain Hardening Exponent
9.2e+08	-0.106	0.213	-0.47	1e+09	0.2

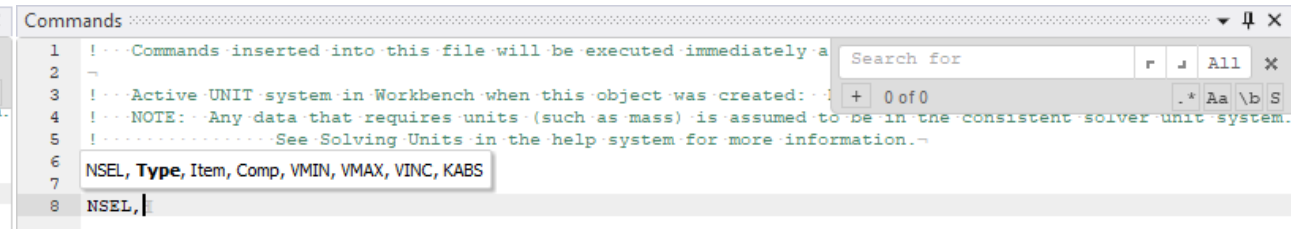
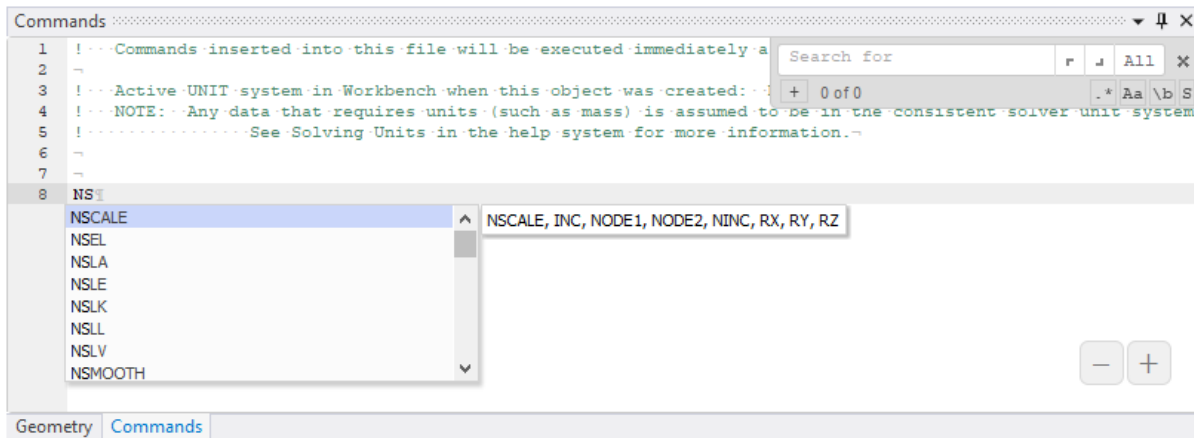
# GRANTA Materials Data for Simulation

- The **GRANTA Materials Data for Simulation** product is now available (in the Mechanical interface only).
- Using the new search capability exposed for Engineering Data in Mechanical, you can quickly search for specific materials in this library



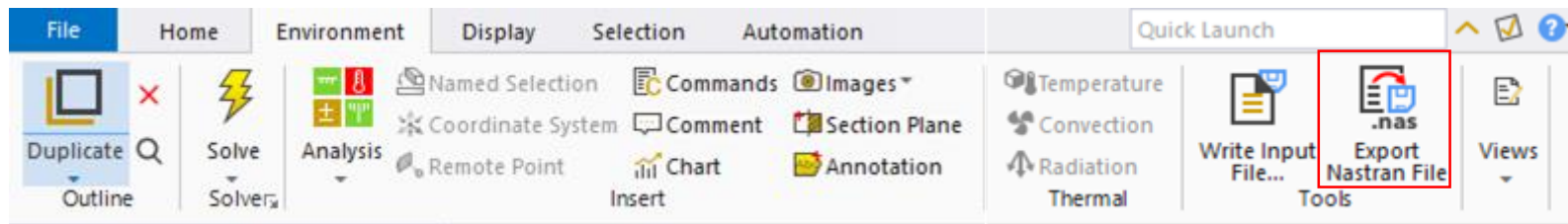
# Commands Object

- The commands object has been updated and now,
  - Provides an **auto-completion** drop-down menu (based on character entry) as well as a tooltip banner that displays the associated command arguments (Mechanical APDL Solver only).
  - Provides a search feature using the **Ctrl+F** key combination.
  - Includes line numbers.
  - Displays hidden characters such as paragraph marks and spaces.
  - Provides **colored syntax highlights** for entries such as fixed values, variables, comments, etc. (Mechanical APDL or Rigid Body Dynamics solvers only).
  - Provides integrated zoom in and zoom out options. Previously, zooming was performed using the mouse wheel.
  - Has a preference category in the **Options** dialog that lets you specify the default behavior of some of the above features.

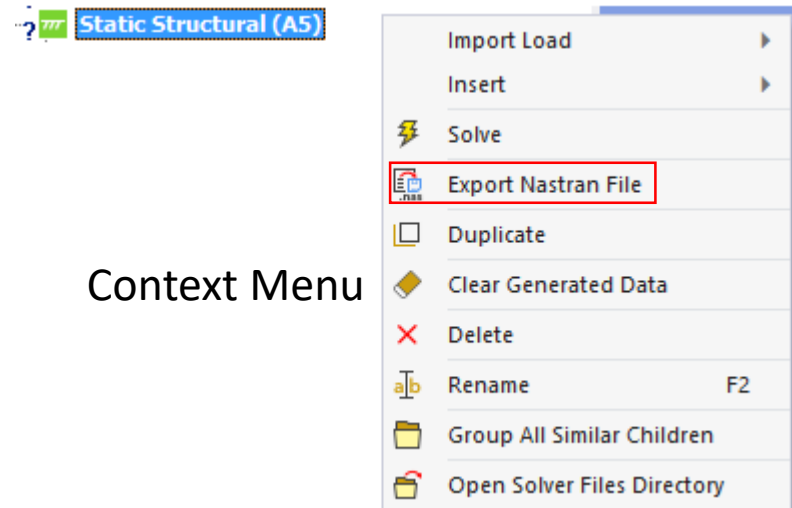


# NASTRAN File Export

- Users can now export your simulation as a NASTRAN bulk data (.nas or .bdf)
- Supported for:
  - Static Structural
  - Modal
- Supported through Export Nastran File option in both context menu and context tab for the environment.



Ribbon Tab



Context Menu

# New Solution Combination

- With the new solution combination, users can:
  - Specify **multiple** combinations.
  - Combine solutions for **Static** Structural, **Transient** Structural, and **Harmonic** Response analyses.
  - Specify solution combinations as either **Linear** or **SRSS** (Square Root of Summation of Squares).
  - Use Tabular Data or a result Set Number to specify which combination you wish to display.
  - **Import** and/or **Export** the Solution Combination Worksheet as a Comma Separated Value (CSV) file.

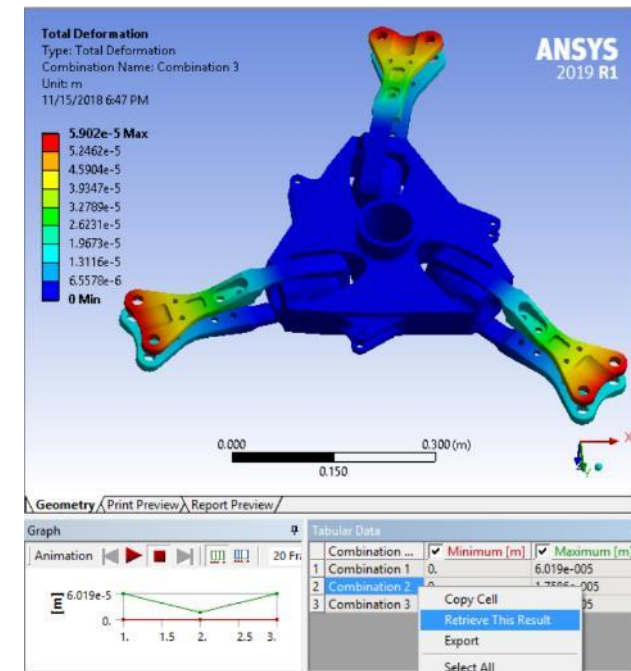
**Solution Combination**

\*Right click on the grid to add/delete a row or a column.

Import... Export...

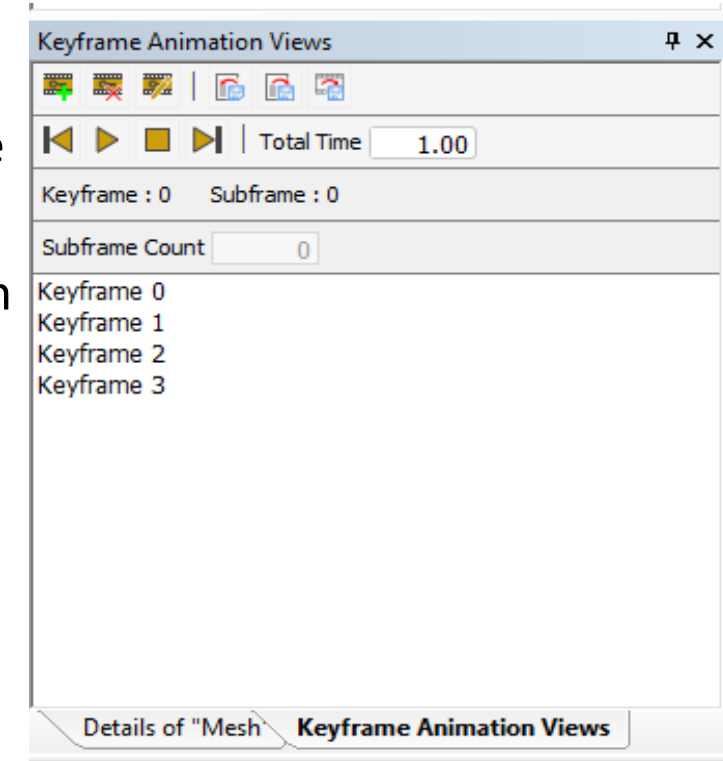
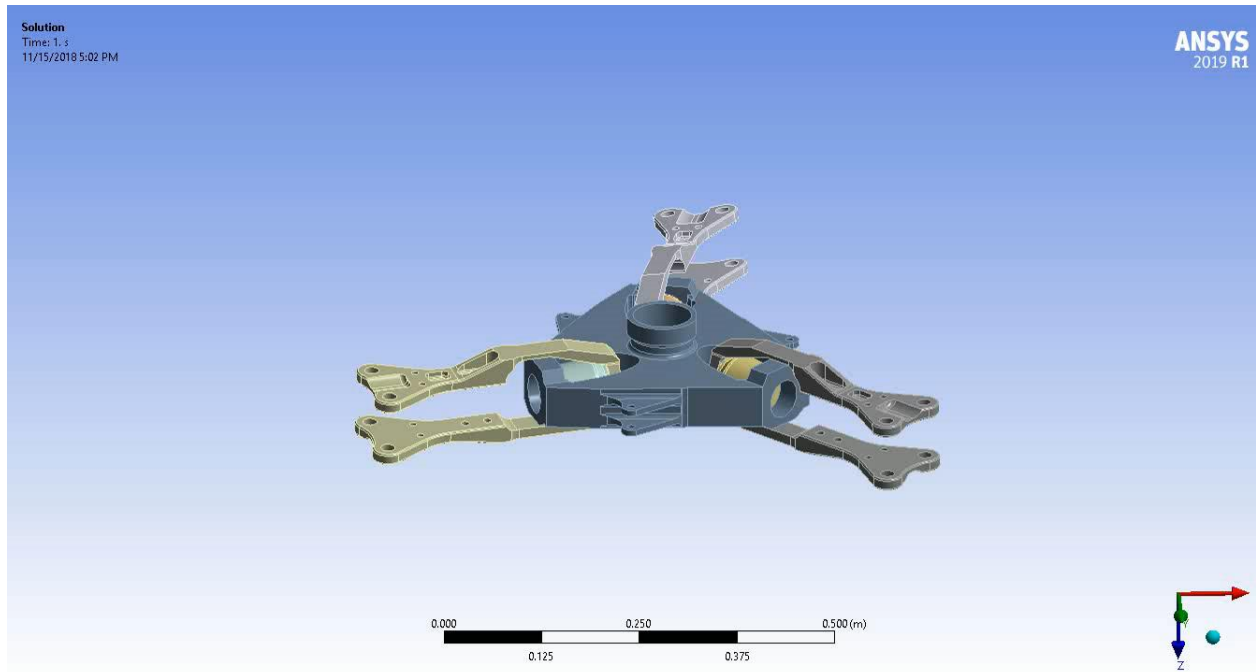
	A	B	C	D	E	F
1		<b>Environment</b>	Static Structural	Static Structural	Harmonic Response	Transient
2		<b>Time/Frequency</b>	1	End Time	85	2
3		<b>Phase Angle</b>			270	
4						
5	<b>Combination ...</b>	<b>Type</b>				
6	Combination 1	Linear	1	1	0	1
7	Combination 2	SRSS	0	0.5	0.5	0
8	Combination 3	Linear	-1	2	0	0.5

+ Add Combination + Add Base Case



# Keyframe Animation

- Keyframe animation enables you to string together different snapshots of the model in the Geometry window to create an animation
  - Keyframes are created by positioning the model in the desired orientation and clicking on Create Keyframe.
  - The application interpolates the transition from keyframe to keyframe to create a smooth animation.

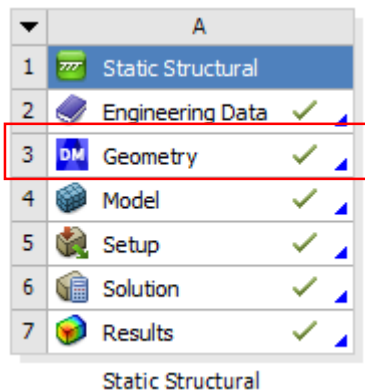


- Export the video in various formats: MP4, WMV , AVI and GIF



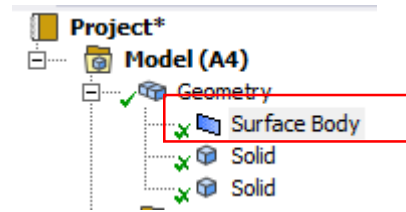
# Mixed-Dimension Analysis

- Mechanical now allows combining two-dimensional (2D) and three-dimensional (3D) bodies in the same model.
- For shell bodies (only) defined in the X-Y plane, the **Dimension** property on the **Body** object lets you set the body as either 3D (default) or 2D. Mechanical then uses this property to write the appropriate element type (3D or 2D) to the input file.



