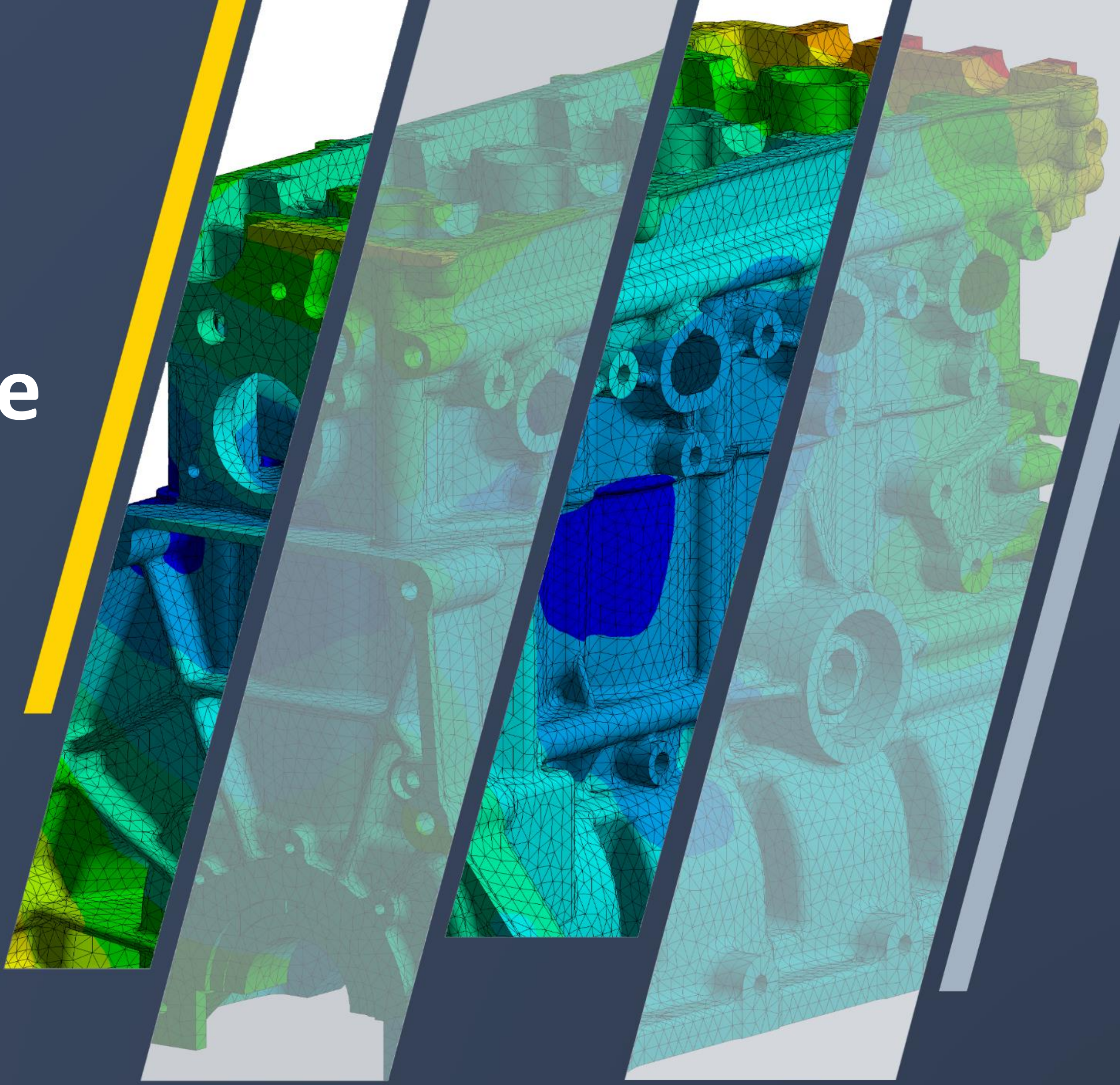
ANSYS

Structures 2020 R1 Update Presentation



Content

Mechanical Core

Expanded Imported Load Support

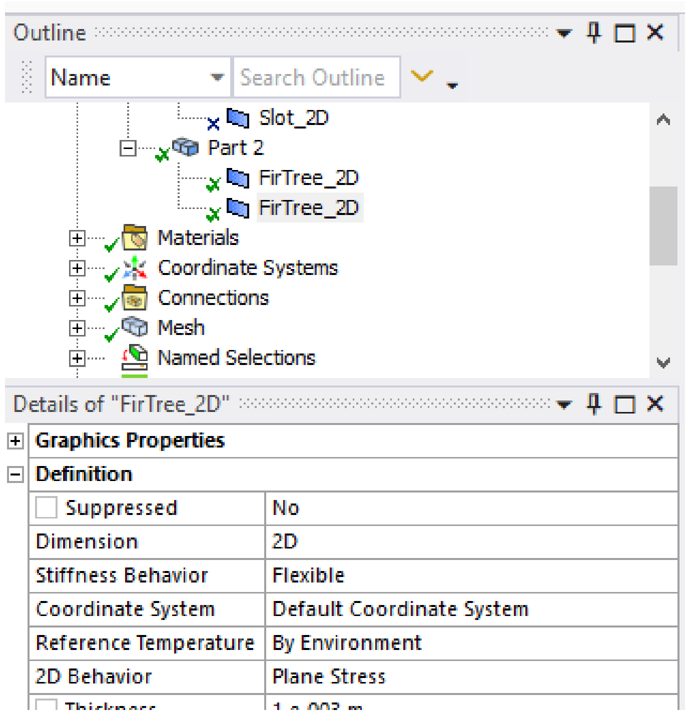
Line Body Treatment

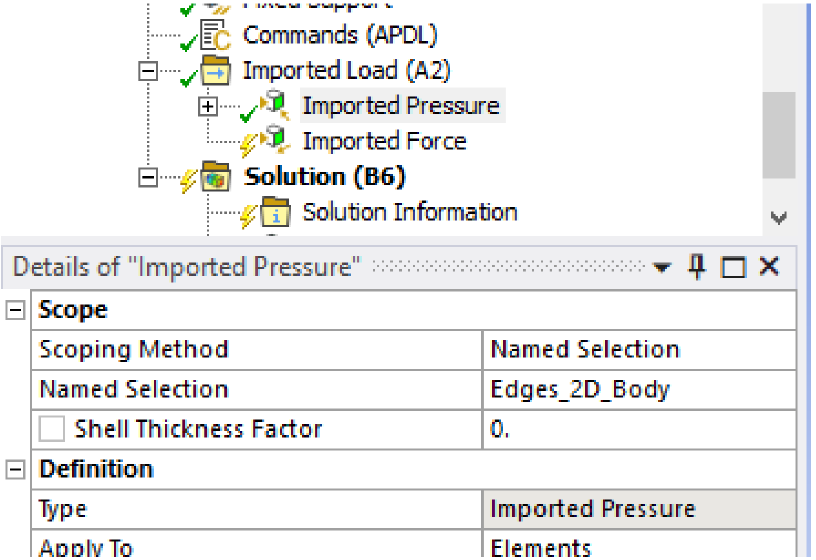
2D Contact Normal Display and Flipping

Editable and User Defined Cross Sections

Expanded Imported Load Support

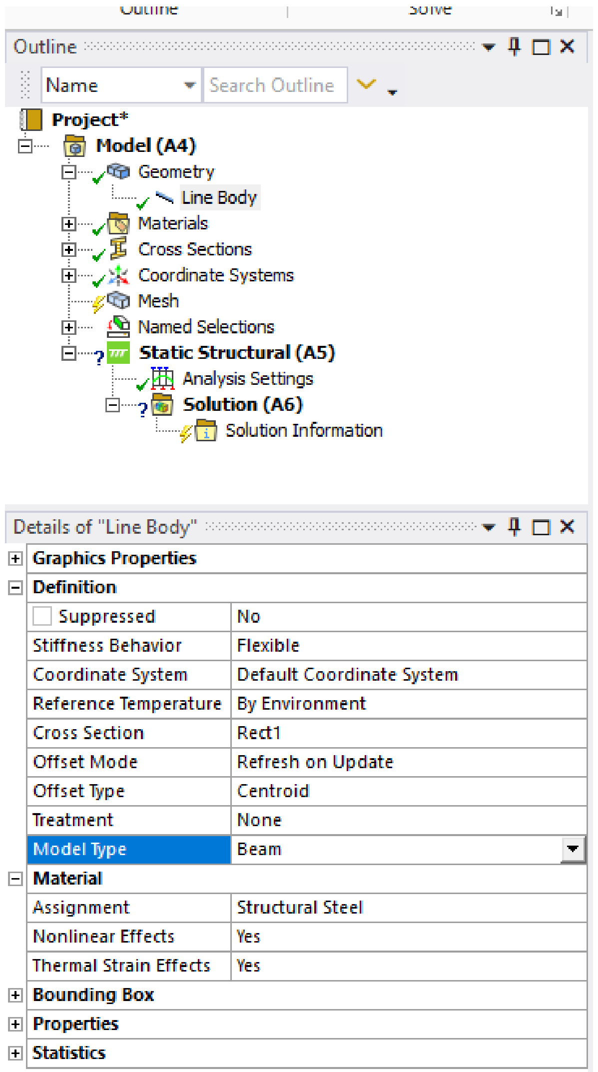
- Imported loads in a 3D analysis can now be scoped to shells with 2D dimension behavior





Line Body Treatment

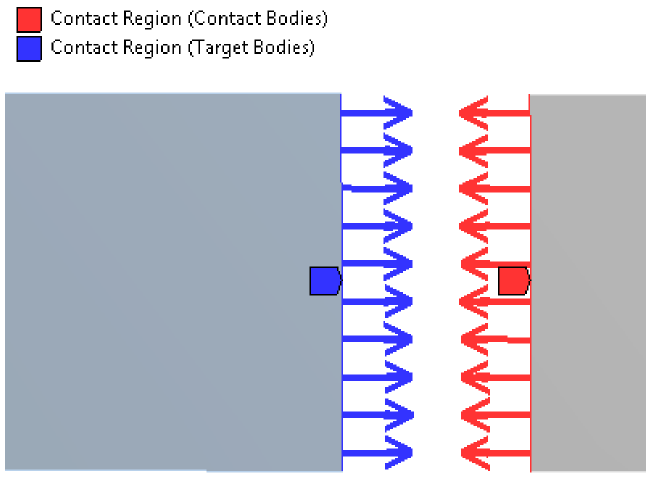
- Line bodies now have access to the Treatment property in the UI



2D Contact Normal Display and Flipping

- For edge contacts on 2D surface bodies, there is a new **“Display”** category that includes the property **“Element Normals”**. This property displays the normal direction of the elements for each edge in contact

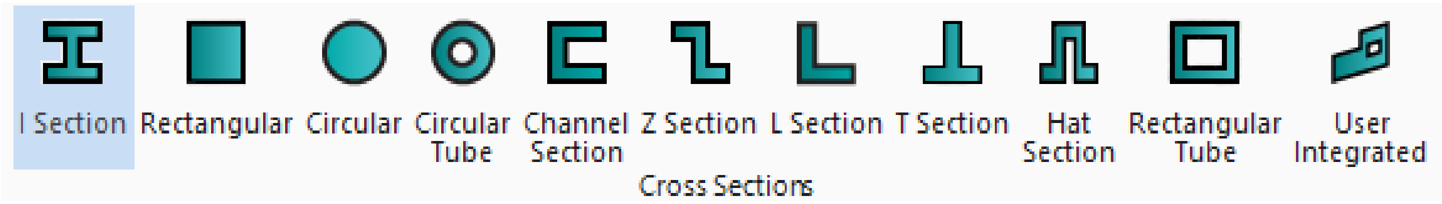
- This display feature works in combination with two additional new properties of the **“Geometric Modifications”** category: *Flip Contact Normals* and the *Flip Target Normals*. These properties enable you to invert or flip the normal direction of the edge elements in contact

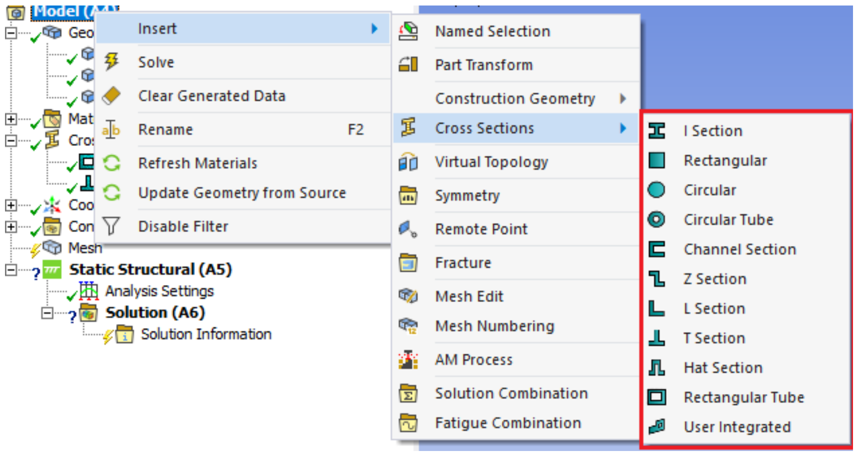


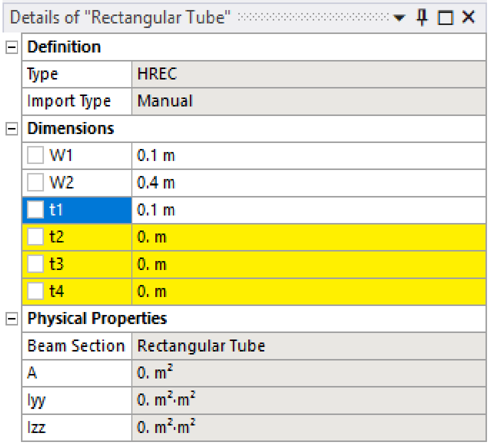
Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	1 Edge
Contact Bodies	cont
Target Bodies	targ
Shell Thickness Effect	No
Protected	No
Definition	
Display	
Element Normals	Yes
Advanced	
Geometric Modification	
Contact Geometry Correction	None
Target Geometry Correction	None
Flip Contact Normals	No
Flip Target Normals	No

Editable and User Defined Cross Sections

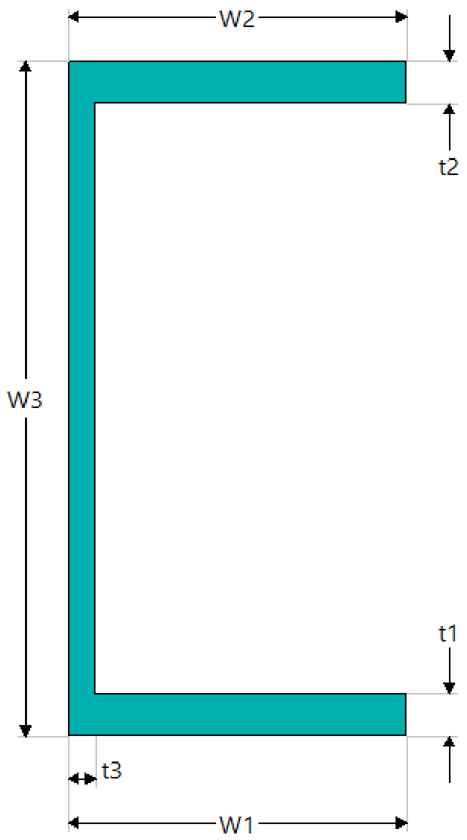
- Mechanical now supports creating, editing and duplicating cross sections





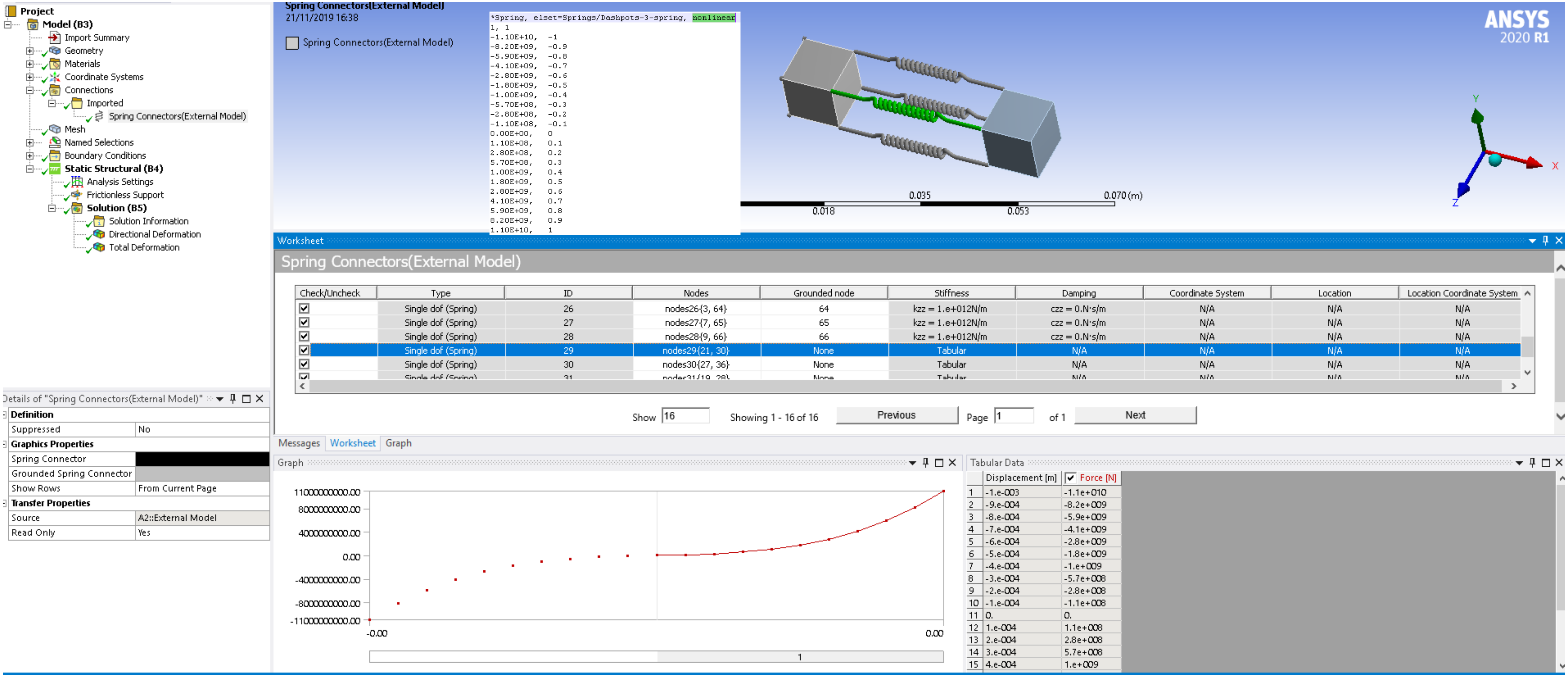


C Channel Section

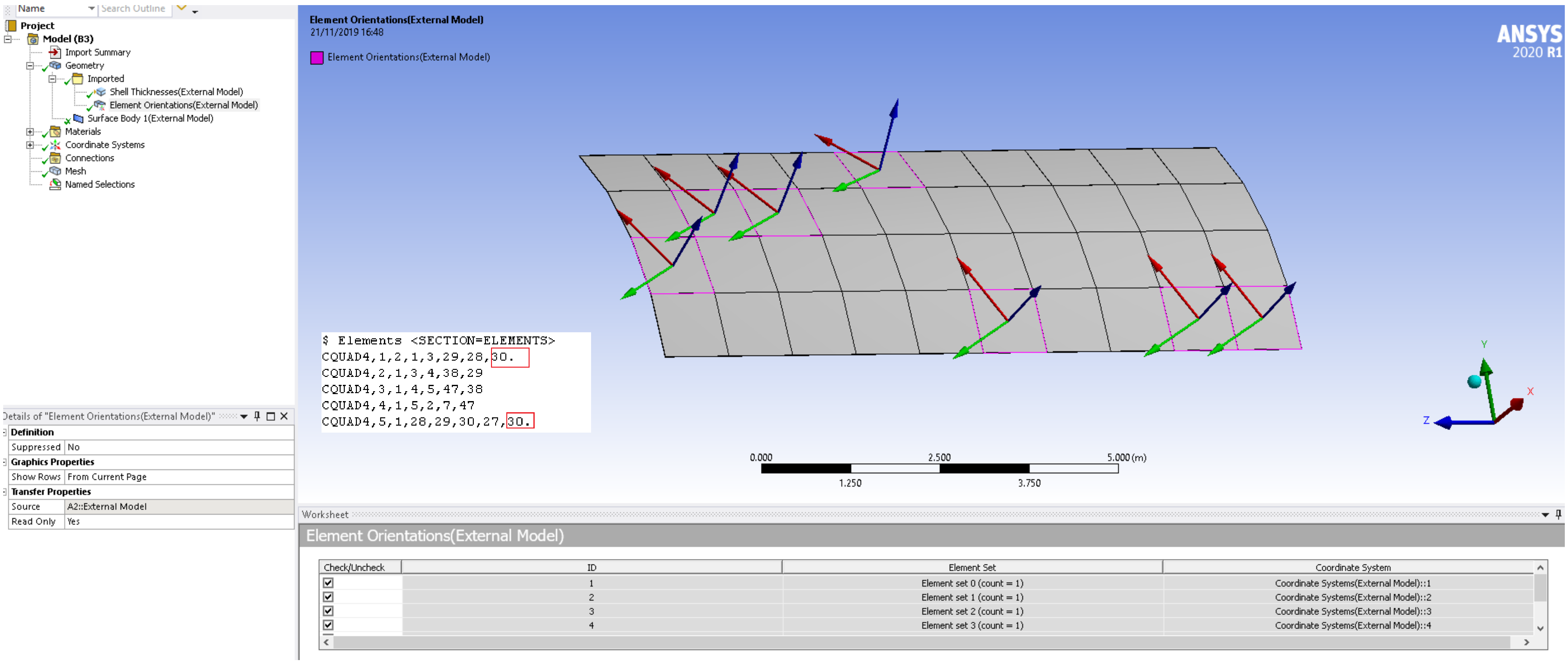


External Model

Support Nonlinear Springs from ABAQUS (SPRING2 with table)

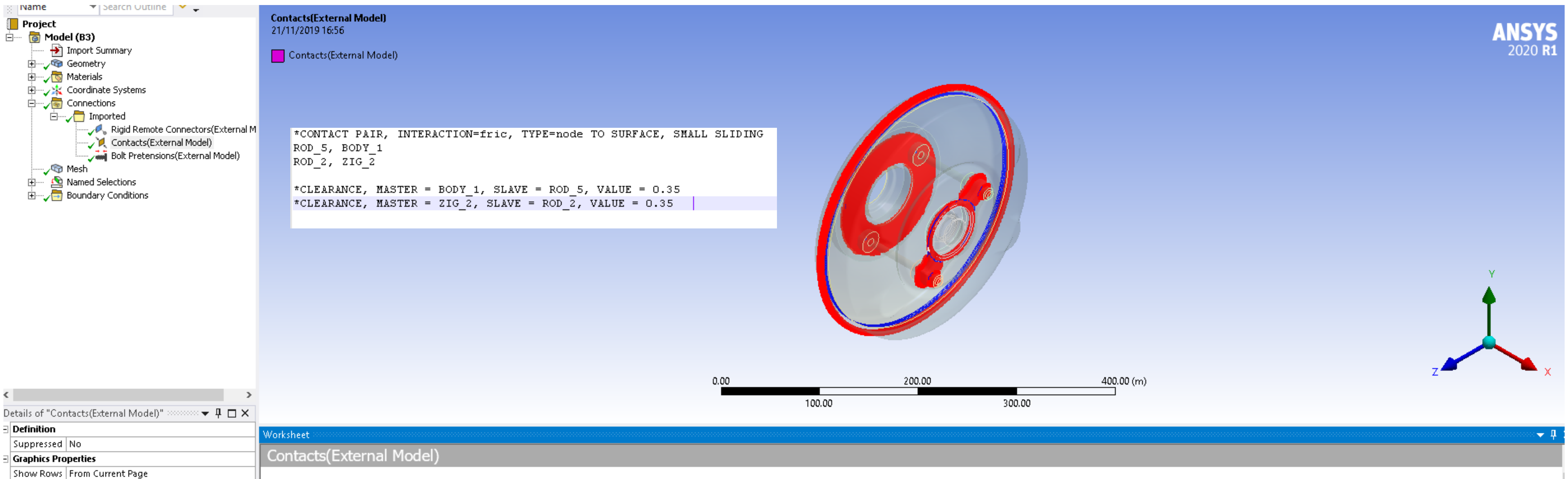


Support Theta Field from NASTRAN CQUAD*/CTRA*/CTRI* Cards



Support ABAQUS *CONTACT INTERFERENCE and *CLEARANCE

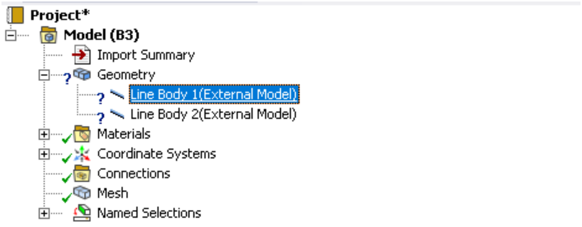
- These are shown in the columns « Interface Treatment » (« Offset Only, No Ramping » and « Offset Only, Ramped ») and « Offset »

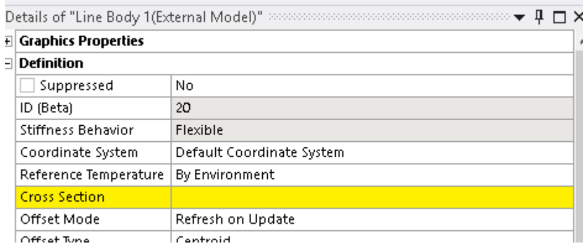


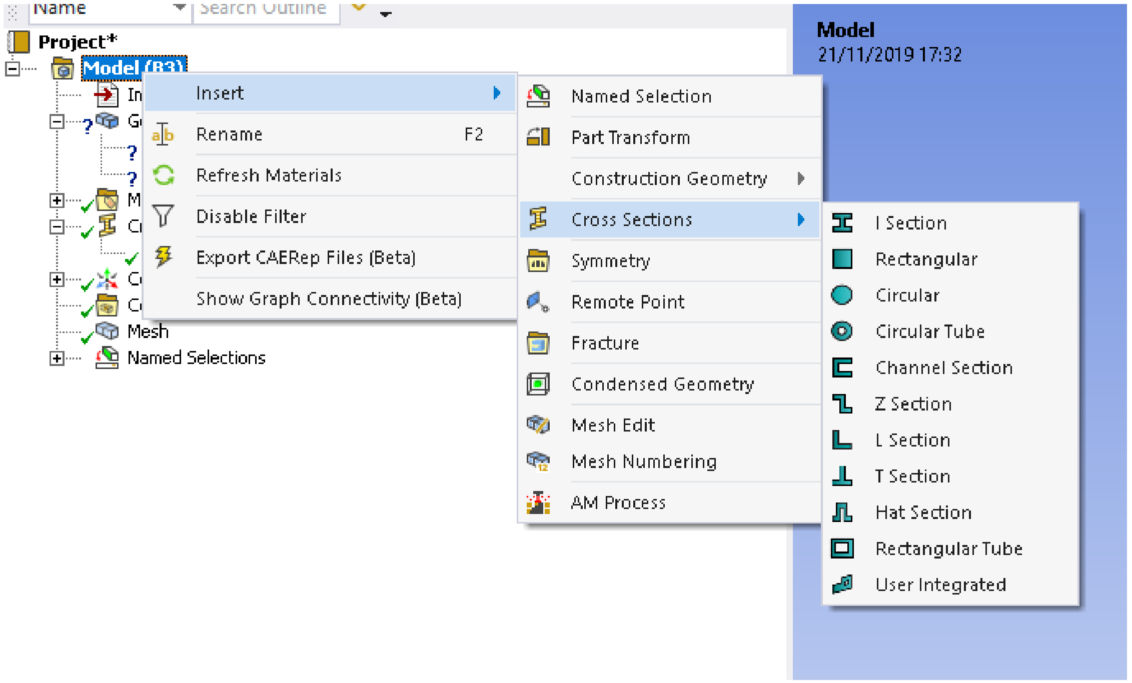
Check/Uncheck	ID	Source	Target	Type	Fricti...	Behavior	Formulation	Ther...	Normal Stiffness	Normal Stiffness Value	Interface Treatment	Offset
☑	1	ROD_5(External Model)	BODY_1(External Model)	Frictional	2.e-002	Asymmetric	Augmented Lagrange	0. W/°C	Program Controlled	N/A	Offset Only, No Ramping	-0.35 m
☑	2	ROD_2(External Model)	ZIG_2(External Model)	Frictional	2.e-002	Asymmetric	Augmented Lagrange	0. W/°C	Program Controlled	N/A	Offset Only, No Ramping	-0.35 m
☑	3	ROD_1(External Model)	ZIG_1(External Model)	Frictional	0.2	Asymmetric	Augmented Lagrange	0. W/°C	Program Controlled	N/A	Offset Only, No Ramping	0. m
☑	4	ZIG_3(External Model)	REAR_1(External Model)	Frictional	0.2	Asymmetric	Augmented Lagrange	0. W/°C	Program Controlled	N/A	Offset Only, No Ramping	0. m

Allow Importing Beam Elements without Cross Sections Assignments

- Beam elements without cross sections assigned to them can now be imported (previously blocked)
- Cross sections can later be assigned to them (on line bodies) using the new Cross Sections objects

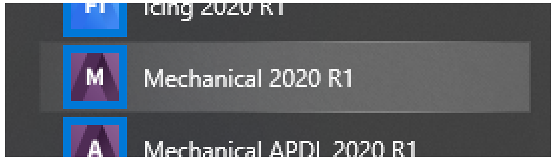






Drag&Drop into Mechanical Standalone

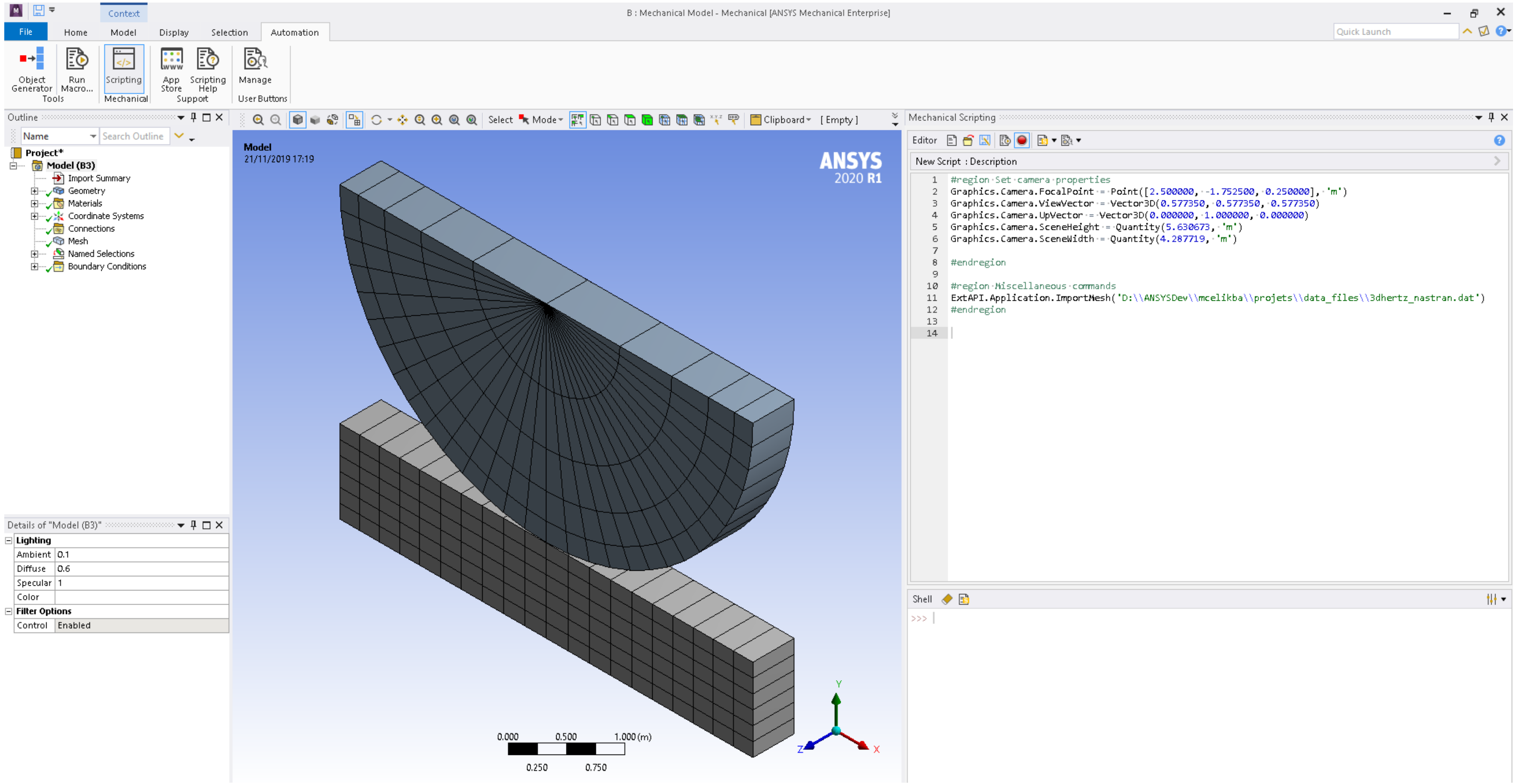
- Launch Mechanical standalone



- Then, **“drag&drop”** one of the supported External Model files into the Mechanical window