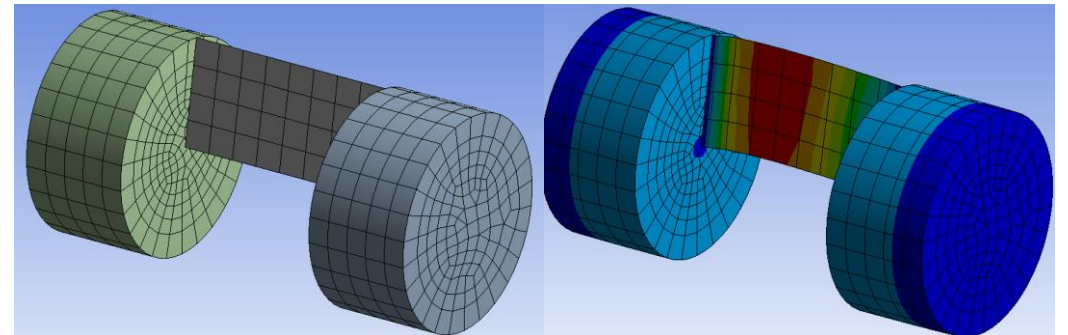
12	Advanced Geometry Options	
13	Analysis Type	3D
14	Compare Parts On Update	No

Analysis type (Geometry type) set to 3D in Workbench



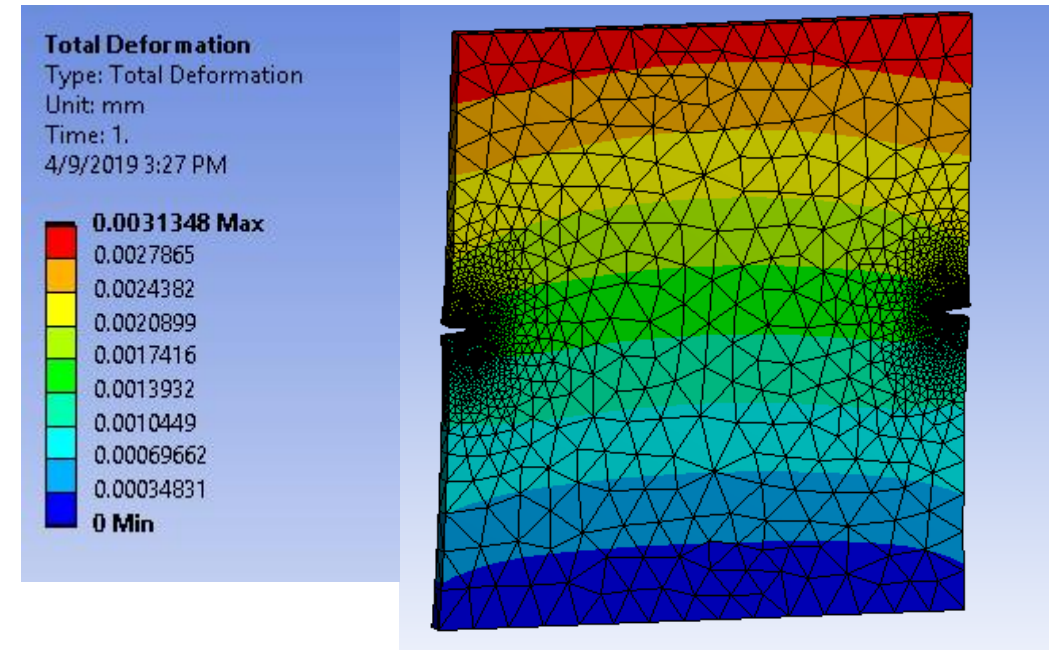
Details of "Surface Body"	
+ Graphics Properties	
- Definition	
<input type="checkbox"/> Suppressed	No
Dimension	2D
Stiffness Behavior	Flexible

Geometry Dimension set to 2D in Mechanical on Sheet Body



# SMART Crack growth features

- Support multiple Crack Growth using SMART methodology
- Evaluation of SIFS and J-integral results for all cracks in the model
- Newly supported options
  - Multiple Load Steps
  - Thermal load



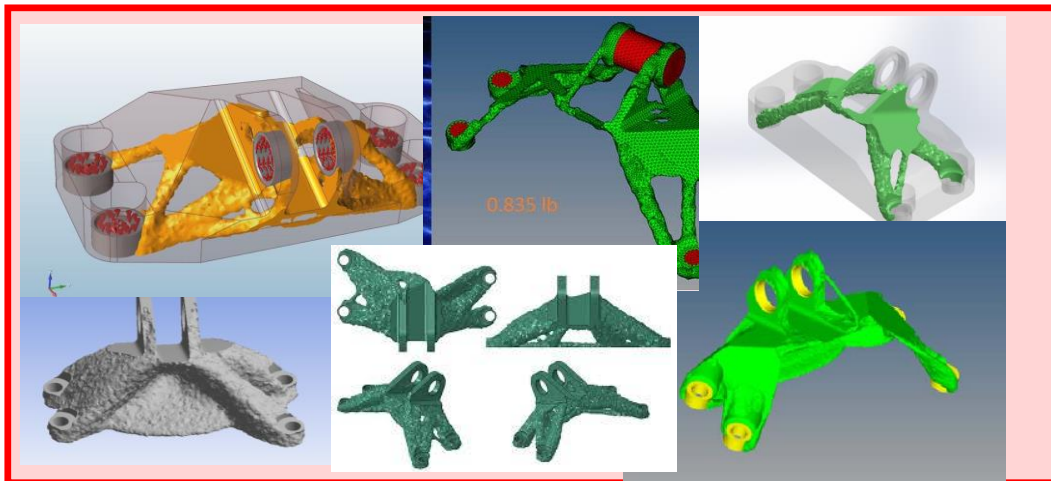
# Topology Optimization - Level-Set

- Density-based method suffer from lack of properly defined boundary
  - Forces the designer to interpretation of a poorly defined geometry
- Level-set-based method “pushes” the boundaries
  - Result is a water-tight smooth surface, ready for re-analysis, printing or re-CADing

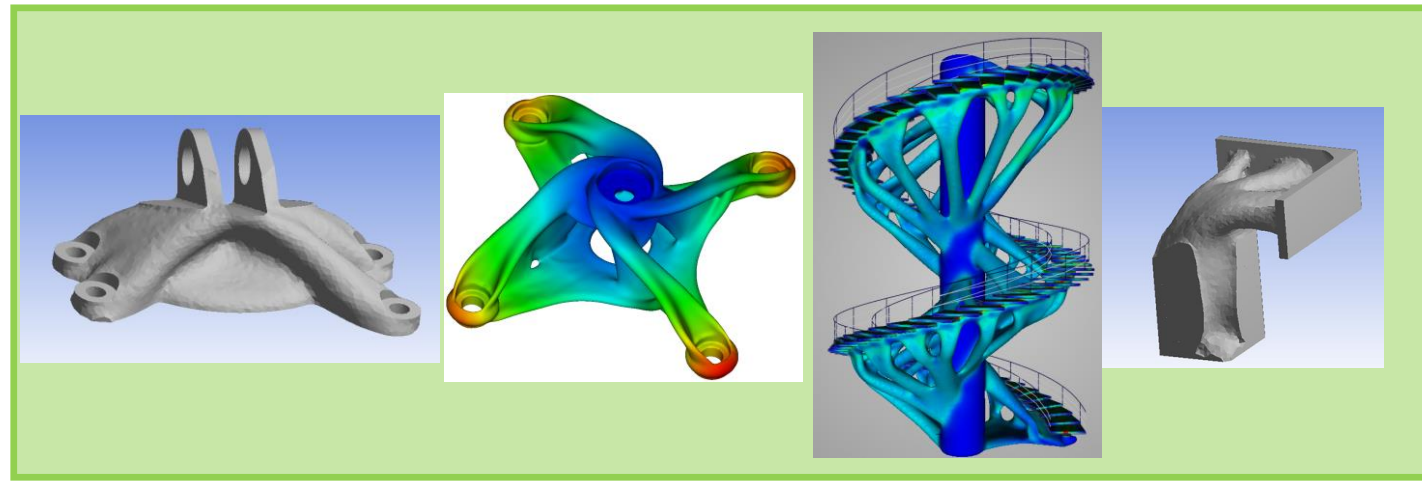
Tets & Hex  
Manufacturing constraints

- Pull out
- Max Thickness

Density-based optimization for various commercial products

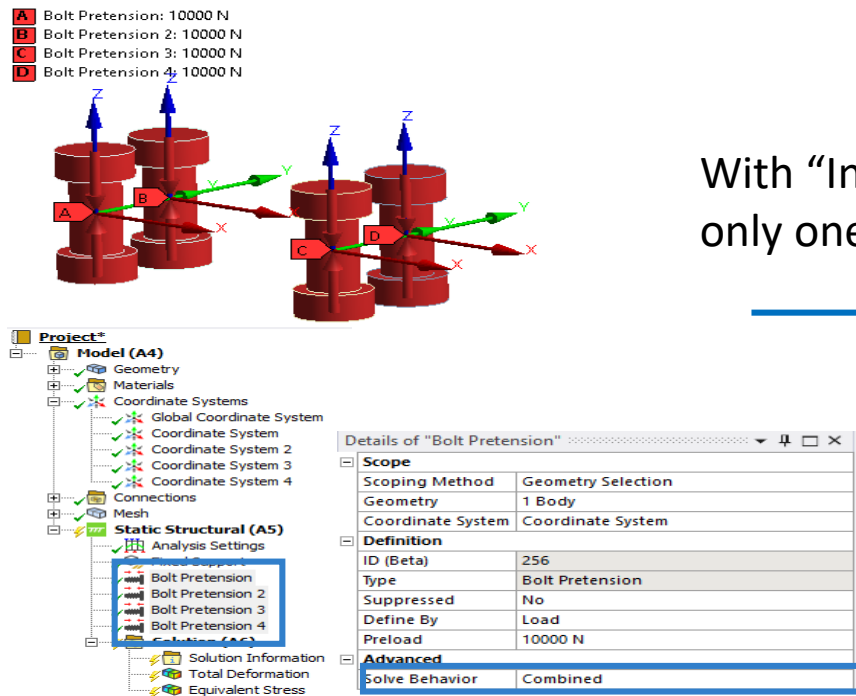


Level Set-based optimization in ANSYS Mechanical and Discovery Live

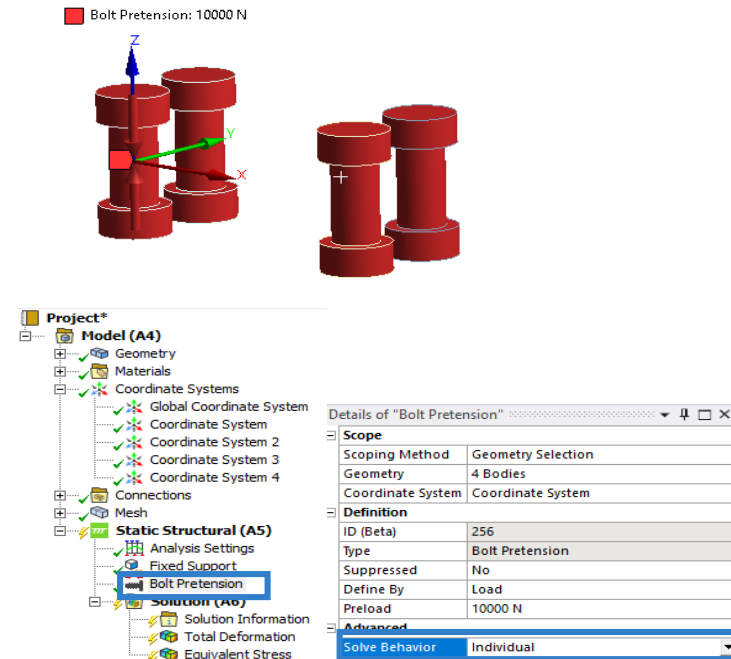


# Bolt Pretension Solve Behavior option

- Solve Behavior option of “Combined” (default) and “Individual” is exposed. With “Individual” option selected, the user can create one Bolt Pretension object for multiple Body selection instead of those many objects

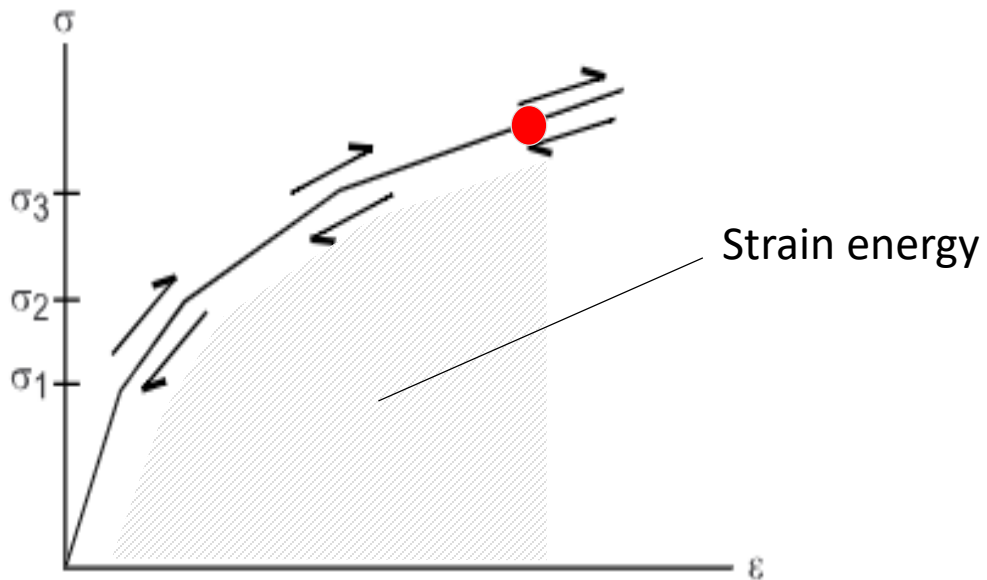


With “Individual” option,  
only one object is required



# Multilinear Elasticity

The program provides a capability to model multilinear elastic materials. This material causes unloading to occur along the same path as loading. The multilinear elastic material model is defined with the TB,MELAS command.



Elements support: PLANE182 (plane strain or axisymmetric), PLANE183 (plane strain or axisymmetric), SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, SOLID273, SOLID285, PIPE288, PIPE289.

# A New Enhanced Lagrange Multiplier Method (2019R2)

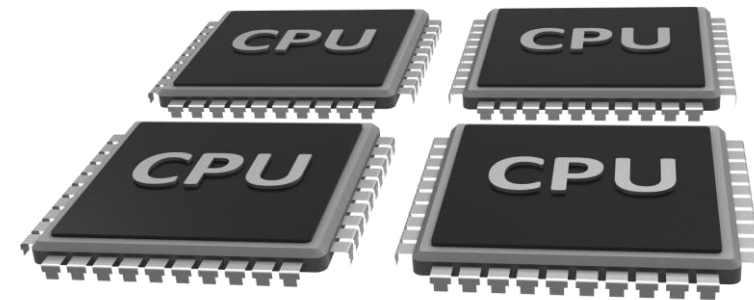
A new Lagrange multiplier (LM) logic is implemented which reduces numbers of the equation ordering and leads to great performance improvements and better scalability in DMP run.

## Overall Observations for 14 customer's Models:

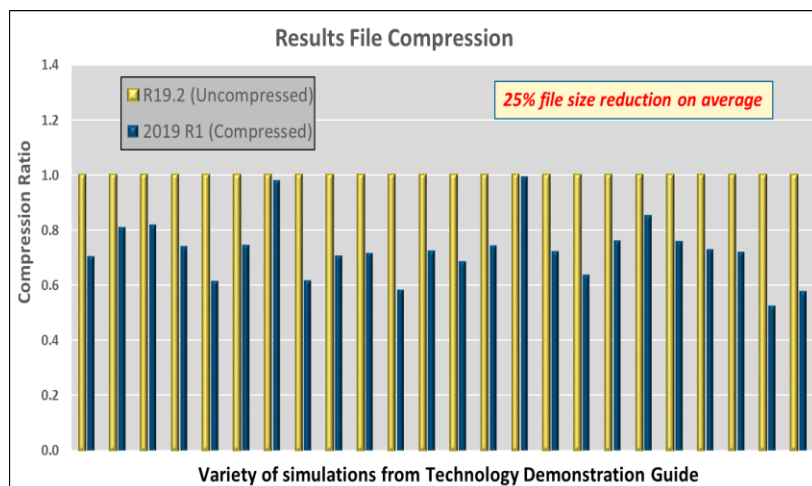
Model Detail (DMP 128 CPU cores)	Version	Solution Time ratio	RST File ratio
M1 : Baseline	2019R1	1.00	1.00
M2 : M1 + LM Logic + OUTRES + KEYO,CID,9,1	2019R2	0.70	0.82
M4 = M2 + /FCOMP,RST,SPARSE	2019R2	0.71	0.59

- LM Logic significantly helps reducing solution time
  - **Solution time ratio (relative) reduced by 42%**
- /FCOMP,RST,SPARSE significantly reduces RST file size without affecting solution time
  - **RST file ratio reduced by 70%**

## Distributed ANSYS – NOW Default

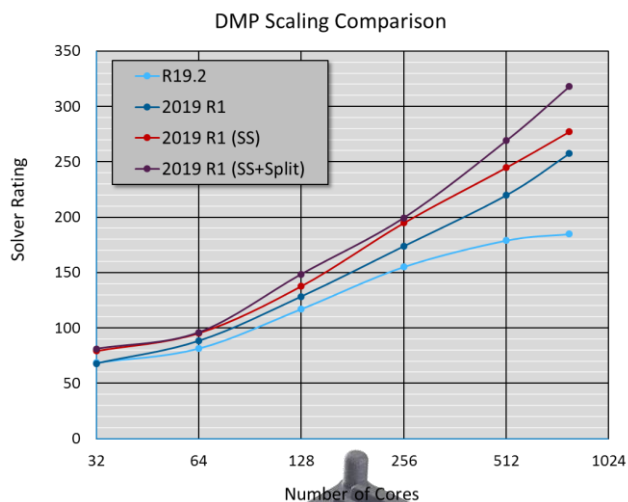


## RESULTS FILE COMPRESSION

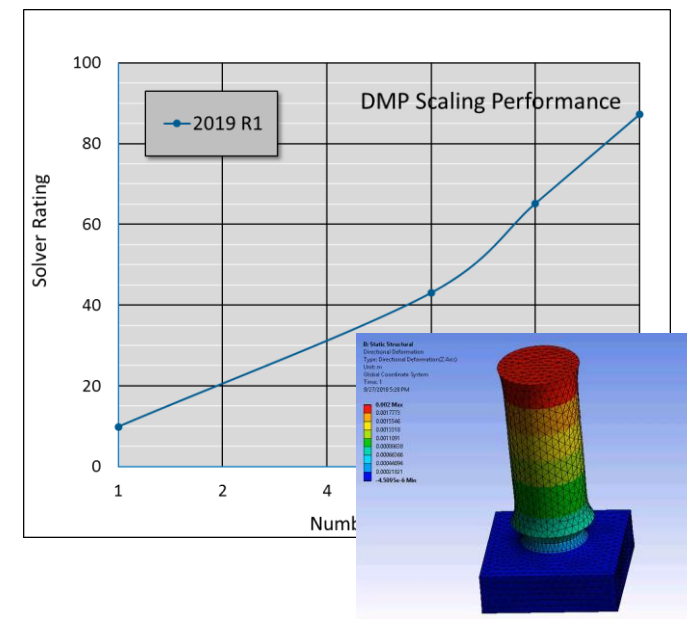


**25% reduction**

## DISTRIBUTED CONTACT

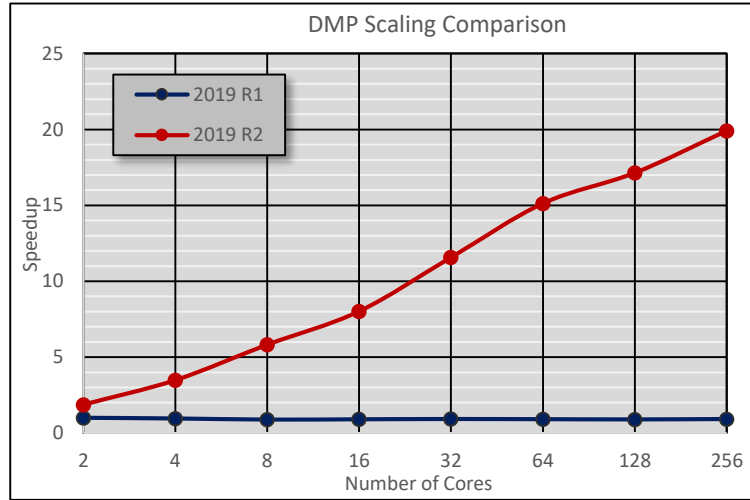


## DANSYS for SMART FRACTURE

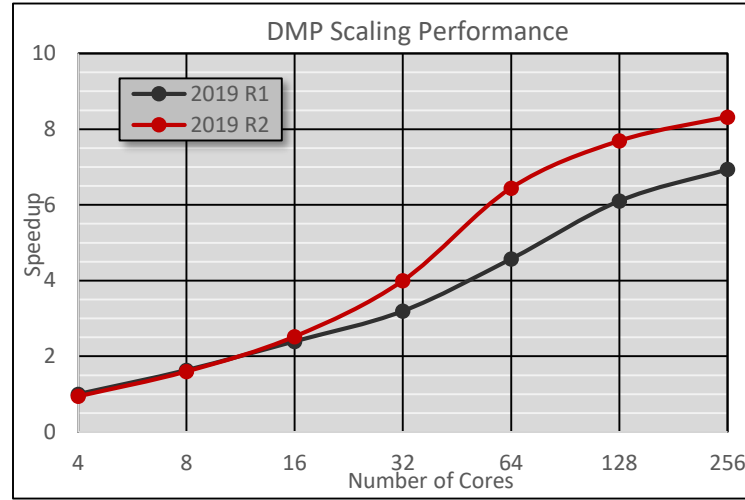




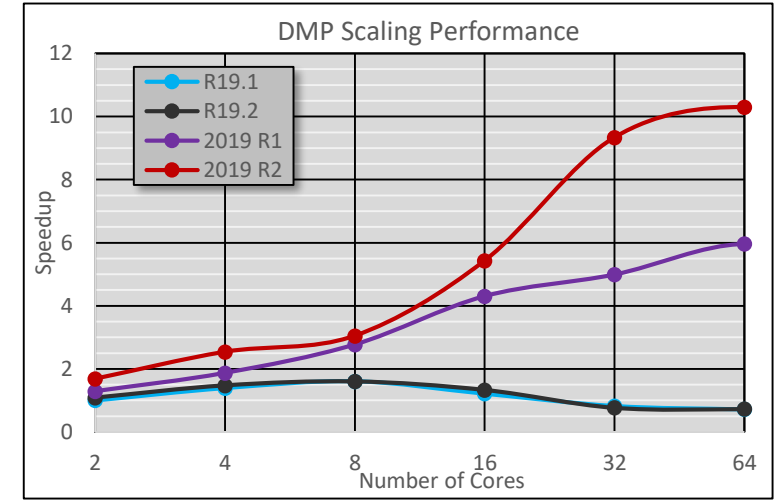
# MAPDL - Performance Improvements



Substructuring – **New!**



SMART



Normal Lagrange sliding contact

- **New compression algorithm (sparsify) on by default**

- Smaller results files (10-50% on average)
- Lossless compression of results file data
- Virtually no performance penalty

- Codes that directly read the MAPDL results file without using binlib.dll will no longer work!

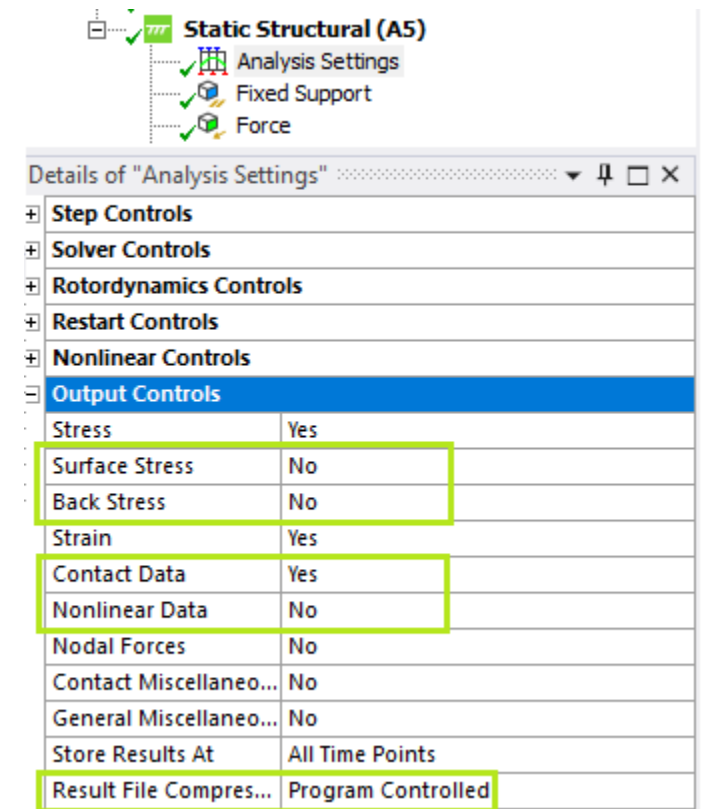


# Output controls and Result file compression

- Additional output controls added which provides granular controls on the outputs written to the result file. The defaults for these controls have been chosen carefully based on analysis and physics type.

- ❖ Surface Stress
- ❖ Back Stress
- ❖ Contact Data
- ❖ Nonlinear data
- ❖ Result File Compression
- ❖ Element Current Density
- ❖ Electromagnetic Nodal Force
- ❖ Heat Generation Rate
- ❖ Result File Compression

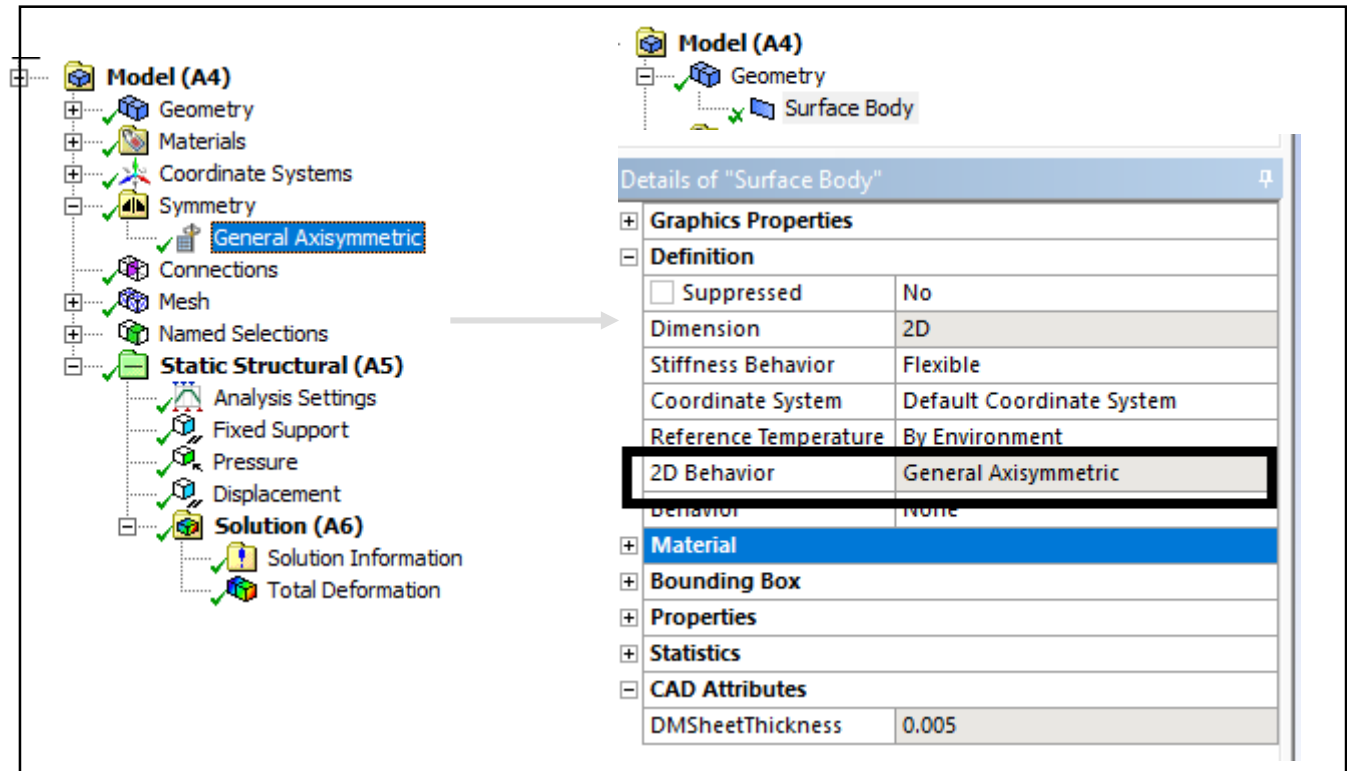
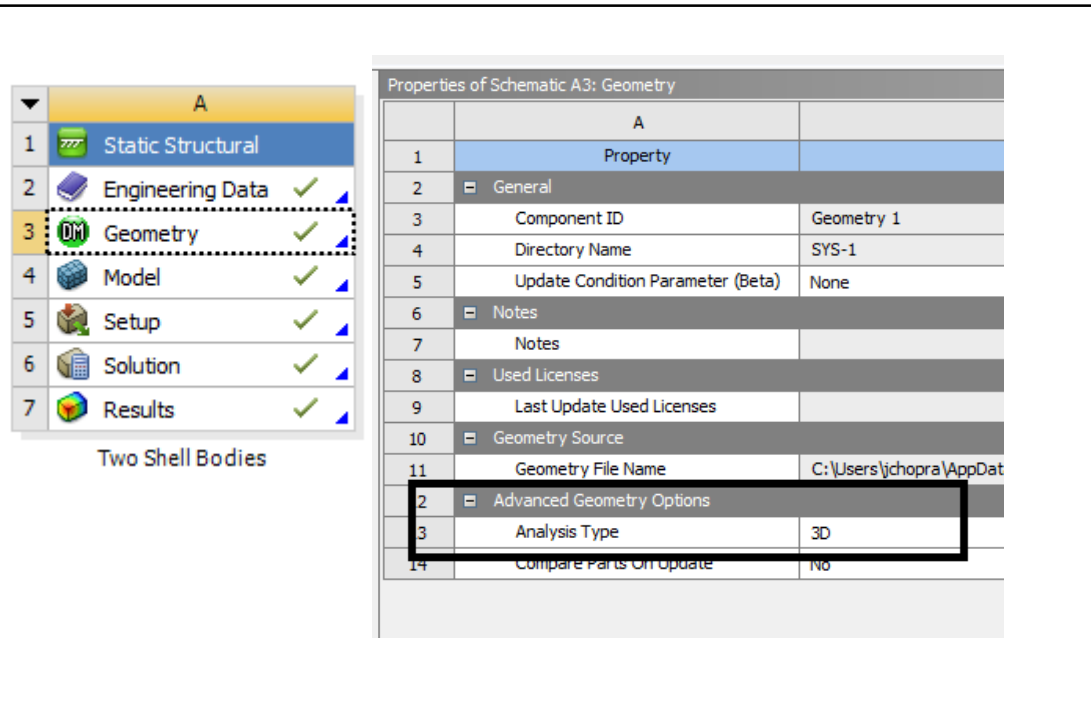
- The **Result File Compression** controls the compression level for the result file with the following options
  - Program Controlled
  - Sparse
  - Off