File/Import and Drag&Drop into Mechanical Standalone Journaling (Beta) and Scripting

- New python Apis to import mesh files using Mechanical Scripting feature



Workbench Post & Graphics

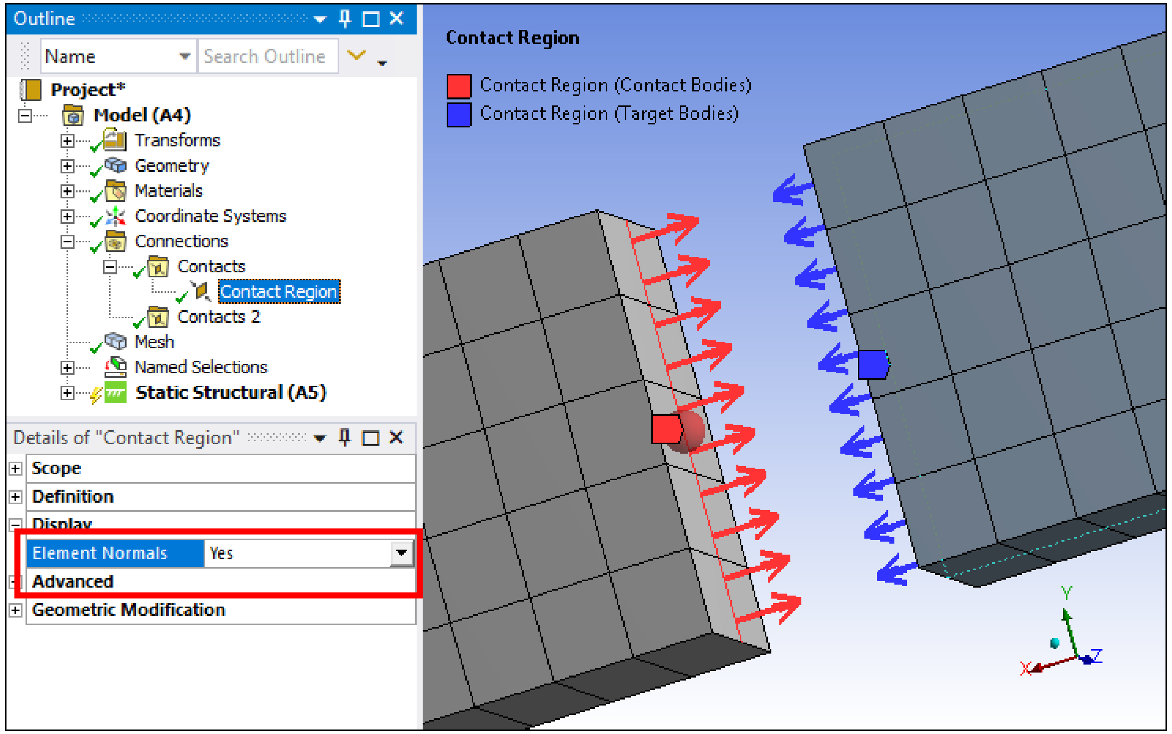
Geometry and Connections

2D Edge Contact Display and Modification

Display of 2D Edge Contact Normals

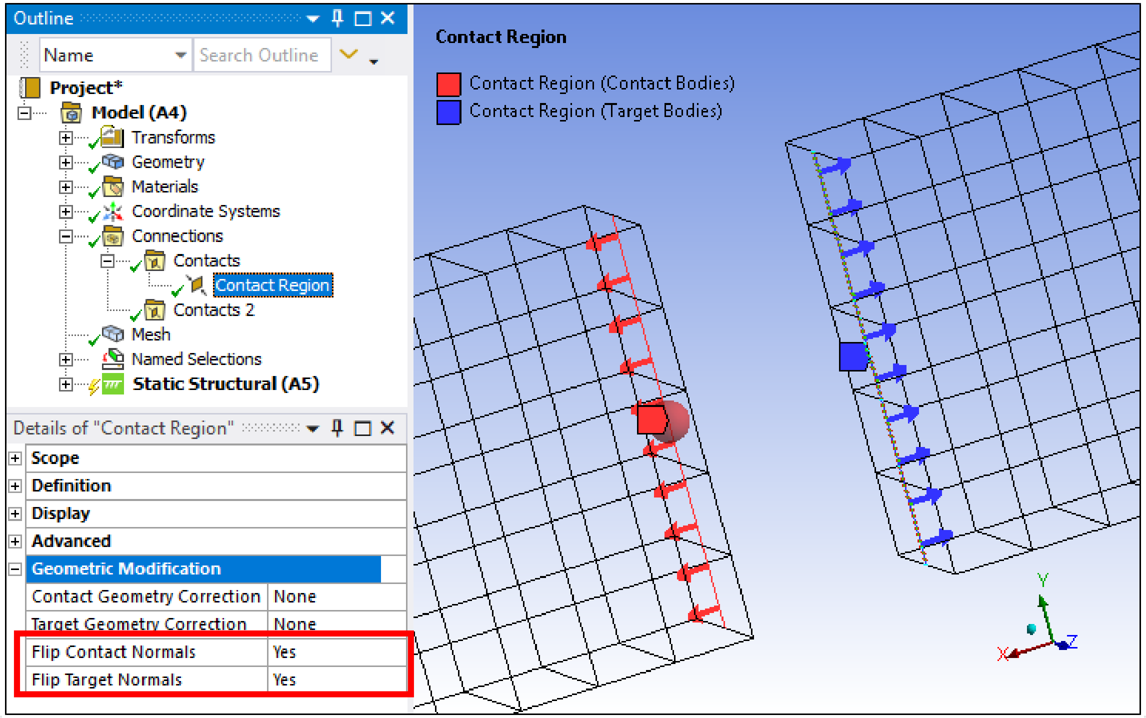
- A new **“Element Normals”** display property has been introduced for contact regions that contain 2D edge scopings

- The option enables or disables the display of the normals for a 2D edge’s expanded mesh element faces
  - Arrows appear in-between mesh nodes (and mid-side nodes, if present)



Re-orientation of 2D Edge Contact Normals

- The new annotation for 2D edge normals can help identify improperly oriented mesh elements, which can lead to improper results from the solver
- Using the newly added **“Flip Contact Normals”** and **“Flip Target Normals”** properties of the contact region, improperly oriented mesh elements can be corrected as they are being sent to the solver

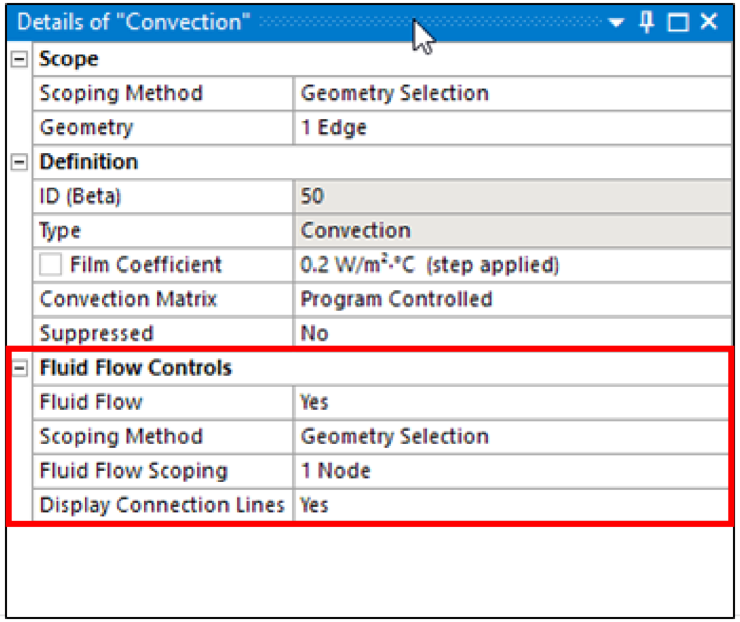


Loading and Boundary Conditions

Convection Load with Fluid Flow Display

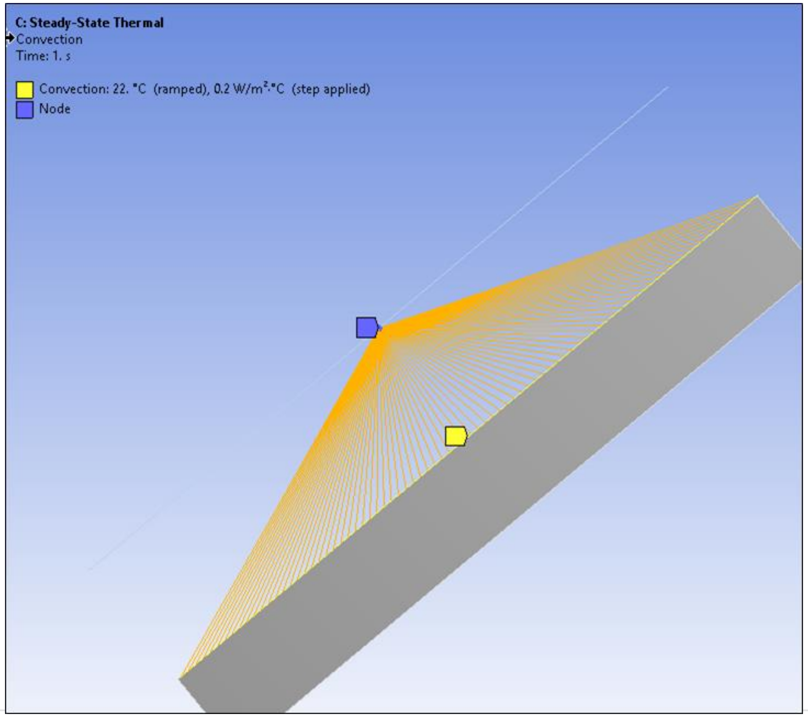
Convection Load with Fluid Flow Display

- When you specify reference temperature using the **“Thermal Fluid”** property of a **“Line Body”**, and you set the **“Fluid Flow”** property of a *Convection* or *Imported Convection Coefficient* objects to *Yes*, there is a new display property: **“Display Connection Lines”**. This property enables the display of connection lines between the centroid of each element face/edge of the convection surface(s)/edge(s) to the corresponding closest node on the fluid flow scoping



Details of "Convection"

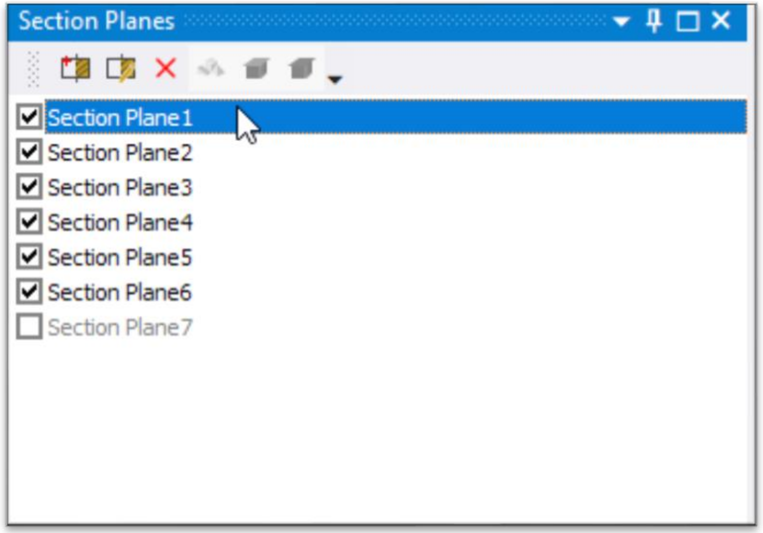
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
ID (Beta)	50
Type	Convection
Film Coefficient	0.2 W/m²·°C (step applied)
Convection Matrix	Program Controlled
Suppressed	No
Fluid Flow Controls	
Fluid Flow	Yes
Scoping Method	Geometry Selection
Fluid Flow Scoping	1 Node
Display Connection Lines	Yes



Graphics: Creating Section Planes

Creating Section Planes

- Section Planes Tool allows you to now create more than six **“Section Planes”**. However, if six planes are active (“checked”), any additional planes cannot be viewed until you have deactivated (“unchecked”) an existing plane and then activated the desired plane. Once you exceed six, unchecking an existing plane enables you to activate any defined planes greater than six



ACT API

Querying Results with PlotData

Result Extraction using PlotData

- Contour results for an evaluated result can be accessed in ACT via PlotData. The result is represented in a table format with **"Independent"** and **"Dependent"** column variables

- Depending on the type of result (nodal, elemental, elemental nodal, path results), the independent variables can be nodes, elements, both or x,y,z coordinates. The result value components are dependent variables for the tabular result

```
Shell
>>> result = Model.Analyses[0].Solution.Children[8]
    s2= result.PlotData
>>> s2
              Node      Element      Values
                                       (Pa)
0                1            1   -0.046019
1                2            2    -0.14894
2                3            3    -0.29478
...
34557         8269          122     0.11835
34558         8270          121    0.091473
34559         8271          133    0.010163
>>> s2.Independents
              Node      Element

0                1            1
1                2            2
2                3            3
...
34557         8269          122
34558         8270          121
34559         8271          133
>>> s2.Dependents
            Values
              (Pa)
0        -0.046019
1         -0.14894
2         -0.29478
...
34557      0.11835
34558     0.091473
34559     0.010163
```

```
>>> elemental = Model.Analyses[0].Solution.Children[9]
>>> elemental.PlotData
       Element          XY          YZ          XZ
                     (rad)       (rad)       (rad)
0            1           0           0           0
1            2           0           0           0
2            3           0           0           0
...
573        574      1.5708           0           0
574        575      1.5708           0           0
575        576      1.5708           0           0
```

```
>>> nodal = Model.Analyses[0].Solution.Children[9]
>>> nodal.PlotData
          Node           X          Y  ...        XY          YZ           XZ
                      (Pa)       (Pa)  ...      (Pa)        (Pa)         (Pa)
0            1  -0.0053892    -24.683  ...  0.0054423   0.044763     0.10038
1            2    -0.11291    -24.712  ...    0.08598  -0.025587    0.059832
2            3    -0.26545    -24.708  ...    0.12777  -0.069434 -0.00044115
...
8278      8279    0.018029    -22.675  ...   0.075221   0.088281  -0.0039558
8279      8280    0.015398    -24.733  ...   0.013573  -0.012347  -0.0050252
8280      8281   0.0020731    -25.473  ...  -0.047386   -0.12531  -0.0049456
```

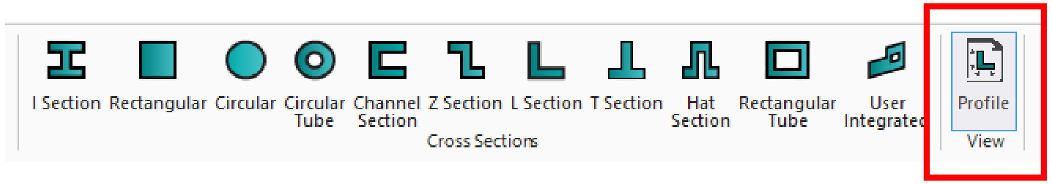
```
>>> path = Model.Analyses[0].Solution.Children[11]
>>> path.PlotData
     X Coordi...   Y Coordi...   Z Coordi...   ...          XY          YZ          XZ
                                               ...     (m)/(m)     (m)/(m)     (m)/(m)
0              1             1             1   ... -2.9161E-11  8.3858E-10  4.8314E-10
1              1       0.97917             1   ...  -2.618E-11  7.1473E-10  4.2688E-10
2              1       0.95833             1   ...   -2.32E-11  5.9088E-10  3.7062E-10
...
46             1      0.041667             1   ... -1.5804E-11   2.986E-10 -3.9267E-10
47             1      0.020833             1   ... -1.8701E-11  3.6569E-10 -4.3135E-10
48             1             0             1   ... -2.1598E-11  4.3278E-10 -4.7004E-10
```

Cross Sections

Profile Option

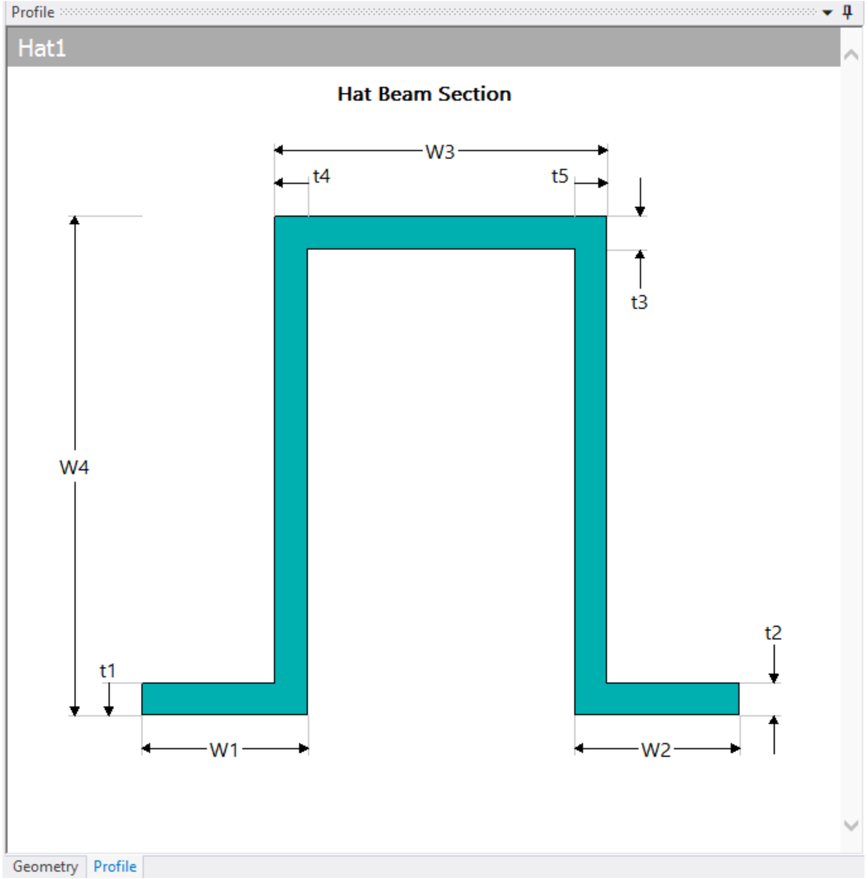
Display a Cross Section using the Profile Option

- Enabling the **“Profile”** option of the **“Cross-Section Context”** menu displays a cross section with its dimensions. Any modifications made to the *Dimensions* in the *Details* of the cross-section object will show in this **“Profile”** view



Details of "Hat1"

Definition	
Type	HATS
Import Type	Manual
Dimensions	
W1	5.e-003 m
W2	5.e-003 m
W3	1.e-002 m
W4	1.5e-002 m
t1	1.e-003 m
t2	1.e-003 m
t3	1.e-003 m
t4	1.e-003 m
t5	1.e-003 m
Physical Properties	
Beam Section	Hat1
A	4.6e-005 m^2
Iyy	1.3478e-009 $m^2 \cdot m^2$
Izz	1.0553e-009 $m^2 \cdot m^2$

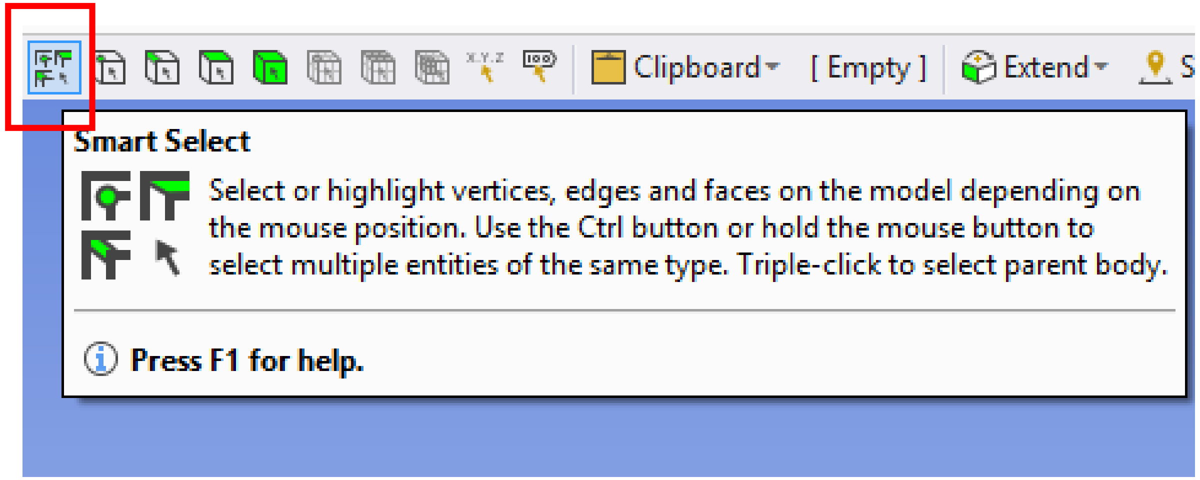


Smart Select

Default Now

Smart Select

- The Graphics toolbar option, **"Smart Select",** is now the default geometry picking option

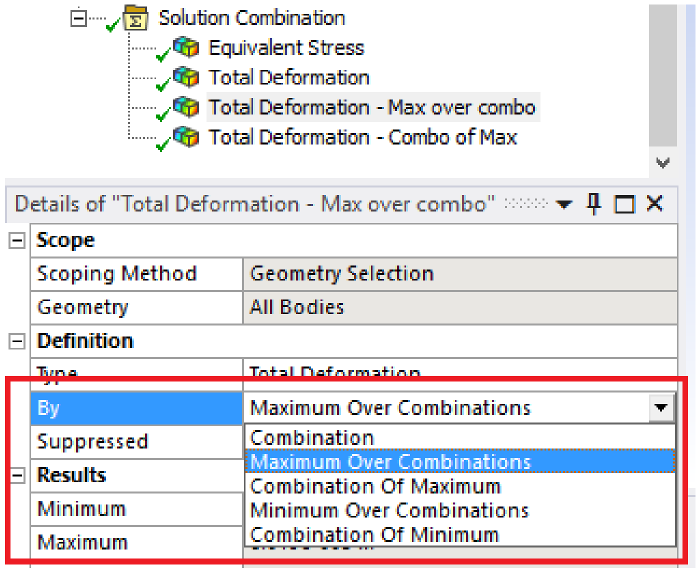


Solution Combination

Envelope Calculations

Envelope Calculations for Solution Combination Results

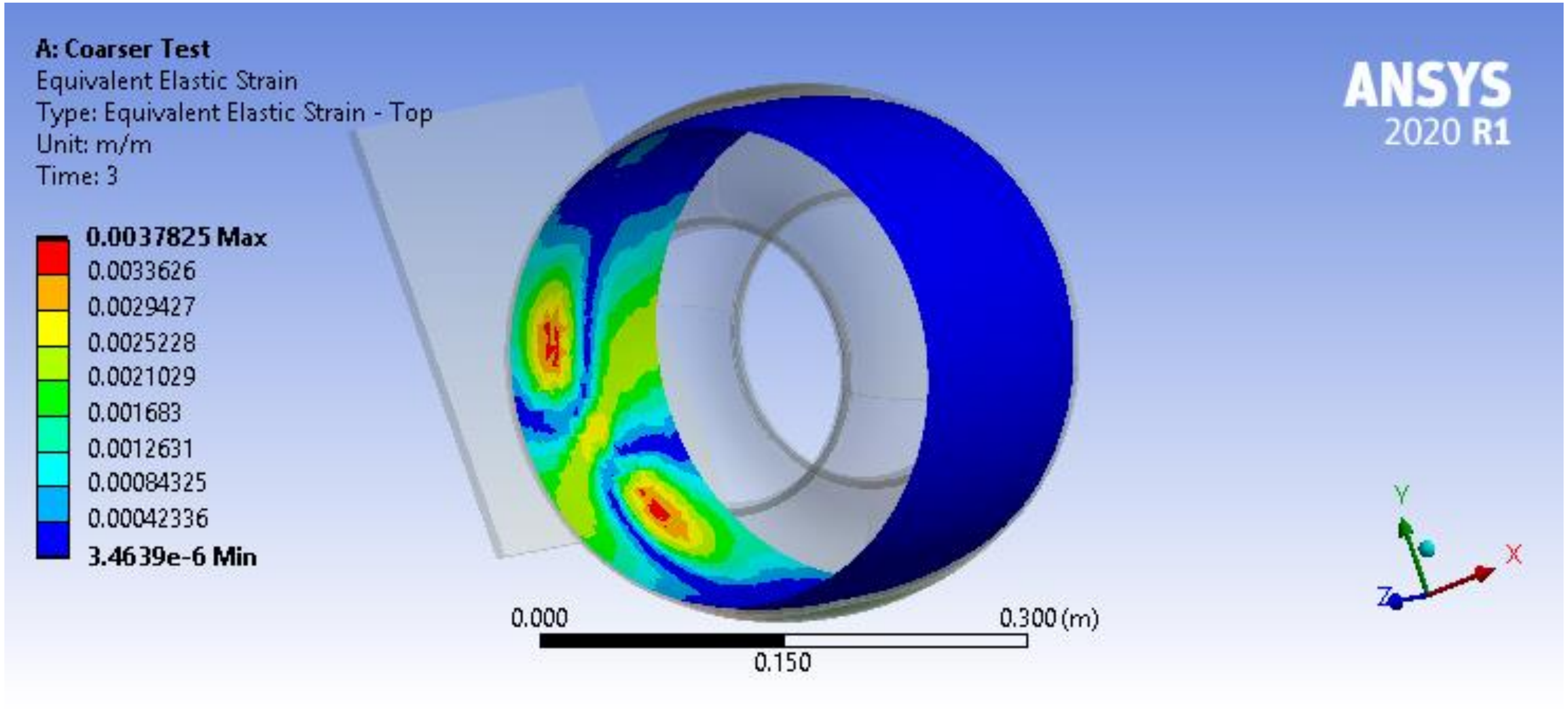
- **"Solution Combination"** results now have additional options to choose the type of calculation:
  - *Combination (default):*
    Specify a desired combination for result evaluation
  - *Maximum/Minimum Over Combinations:*
    Each node, element, or sample point is swept through the combinations to find its maximum/minimum result
  - *Combination of Maximum/Minimum:*
    Each node, element, or sample point is swept through the combinations and the combination at which the maximum/minimum occurs is reported



Post-processing Reinforcement Elements

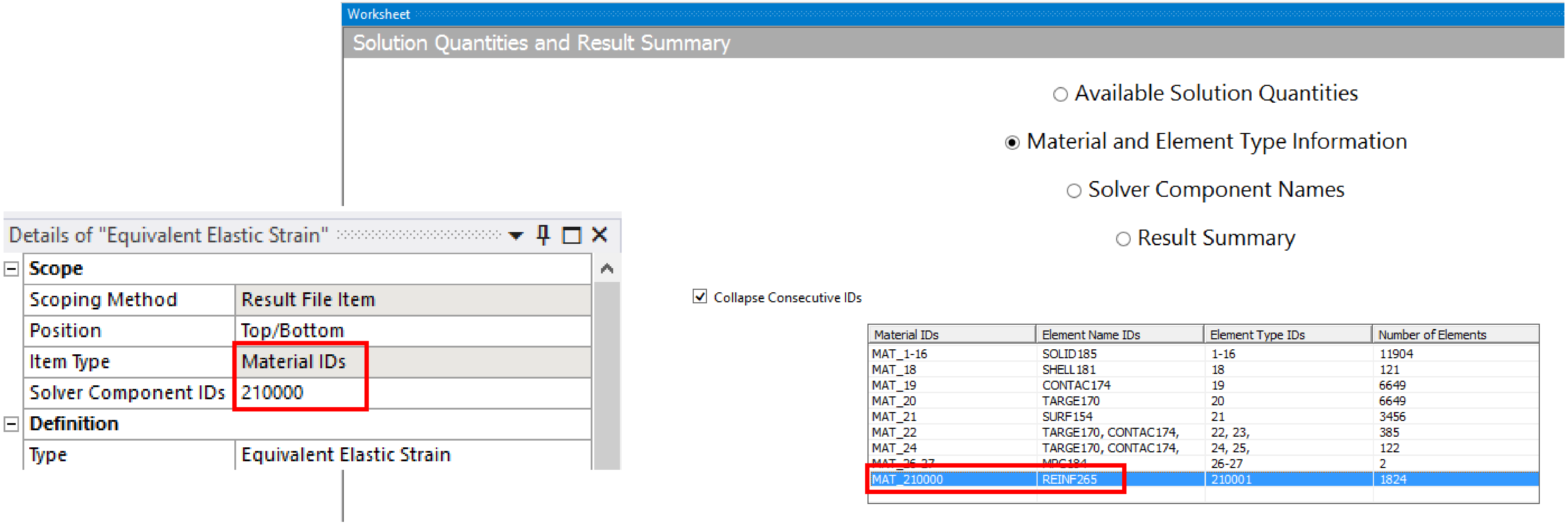
Results for Reinforcement Elements

- When you have reinforcement elements (REINF263, REINF264, and REINF265) in your model, you can now view results for these elements using the **“Result File Item”** scoping method for result objects



Results for Reinforcement Elements

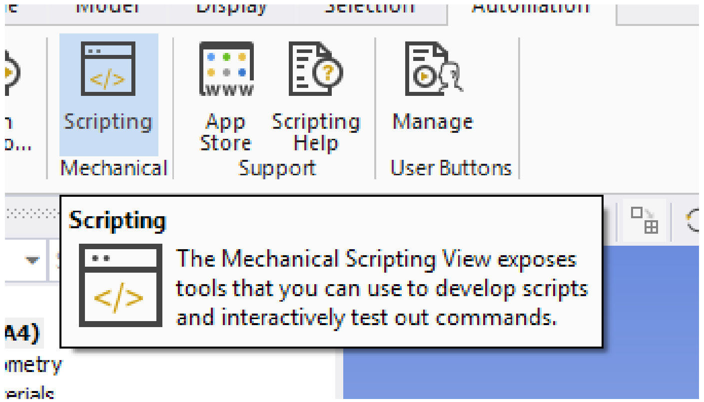
- The *Material* and *Element Type Information* selection of the *Solution Quantities* and *Result Summary* worksheet page allows you to quickly create and view results on these elements

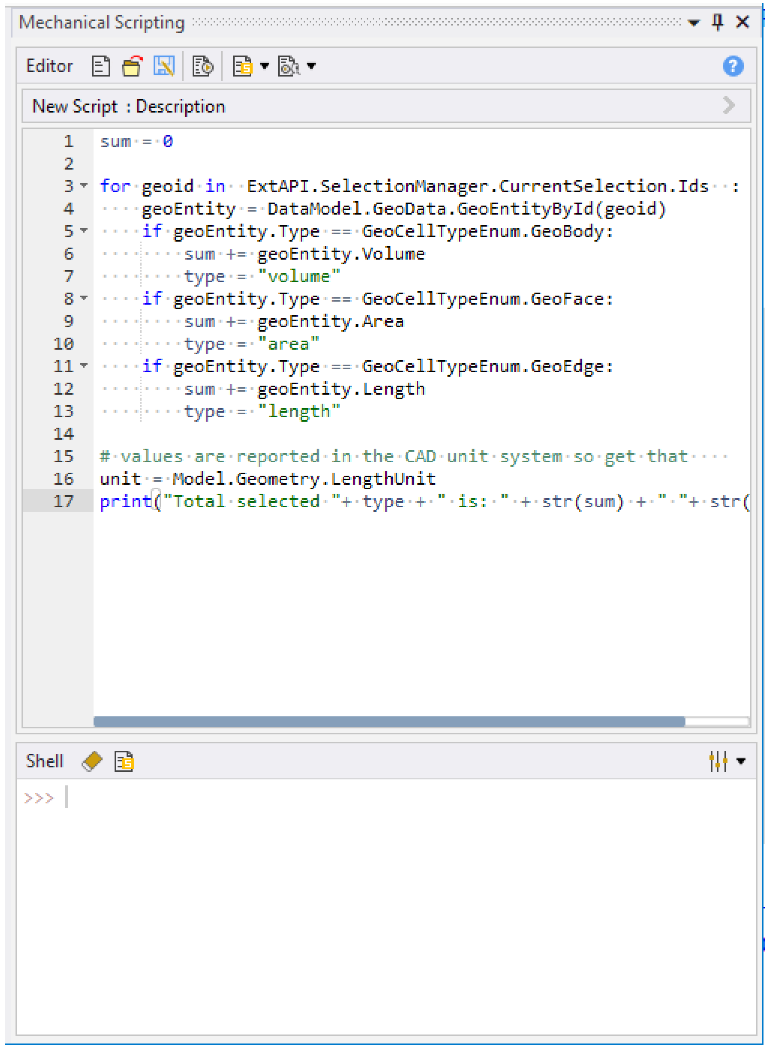


Mechanical Architecture

Mechanical Scripting

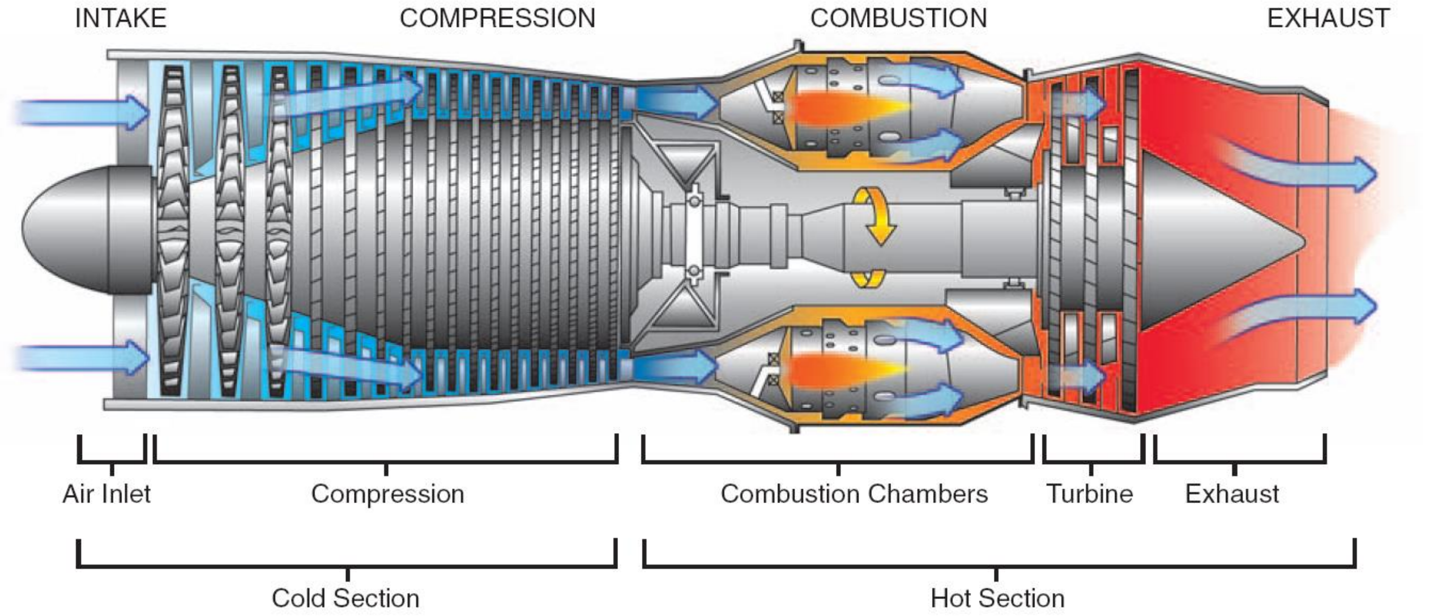
- ACT Console replacement
- Maximize real-estate
- Scripting editing and execution
- Integration of button editor





Thermo Mechanical Fatigue Material Models

- Nonlinear Isotropic Hardening (Power Law and Voce Law)
- Chaboche Kinematic Hardening with Static Recovery
- Exponential Visco-Hardening (EVH) Viscoplasticity
- Perzyna and Peirce Viscoplasticity
- Multilinear Isotropic Hardening Static Recovery
- Hill Yield, f(T)



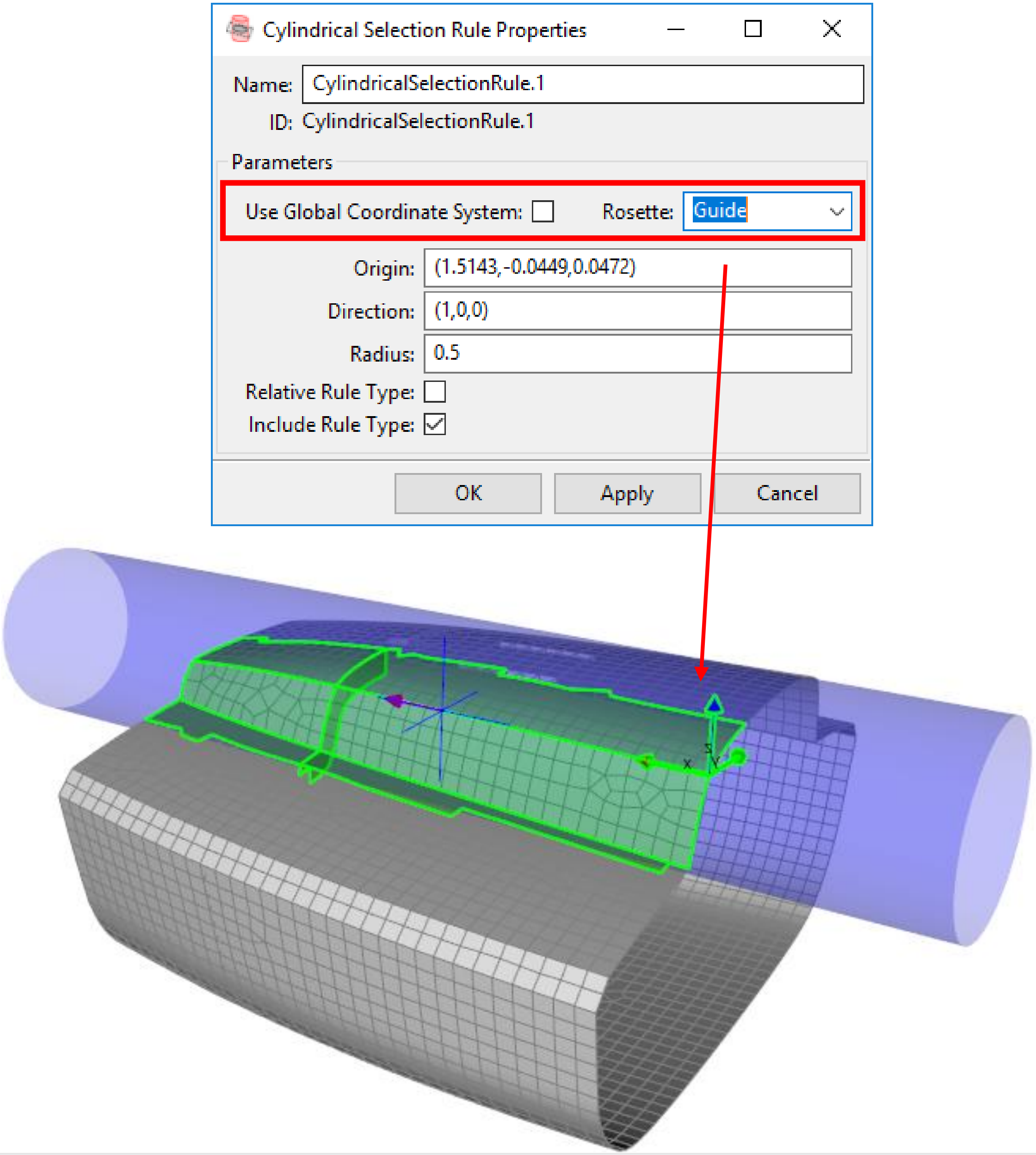
https://www.sciencemuseum.org.uk/

Composites

Selection Rules Based on Rosettes

- **"Selection Rules"** can now be defined relative to Rosettes. When enabled, the origin and directions of the Selection Rule are relative to the selected Rosette instead of the global coordinate system. This improves associativity, makes the modeling more accurate and more convenient.

- This feature is implemented for the Parallel, Cylindrical and Spherical Selection Rule



Python and other Third-Party Software Upgrades

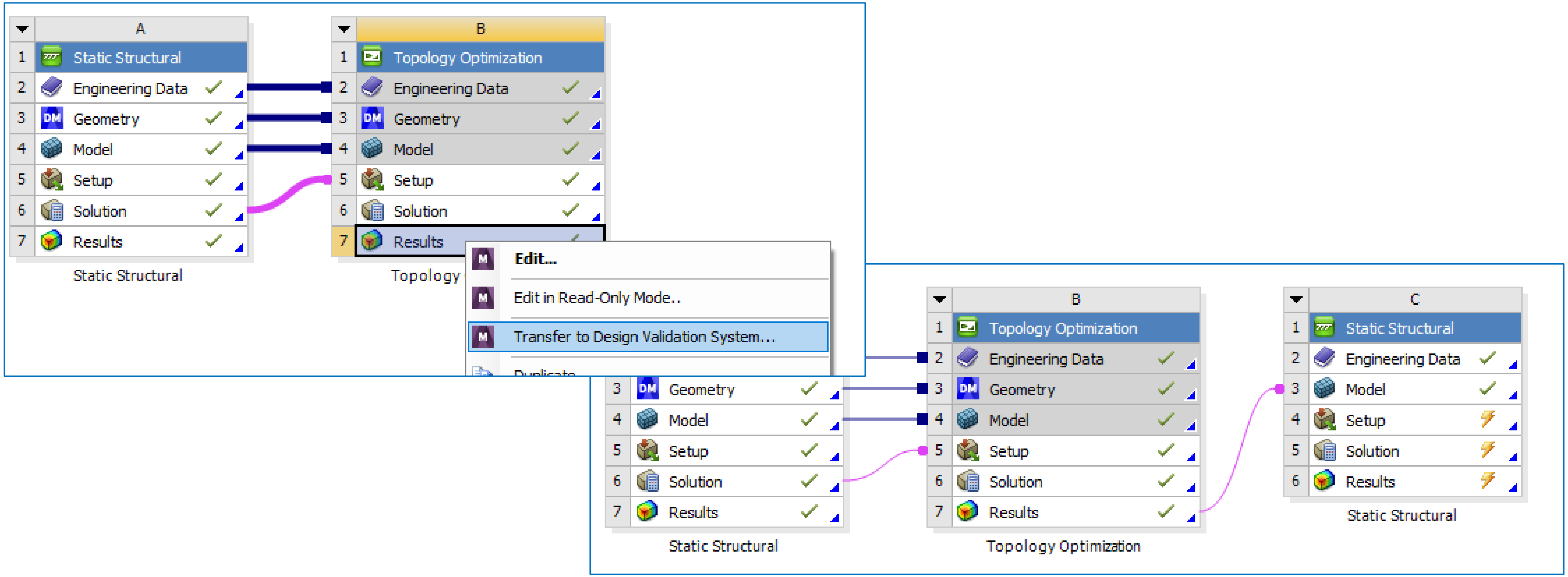
- The ACP user interface is now based on Python 3.7.4. In addition, many third-party software packages have also been upgraded. For this reason, the handling of the Python Shell used for scripting and the appearance change slightly.

- Note that when you are using the Python scripting capability of ACP, you must ensure that your scripts are compatible with Python 3.

Level-set <u>and</u> Density Based Topology Optimization

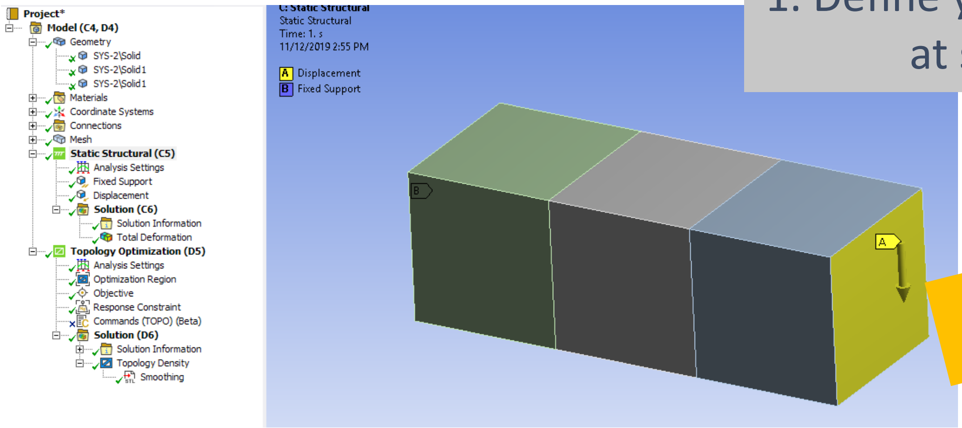
New Topology Optimization Validation Workflow

- Goes directly from tessellated optimal shape to mesh and validation
- Automatic re-scoping of boundary conditions

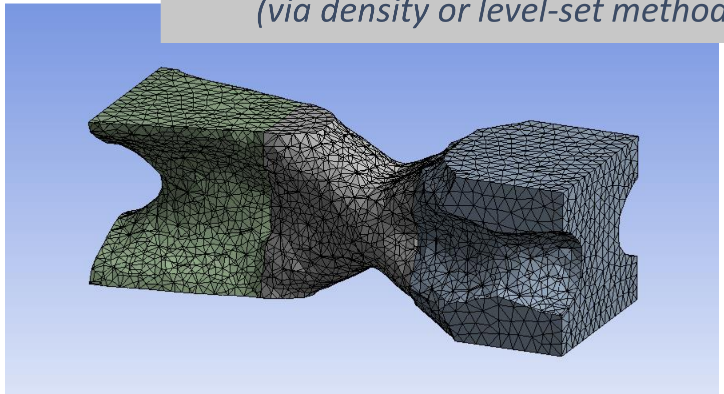


From Topology Result to Validation

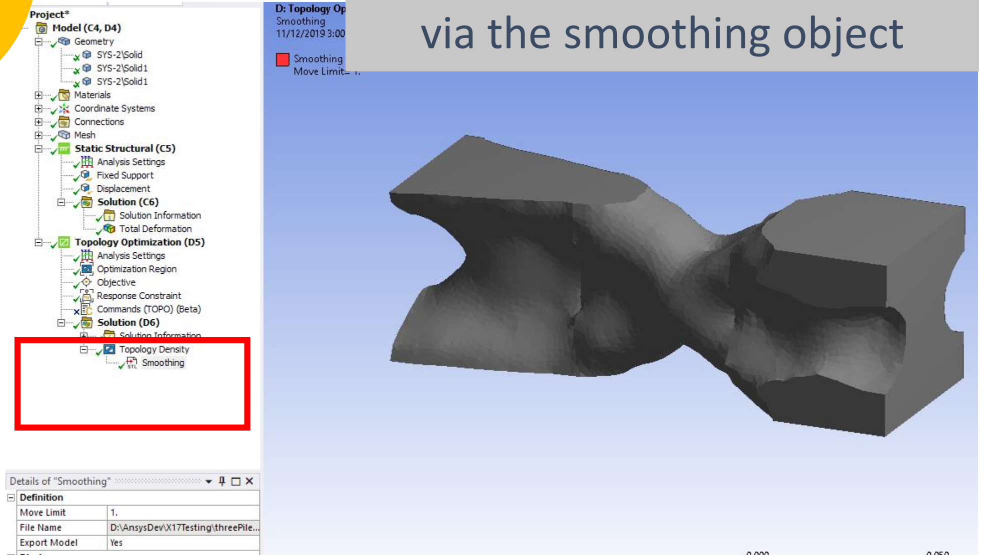
1. Define your analysis at stake



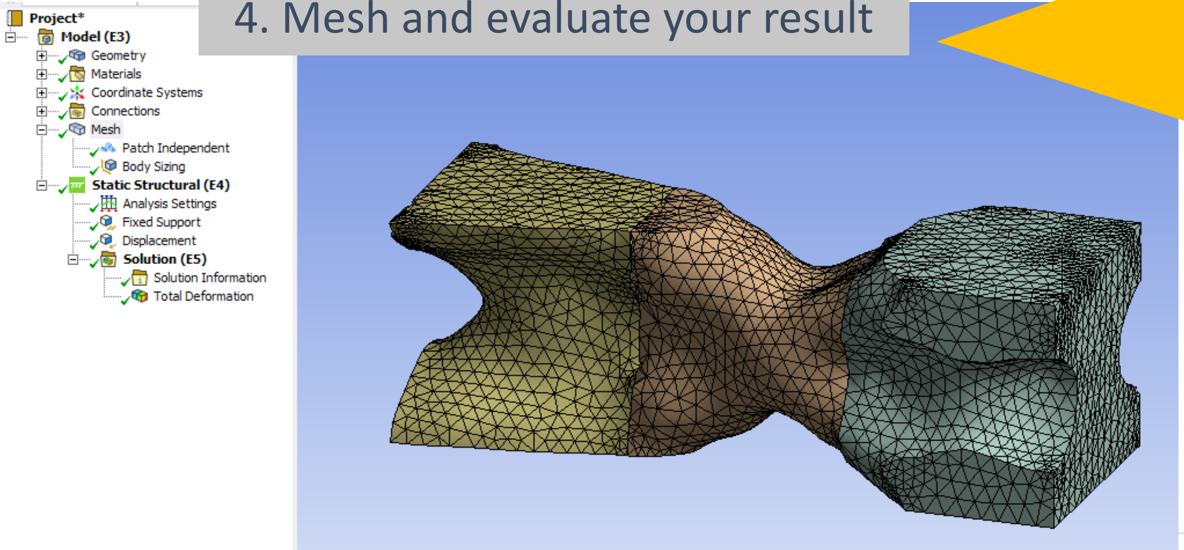
2. Define and solve your topology optimization problem *(via density or level-set methods)*



3. Create and export a model via the smoothing object

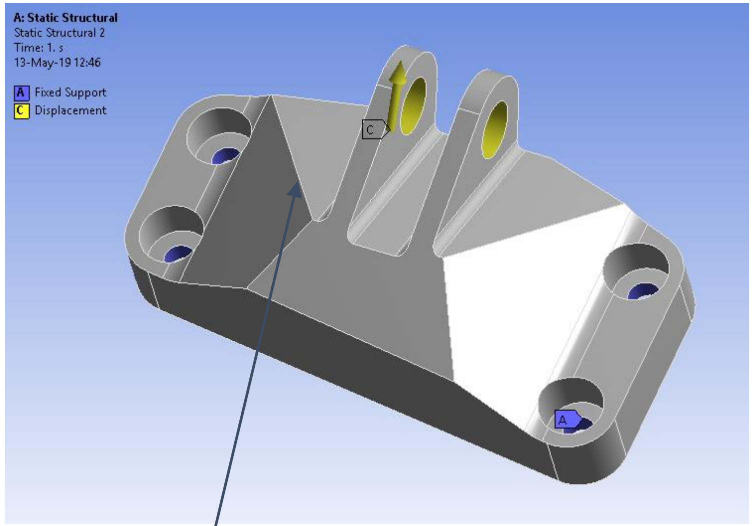


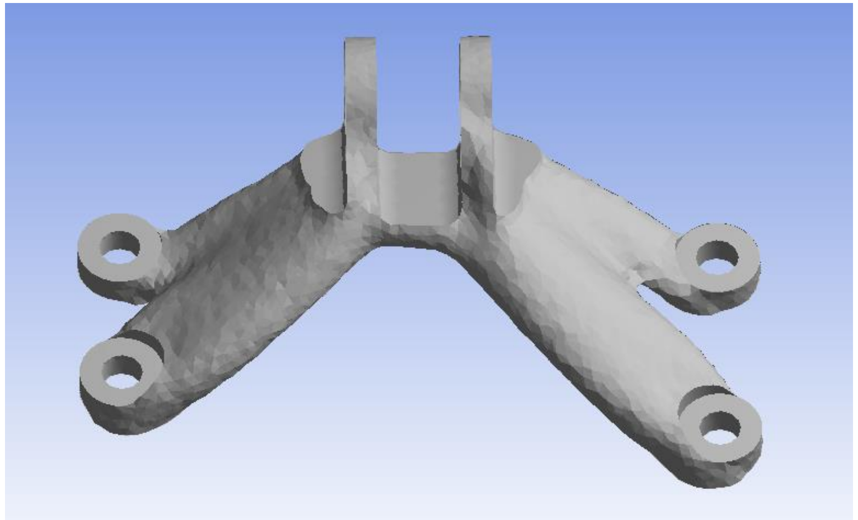
4. Mesh and evaluate your result



Reaction-Force Constraint

- Besides compliance or displacement criterion, the RF is a stiffness criterion that is very convenient, *especially in the context of prescribed displacement*



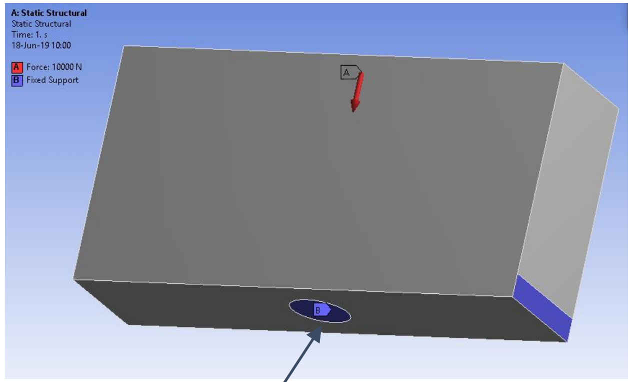


The reaction force is computed over The **FACE A** (ie prescribed disp)

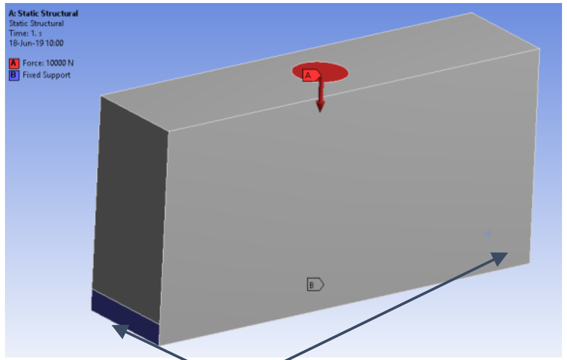
Minimize volume
st reaction force constraint

$$\begin{cases} \min\limits_{\Omega} Vol(\Omega) \\ st : RF(\Omega) = \int_A \lambda_z ds \geq 22kN \end{cases}$$

Reaction-Force Criterion



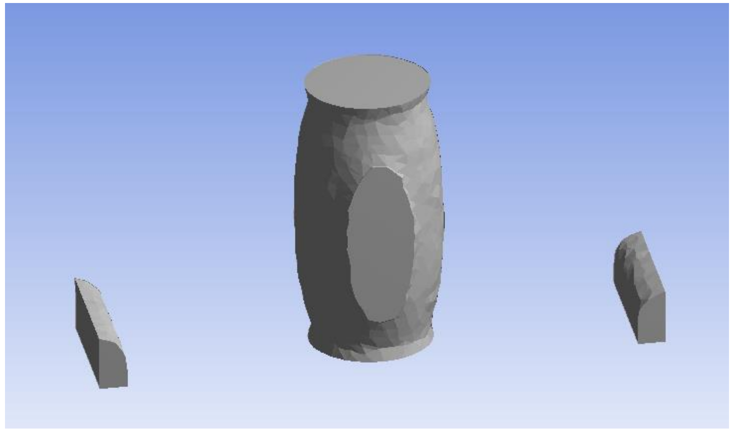
FACE A



FACEs B

min compliance st volume (20%)

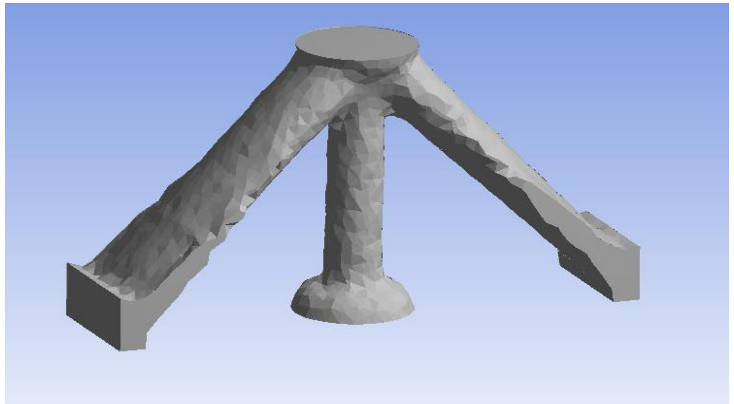
(the clamped parts (B) are not crucial so they have been disconnected)



(a) min compliance st vol and RF

$$\begin{cases} \min\limits_{\Omega} compliance(\Omega) \\ vol(\Omega) \leq 20\%, \quad RF_A \leq 5kN \end{cases}$$

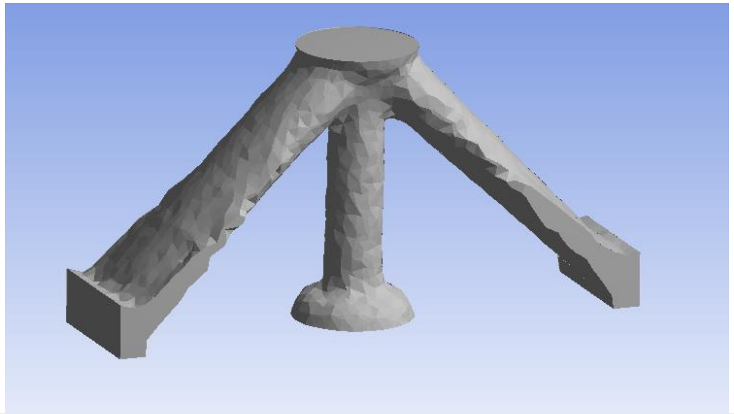
(this constraint aims to limit the force that goes through the face A)



(b) min compliance st vol and RF

$$\begin{cases} \min\limits_{\Omega} compliance(\Omega) \\ vol(\Omega) \leq 20\%, \quad RF_B \geq 5kN \end{cases}$$

(by contrast, this constraint aims to have a minimal force that goes through the faces B)



User Defined Criteria

- Support of *Primary and Composite Criteria*: available with static system, independently of topology optimization
- Can be used in constraint or as objective in both level-set and density-based topology optimization

Static Structural (A5)
- Analysis Settings
- Fixed Support
- Force
- Solution (A6)
  - Solution Information
  - Total Deformation
  - Directional Deformation
  - Deformation Probe
  - Primary Criterion
  - Primary Criterion 2
  - Composite Criterion

Details of "Primary Criterion"

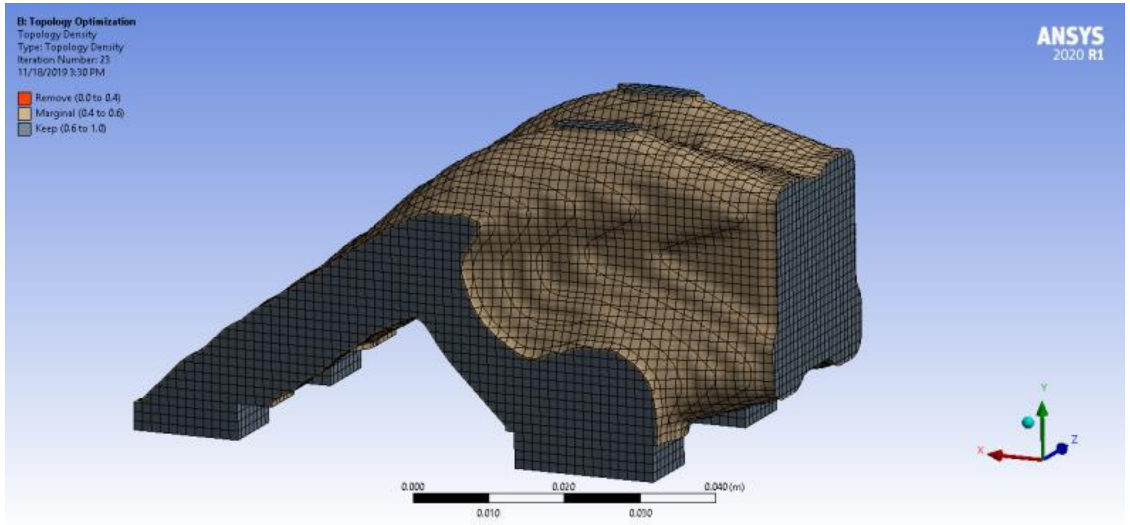
Definition	
Base Result	Displacement
Suppressed	No
Results	
Value	1.4784e-004 m
Scoping	
Scoping Method	Named Selection
Named Selection	Upper Points
Load Step Selection	
Step	1
Vector Reduction	
Coordinate System	Nodal Coordinate System
Vector Reduction	X
Spatial Reduction	
Spatial Reduction	Average
Method	Discrete

Worksheet

Composite Criterion

Add | Delete | Delete All

Primary Criterion Selection	Coefficient
Primary Criterion	1
Primary Criterion 2	-1



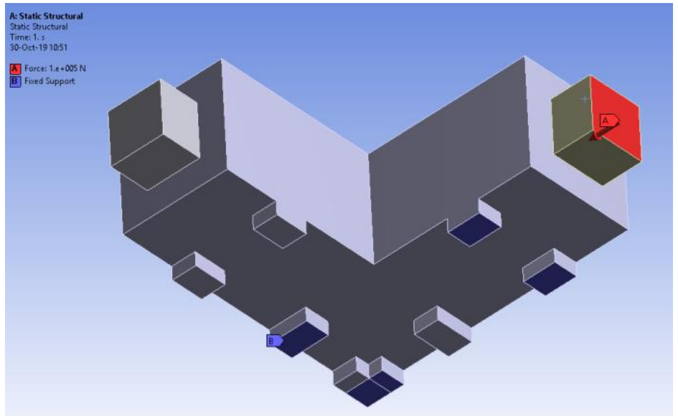
Topology Optimization 2 (C5)
- Analysis Settings
- Optimization Region
- Objective
- Response Constraint
- Response Constraint 2
- Solution (C6)
  - Solution Information
  - Topology Density
  - Displacement Topology Optimization

Details of "Response Constraint 2"

Definition	
Type	Response Constraint
Response	Criterion
Criterion	Composite Criterion
Initial Value	5.4384e-005 m
Lower Bound	Free
Upper Bound	0. m
Suppressed	No