# General Axisymmetry

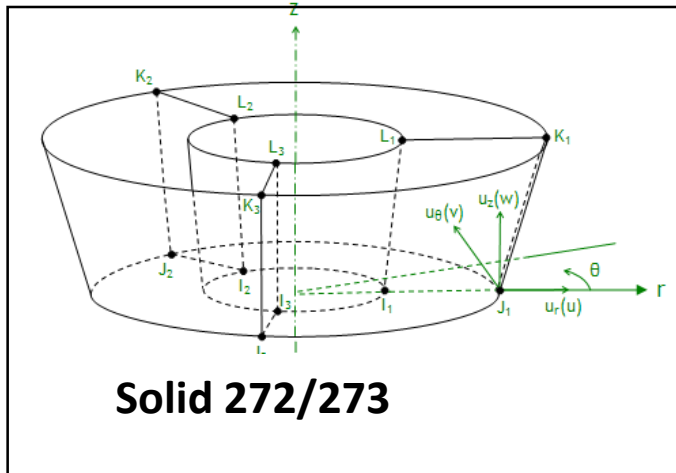
# General Axisymmetric in WB-Mechanical

- Applicable for 3D Static Structural analysis. General Axisymmetric definition is added under Symmetry folder and when scoped to 2D/Surface body makes the behavior of that Body as General Axisymmetric



# General Axisymmetric Definition

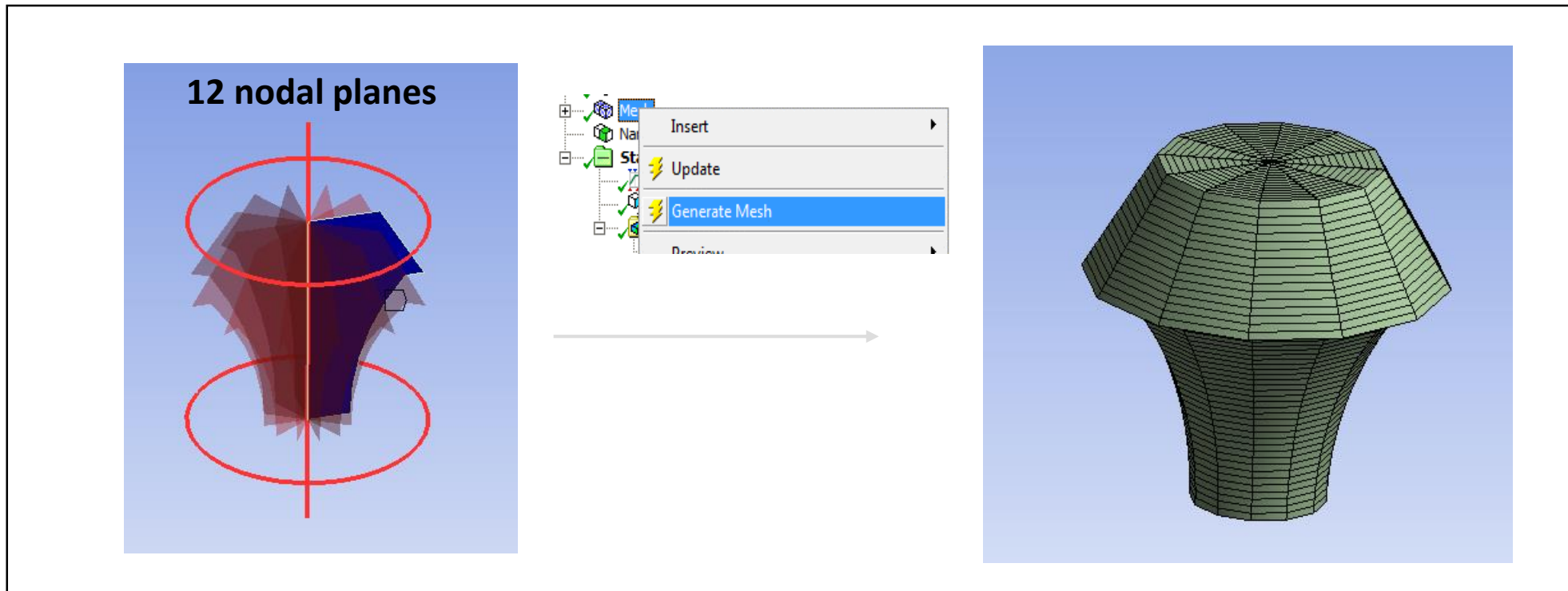
1. General Axisymmetric (Solid 272/273) introduces Fourier series into interpolation function to describe the change of displacements in the Circumferential ( $\theta$ ) direction.
2. General Axisymmetric object in mechanical takes Nodal planes input to define number of planes and Coordinate system with Axis input to specify the circumferential direction.
3. The graphics view shows the Axis and Orientation by drawing the line and circle. And it shows the number of nodal planes by showing the transformed geometry in different nodal planes



**Nodal planes value range from 1 to 12 excluding 2**  
**Axis definition must not cut through the body**

# General Axisymmetric Mesh

1. General Axisymmetric mesh is generated using Generate Mesh/Update action on Mesh folder
2. The base mesh is created on the surface body and then General Axisymmetric mesh is generated on all the nodal planes as post operation on base mesh.

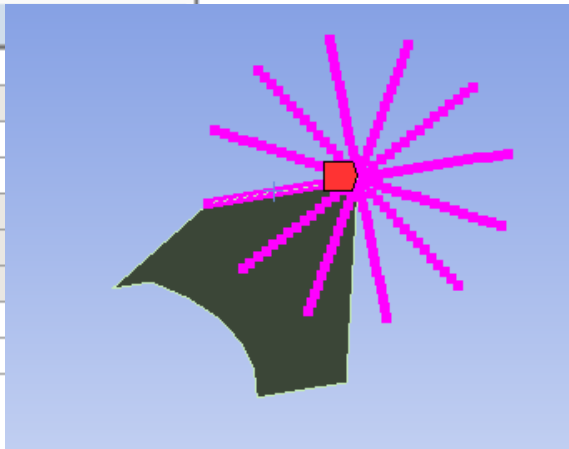


# Loads and Boundary Conditions

1. Nodal loads are directly applied through the Named selections scoped to Nodes. These loads can be non-axisymmetric loads as nodes can be picked from any nodal plane
2. Pressure, Remote force, Moment and Displacement load can be applied to the geometric scoping which can be edge or vertex. If edge of General Axisymmetric body is selected for load application, then load is applied to all the nodal planes. For Pressure load using Surface effect option, SURF159 is created to apply the load

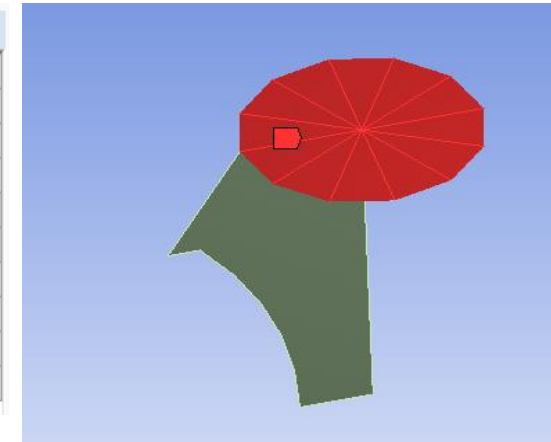
**Load applied to nodes**

Details of "Nodal Pressure"	
[-] Scope	
Scoping Method	Named Selection
Named Selection	Selection 2
[-] Definition	
ID (Beta)	232
Type	Pressure
Define By	Normal To
<input checked="" type="checkbox"/> Magnitude	-1.e+006 Pa (ramped)
Suppressed	No



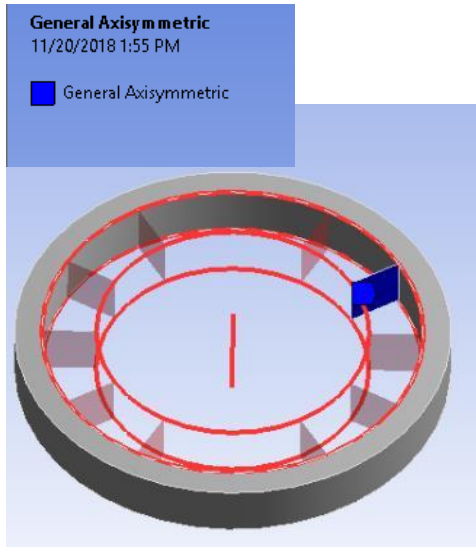
**Graphics shows the face selected in  $\theta$  direction where load is applied when Pressure is scoped to edge**

Details of "Pressure"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
[-] Definition	
ID (Beta)	212
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
<input type="checkbox"/> Magnitude	-1.e+006 Pa (ramped)
Suppressed	No



# General Axisymmetric behavior with Contacts

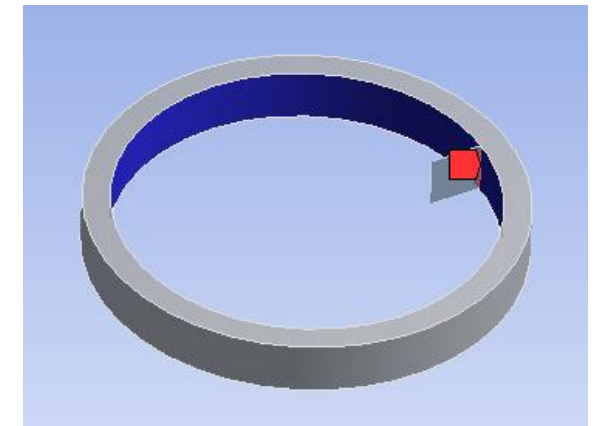
1. Node to surface contact is created between General Axisymmetric body in contact with other bodies. CONTA175 is created for the contact side which will be General Axisymmetric body and TARGET170 element is created for 3D target surface
2. Only Bonded contact is supported when Nodal plane 1 is defined
3. Number of nodal plane should be same when General Axisymmetric body is in contact with other General Axisymmetric body



Edge to Face contact  
between General  
Axisymmetric body and  
Solid body

Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	1 Face
Contact Bodies	Inner_Ring_2D
Target Bodies	Outer_Ring
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Target Geometry Correction	None

Nodes in all nodal planes for the General Axisymmetric scoped edge is considered for contact

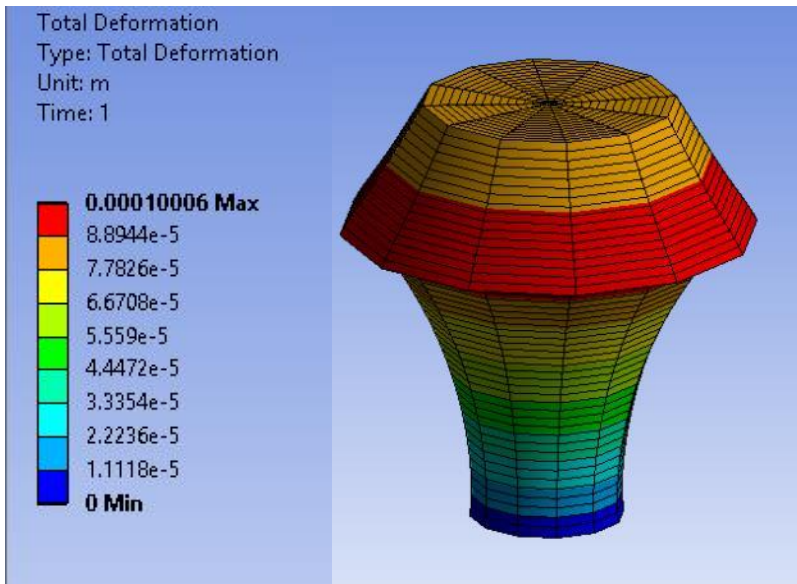




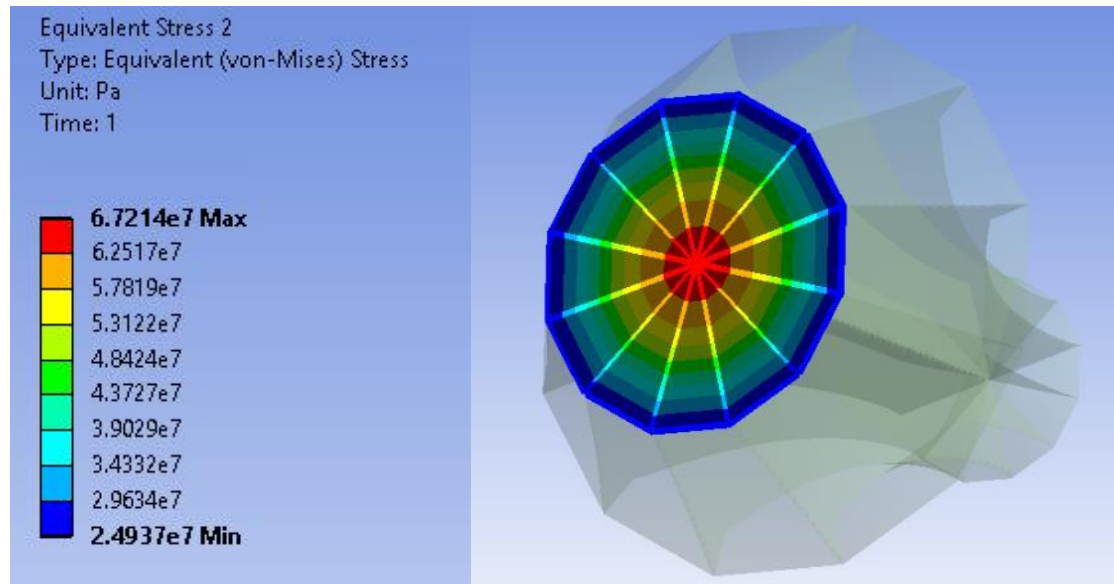
# Results

1. All regular results can be extracted on the General Axisymmetric body scoping or Mesh. The results shown below are Deformation and Stress which are symmetric in the circumferential direction in the presence of Axisymmetric loading applied in this case

## Total deformation scoped to All Bodies



## Stress shown in all nodal planes when scoped to an edge

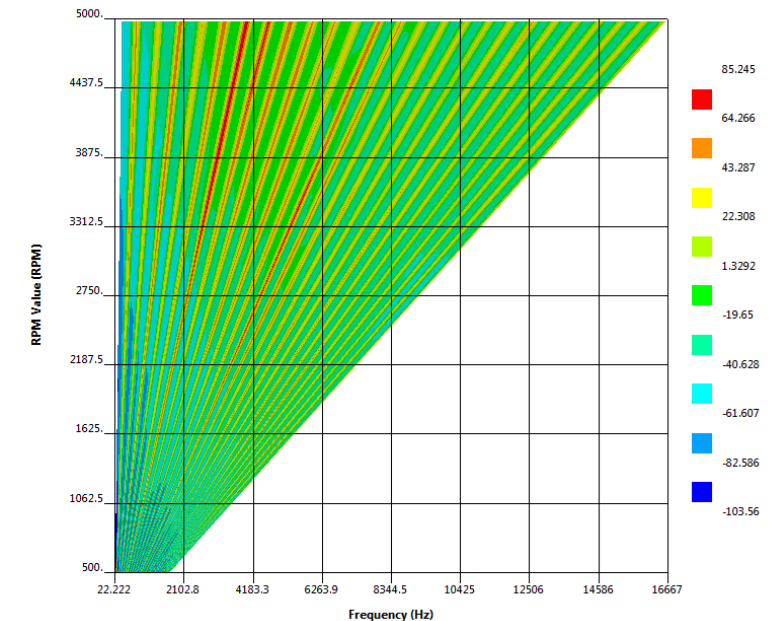
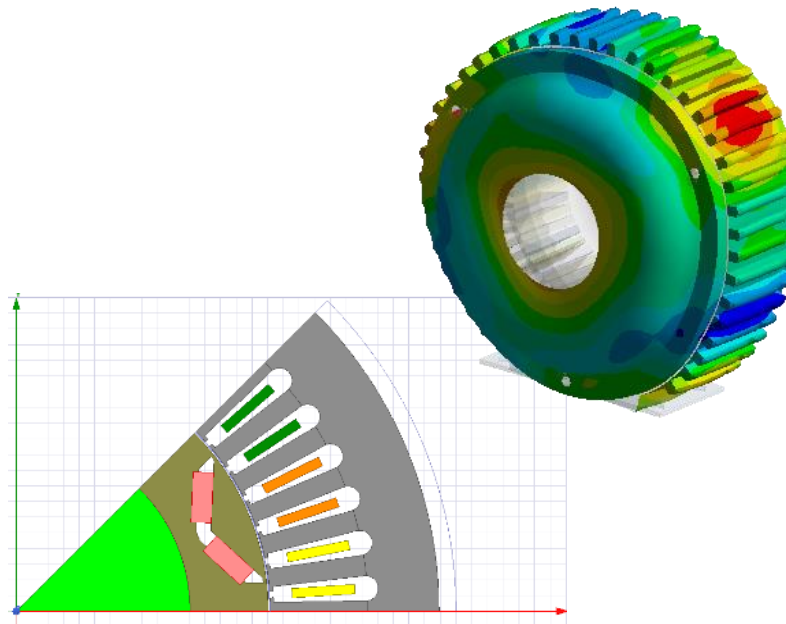
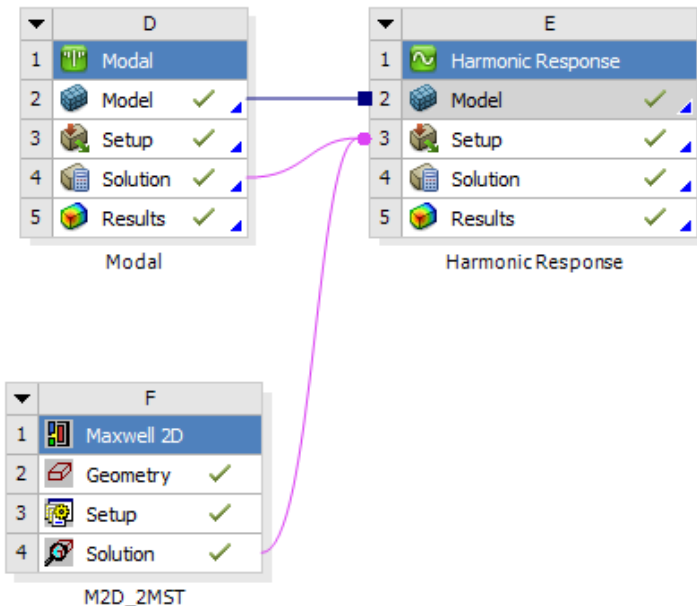




# Acoustics & NVH

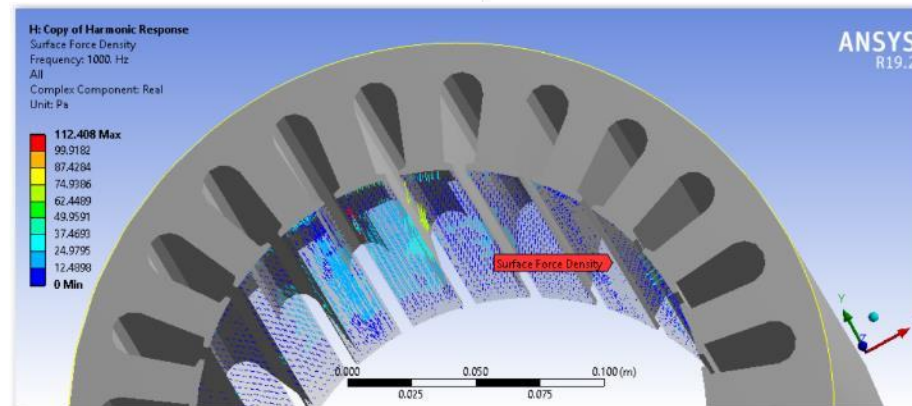
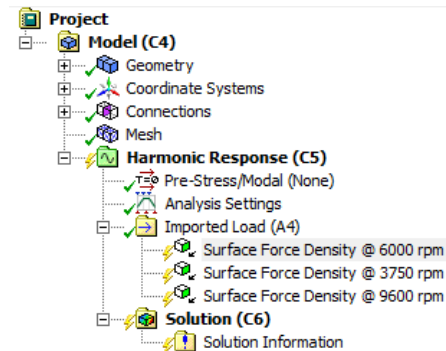
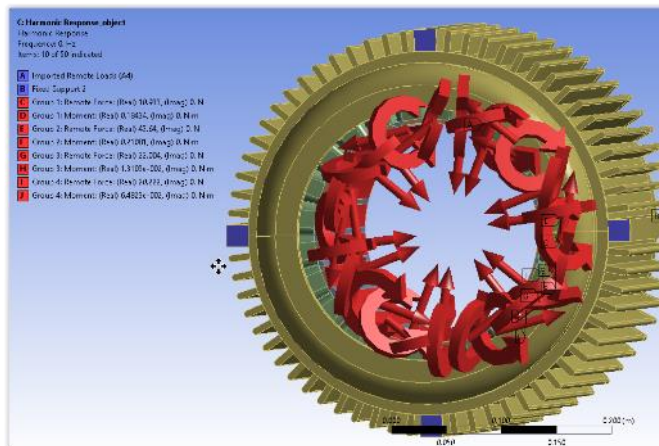
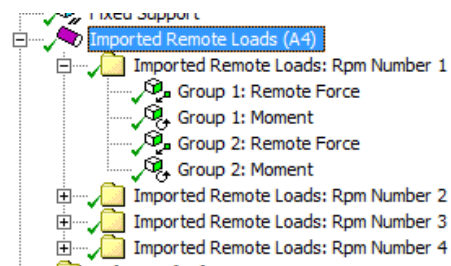
# Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

- For Electric Machine design, it is important to analyse the acoustic signature of the system. In that goal, Equivalent Radiated Power can be calculated for a range of Rotational Velocities and Frequencies. We have develop a fully automated workflow to reach this goal.
  - ✓ Perform DX Parametric study in Maxwell
  - ✓ Transfer EMAG forces to Mechanical for all RPMs
  - ✓ Plot ERP Waterfall Diagram



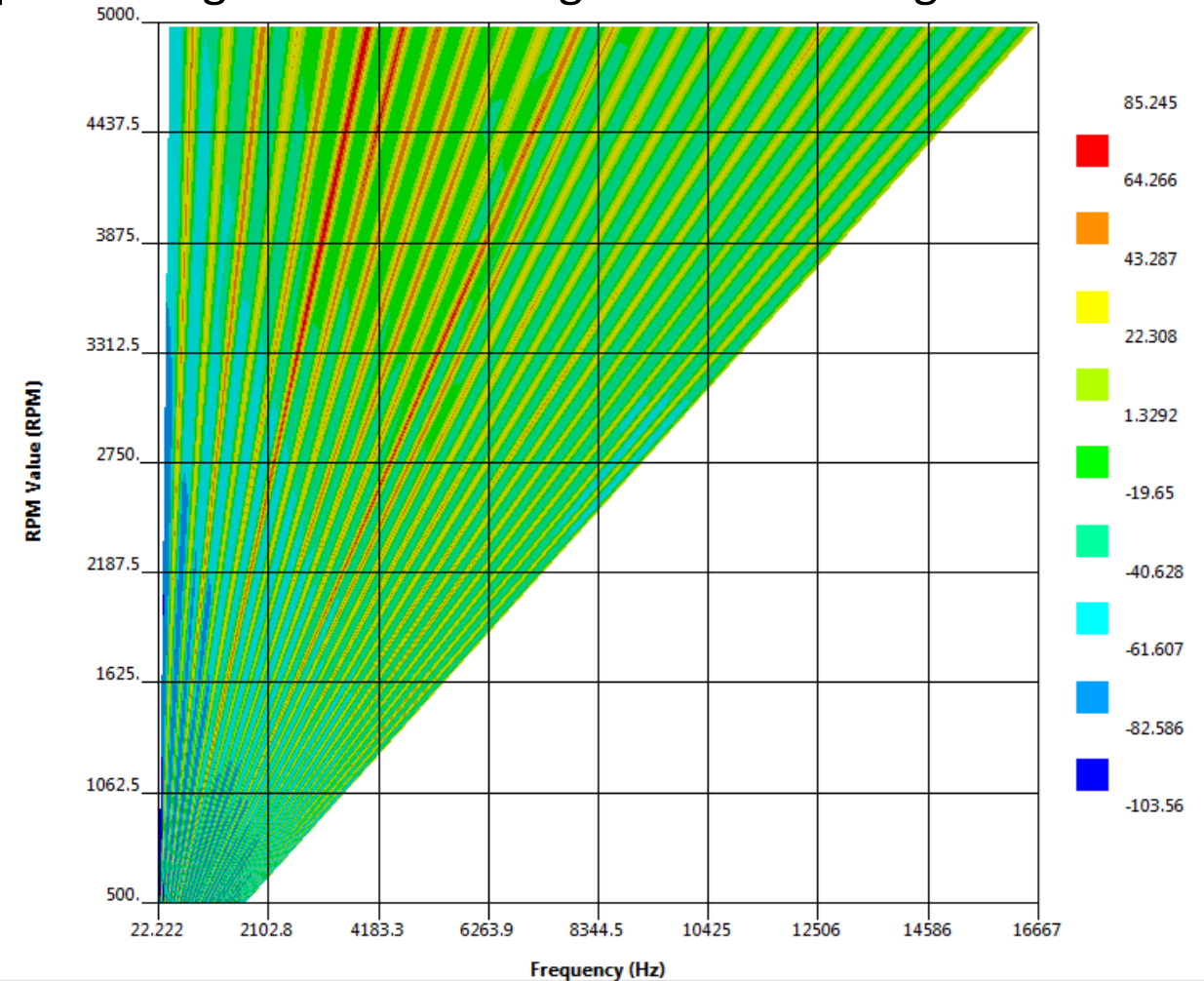
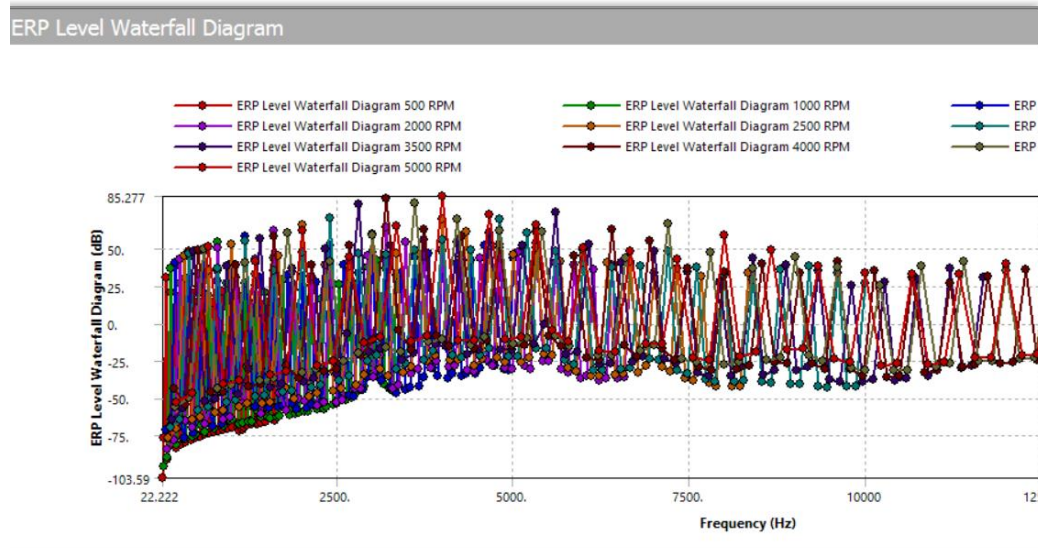
# Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

- Two mapping strategies available depending on the geometry compliance:
  - ✓ Object Based: Integrated Forces / Moments
  - ✓ Mesh Based: Surface Force Densities



# Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

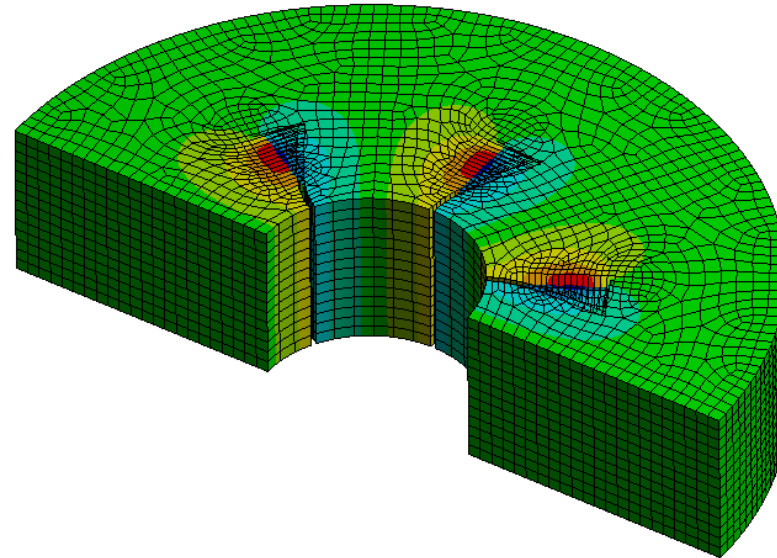
- Equivalent Radiated Power waterfall diagram results provide a global acoustic signature for a range of RPMs and frequencies.