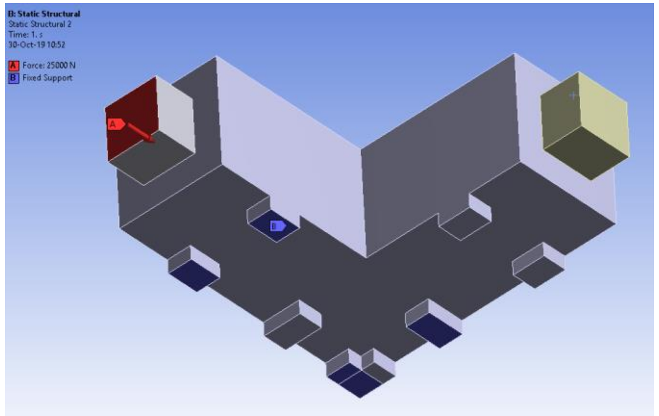
Level-set Topology Optimization

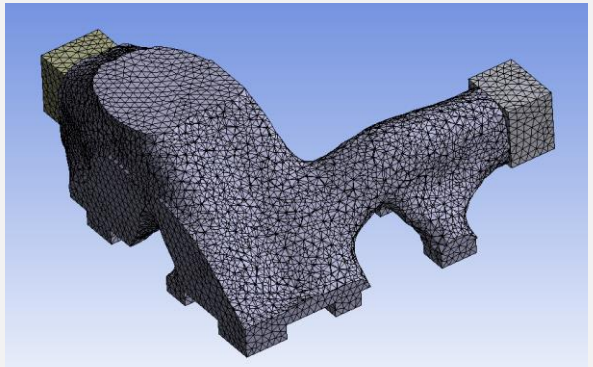
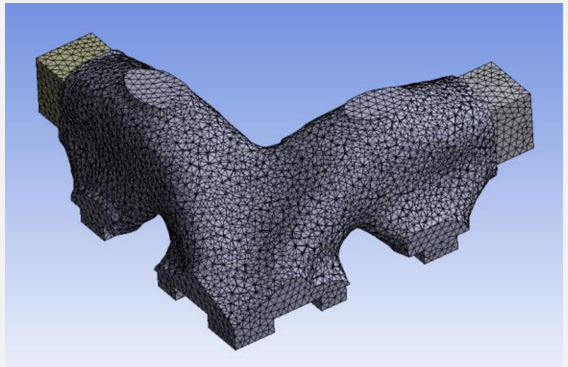
Multi-Objective: Standard versus Normalized Weighted Sum



1st static linear analysis: F=1.00e5 N

2nd static linear analysis: F=0.25e5 N



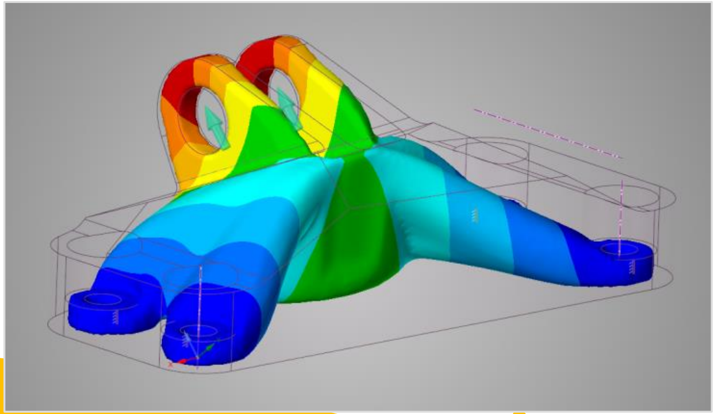
	Standard	Normalized
Description	Summation without any scaling $\sum_i \alpha_i J_i$	Automatic scaling by the initial value $\sum_i \alpha_i \frac{J_i}{\left\lvert J_i^{k=0} \right\rvert}$
Remarks	Relevant when it comes to aggregate the same quantities	Relevant to sum different sort of quantity
Result for the example	 The first load (i.e. the most important) naturally gets more material	 The 2 loads now receive the same amount of material (no matter their magnitude)

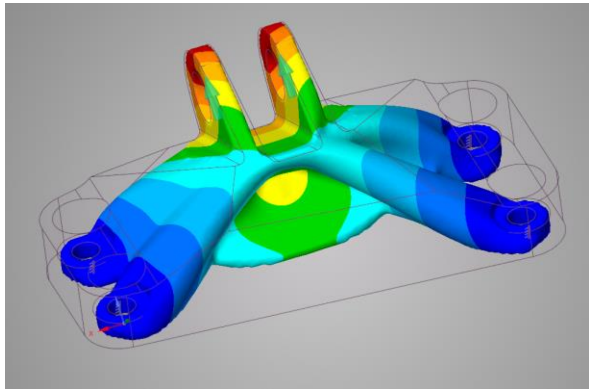
Miscellaneous

- Reaction Force: lower/upper limit
  - Available both for density and level-set methods
- User Defined Function
  - UDF aims to pave the way to a more generic formulation of optimization problem
  - More complex criteria can be created and then used as objective and/or constraint
  - Available both for density and level-set methods
- Overhang
  - Improvements have been made to consolidate the method
- Storage: less space is necessary to perform an optimization
- Multi-objective optimization
  - Standard or normalized formulation is available
- Discovery Live
  - Modal analysis and multi-analysis are now available
  - Max thickness and bi-directional pull-out constraints are also linked

Discovery Live

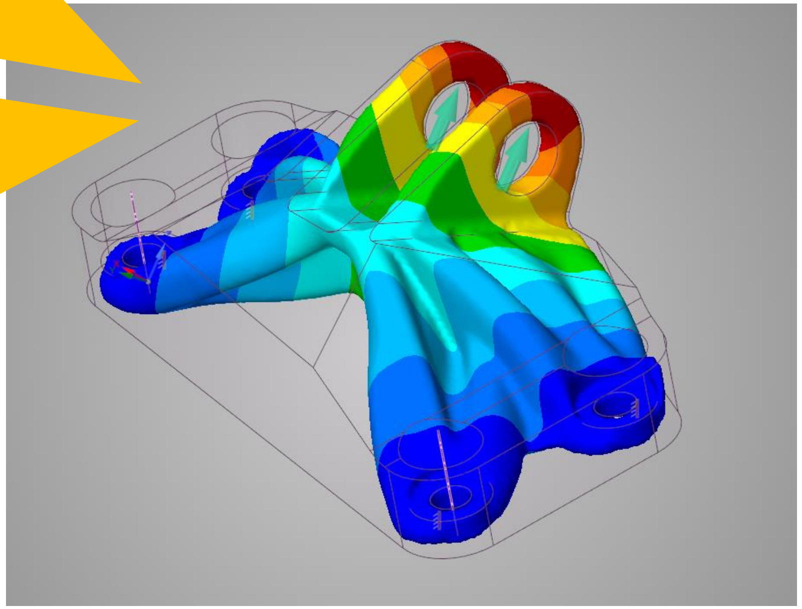
+ Bi-directional pull out constraint

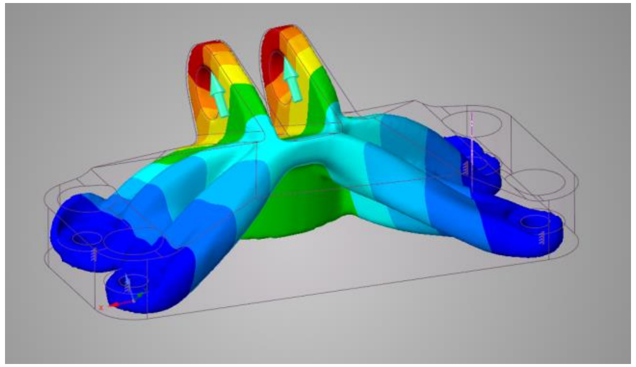




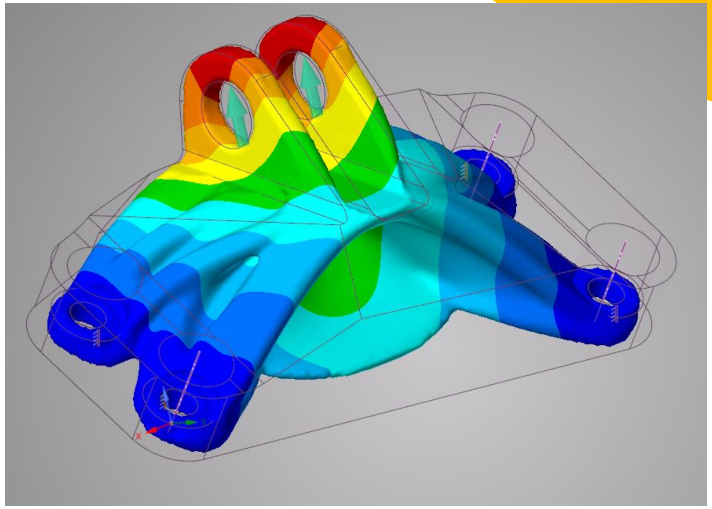
Minimum compliance st volume

Minimum compliance
st volume, max thickness and pull out constraints





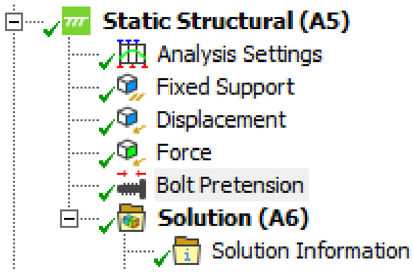
+ Maximum thickness constraint



Density-Based Topology Optimization

Bolt Pretension

- Density-based topology optimization now supports **“Bolt Pretension”** with multiple load steps, where the state is changed from *Load* to *Lock*
- Limitation removed for Lock load steps

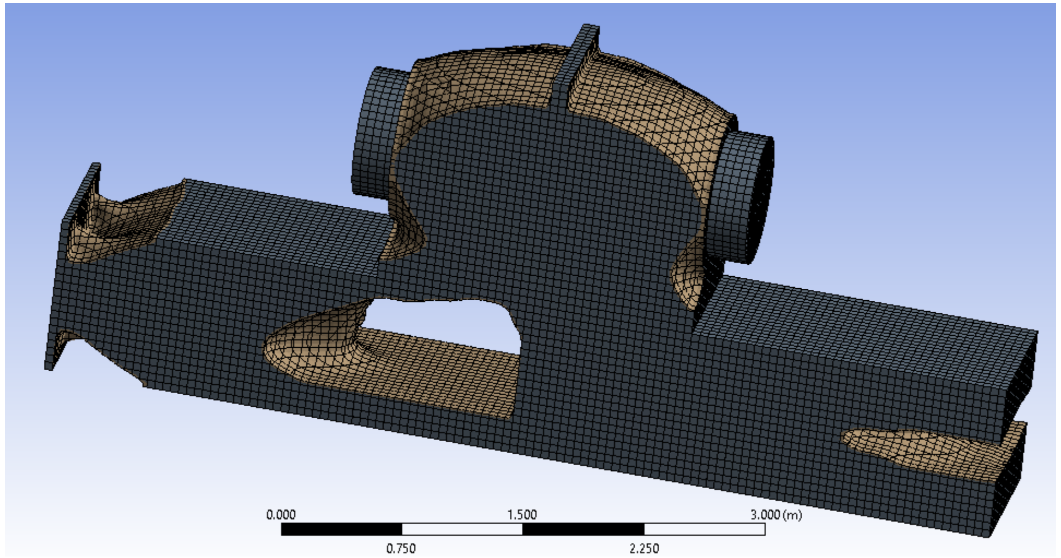


Details of "Bolt Pretension"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
ID (Beta)	72
Type	Bolt Pretension
Suppressed	No
Define By	Load
Preload	1.e+006 N
Advanced	
Solve Behavior	Combined

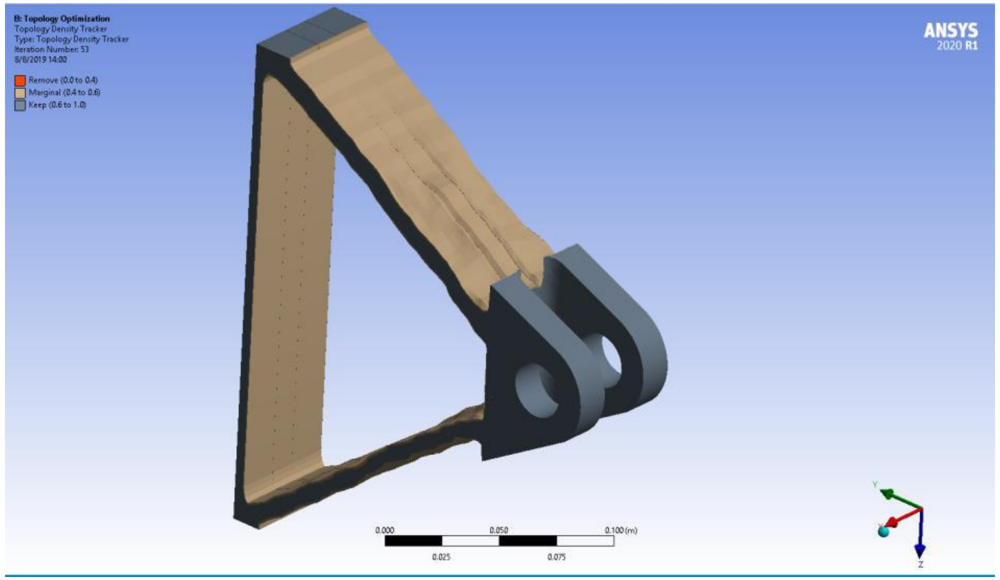
Details of "Bolt Pretension"

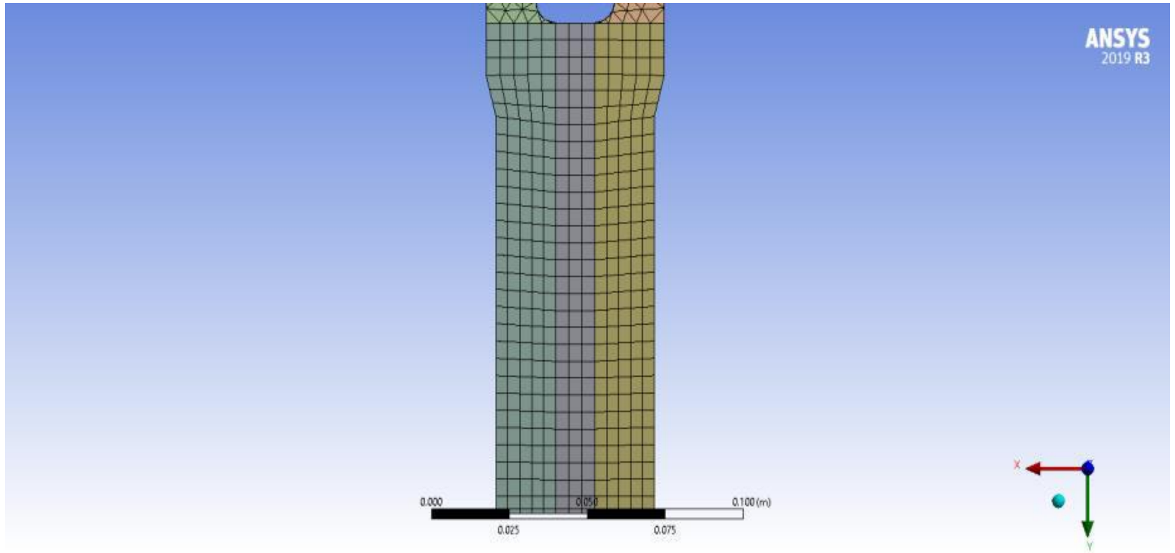
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
ID (Beta)	72
Type	Bolt Pretension
Suppressed	No
Define By	Lock
Advanced	
Solve Behavior	Combined



Extrusion with Less-restrictive Meshes

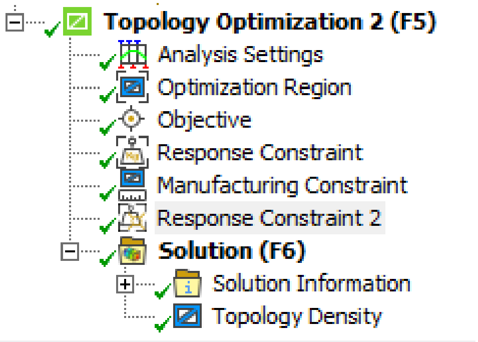
- Extrusion manufacturing constraint is less-restrictive in rejecting meshes that do not satisfy extrusion





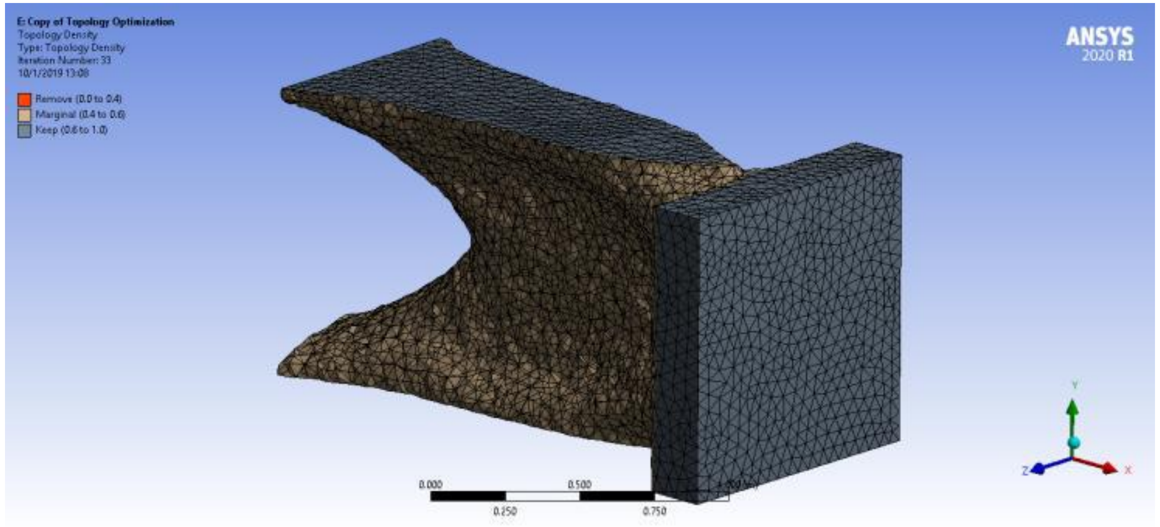
Center of Gravity Constraint (Beta)

- Density-based topology optimization supports restricting the center of gravity (already supported by the level-set optimization)



Details of "Response Constraint 2"

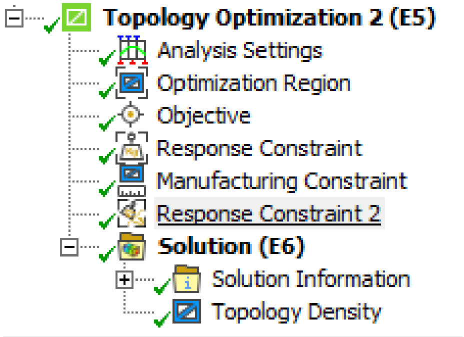
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Response Constraint
Response	Center Of Gravity
Minimum Value	1.25 m
Maximum Value	Free
Suppressed	No
Location and Orientation	
Axis	X Axis



Optimizer puts material on the right to satisfy the CoG constraint

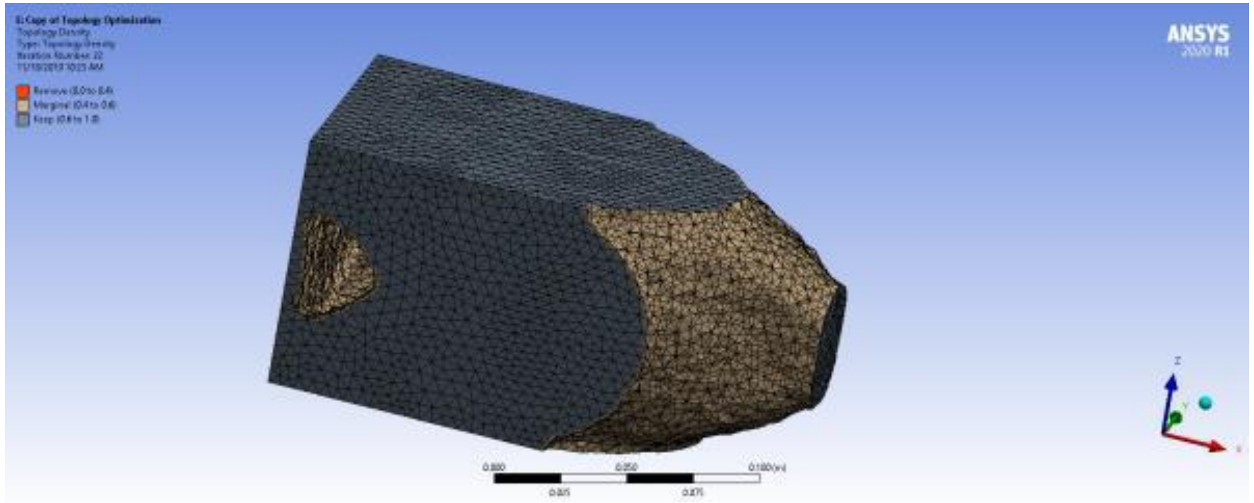
Moment of Inertia Constraint (Beta)

- Density-Based topology optimization supports constraints on the Moment of Inertia (already supported by the level-set optimization)



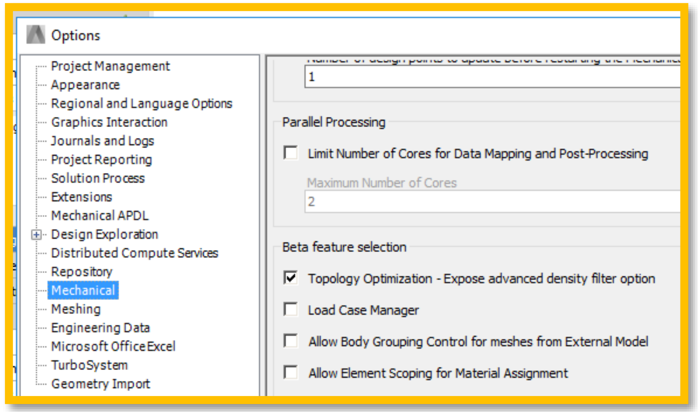
Details of "Response Constraint 2"

Scope	
Scoping Method	Geometry Selection
Definition	
Type	Response Constraint
Response	Moment Of Inertia
Define By	Absolute Range
Minimum Value	1.5e-002 kg·m²
Maximum Value	1. kg·m²
Suppressed	No
Location and Orientation	
Coordinate System	Coordinate System
Axis	X Axis

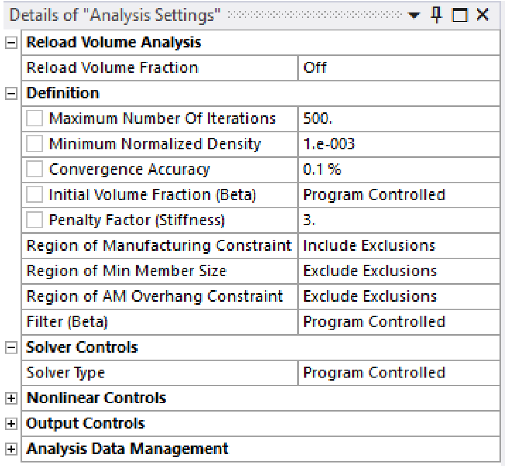


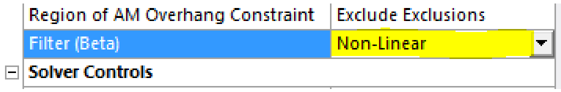
Advanced Filter (Beta)

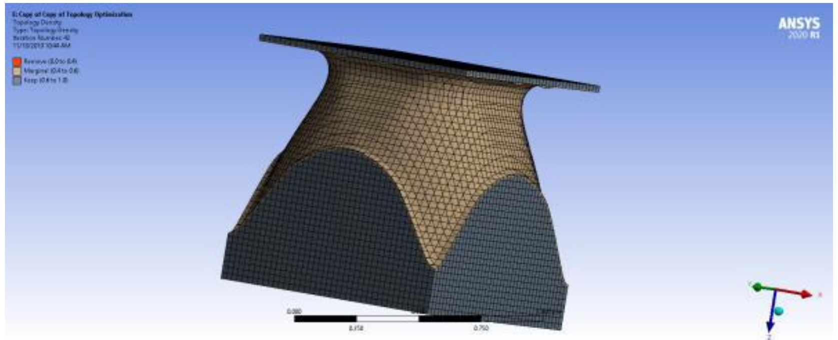
- A new non-linear filter is available to help the optimizer to obtain better defined shapes
- Turn Beta on from WB Project Schematic and then turn on the dedicated beta feature



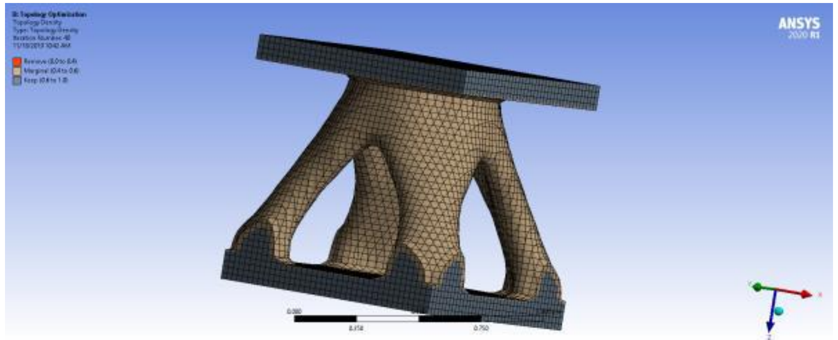
Select **“Non-Linear”** as filter in **“Analysis Settings”**







Default, linear filter



Non-linear filter

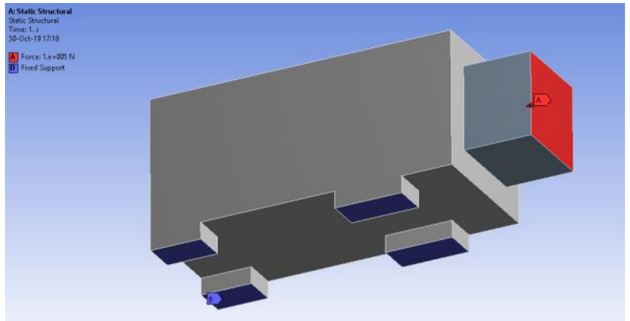
Node-Based Shape Optimization (Beta)

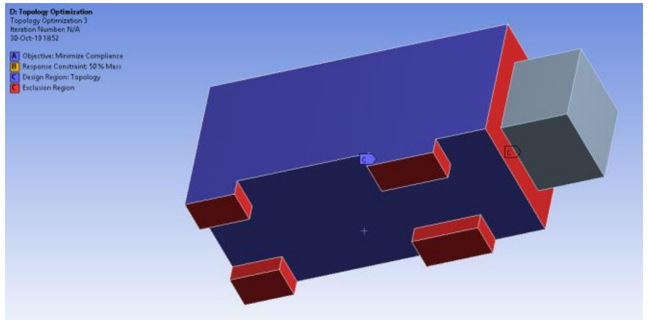
Capabilities of the Node-Based Shape Optimization

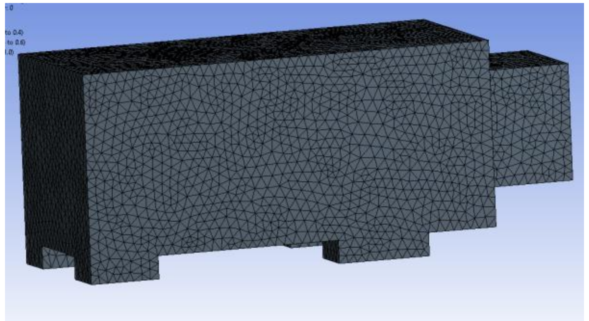
Area	Concern	Features
Geometry	Element type in the optimizable region	3D elements (only tetra) - Linear or Quad
	Element type for the non-optimizable region	No restriction
Geometric Analysis	Criteria	Mass, volume, Center of gravity, moment of inertia
Static Linear Analysis	Boundary condition	Fixed displacement, prescribed displacement
	Loads	Nodal force, surface force (pressure), volume (gravity, acceleration, rotational accel) Thermal load
	Criterion (available for any BC/load)	Generalized compliance Displacement-based criterion Reaction force
Modal Analysis	Boundary condition	None, fixed displacement
	Criterion	Eigenfrequency
Manufacturing Constraints	Pull-out	Not yet
	Overhang	Not yet
	Maximum thickness	Not yet
Optimization	Objective	Single objective. Minimum or Maximization. Any criterion can be selected.
	Constraint	None, one or many. Any criterion can be selected.
	Design variable	Manage multi DV
Miscellaneous	OS	Windows, Linux, RSM

Example: Bull

Optimizable faces in blue



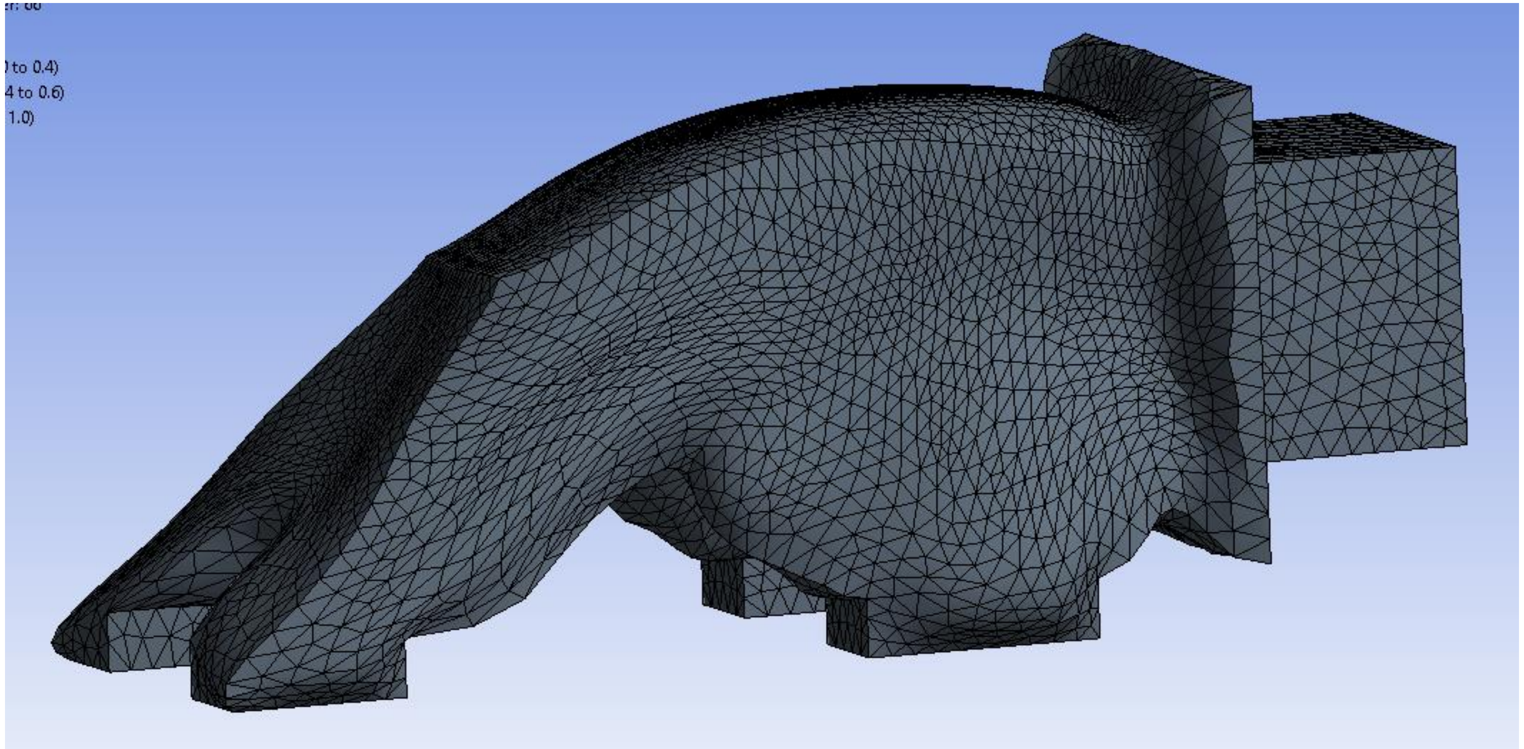




Remarks:
- Static linear analysis
- Only the blue faces are optimizable
- The optim problem reads:

$$\begin{cases} \min_{\Omega} compliance(\Omega) \\ st : vol \leq 50\% \end{cases}$$

Result:
- Large mesh deformation
- Smooth shape



Topology versus Node-based Shape Optimization

	Topology Optimization	Node-Based Shape Optimization
Description	• Immersed boundary method: i.e. the shape is approximated thanks to a density-field or a level-set method • The mesh is fixed	• Body-fitted method: the shape is exactly defined • The mesh is not fixed anymore, nodes location will change
Strength	• Manages topology changes (nucleation, merge, ..) • Easy setup: crude design domain, nothing to parametrize	• No shape approximation • Accurate computation of local quantities (stress, strain, thickness, etc.) on the boundary • Easy setup: just select the optimizable faces
Weakness	• The shape is approximated • Local quantities are not accurately computed at the interface (void/solid)	• No topology change • Moderate deformations are expected

Contact, NLAD, Fracture

Contact Enhancements in Mechanical

- **"Advanced Analysis Settings"** now supports a new setting, *Contact Split (DMP),* that allows for better solver performance in distributed mode. When turned on, the solution process of models involving large number of contact elements speeds up. This is achieved by distributing the contact calculations across specified number of cores and improving the load balance ratio. The default for Contact Split (DMP) option is set to off.