# Cyclic expansion option for FSI cyclic symmetry

- Cyclic Expansion options for available in Acoustics:
  - ✓ Number of sectors to display
  - ✓ Starting sector

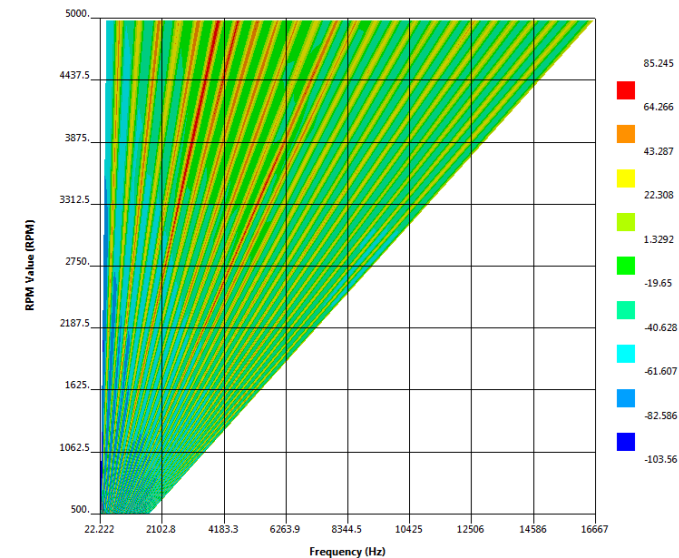
Details of "Solution (B6)"	
+	<b>Solution</b>
+	<b>Adaptive Mesh Refinement</b>
+	<b>Information</b>
-	<b>Cyclic Solution Display</b>
	Number of Sectors 3.
	Starting at Sector 1.
	Expansion Method (Beta) New



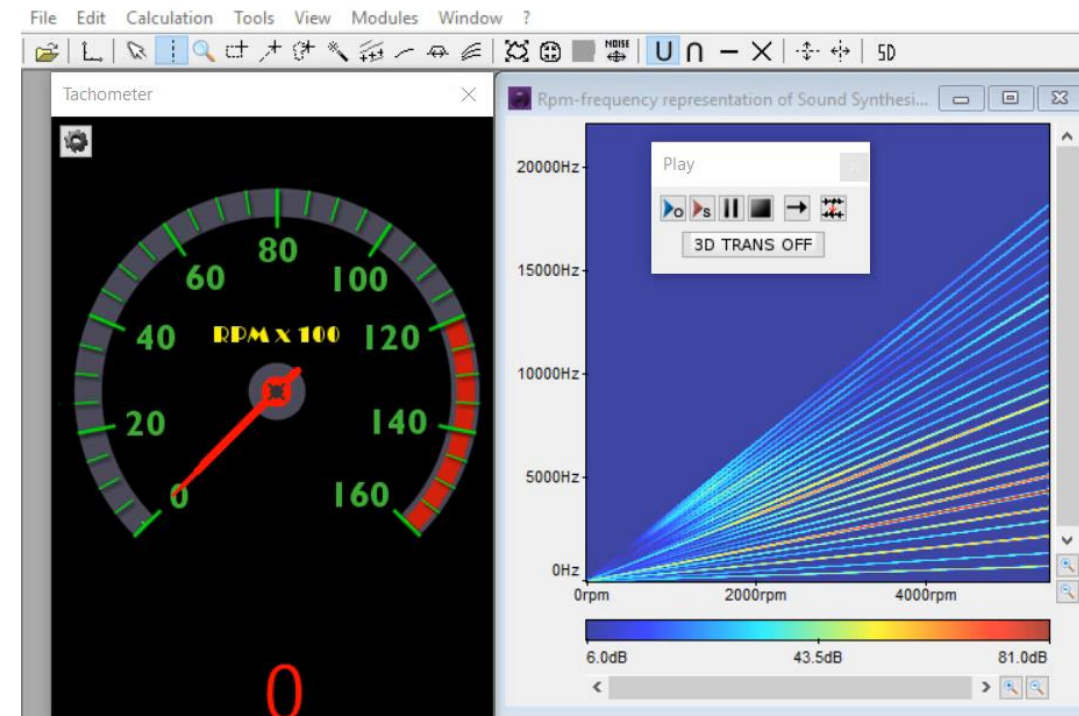
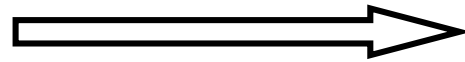


# Export Results to XML format for VRXP Sound Dimension (Beta)

- Allows XML export for Equivalent Radiated Power and Sound Power Level. XML file can be read in Optis VRXP Sound Dimension to synthesize the sound and create a .WAV file.



Listen to **computed** ERP



# Option to ignore damping in modal Acoustics (Beta)

- Available option to ignore acoustic damping material properties (viscosity...) which are defined by default for Air and Water materials. That allows to avoid using Damped solver without deleting those properties in Engineering Data.

Details of "Analysis Settings"	
[-] Options	
Max Modes to Find	6
Limit Search to Range	Yes
Range Minimum	1.e-002 Hz
Range Maximum	1.e+006 Hz
[-] Solver Controls	
Damped	No
Solver Type	Program Controlled
[+] Output Controls	
[-] Damping Controls	
Ignore Acoustic Damping (Beta)	Yes

# ANSYS Cloud



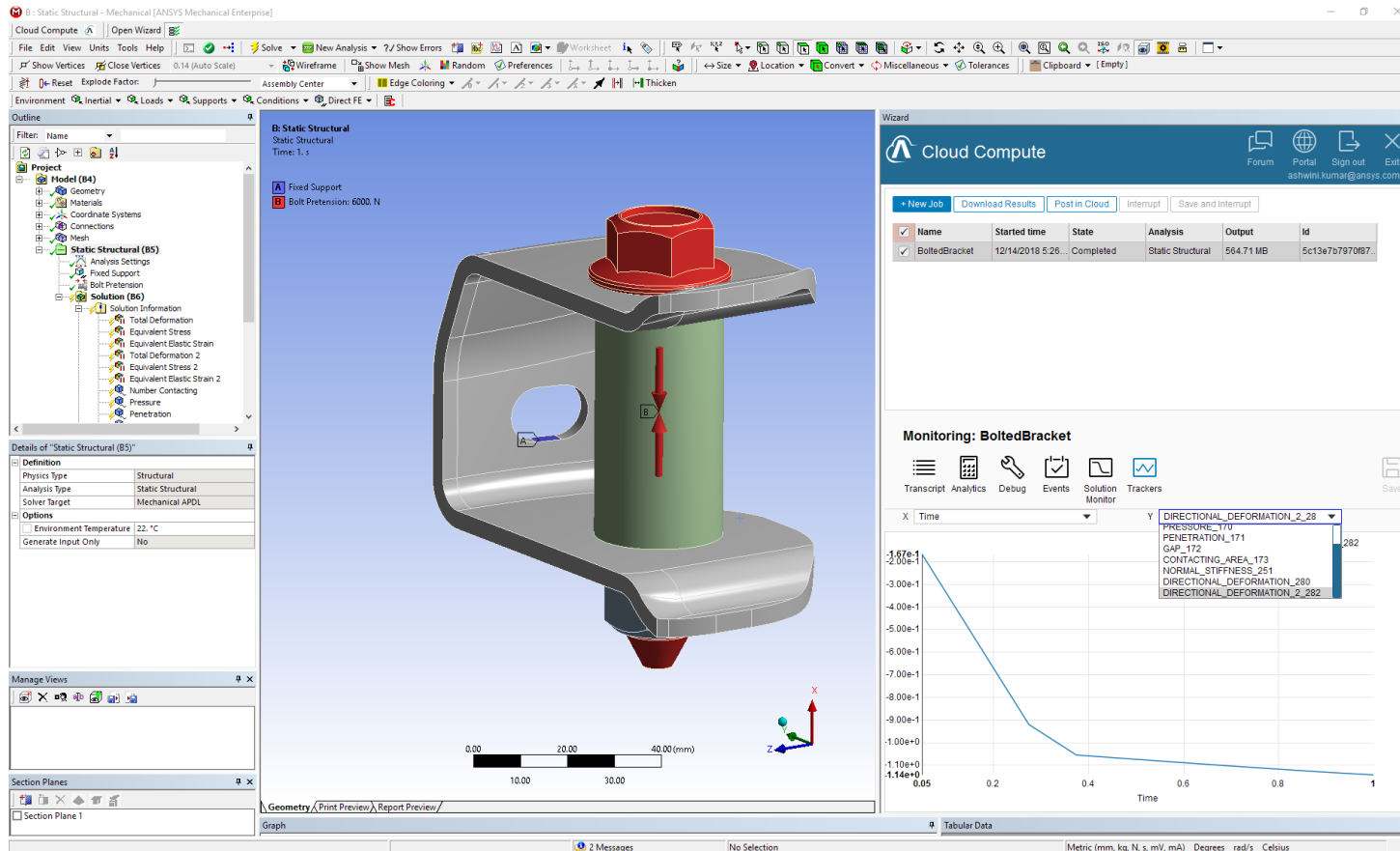
# ANSYS Cloud



**Cloud-based service that provides easy access to on-demand HPC directly from ANSYS Mechanical & Fluent solver**



# ANSYS Cloud Compute enables “Solve on Cloud”



Pre



Preprocessing on desktop – geometry prep, meshing, physics, boundary conditions

Solve



Files uploaded to cloud for solve, including automated optimization and DOE

Check



Validate results with remote 3D viewer. Leave heavy weight results in cloud

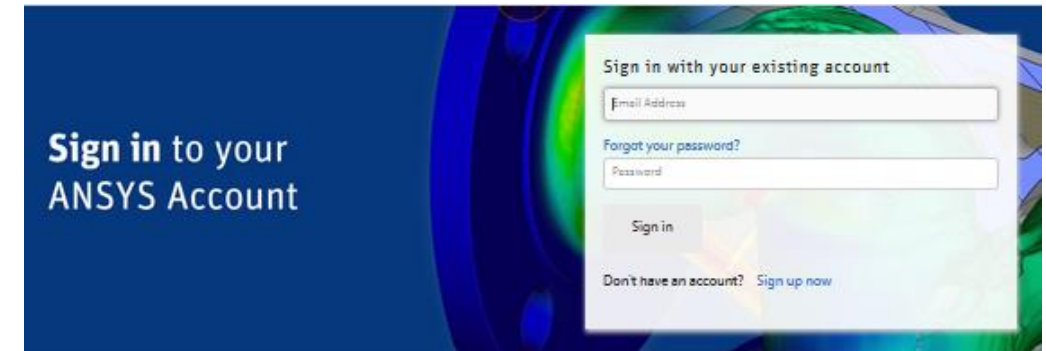
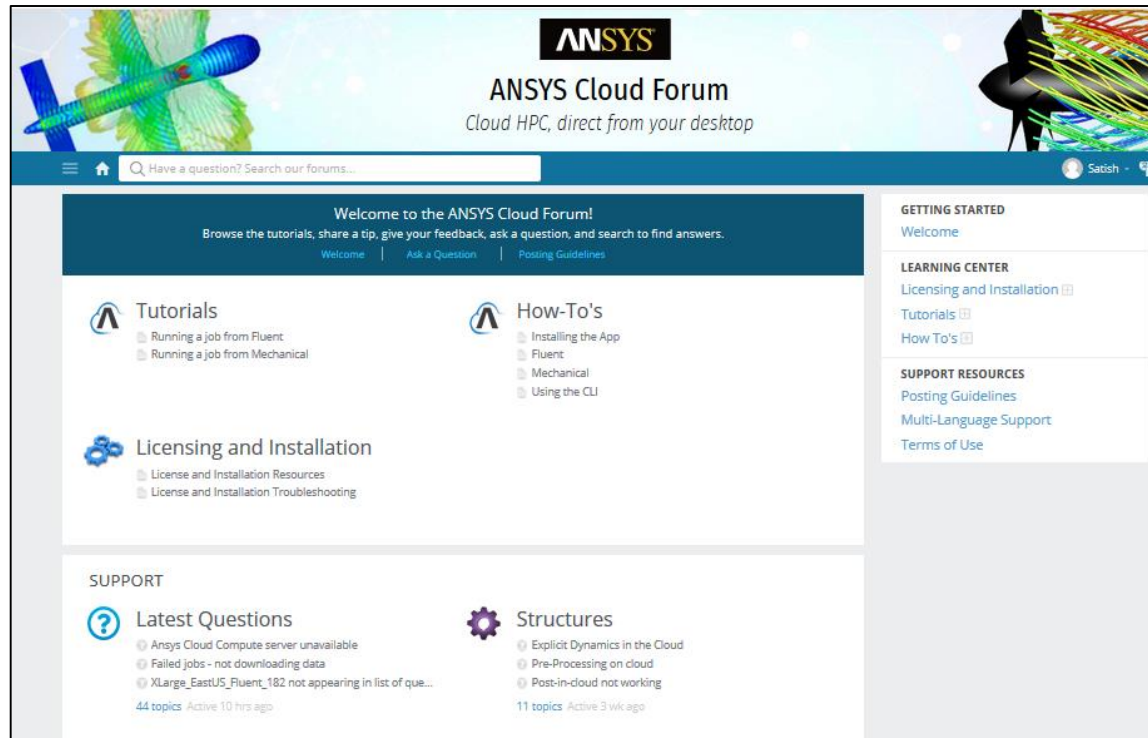
Post



Download results to desktop for detailed post-processing and data management

# Accessing ANSYS Cloud

- Register for ANSYS Account
- Forum to get help and provide feedback
- Download ANSYS Cloud Compute ACT App from ANSYS Cloud Portal



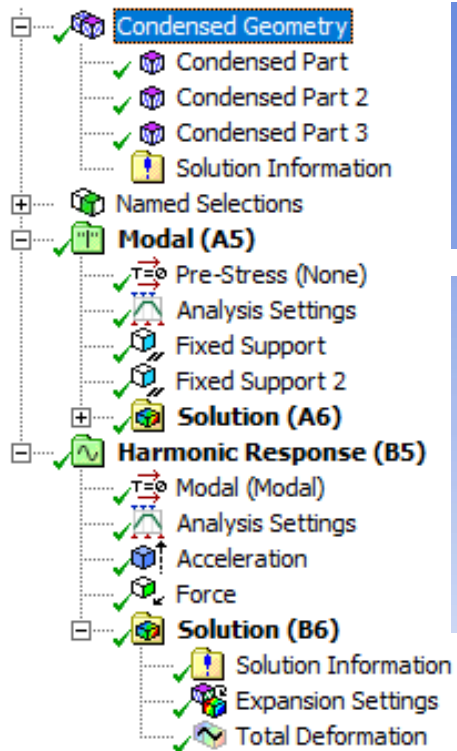
# Component Mode Synthesis aka Super Element

# Outline

- CMS exposure for MSUP harmonic response analysis
- Bushing formulation exposure for Bushing joint which can be internal to Condensed Part
- Support of Acceleration loads in MSUP harmonic using Substructure restart procedure
- Improve disk space and performance by performing file reference for use pass and expansion pass instead of file copy

# CMS exposure for MSUP harmonic analysis

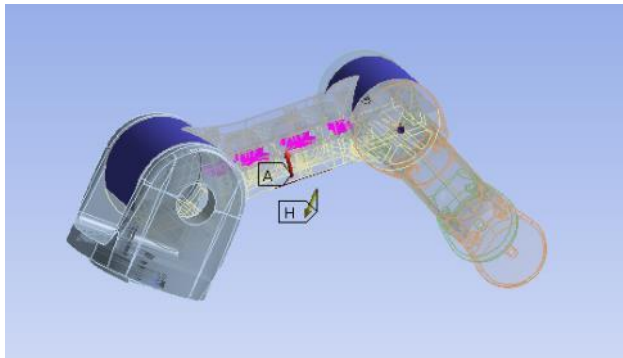
- Standalone and Linked MSUP Harmonic analysis can now use CMS based matrix reduction method to work with Substructures.