Details of "Analysis Settings"

Large Deflection	On
Inertia Relief	Off
Rotordynamics Controls	
Restart Controls	
Nonlinear Controls	
Advanced	
Inverse Option	No
Contact Split (DMP)	Off
Output Controls	Program Controlled On Off
Stress	
Surface Stress	No
Back Stress	No
Strain	Yes

Cantilever Beam Model with number of contacts = 228

Solution time with 12 cores, without contact splitting

```
CP Time      (sec) =        15.766
Elapsed Time (sec) =        24.000
```

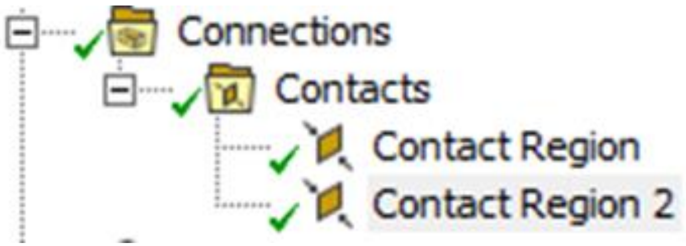
Solution time with 12 cores, with contact splitting, Number of maximum splits per contact = 12

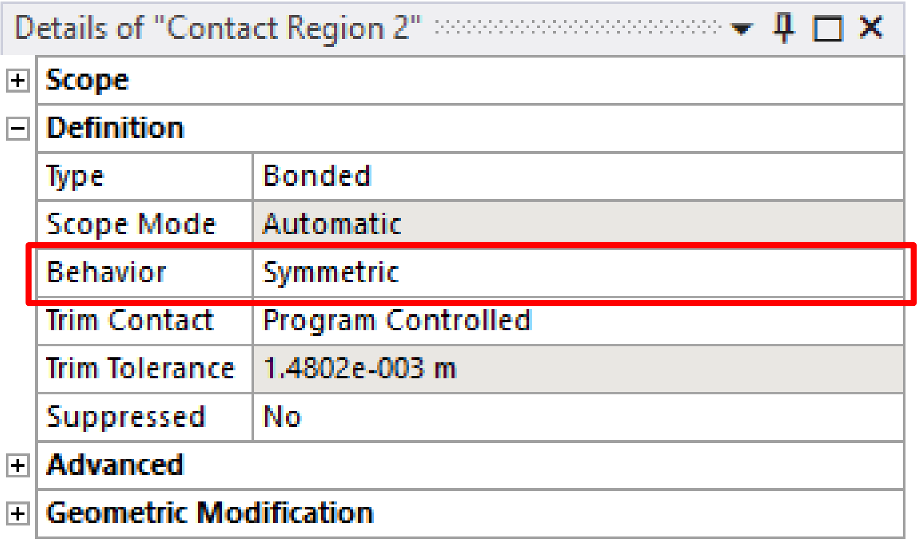
25 % less Elapsed Time

```
CP Time      (sec) =        14.062
Elapsed Time (sec) =        18.000
```

Contact Enhancements in Mechanical

- Symmetric Contacts Similar Characteristics
  - The **"Contacts"** for which the *Behavior* is specified as *Symmetric* in Mechanical, the contact will now keep the same contact characteristics for symmetric pairs (KEYOP(8)=1) as opposed to previous behavior, where each contact pair had its own contact characteristics. This helps the users with much better results for symmetric contacts





Contact Enhancements in Mechanical

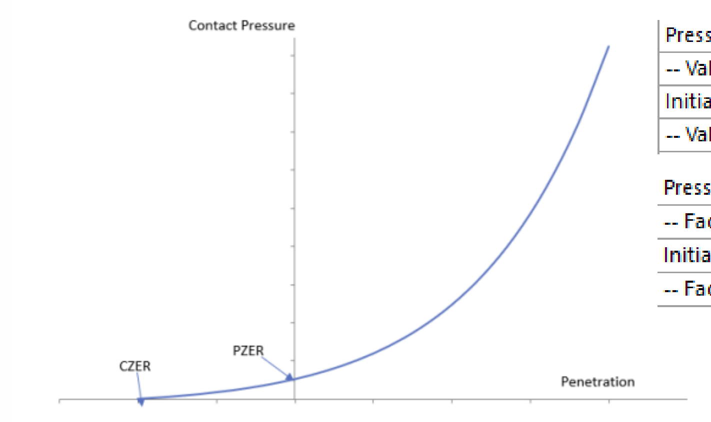
- Mechanical now supports a new option for contact region (Each Iteration, Exponential) in **"Update Stiffness"**
- This option is only valid for Frictional/Frictionless contact with Pure Penalty formulation
- This option will update contact stiffness based on exponential pressure-penetration relationship

Details of "Contact Region"

Definition	
Type	Frictional
Friction Coefficient	0.2
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	1.1979e-003 m
Suppressed	No
Advanced	
Formulation	Pure Penalty
Small Sliding	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	Program Controlled
Pinball Region	Never
Time Step Controls	Each Iteration
Geometric Modification	Each Iteration, Aggressive
Interface Treatment	Each Iteration, Exponential
	Add Offset, No Ramping

- Once this option is selected, two more properties appear:
    - Pressure At Zero Penetration → PZER in MAPDL
    - Initial Clearance → CZER in MAPDL

Figure 3.13: Pressure-Penetration Relationship



Pressure At Zero Penetration	Value
-- Value	2. MPa
Initial Clearance	Value
-- Value	0.1 mm

Pressure At Zero Penetration	Factor
-- Factor	1.6
Initial Clearance	Factor
-- Factor	2.e-002

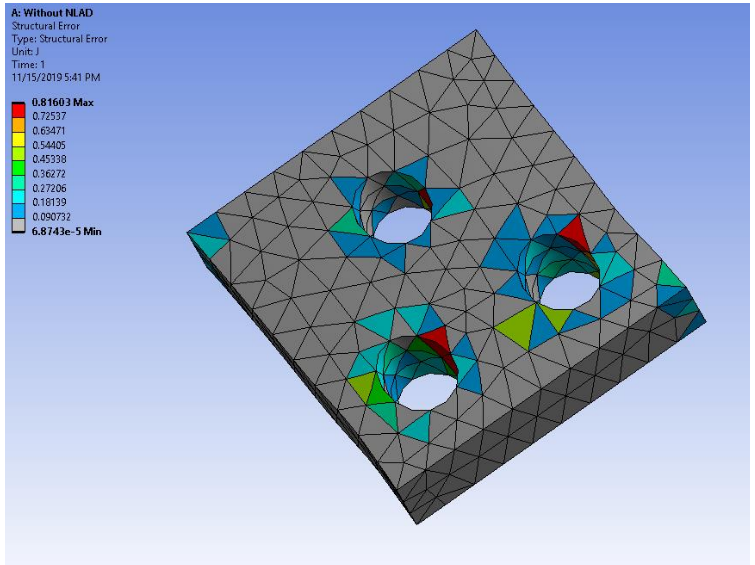
- Both the properties have three dropdown options:
    1. Program Controlled (default): Solver computes the default values
    2. Value: User can define any positive value
    3. Factor: User can define the factor of solver computed default

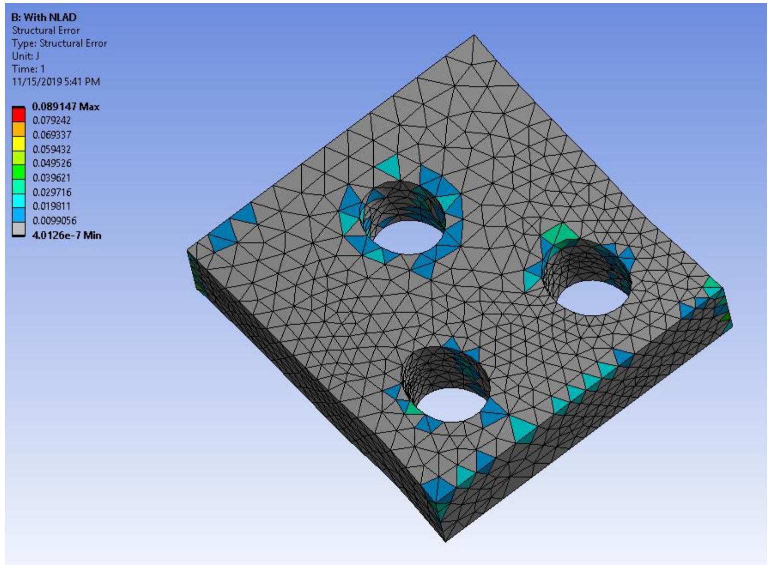
Update Stiffness	Each Iteration, Exponential
Pressure At Zero Penetration	Program Controlled
Initial Clearance	Program Controlled

Program Controlled
- Program Controlled
- Value
- Factor

NLAD Enhancements in Mechanical

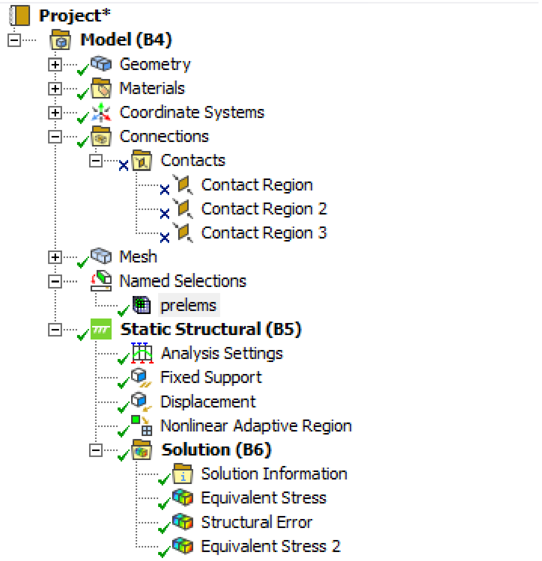
- Mechanical now supports Non-linear Adaptivity Region with Large Deflection off
- This can be useful in situations where deformations are not large, but structural errors are large and can be corrected by adaptively refining mesh
- Mechanical will now also overwrite restart files in NLAD, after maximum number of files reaches 999

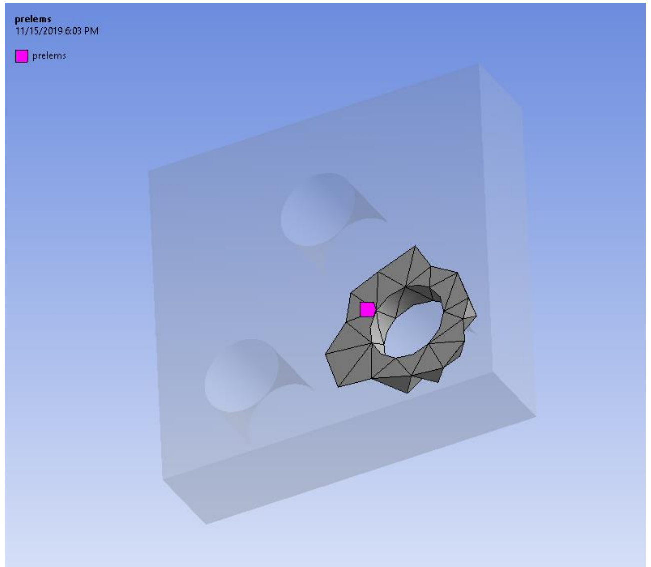


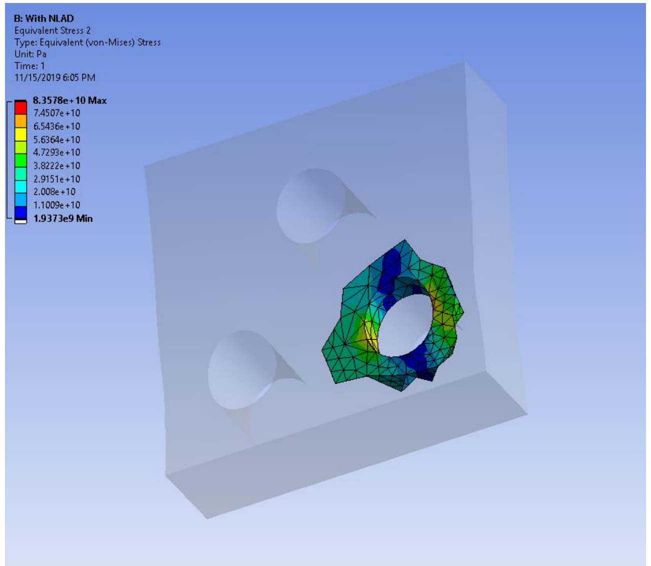


NLAD Enhancements in Mechanical

- **“Preserve During Solve (Beta)”** in 2020 R1
    - Mechanical now allows to preserve named selections during NLAD solve. This can help a user to evaluate results on the elemental named selection region, which was defined before the adaptive refinement





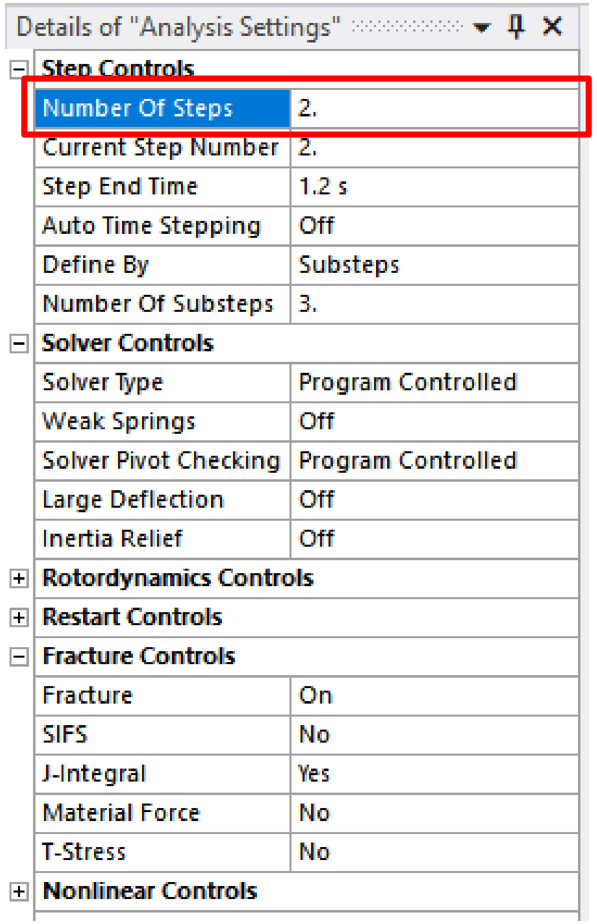


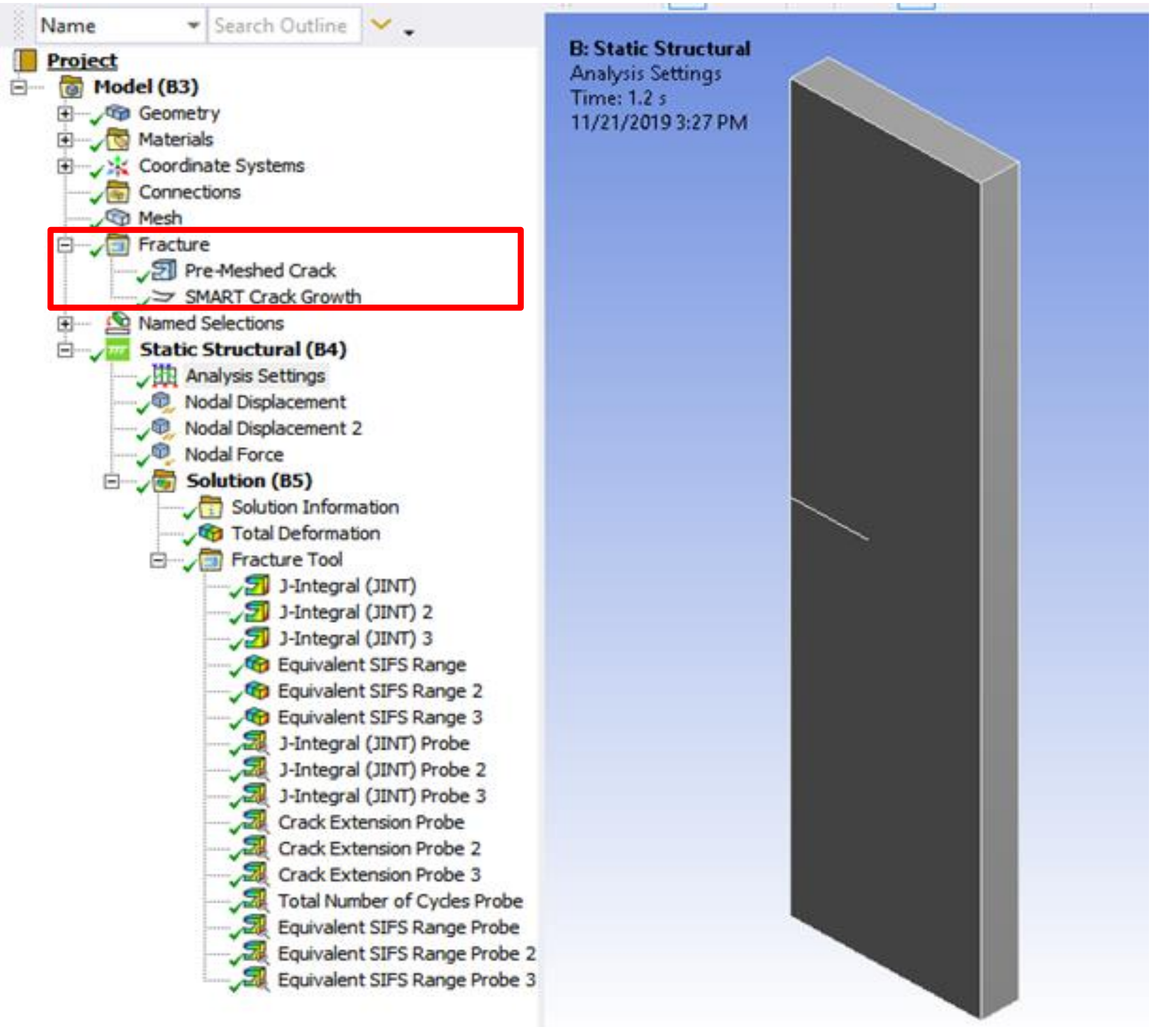
Details of "prelems"

Scope	
Scoping Method	Geometry Selection
Geometry	30 Elements
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Preserve During Solve (Beta)	Yes
Statistics	
Type	Manual
Total Selection	30 Elements
Suppressed	0
Used by Mesh Worksheet	No

Fracture

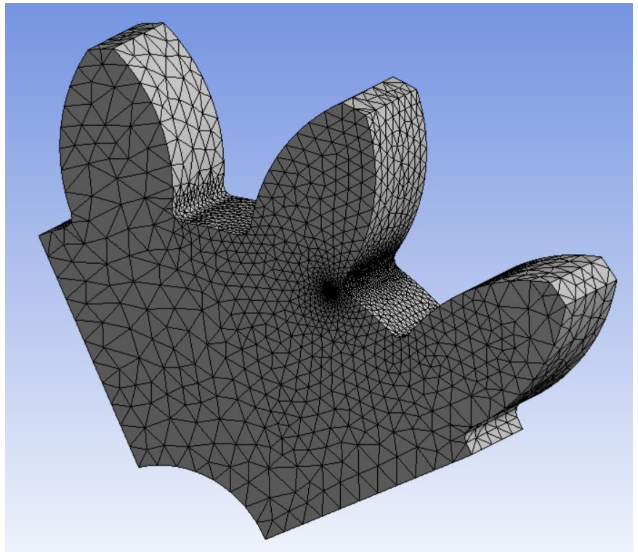
- SMART crack growth now supports multiple load steps



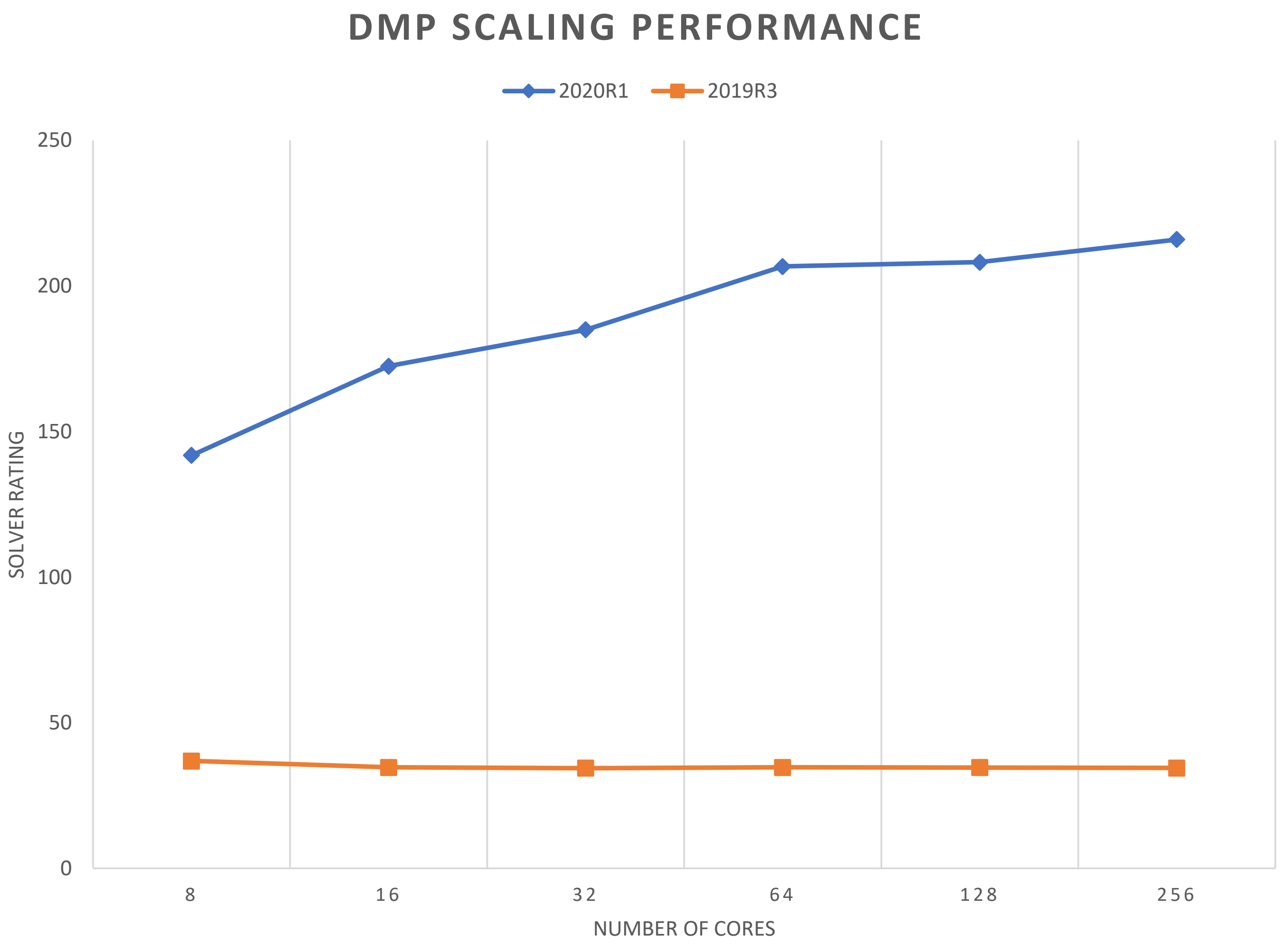


SMART

Distributed ANSYS for Fracture Parameter Calculations

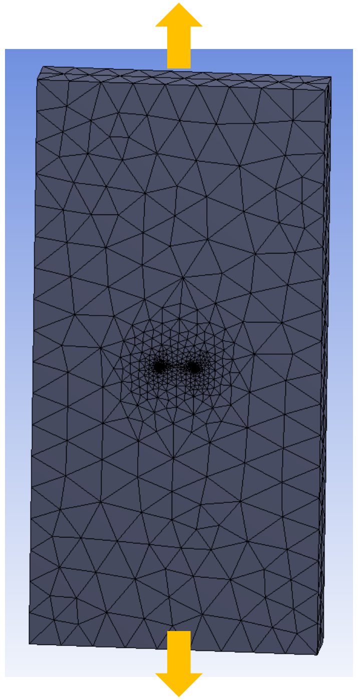


- 2.2 million DOF; PCG solver
- Static analysis with fracture calculation
- Linux cluster
  - CPU: 2x Intel Xeon E5-2690 v4 2.6GHz,35M Cache,9.60GT/s QPI,Turbo,HT,14C/28T (135W) Max Mem speed 2400MHz
  - Ethernet speed: 10Gbps
  - OS: CentOS release 6.7

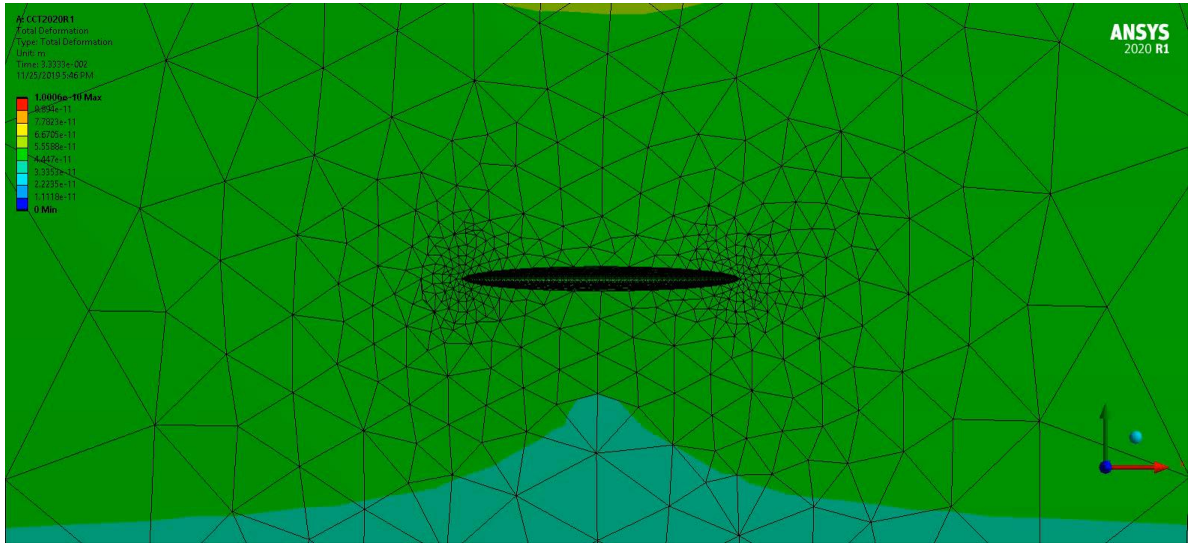


SMART Crack Growth Enhancement

- Robustness Enhancement
  - Continued solver and meshing improvement in remeshing handling
    - Substantially reduce number of elements in the remeshing
    - Improvement in the remeshing success rate
    - Improved remeshing with crack growing into corner
    - Improved remeshing with crack growing cut through part
    - Improved remeshing with crack growing cut through edge
  - Continued improvement in solver solution
    - Fracture parameters calculation
    - Crack direction prediction
    - Crack extension prediction



Problem Description:
- Center cracked tension panel subjected to remote tension pressure load
- Elliptic surface crack
- Fatigue crack growth with Paris Law



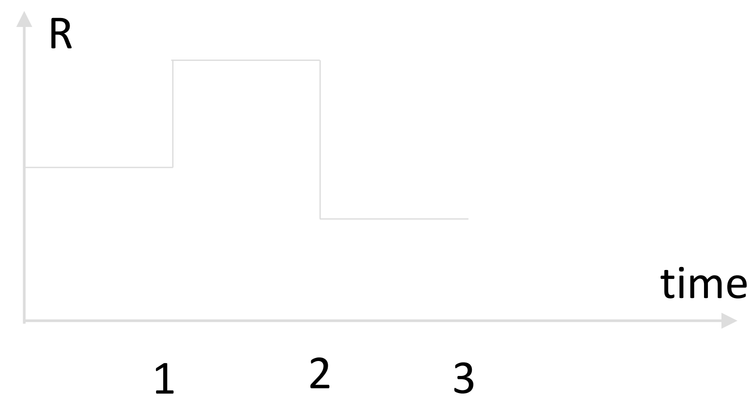
SMART Crack Growth Enhancement

- Tabular stress ratio for fatigue crack growth
  - Define tabular table for stress ratio as function of time

    CGROW,FCG,SRATIO,%rtable%

  - Complex loading pattern can be modeled by using tabular load and tabular stress ratio table

```
*dim,rtable,table,6,1,,TIME     ! R ratio table
rtable(1,0) = 0
rtable(1,1) = 0.3
rtable(2,0) = 1.0
rtable(2,1) = 0.3
rtable(3,0) = 1.0001
rtable(3,1) = 0.5
rtable(4,0) = 2
rtable(4,1) = 0.5
…
```



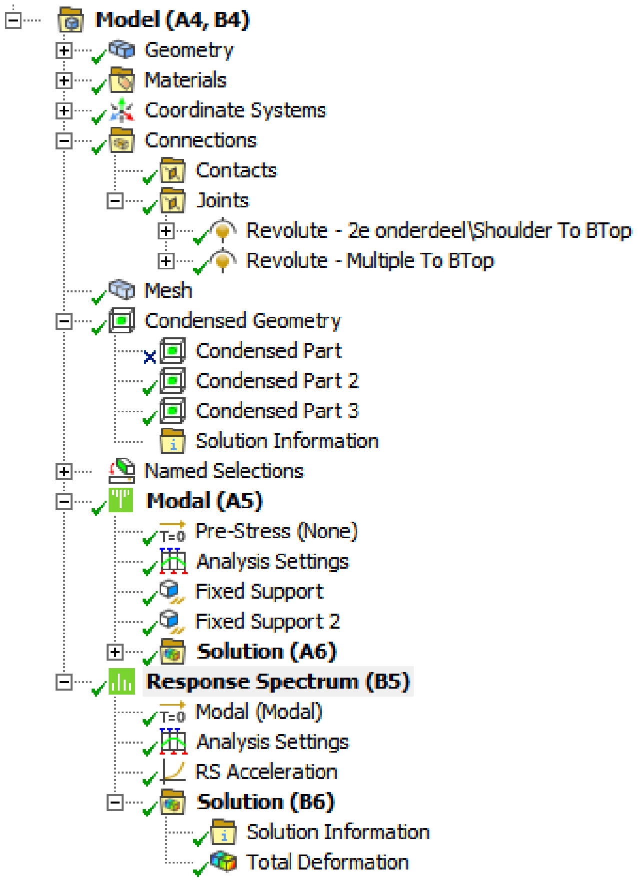
Linear Dynamics

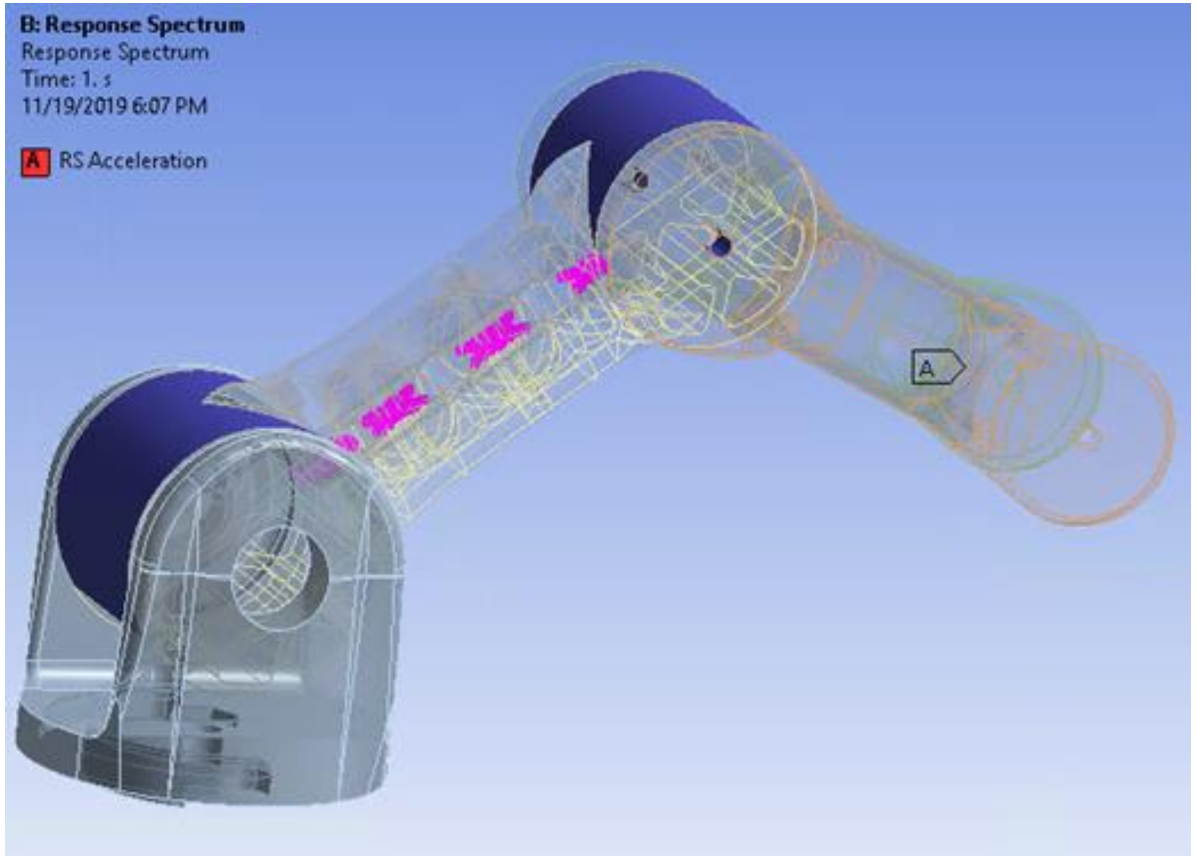
Linear Dynamics Enhancements in Mechanical

- The Linear dynamics features enhancements in 2020 R1 release of mechanical are as follows:
  - Top down CMS model reduction method for Response Spectrum analysis
  - Volumetric Force Density transfer from Maxwell
  - On Demand result calculation for Mode Superposition (MSUP) harmonic and transient analysis

CMS Enhancements in Mechanical

- Top down CMS based method for generation of super elements is now supported for Response Spectrum analysis



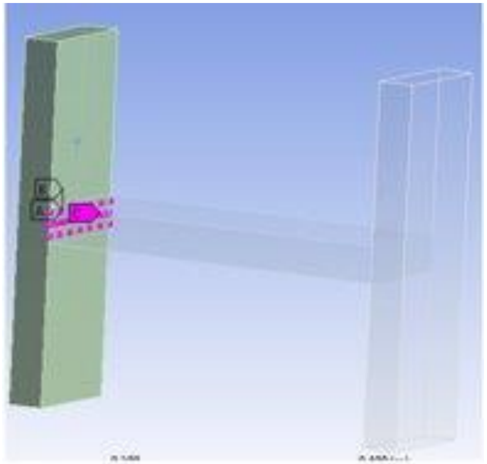


CMS Enhancements in Mechanical

- When **"Trim Contact"** option of *On* is selected, then the master degrees of freedom is also trimmed at the contact interface during generation pass. This leads to a reduction of the master degree of freedom and will improve the solution times of the generation pass

Details of "Contact Region 2"

Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	On
Trim Tolerance	Program Controlled
Suppressed	On
Advanced	Off
	Face Based (Beta)



Before Trim

Interfaces	
Number of Interfaces	2.
Number of Master Nodes	476.

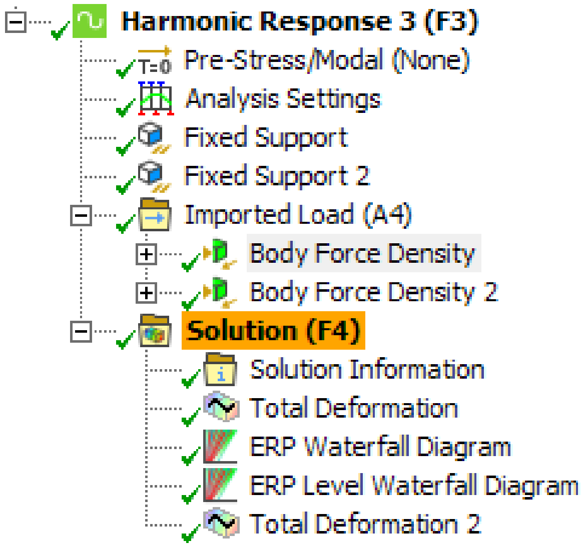
After Trim

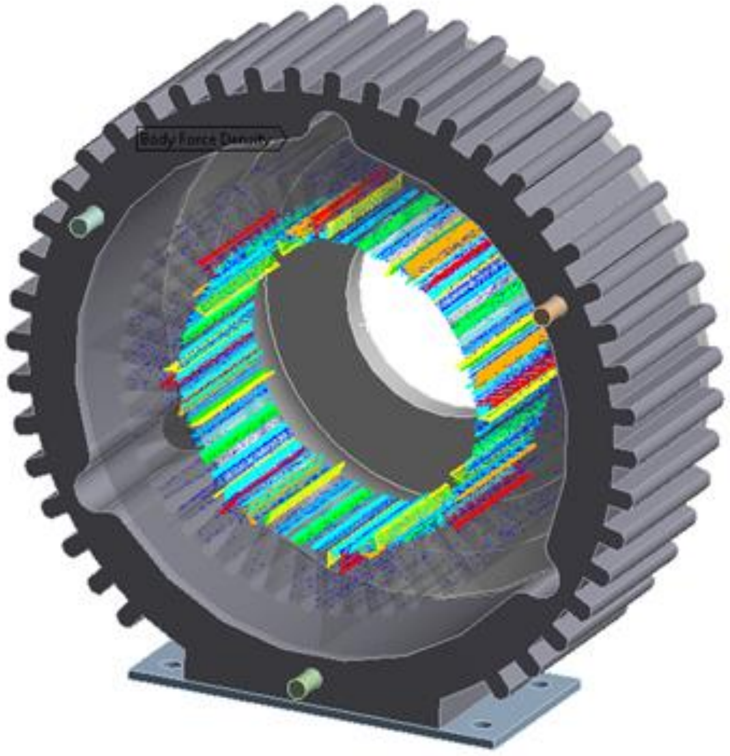
Interfaces	
Number of Interfaces	2.
Number of Master Nodes	80.
Automatic	Bonded - rectbar3 To rectb...
Automatic	Fixed Support

Statistics	
Number of Master Nodes	80.

Volumetric Force Density Transfer from Maxwell

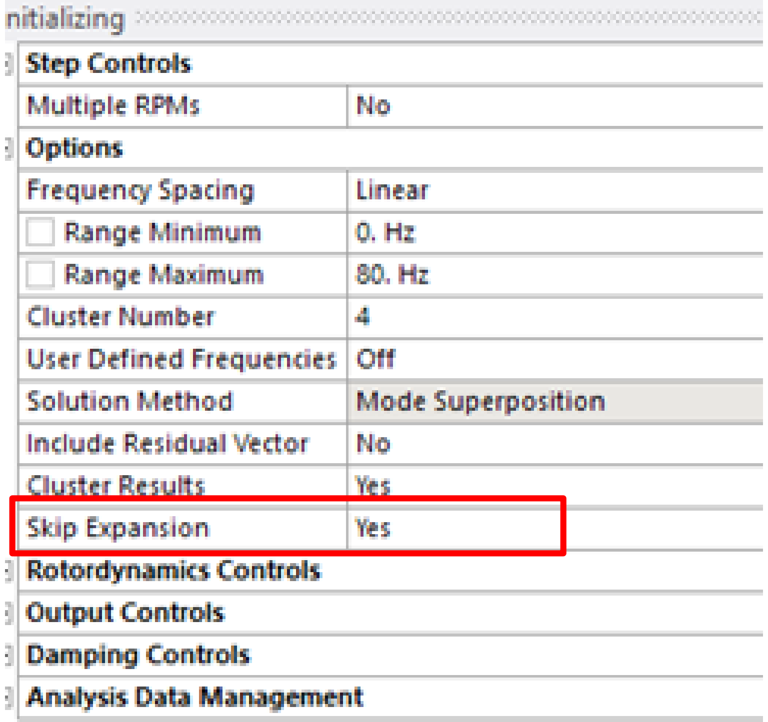
- Support frequency varying body force density in FULL harmonic
- Applications: Electric Transformers, Electric Motors





On Demand Result Calculation in MSUP

- To improve performance, expansion pass can be skipped in MSUP harmonic and transient analysis by using *Skip Expansion* setting under **“Analysis Settings”**
- Displacement, Velocity, Acceleration, Stress, Strain and ERP can be evaluated on demand in this case saving solution time and I/O
- Residual vector are supported
- Using *Skip Expansion* option, the solution times and I/Os can be highly improved as shown below (numbers obtained with medium size model)



Modes	Frequencies	Time Standard	Time Skip Expansion	IO Standard	IO Skip Expansion
100	100	245	94	13.2	5.4
100	200	402	97	20.9	5.4
100	400	729	100	36.4	5.4
100	1000	1669	101	82.7	5.4
1000	1000	6073	867	130.2	52.9

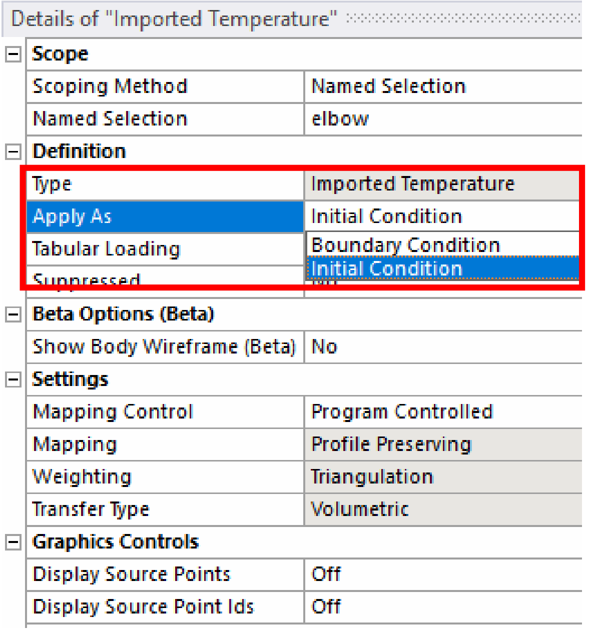
Coupled Field Analysis

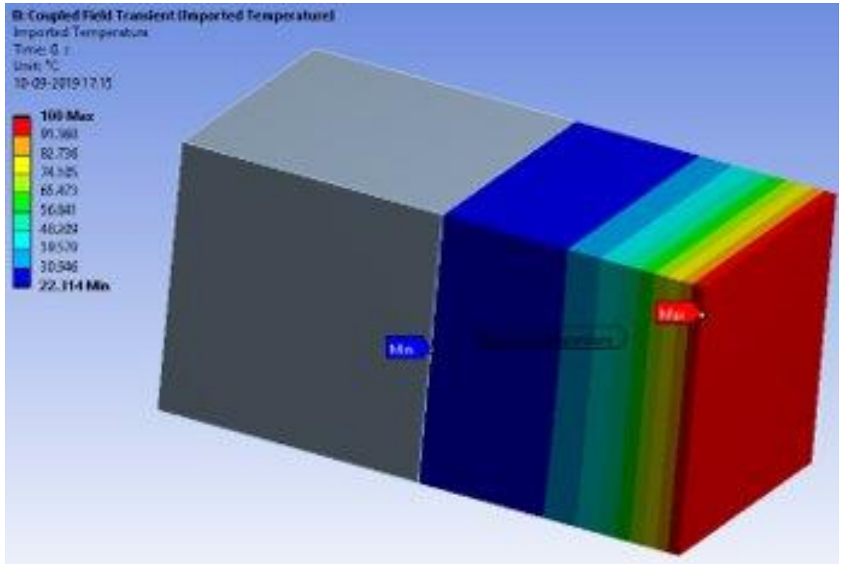
Coupled Field Analysis in Mechanical

- In 2020 R1 release, Mechanical supports these additional features for Coupled Field Static and Coupled Field Transient Analysis
  - Apply Imported Temperature as Initial Condition
  - Supporting External model for Coupled Field Analysis
  - Spot welds
  - Constraint equations and coupling conditions
  - Global temperature Tracker

Coupled Field Analysis in Mechanical

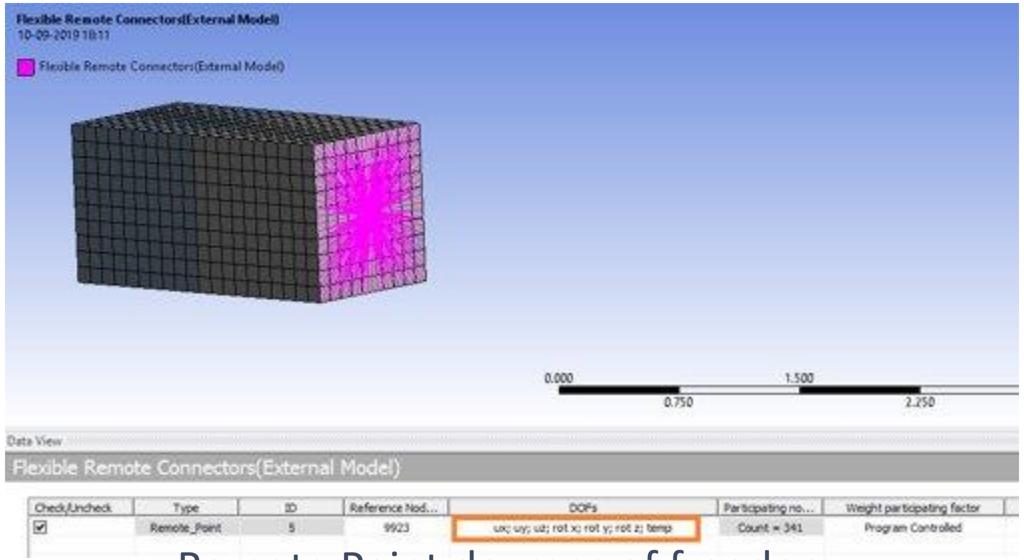
- Mechanical supports option on **“Imported Temperature”** load to apply it as *Initial Condition* or *Boundary Condition*. The default is to apply it as *Boundary Condition*. The **“Imported Temperatures”** were applied as *Boundary Condition* in prior releases when this property was not exposed.



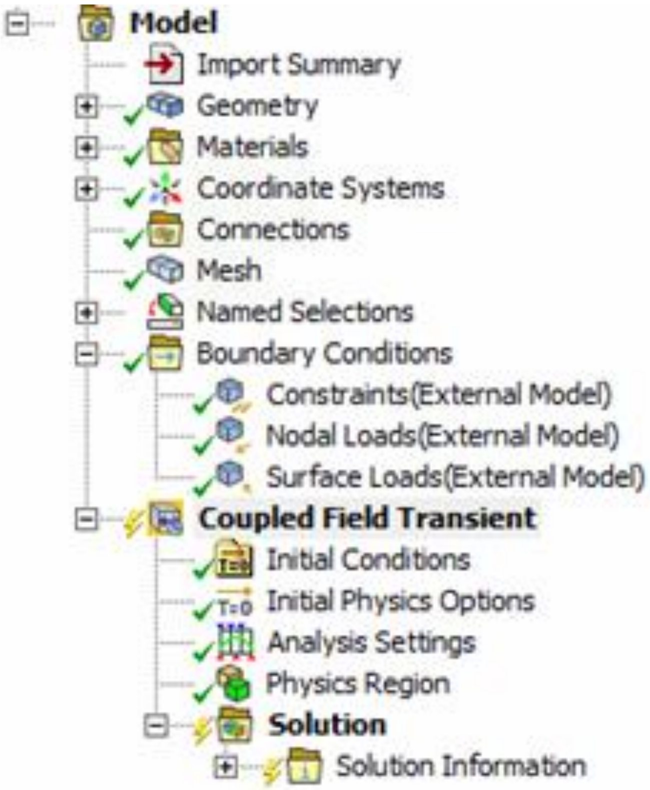


Coupled Field Analysis in Mechanical

- Users can now import CDB, Nastran and Abaqus files into Coupled Field Analysis. Remote point degrees of freedom are picked based on the user selection or physics type of the participating nodes. The missing properties are automatically made invalid to get user's attention (contact thermal conductance)

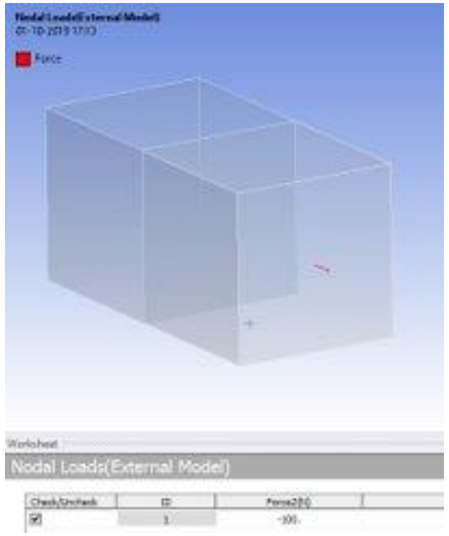


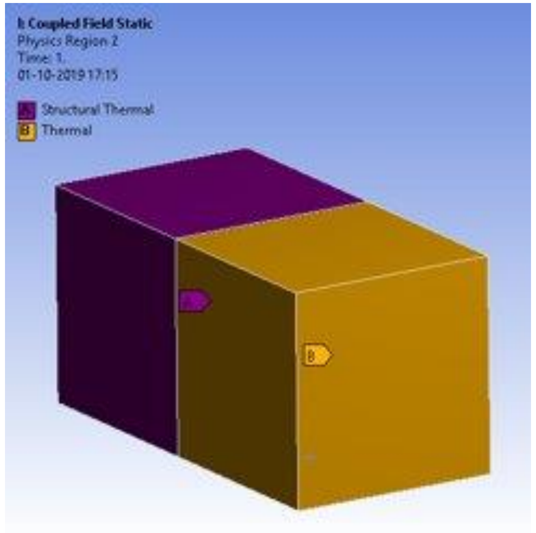
Remote Point degrees of freedom

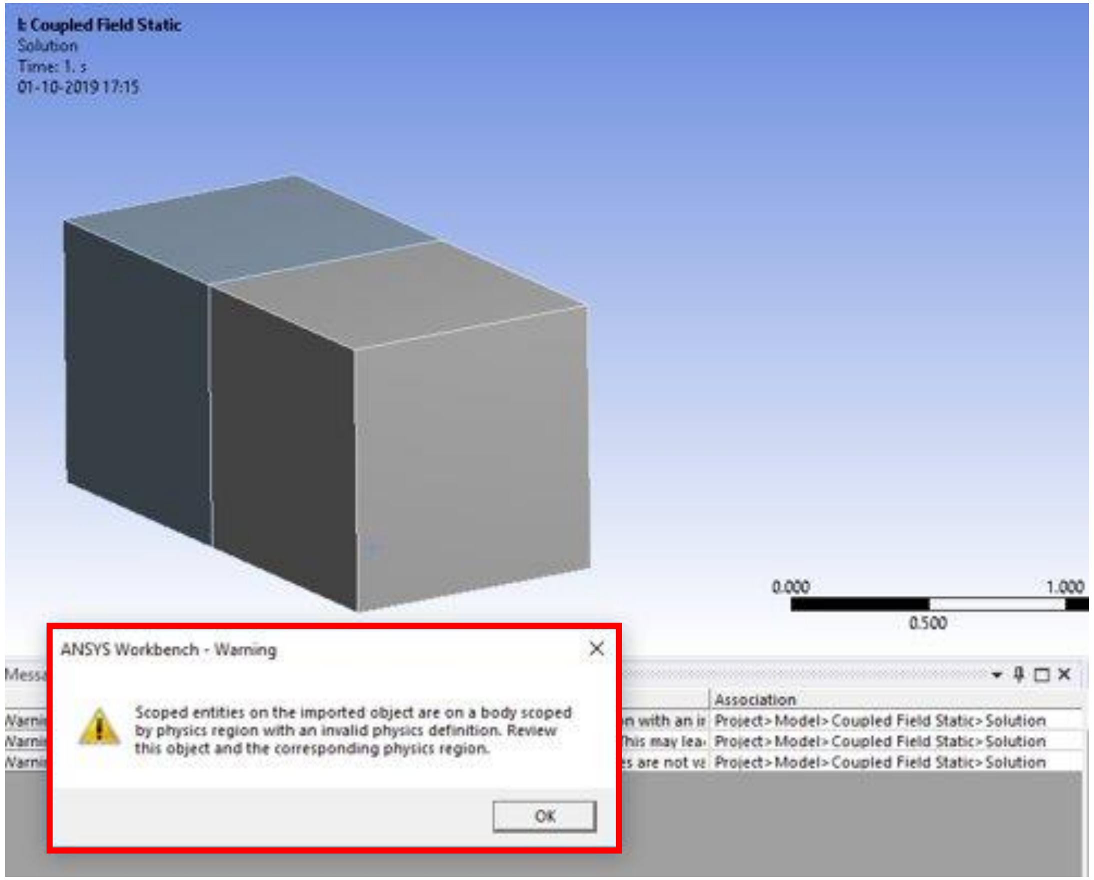


Coupled Field Analysis in Mechanical

- The users will be guided by displaying appropriate warnings when any of the imported objects are not supported by the physics type of the body

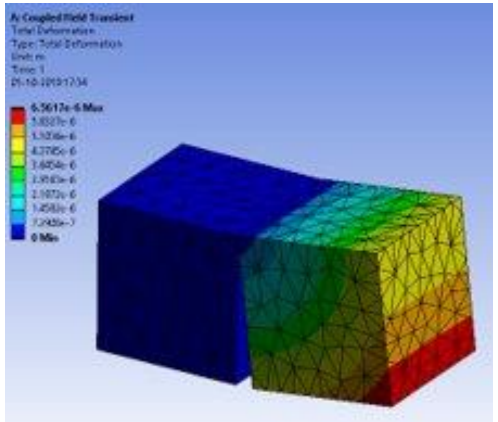


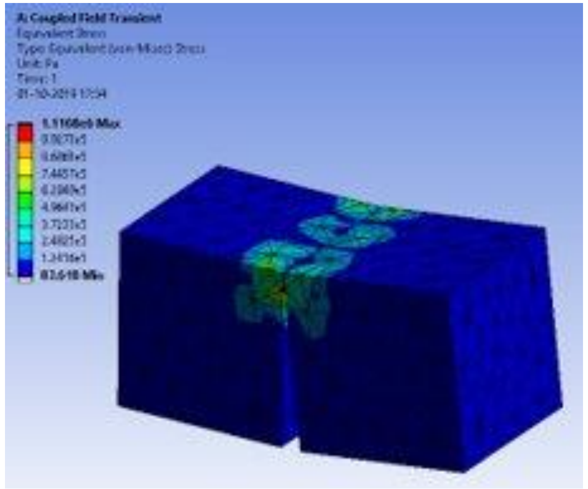


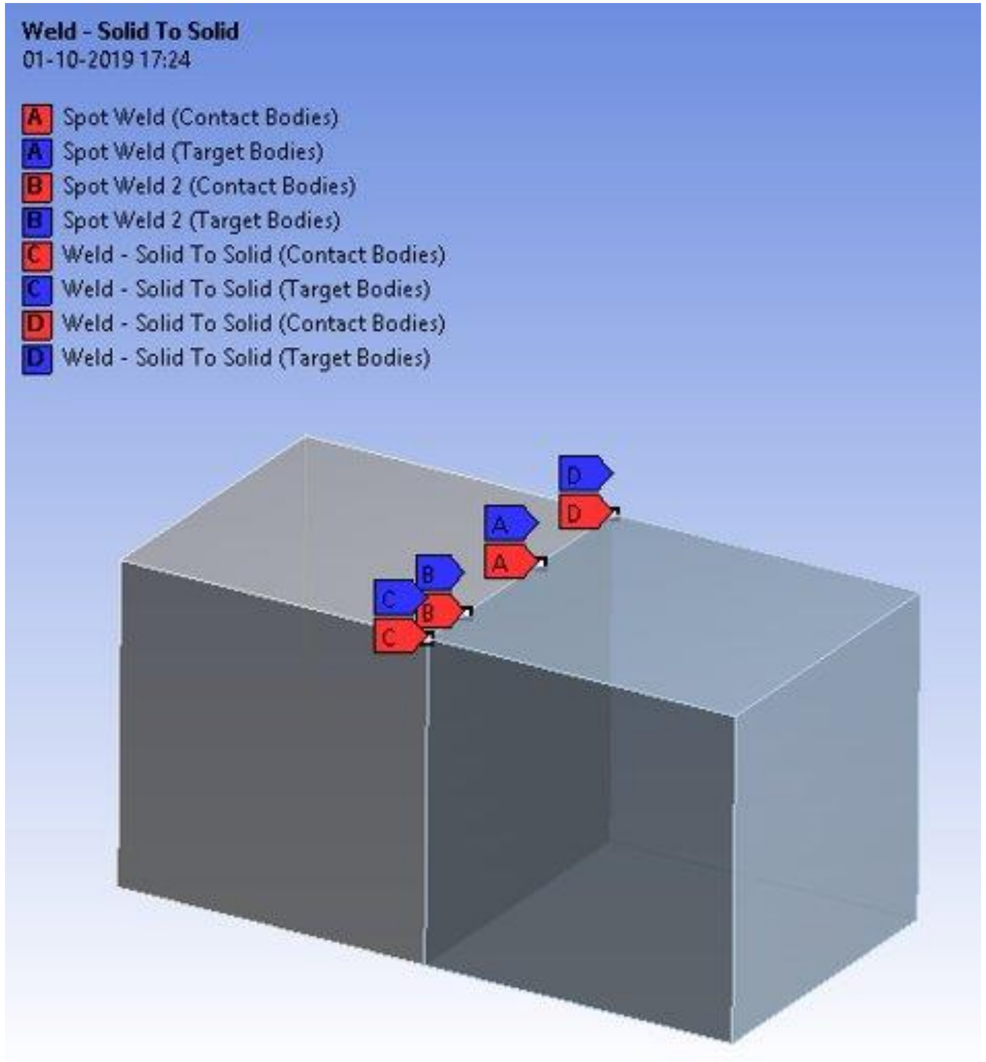


Coupled Field Analysis in Mechanical

- Mechanical supports spot welds for structural only and thermal only contacts in coupled field analysis. It is chosen automatically based on the physics

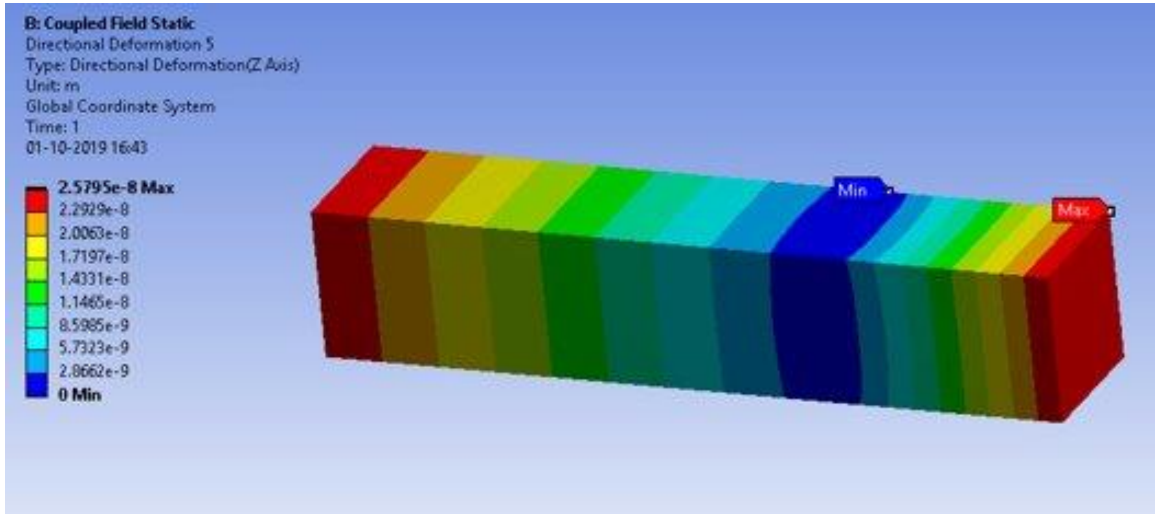






Coupled Field Analysis in Mechanical

- Constraint equations can be used to couple the degrees of freedom between remote points



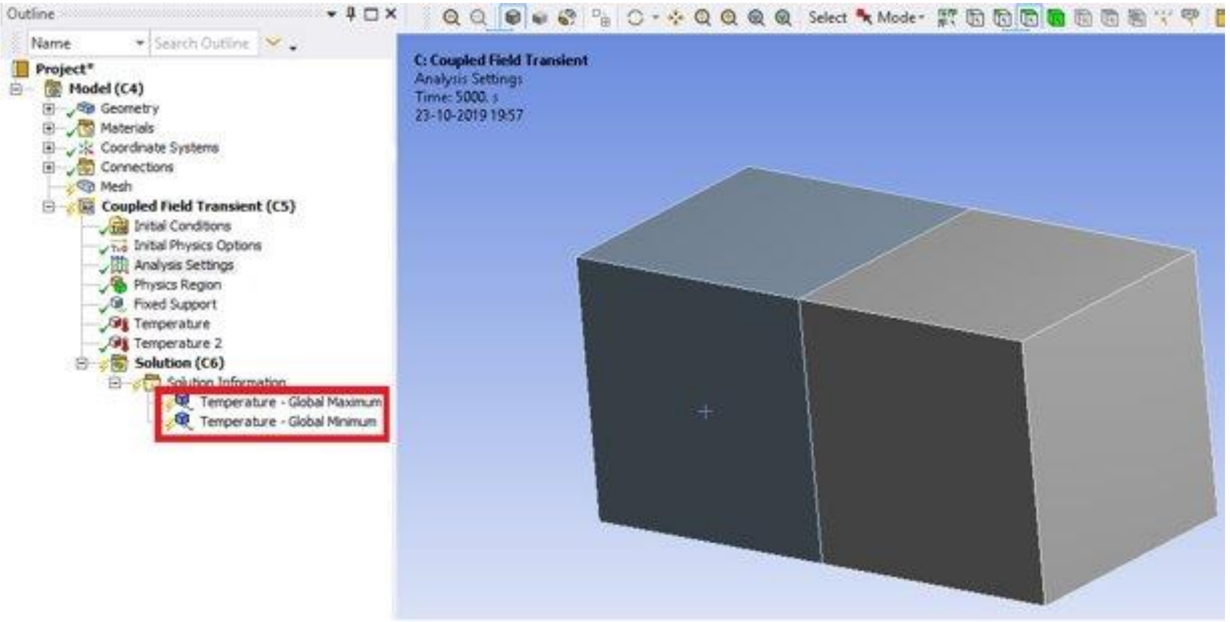
Constraint Equation

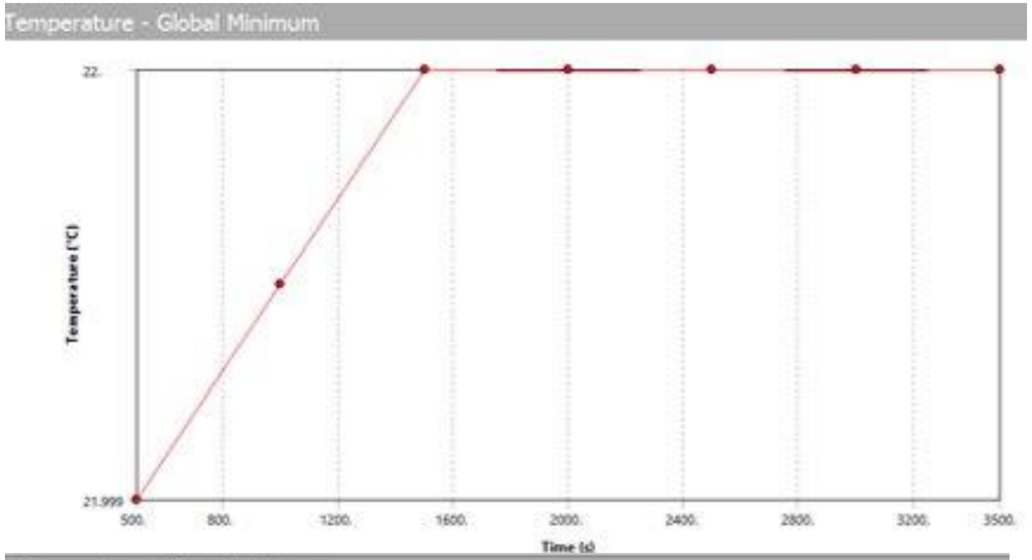
0 = -1000000 (1/m) * Remote Point(X Displacement) + 1000000 (1/m) * Remote Point 2(X Displacement)