## B: Harmonic Response

Harmonic Response  
Frequency: 0. Hz

**A** Force: (Real) 1.e+005, (Imag) 0. N  
**H** Acceleration: 100. m/s<sup>2</sup>



Graphics are made transparent for condensed part and MDOF is displayed on condensed parts

Expansion Settings Worksheet

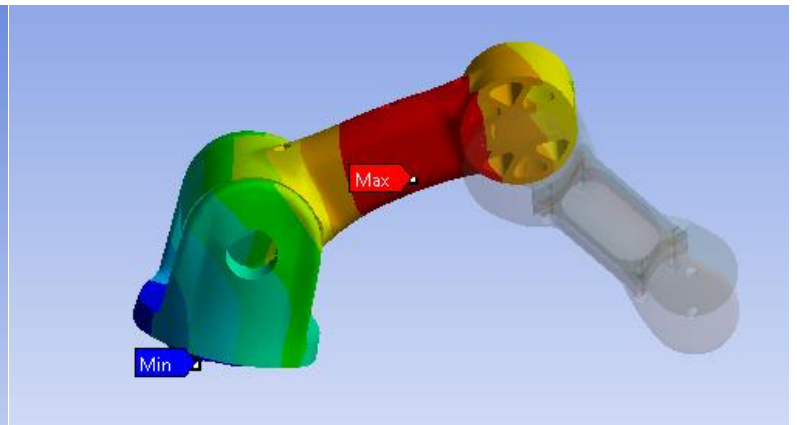
## Expansion Settings

	Condensed Part	<input type="checkbox"/> All Results	<input type="checkbox"/> Displacement
✓	Condensed Part 2	<input type="checkbox"/>	<input checked="" type="checkbox"/>
✓	Condensed Part 3	<input type="checkbox"/>	<input type="checkbox"/>

Expansion enabled for Condensed Part 2 and seen in the deformation results

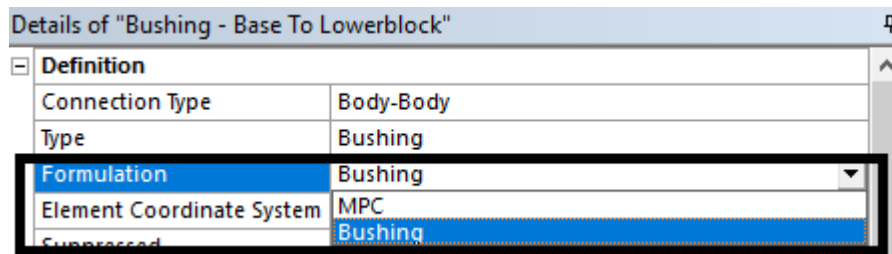
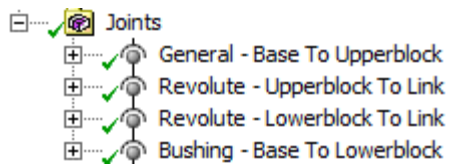
B: Harmonic Response  
Total Deformation  
Type: Total Deformation  
Frequency: 2000. Hz  
Sweeping Phase: 0. °  
Unit: m

0.00043356 Max  
0.00038539  
0.00033721  
0.00028904  
0.00024087  
0.00019269  
0.00014452  
9.6347e-5  
4.8173e-5  
0 Min



# Bushing formulation for CMS based Modal + Harmonic

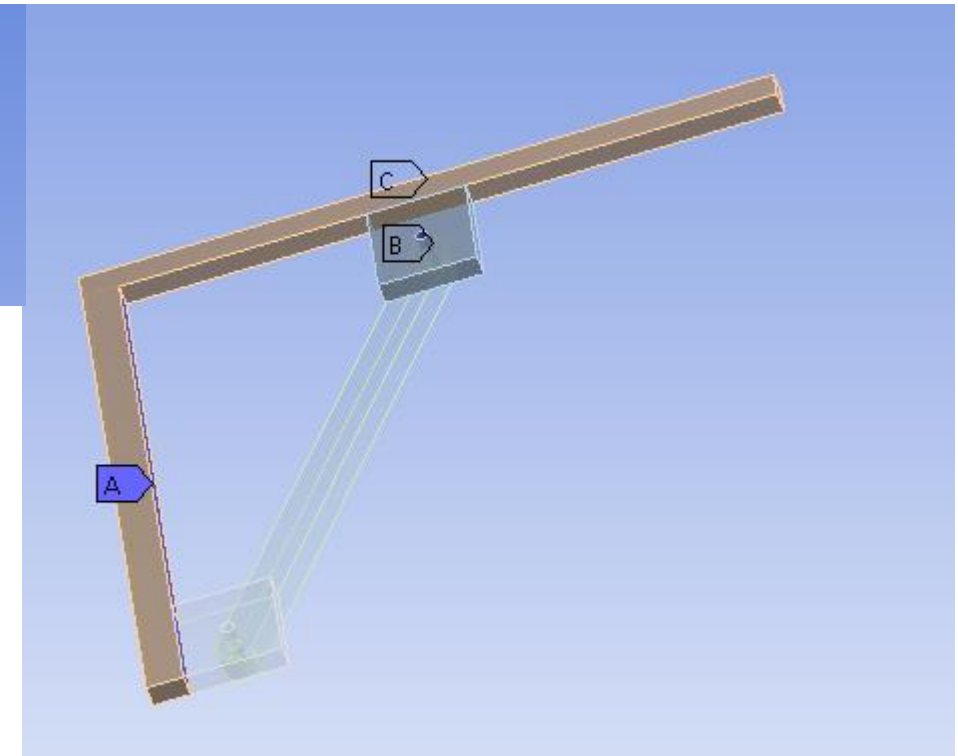
- Bushing joint now supports Bushing formulation which can be internal to Condensed Part and can be included in Generation Pass for Modal and Harmonic analysis



## Condensed Part

- A** General - Base To Upperblock
- B** Revolute - Lowerblock To Link
- C** Fixed Support

**Bushing joint is internal to condensed part and all other joints are in the interface**





# Acceleration load support for Harmonic analysis

- CMS based MSUP Harmonic analysis supports Acceleration load. Since Acceleration is scoped to All Bodies in the analysis, it needs to be applied to both Condensed Part which are within substructures as well as Non-Condensed parts. To support the Acceleration load, substructure restart procedure is performed in Harmonic analysis during use pass.

```
/com,***** Performing Substructuring Re-start *****
/gopr
/clear
/FILNAME,cp133          ! Jobname = cp133
resume,cp133,db
/solu
antype,subs,restart      ! restarting the substructuring analysis
/com,***** Create Acceleration *****
acel,1,0,0               ! generate reduced LV1
solve
acel,0,1,0               ! generate reduced LV2
solve
fini
/clear
```



# Improve disk space and performance

- MODDIR is activated in the Super-element Generation pass of MAPDL solution. This enables the Use pass and Expansion pass to do file reference, when required. This avoids file copy of the LN22 files and other files shown below to improves disk space and performance due to file copy

```
cp128.rst  
cp128.LN22/LN09/LN20  
cp128.full  
cp128.cms  
cp128.emat  
cp128.bcs
```

**These files are read remotely  
through file reference**

```
cp128.sub (c)  
cp128.bclv (c)  
cp128.seld/mlv (c)  
cp128.db (c)  
cp128.esav (c)
```

**These files are copied from  
condensed part folder during  
use pass and expansion pass**

# Distributed ANSYS Enhancements

- **New features**

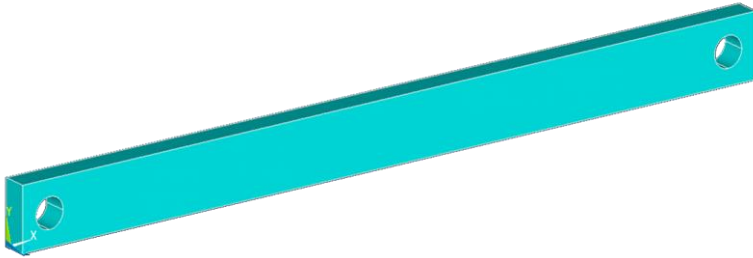
- Support for substructuring expansion passes

- **Improved scaling**

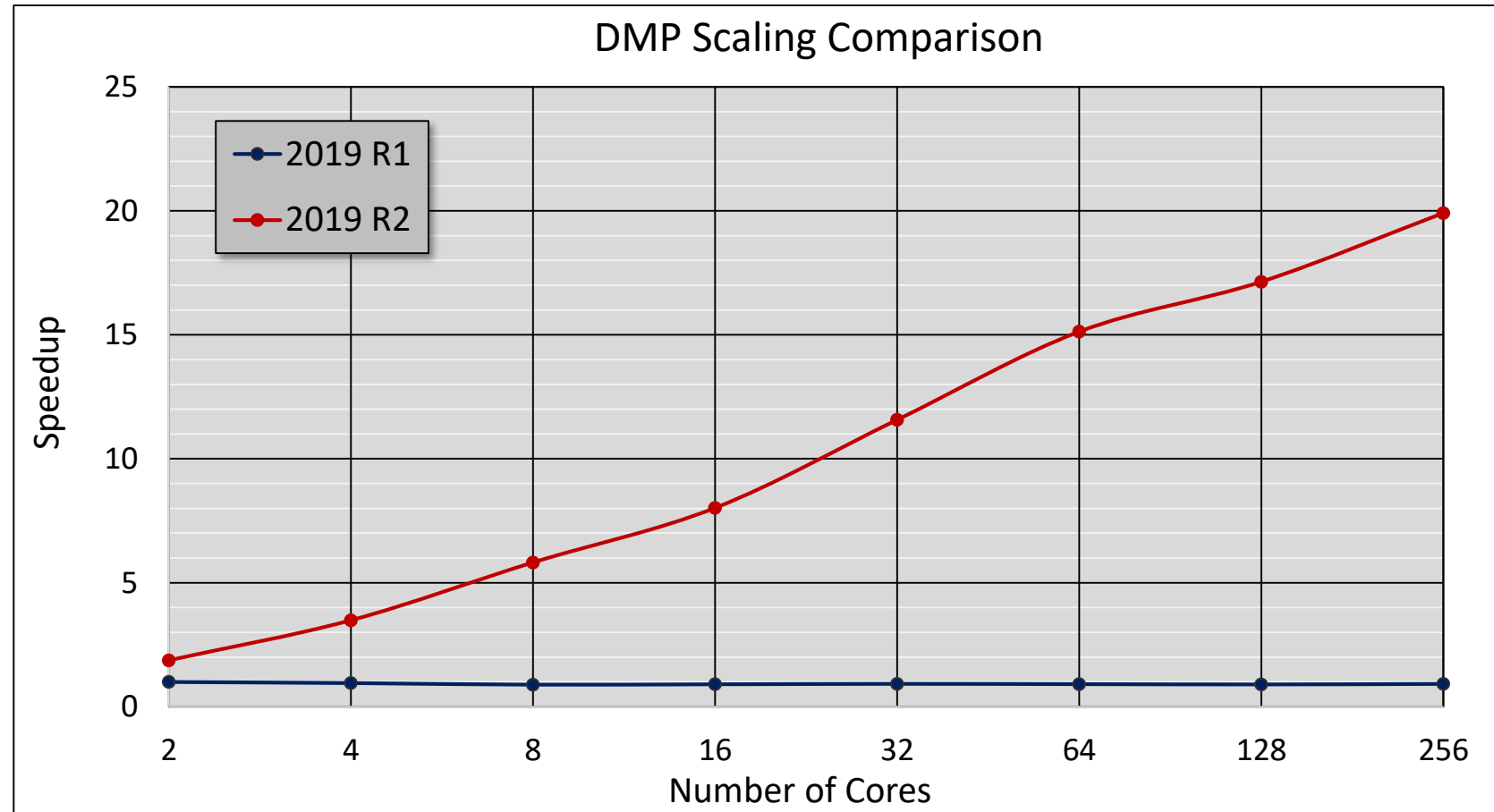
- Significantly improved scaling for SMART fracture simulations
- Improved scaling for models involving contact elements which use the Normal Lagrange algorithm ( $\text{KEYOPT}(2) = 3$ ) while enabling small sliding behavior ( $\text{KEYOPT}(18) = 1$ )
- Upgraded to Intel MPI 2018
  - Faster performance at higher core counts, particularly on Intel Omnipath interconnect

# Distributed ANSYS Enhancements

- Substructure expansion passes now supported



- 1.7 million DOF; sparse solver
- 3D solid elements
- 12 master DOF
- Linux cluster; each compute node contains 2 Intel Xeon E5-2695v3 processors, 256GB RAM, SSD, CentOS 7.2
- Mellanox FDR Infiniband



# Additive

# Additive Prep in SpaceClaim

File

Design

Sketch

Display

Assembly

Measure

Facets

Additive

Repair

Prepare

Workbench

Detail

Sheet Metal

Tools

KeyShot

Create

Add Part

Settings

Orientation Map

Create Regions

Create Support

Export

About

Manual

Build Volume

Orientation

Regions

Support Generation

Export

About

Help

Structure

Build Volume\*

Baseplate

Workspace Stem

Stem (Part)