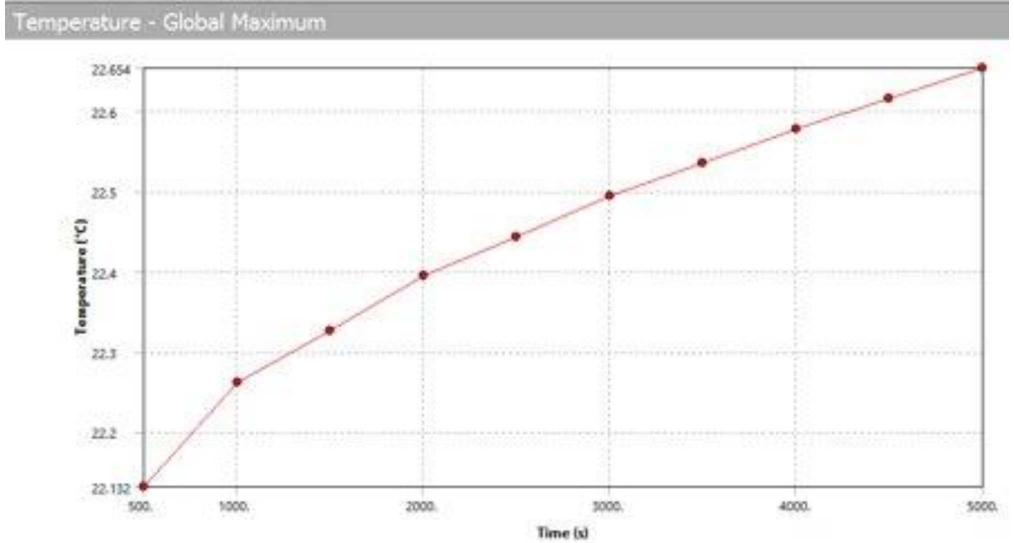
Coefficient	Units	Remote Point	DOF Selection
-1000000	1/m	Remote Point	X Displacement
1000000	1/m	Remote Point 2	X Displacement

Coupled Field Analysis in Mechanical

- *Global minimum* and *global maximum temperature* tracker are now automatically added in a Coupled Field Transient analysis

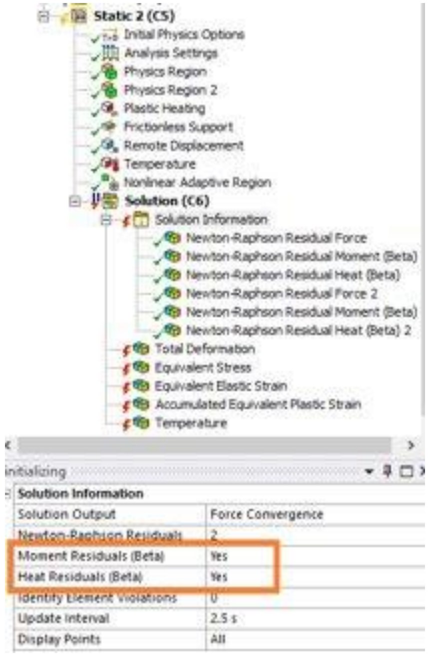


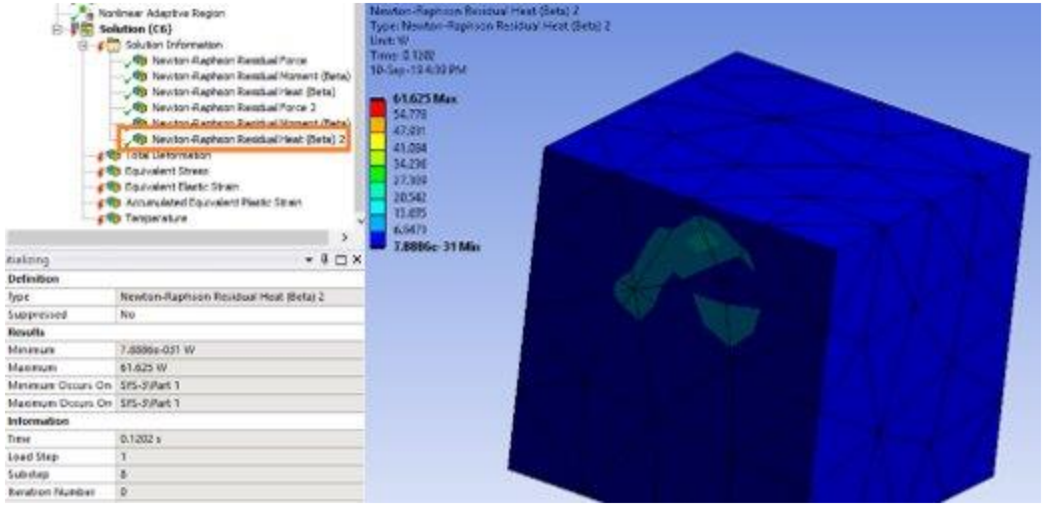


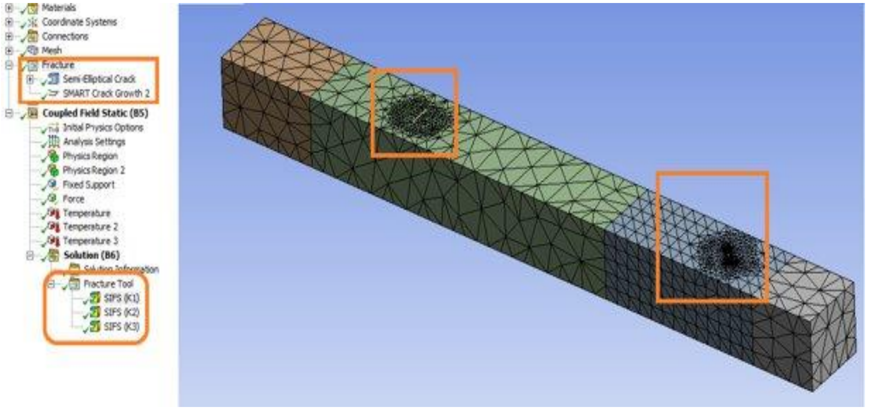


Coupled Field Analysis in Mechanical (Beta)

- Newton-Raphson Residuals for Heat and Moment can be added by setting them to Yes on the Solution Information Tool. Fracture with crack is supported on coupled field analysis if structural physics is selected on the bodies with cracks on it







Advanced Mechanical Features

Advanced Features Supported in Mechanical

- The advanced features supported in 2020 R1 release are
  - Direct pressure without creating surface effect elements
  - Application based transient solution settings for Transient Structural Analysis
  - Output controls for Euler Angle and Volume and Energy

Direct Pressure without Surface Effect Elements

- The direct pressure loading is supported in Mechanical for 3D solids. It will not create any new surface effect elements for applying pressure loads. Instead, it will apply the loads directly to the solid element faces. It is developed using the SFCONTROL command exposed in MAPDL. It applies to Pressure (Normal To/Vector/Components), Force and Hydrostatic Pressure based loading in mechanical.
- Direct pressure option is exposed using **“Applied By”** property which can be set to *Surface Effect* (default) and *Direct*. *Surface Effect* creates surface effect elements which is the default behavior in prior releases

Details of "Pressure"

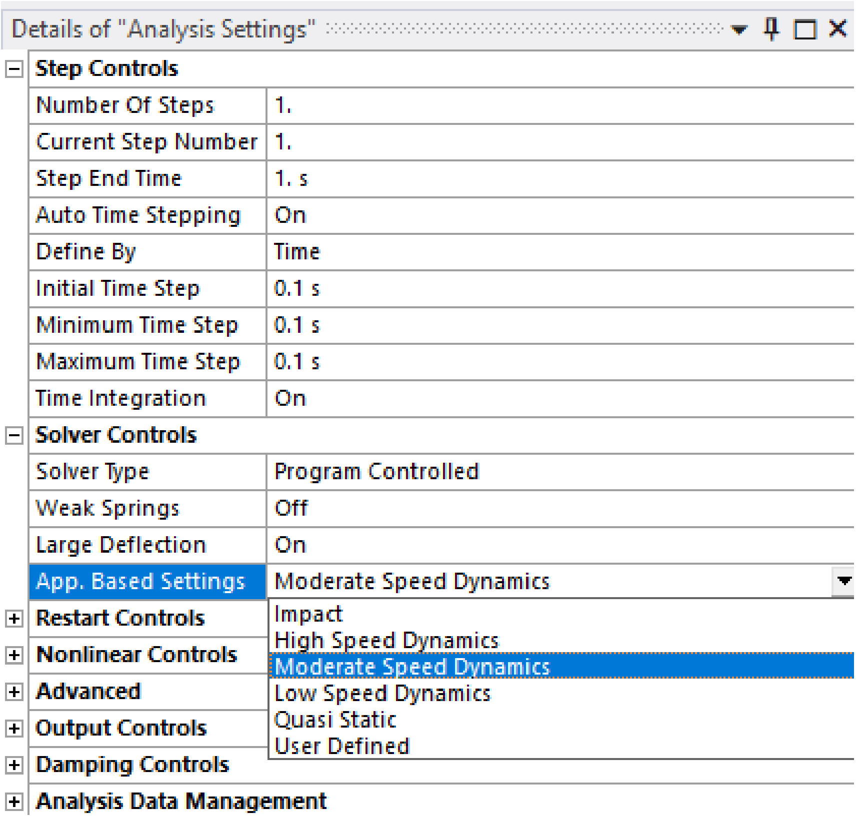
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Vector
Applied By	Direct
Magnitude	1.e+006 Pa (ramped)
Direction	Click to Change
Suppressed	No

Direct Pressure without Surface Effect Elements

- The implementation uses element faces of SOLID elements instead of nodes to avoid issues with shared edges/nodes with other element types and is only applicable for 3D solids in the current release. Multiple loads applied on same scoping and same direction will not have cumulative effect as the last one will overwrite the previous ones. Direct pressure is not supported in presence of Cracks, SMART crack growth, NLAD, CMS and Cyclic Symmetry features

Application Based Transient Solution Settings

- The **“Solver Controls”** group under Analysis settings of *Full Transient Structural Analysis* now make it easier for users to instruct the program to choose the best solution settings based on the intended application of the transient simulation, which could be numerical integrations scheme, integration constants etc.
- The supported application options include *Impact, High Speed Dynamics, Moderate Speed Dynamics, Low Speed Dynamics, Quasi-Static and User Defined*. When *User Defined* option is selected, the user is instructed to specify value of *amplitude decay factor setting*, which was exposed as numerical damping value under “**Damping Settings**” in prior releases



Workbench Mechanical

Acoustics & NVH

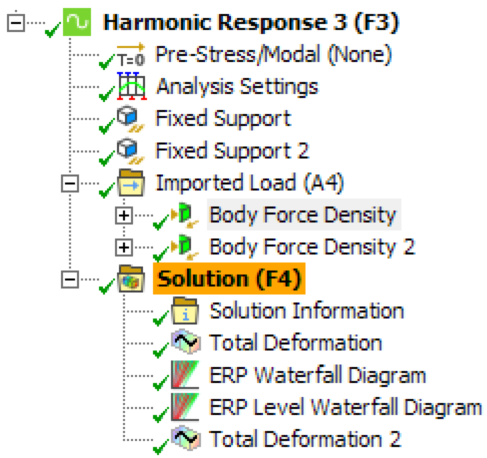
ANSYS Motion

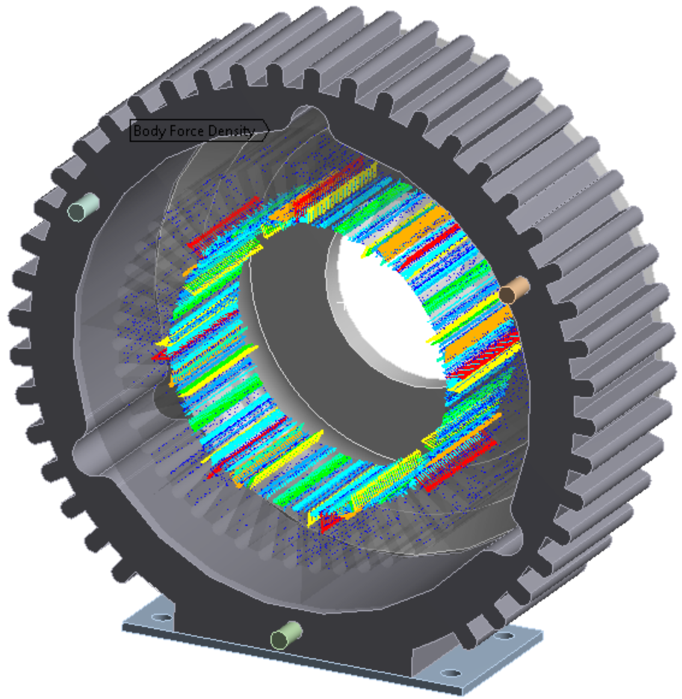
Discovery Live & Autodesk Fusion 360 conversion to Mechanical

External Study Importer

Volumetric Force Density Transfer from Maxwell

- Support frequency varying body force density in FULL harmonic
- Applications: Electric Transformers, Electric Motors





On Demand Result Calculation in MSUP

- To improve performance, expansion pass can be skipped in MSUP harmonic and transient analysis
- Displacement, Velocity, Acceleration, Stress, Strain and ERP can be evaluated on demand in this case saving solution time and I/O
- Residual vector are supported
- **“Skip Expansion”** option shows solution times and IOs can be highly improved (numbers obtained with medium size model)

Modes	Frequencies	Time Standard	Time Skip Expansion	IO Standard	IO Skip Expansion
100	100	245	94	13.2	5.4
100	200	402	97	20.9	5.4
100	400	729	100	36.4	5.4
100	1000	1669	101	82.7	5.4
1000	1000	6073	867	130.2	52.9

nitializing

Step Controls	
Multiple RPMs	No
Options	
Frequency Spacing	Linear
Range Minimum	0. Hz
Range Maximum	80. Hz
Cluster Number	4
User Defined Frequencies	Off
Solution Method	Mode Superposition
Include Residual Vector	No
Cluster Results	Yes
Skip Expansion	Yes
Rotordynamics Controls	
Output Controls	
Damping Controls	
Analysis Data Management	

ANSYS Motion ACT App

- An integration of the Ansys Motion solver technology into the ANSYS Mechanical GUI
- Compatible with ANSYS 2019 R3 upwards
- Provides the power of the ANSYS Workbench and Mechanical Environments to facilitate the pre-processing of Ansys Motion models.

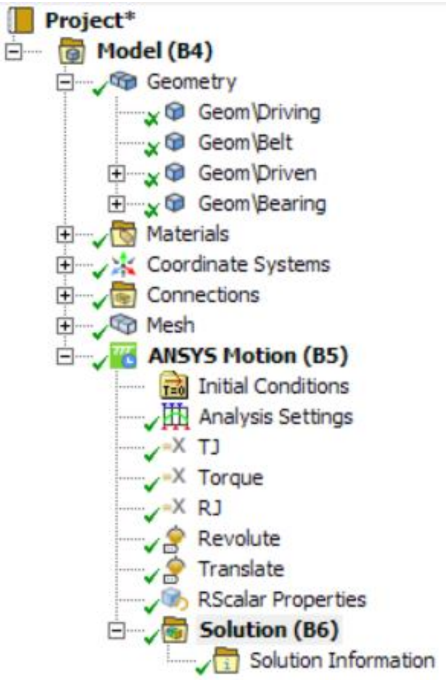
ANSYS Store

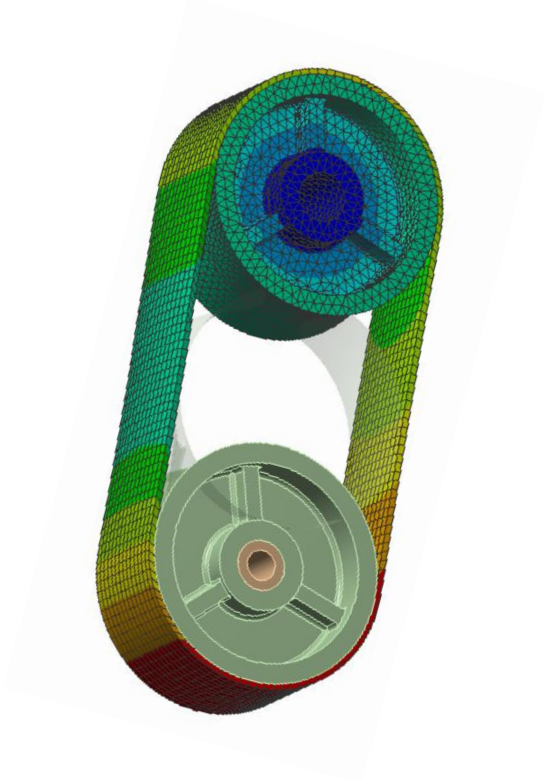
Back to Apps

ANSYS Motion v2019.3

Support ANSYS: 2019 R3
Target Application: Mechanical

MO

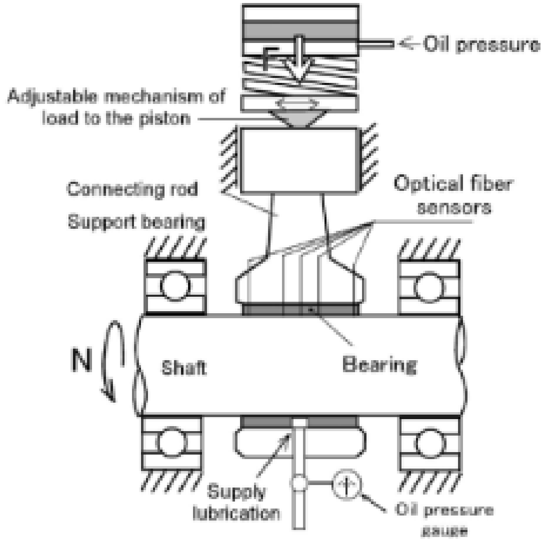




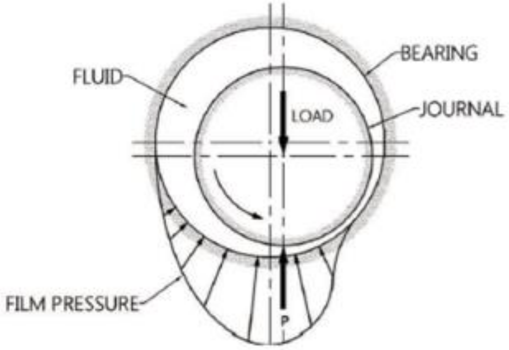
ANSYS Motion - ElastoHydro Dynamic (EHD) Bearing

- Purpose of development
  - EHD is crucial to modelling many mechanical systems
  - Engine, Shaft in a ship, HDD and lots of application requires EHD capabilities
- Application
  - Crank Shaft – Connecting rod in Engine, High-speed compressor and so on
  - Drivetrain toolkit is required for the element
- Properties:

	Characteristics
Basic	- Oil film pressure depend on eccentricity & speed - Surface roughness
Body type	- Rigid, FE (Nodal, Modal)
Output	- Oil pressure force display



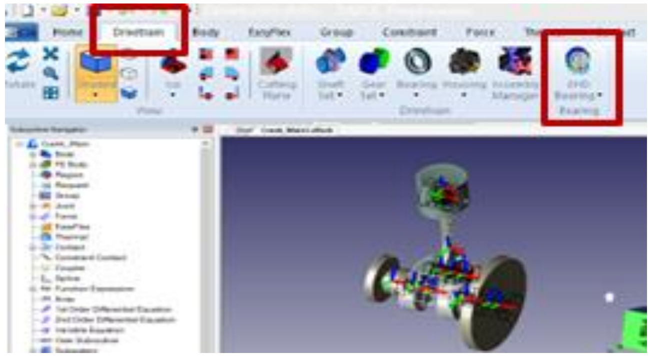
< EHD bearing >

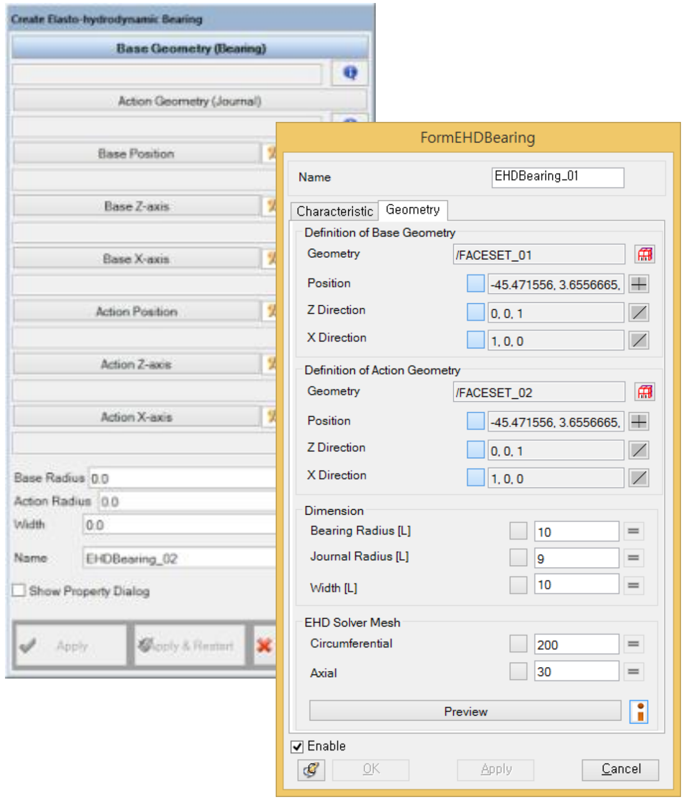


< Applications >

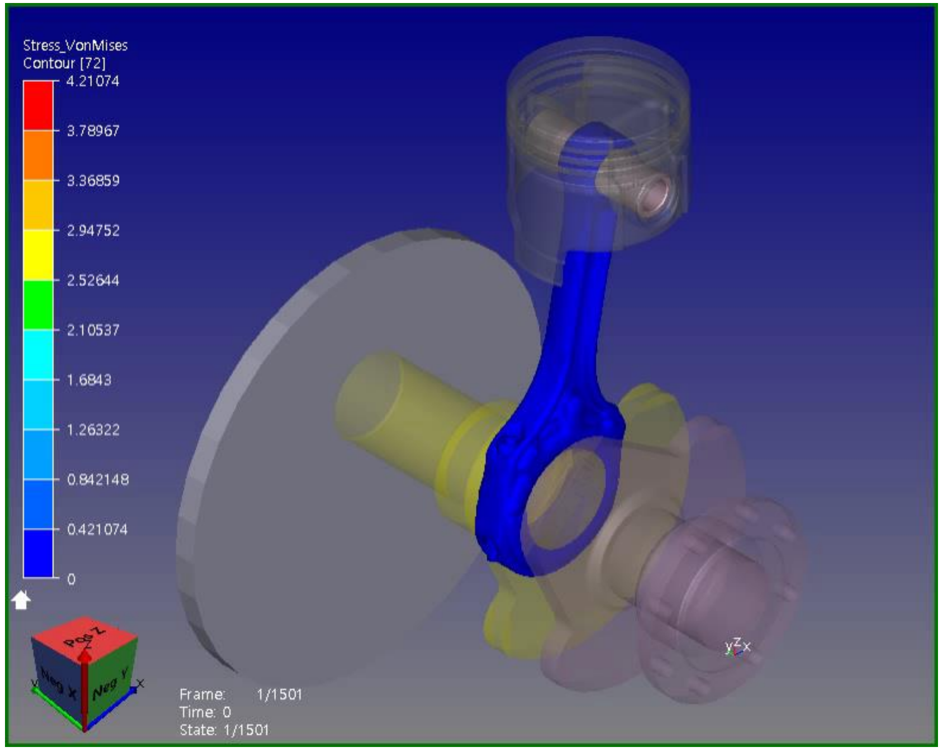
ANSYS Motion - EHD Bearing

- Crank Shaft & outputs
  - Drivetrain toolkit support the EHD element

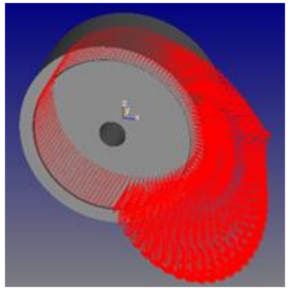




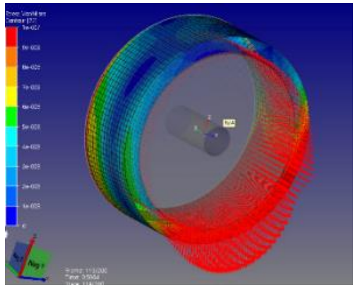
Piston Crank Movie



Supported results



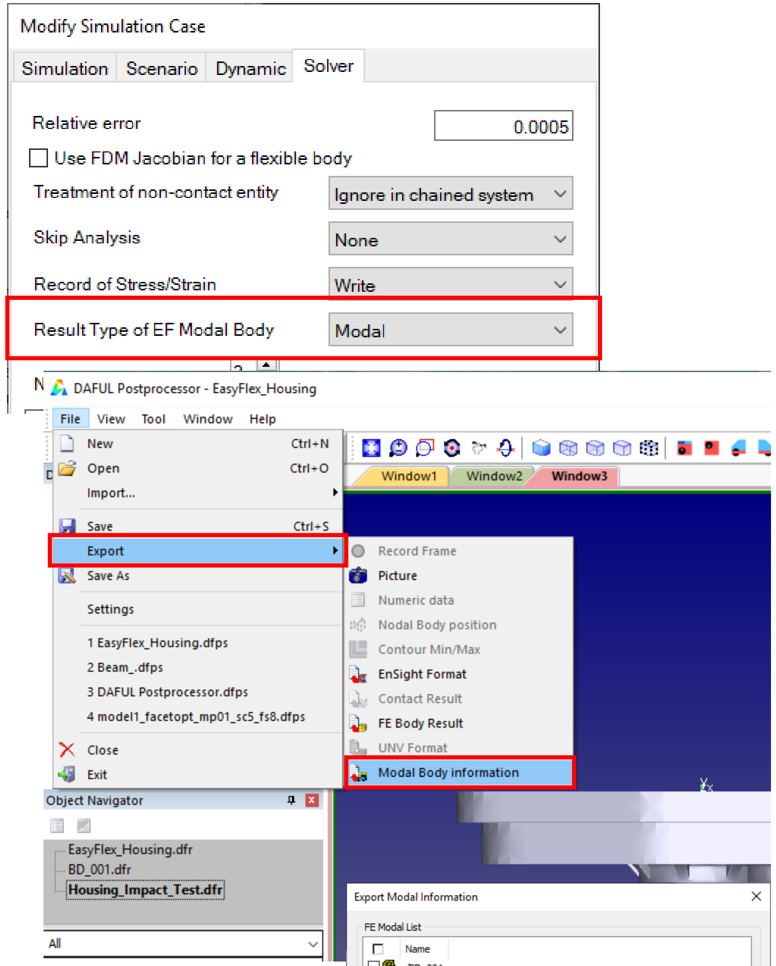
Pressure pattern

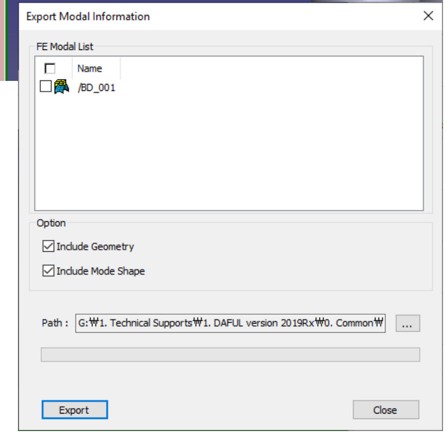


Force & Stress

ANSYS Motion - Modal Data Export

- Purpose of development
  - In order to calculate sound pressure, surface acceleration must be recovered
  - The size of recovered data in the time domain is too big to use
  - An alternative way to calculate these data was required
- Application
  - Sound Pressure Calculation
  - Other applications that need to use surface position, velocity or acceleration
- Operation concept
  - After activating an animation window, you can access the export menu
  - File → Export → Modal Body Information
- Remark
  - Before the simulation, make sure that you are using the "Modal" option for result type
  - If you know all the below information, you can save disk space by turning off the option
    - "Include Geometry" → Node & connectivity information.
    - "Include Mode shape" → Mode shape data



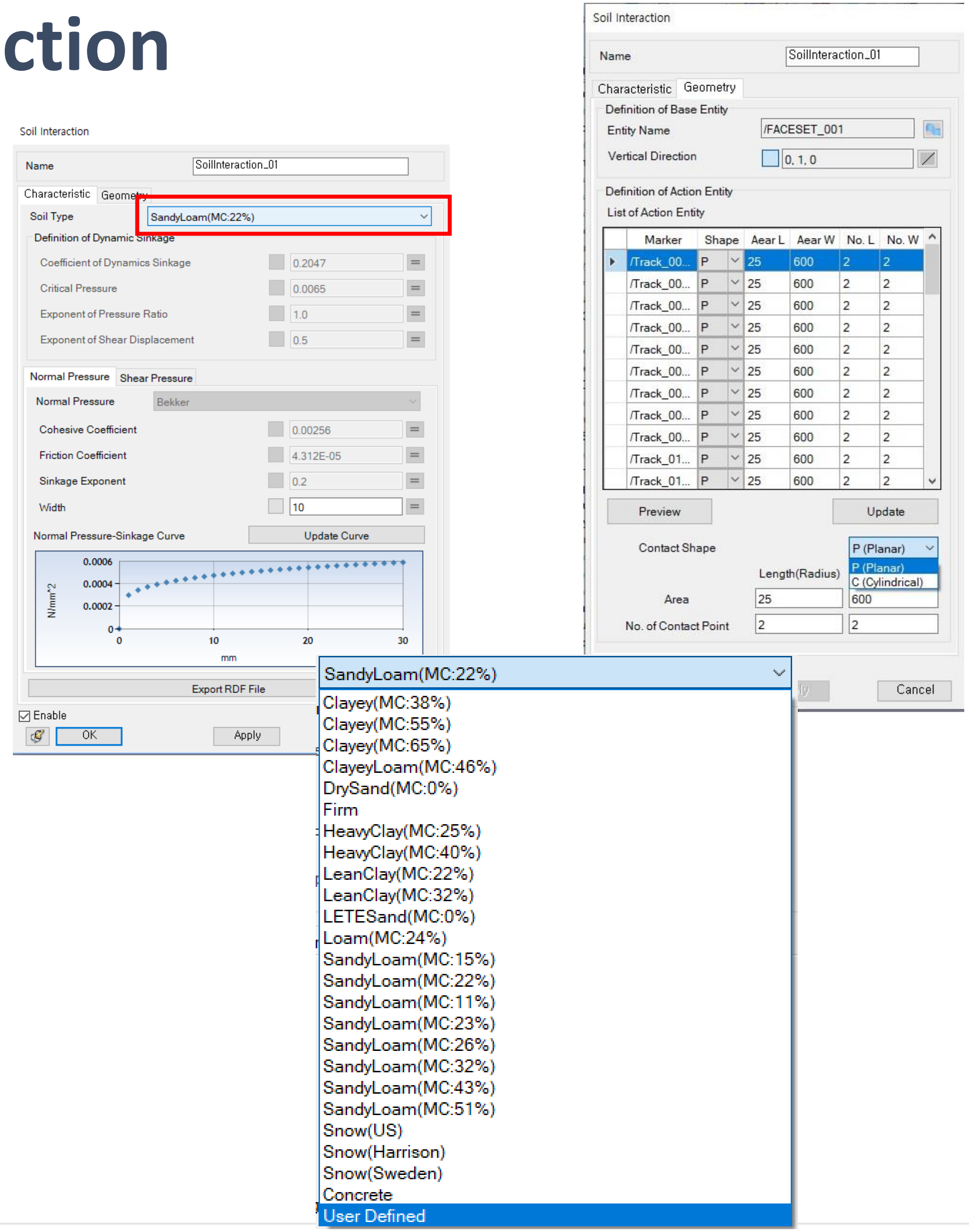


ANSYS Motion Links Toolkit - Soil Interaction

- Purpose of development
  - Soft soil conditions must be considered for tracked vehicle operation environment
  - Bekker's formulation is used