Support Regions

Surface\_SupportRegion

Surface\_SupportRegion

Surface\_SupportRegion

Supports

Structure

Layers

Selection

Groups

Views

Options

Properties

Created

Creator

Description

Identifier

Keywords

Language

Last Modified By

Last Printed

Modified

Revision

Subject

Title

Version

Material

Material Name

Fluid

Density

Name

Instance Suffix

Sheet Metal

Unknown Material

False

None

Properties

Appearance

Showing orientationmap of Facets

Heartcell support

Block with angled support

Rod support

ANSYS

2019 R2

Orientation Map

Current Settings

Z-Offset

Z

270

180

90

0

0

45

90

135

X

Rotation: 90° 231° 5 mm

Prioritization

34%

33%

33%

Support

Z

X

Build Time

Z

X

Distortion Tendency

Z

X

Showing orientationmap of Facets

Stem\*

Build Volume\*

1 Component

# ANSYS SpaceClaim Additive Prep Work Flow

Load parts

Define build envelope

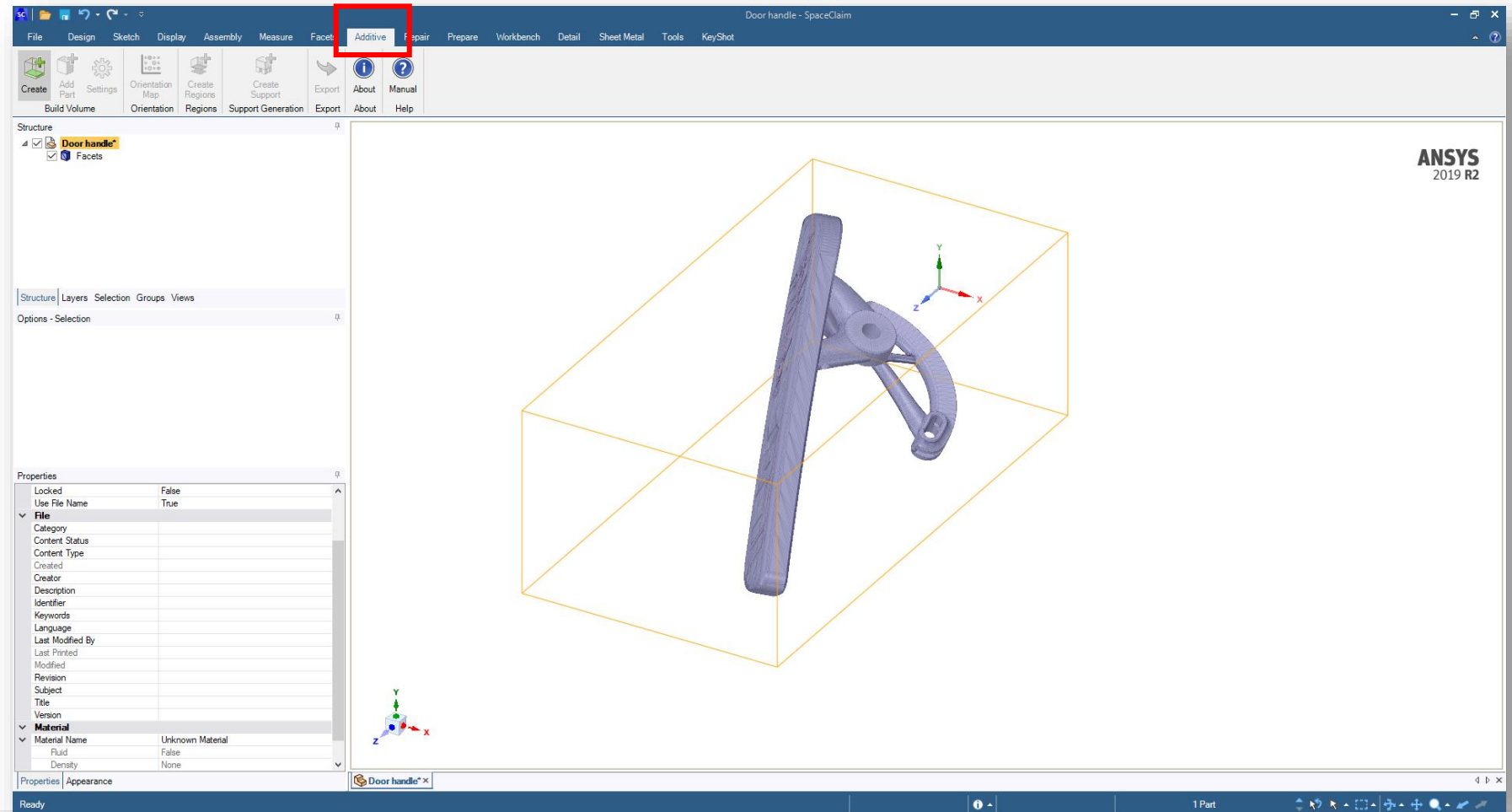
Optimize orientation

Create support regions

Create supports

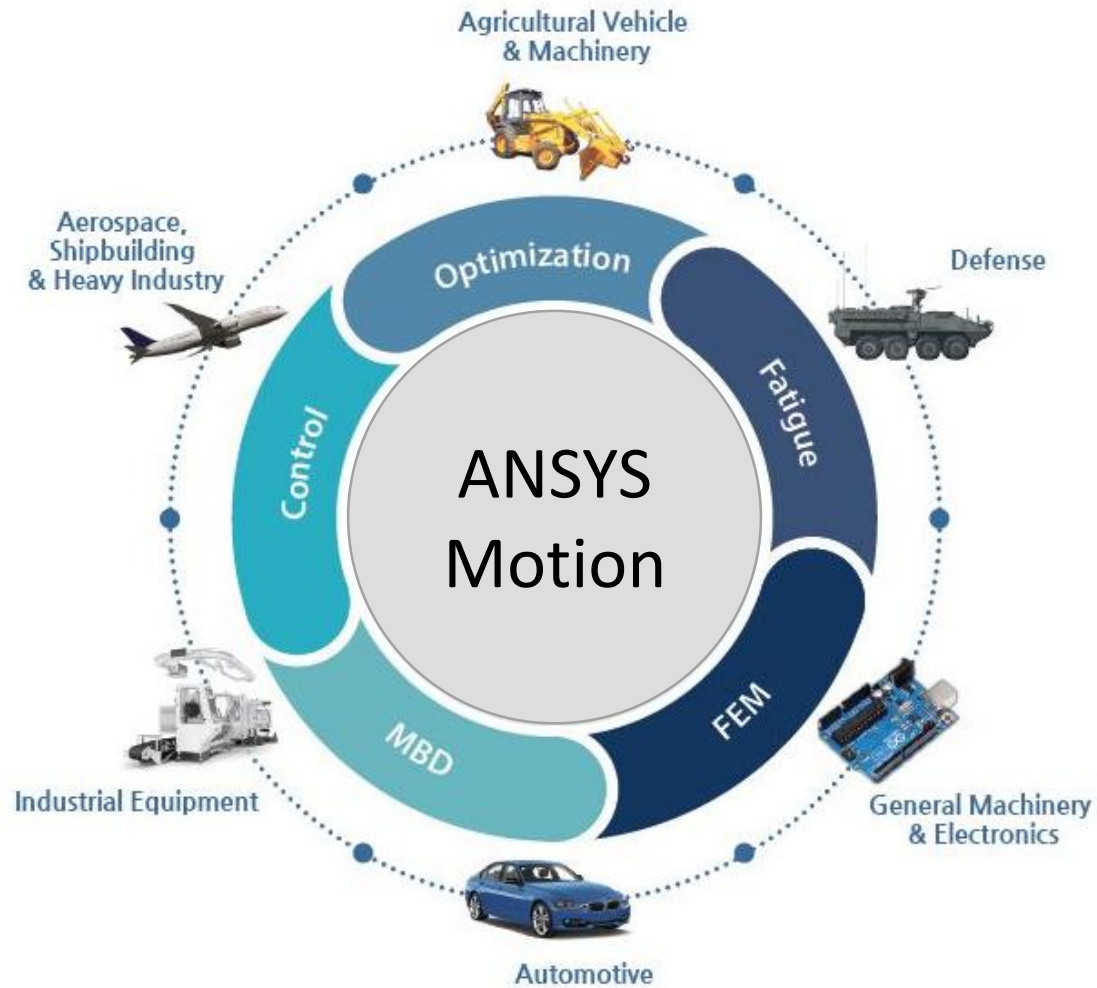
Export & print

- Import any file format
- Access to all SpaceClaim features for file manipulation & repair
- Design parts from scratch



# Ansys Motion

# ANSYS Motion



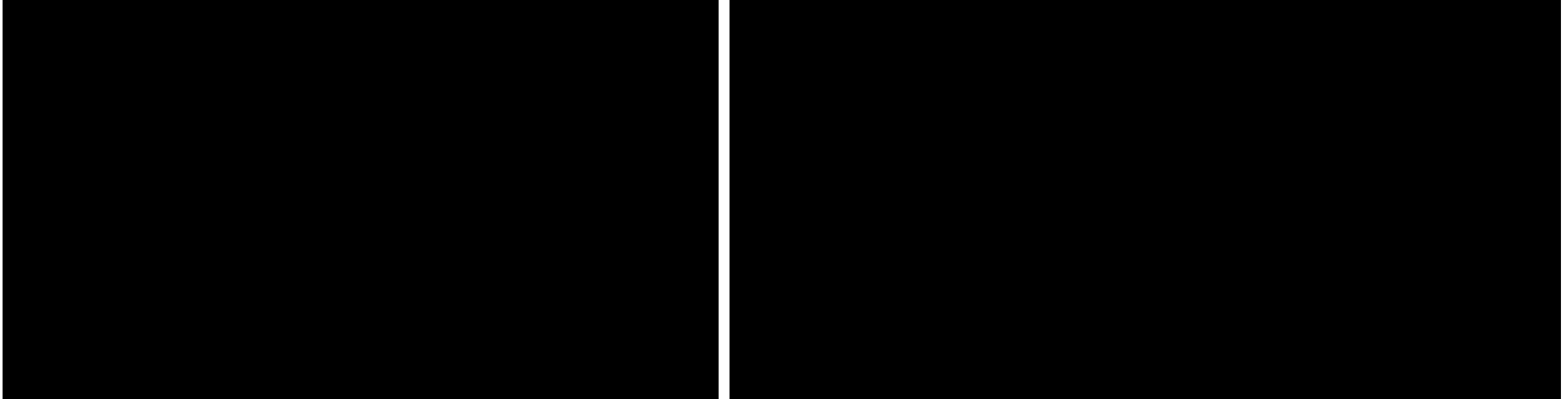
**ANSYS Motion: A dedicated Multi Body Dynamics tool.**

**ANSYS Motion: A software suite consisting of a baseline package and four toolkits**

- **ANSYS Motion**
- **ANSYS Motion Links Toolkit**
- **ANSYS Motion Drivetrain Toolkit**
- **ANSYS Motion Car Toolkit**
- **ANSYS Motion EasyFlex Toolkit**



# Ansys Motion – a new paradigm in Multibody Dynamics



Ansys Motions' advanced contact logic and tightly integrated rigid and flexible solvers give a capability that is unique.

# Modules Contained Within ANSYS Motion

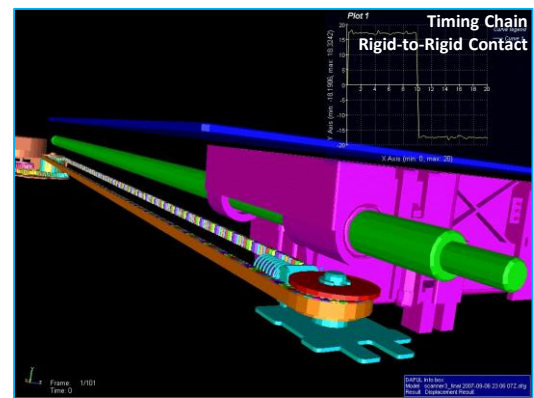
Product	Module	Description
ANSYS Motion	MBD Pro	Multi-Body Dynamics Analysis Package
	FE Dynamics	Multi-Flexible Body Dynamics Analysis Package based on FEM
	Linear	Mode and Natural Frequency Analysis Module
	Fatigue	Fatigue Analysis Module with S/N and E/N curves
	Modal	Modal Flexible Body Module
	SMP	Parallel Processing Module for Solver
	MATLAB Interface	Co-simulation with MATLAB Simulink
	FMI	Co-simulation based on FMI
	API DEV. Execution	API module for developers
Toolkits	CAR Toolkit	Vehicle Dynamics Analysis Toolkit
	LINKS Toolkit	Tracked Vehicle Dynamics Analysis Toolkit, Belt and Chain Dynamics Analysis Toolkit
	DRIVETRAIN Toolkit	Power Transfer System Dynamics Analysis Toolkit for NVH
	EASYFLEX Toolkit	Multi-Flexible Body Dynamics Analysis Package based on MeshFree technology

The ANSYS Motion package contains all of the these modules as standard

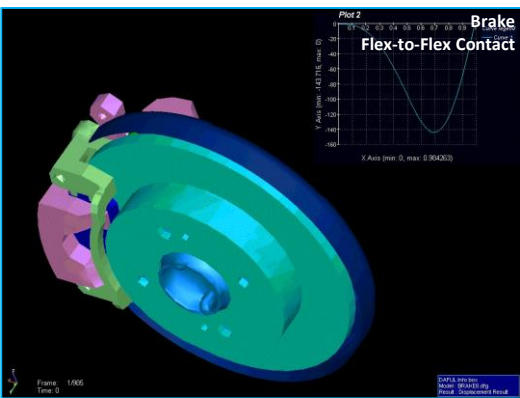
Toolkits can be used to aid preprocessing for specialist applications

# ANSYS Motion distinctions

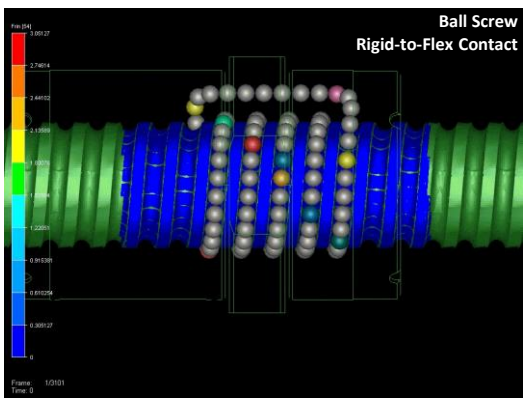
- User wants to get more accurate boundary conditions for static analyses through analysis of complete system dynamics.
- ANSYS Motion has the most advanced solver of any commercially available multi-body dynamics tool.
- ANSYS Motion is strong when it comes to contact of complex geometry and high speed rotating problems.
- ANSYS Motion can solve large deformation problems in the time domain while also considering non-linear material properties



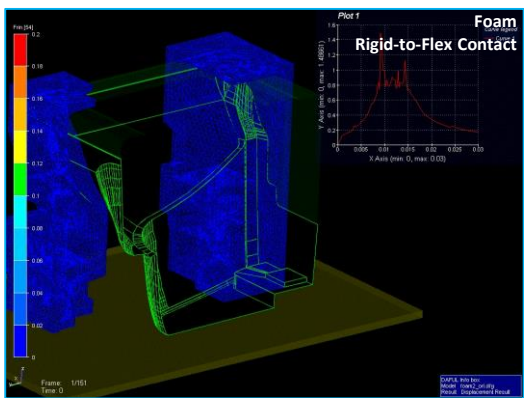
ADAMS	3,000 min
Ansyp Motion	46 min.



RecurDyn	4.7 days
Ansyp Motion	6 min.



RecurDyn	Not Solved
Ansyp Motion	40 min.



RecurDyn	780 min.
Ansyp Motion	45 min.

# Thank You and Questions