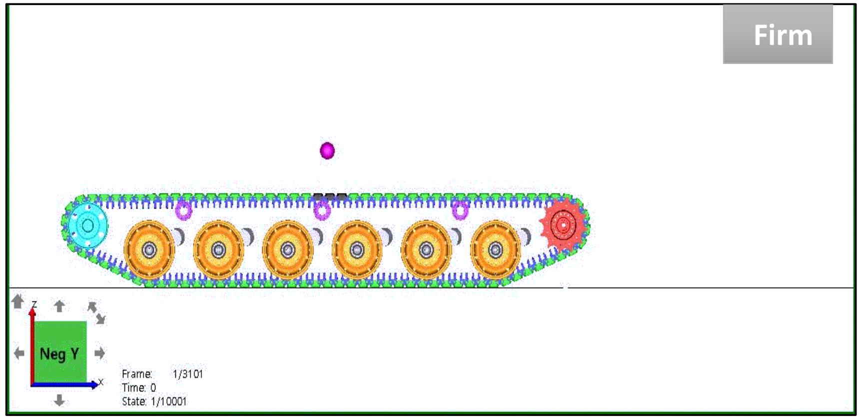
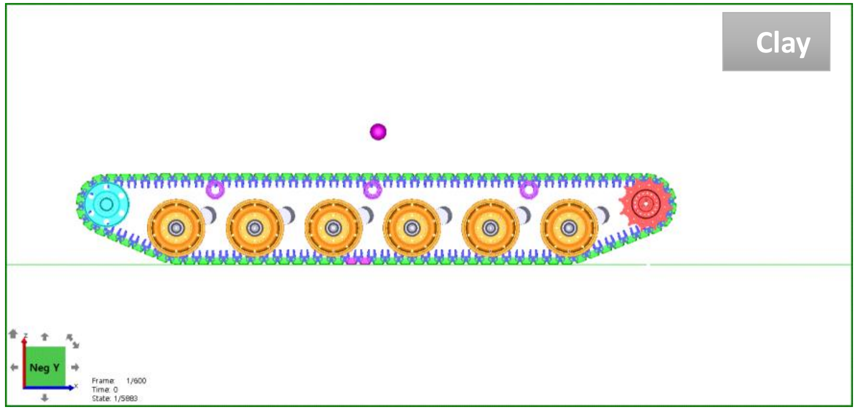
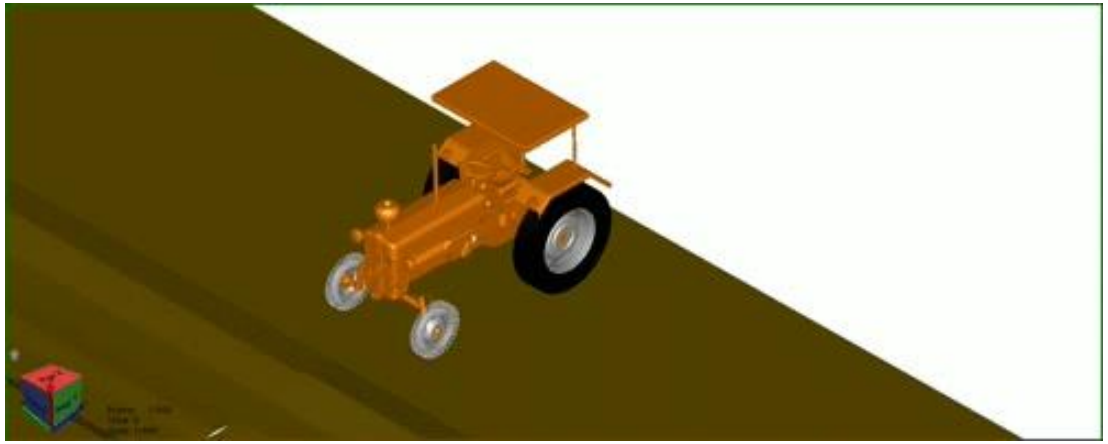
- Application
  - Tracked or wheeled vehicle drive on soft ground condition

- Theory & Operation concept
  - Contact forces with soil can be calculated by Bekker's normal pressure and Wong's shear pressure
  - Soil property and road contact are supported with soil interaction



ANSYS Motion Links Toolkit - Soil Interaction Example

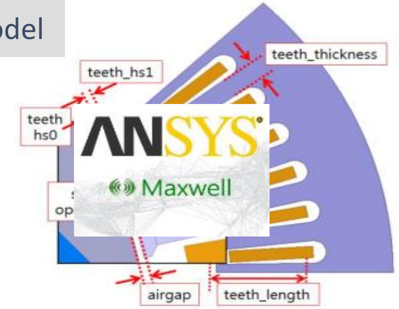
- Example models
  - Tracked vehicle: Penetration pattern is different between two model
  - Wheeled Vehicle: Front wheel is made of “cylinder type” and rear one is “planar type”

Tracked vehicle	Wheeled Vehicle
 	

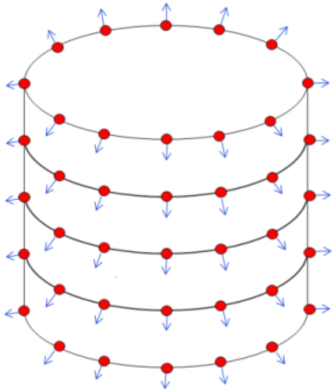
ANSYS Motion - Electro-Magnetic (EM) Force & UNV file

- Purpose of development
  - Consideration of electro-magnetic force for the motor driven power transmission mechanism
  - An electro-magnetic force variation is a key for the NVH analysis of the motor-based system
- Operation concept
  - Motor structure model in MAXWELL → Export electro-magnetic force pattern to Universal file for each stator & rotor
  - Import the file into DAFUL for each RPM (EM force)
  - DAFUL Solver apply these force while a dynamic simulation
- Application
  - A system which needs to be considered an Electro-Magnetic force

Maxwell Model



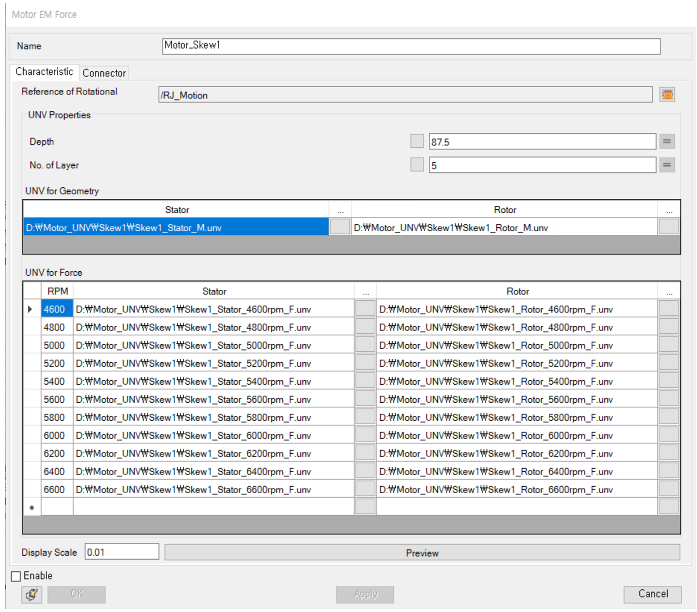
UNV file of Electro-magnetic force



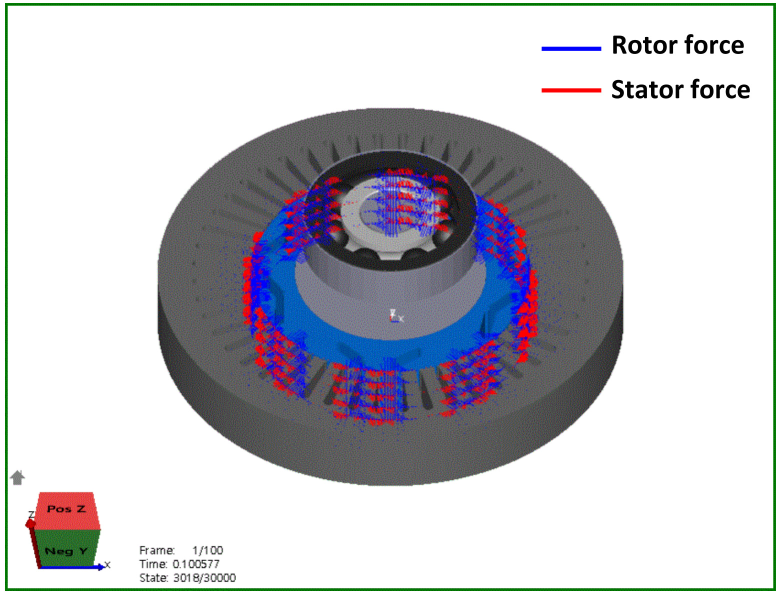
3D Force by using EM Force

ANSYS Motion - EM Force

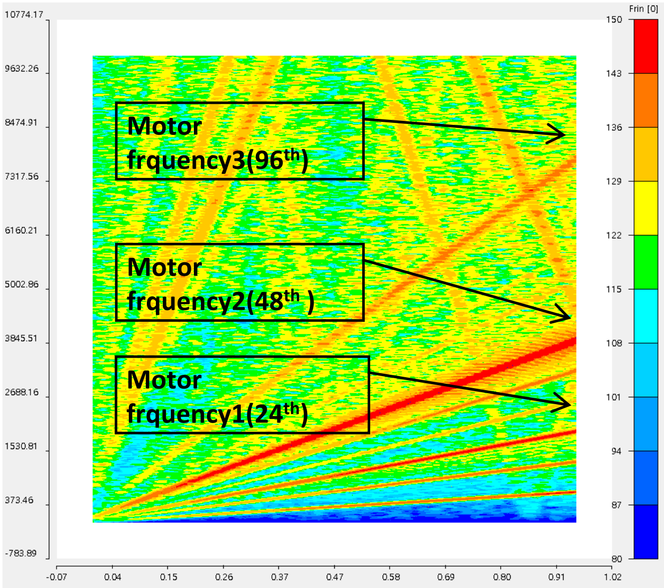
- Motor Dynamics
  - A FE nodal or modal body should be set to create force. Select “stator” and “rotor” body
  - Can conduct a dynamic analysis considering the electromagnetic force and flexibility of the motor



Property dialog

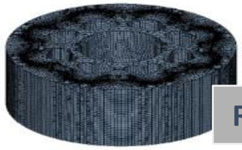


Motor Dynamics



Vibration Color Map

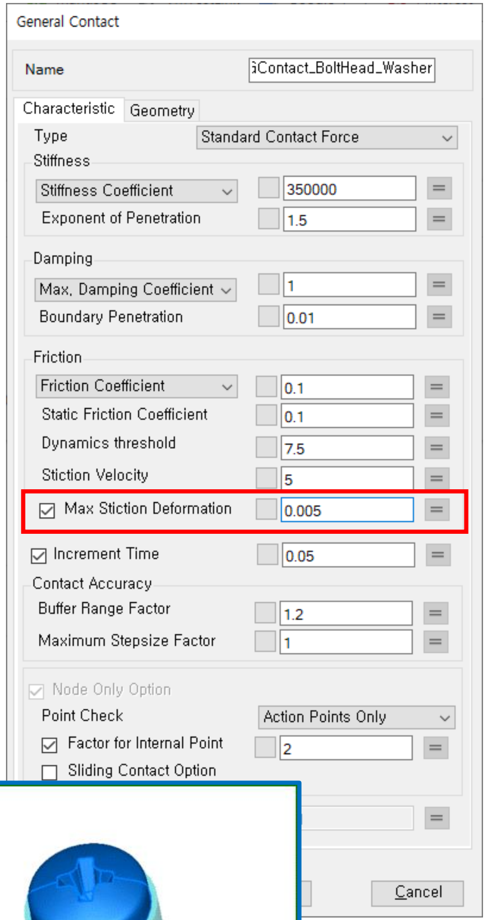
Stator

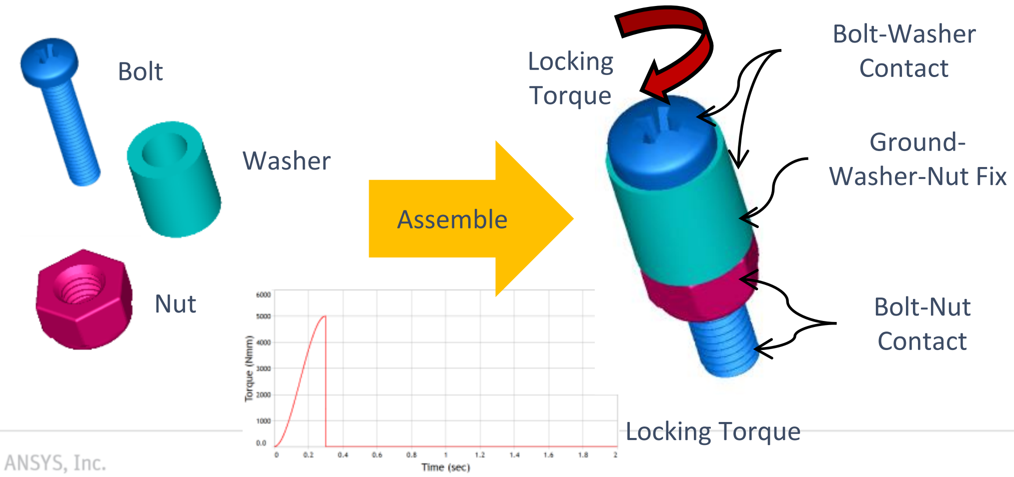


Rotor

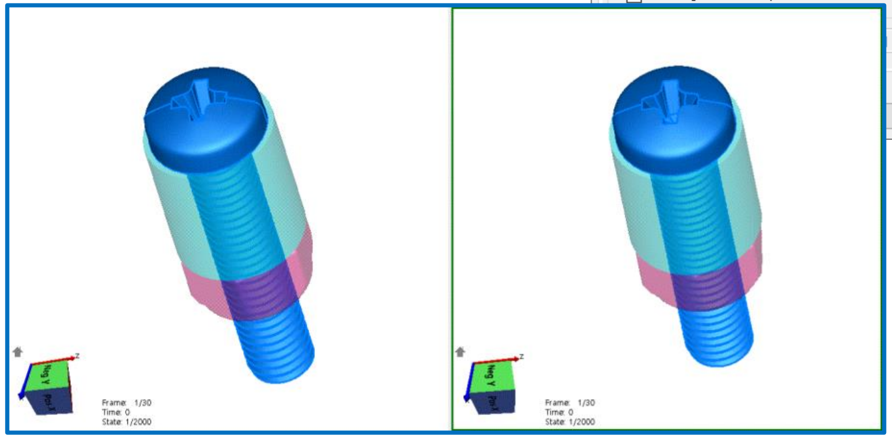
ANSYS Motion Stiction Improvement

- Purpose of development
  - Stiction model of contact friction had been developed in the previous version, but it had a problem to solve a slip phenomenon if a sliding distance exceeds the "Max Stiction Deformation" distance
  - The algorithm has been improved to resolve the stick/slip problem
- Operation concept
  - Same as before and need to be careful to set the value
- Application
  - Bolt locking simulation.
  - Slip stop simulation by friction on the slope





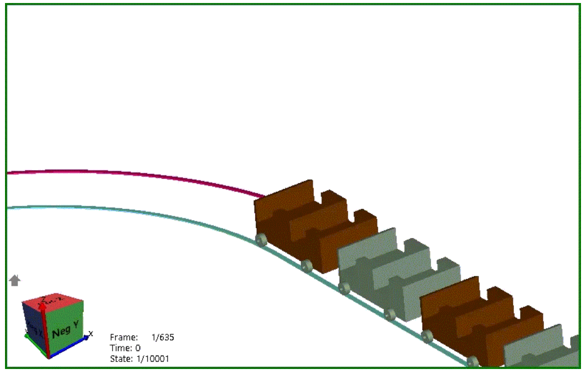
Locking Torque

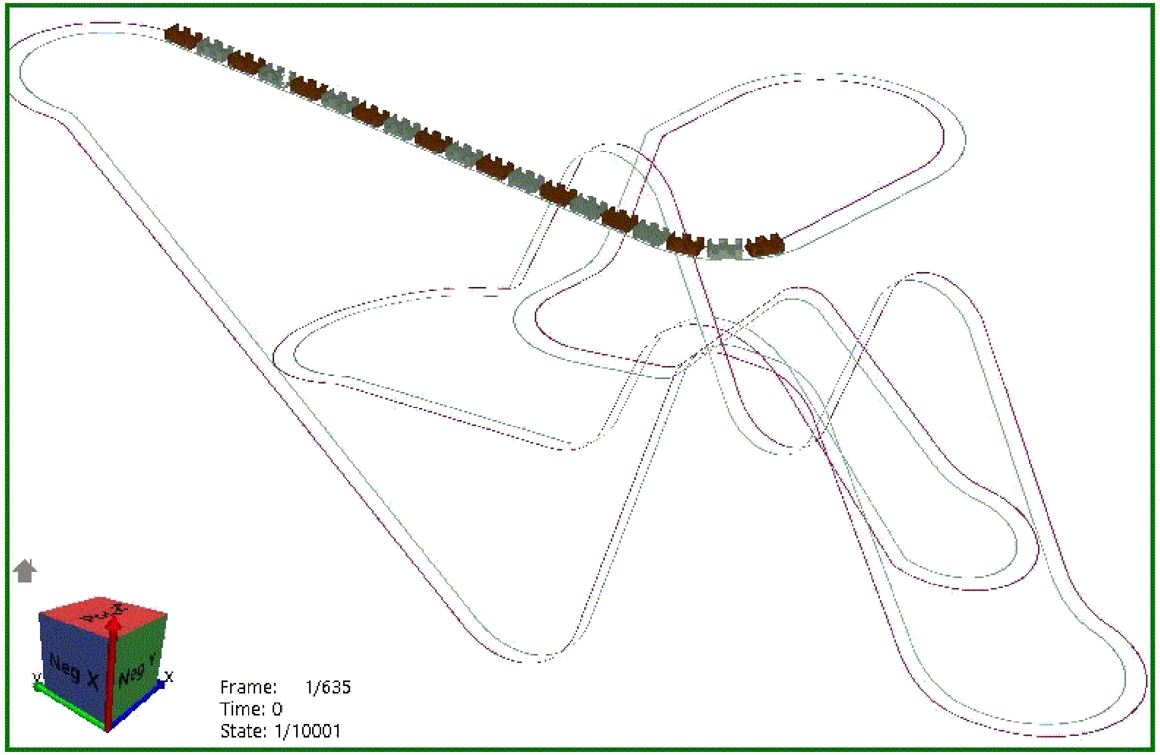


2019 R1 2019 R3

ANSYS Motion Special API for Roller-Coaster

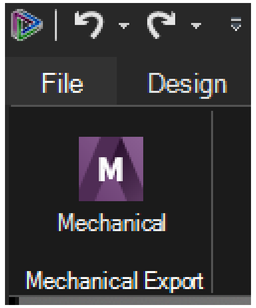
- Purpose of development
  - Designed for building Roller-Coaster model
  - SPECIAL API has been supported to assemble a series of bodies on the curved path
- Application
  - Roller-Coaster assembly or similar application which must be assembled on predefined curve
- Remark
  - Refer “Links 3D API.pptx” for detail operation

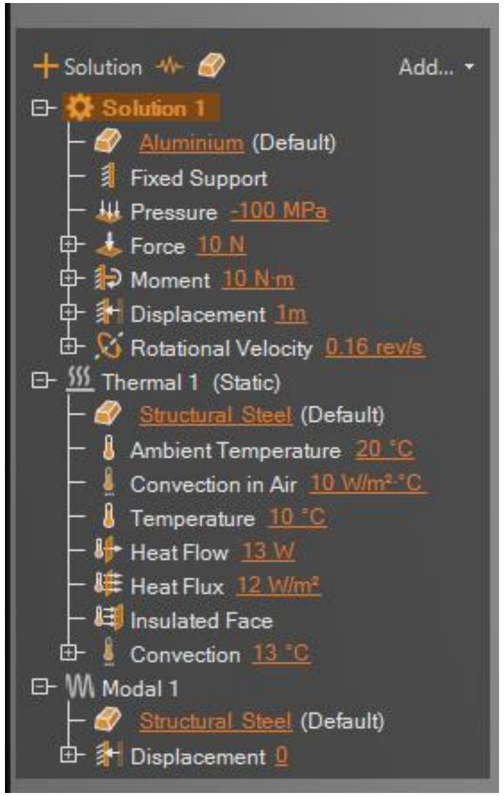




Discovery Live Add-In for Mechanical Export

- Add-in added to Discovery Live to export the database:
  - Geometry, materials, studies, joints, boundary conditions, loads

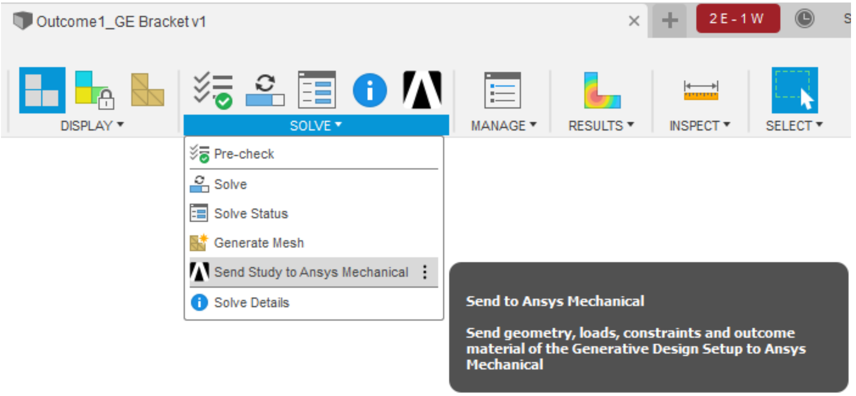




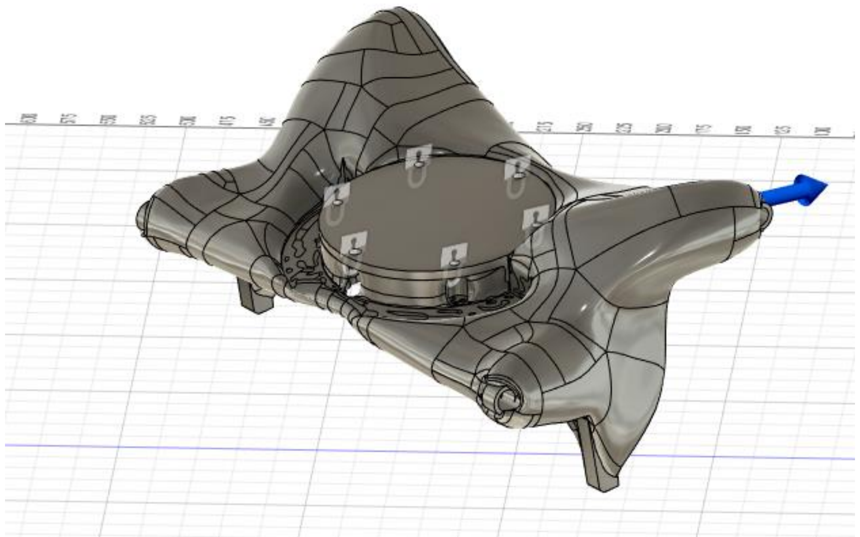
- Exported file can be imported using External Study Imported ACT App
- Double-clicking on the **"Exported File"** runs the Mechanical import

Autodesk Fusion 360 Generative Design Export

- Autodesk Fusion 360 allows Mechanical Export for Generative Design:
  - Geometry, materials, load cases, boundary conditions, loads

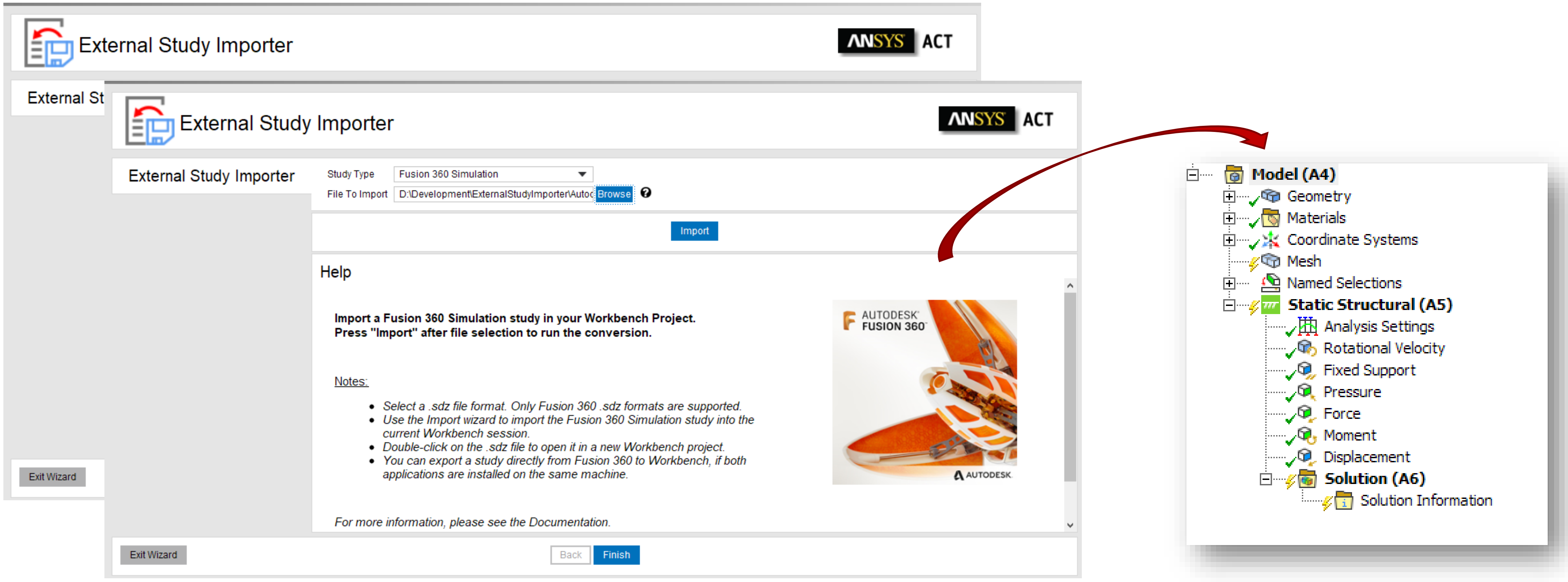


- If Mechanical is installed conversion is launched, otherwise a file is exported
- Exported file can be imported using External Study Imported ACT App
- Double-clicking on the **"Exported File"** runs the Mechanical import



External Study Importer Mechanical App

- Create and pre-install ACT App to import Discovery Live and Fusion 360 studies
- Conversion is also done automatically by clicking on the exported files



MAPDL Linear Dynamics

Component Mode Synthesis (CMS) – New Method

Objective:

Obtain better convergence and improve performances when the master nodes are defined at locations other than the interfaces.
Example: observation nodes where the displacement solutions are requested without expansion pass.

Feature:

For the free-interface CMS analysis (CMSOPT,FREE), by issuing “SUPPORT = ON” on the M command, pseudo-constraints are specified on some master nodes to enforce constrains during the mode-extraction analysis done in generation pass.

Command option:

M,*NODE,Lab1,NEND,NINC,Lab2,Lab3,Lab4,Lab5,Lab6,***SUPPORT**

Note: This CMS method is also called mixed method

Component Mode Synthesis (CMS) - Equations

DOFs partition

- Master DOFs: $m = [m_1\ m_2]$
- Slave DOFs: s

Component modes are normal modes obtained with:

- **FIX method: all master DOFs in m are constrained (fixed)**

$$[\Phi] = \begin{bmatrix} [0] \\ [\Phi_s] \end{bmatrix}$$

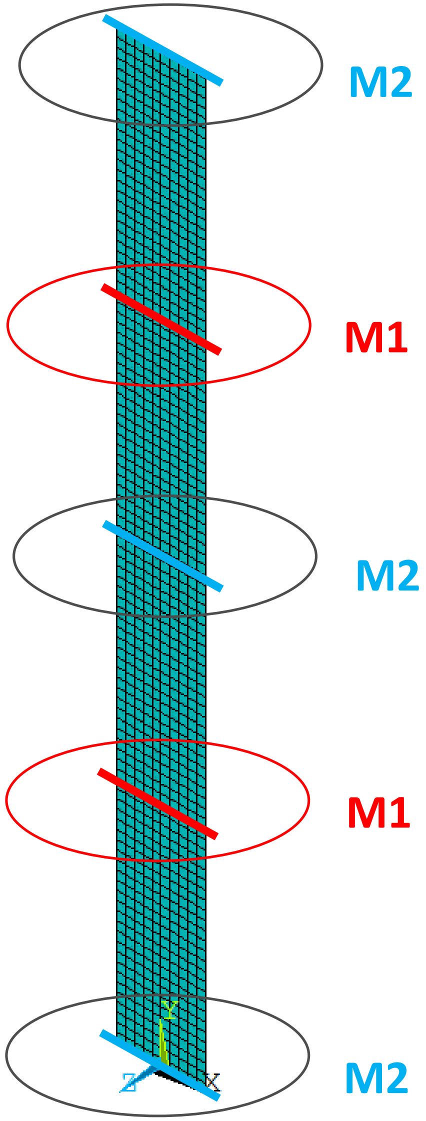
- **FREE and RFFB methods: all master DOFs in m are unconstrained (free)**

$$[\Phi] = \begin{bmatrix} [\Phi_m] \\ [\Phi_s] \end{bmatrix}$$

- **FREE method with SUPPORT = ON applied on m_1 DOFs**
  m_1 DOFs are constrained, m_2 DOFs are unconstrained

$$[\Phi] = \begin{bmatrix} [0] \\ [\Phi_{m_2}] \\ [\Phi_s] \end{bmatrix}$$

Component Mode Synthesis (CMS) - Example



Modal analysis with DOFs of m_1 constrained

```
/solu
antype,7
m,M1,all,,,,,,,,on
m,M2,all
allsel
seopt,plate,2
cmsopt,free,25
solve
finish
```

CMS use pass

MODE	FREQUENCY (HERTZ)
1	0.9971905778302
2	1.010560003321
3	1.627142147610
4	4.443753809167
5	5.479559213616
6	5.587669504940
7	5.718158942417
8	6.361381730613
9	6.471946425180
10	8.961885916441
11	11.57222771454
12	14.72991726365
13	17.58170060619
14	17.81447941803
15	18.19534007751
16	18.47085906064
17	18.51194745337
18	22.64473712268
19	25.81880894079
20	31.46612557674
21	32.75978755578
22	33.05480025529
23	34.88231560301
24	36.37473404422
25	36.89574458328

FULL analysis

MODE	FREQUENCY (HERTZ)
1	0.9971905870709
2	1.010560012418
3	1.627142153559
4	4.443753810965
5	5.479559215506
6	5.587669506946
7	5.718158944216
8	6.361381731923
9	6.471946426605
10	8.961885917475
11	11.57222771526
12	14.72991726419
13	17.58170060666
14	17.81447941842
15	18.19534007798
16	18.47085906115
17	18.51194745368
18	22.64473712306
19	25.81880894109
20	31.46612557710
21	32.75978755601
22	33.05480025552
23	34.88231560332
24	36.37473404440
25	36.89574458346

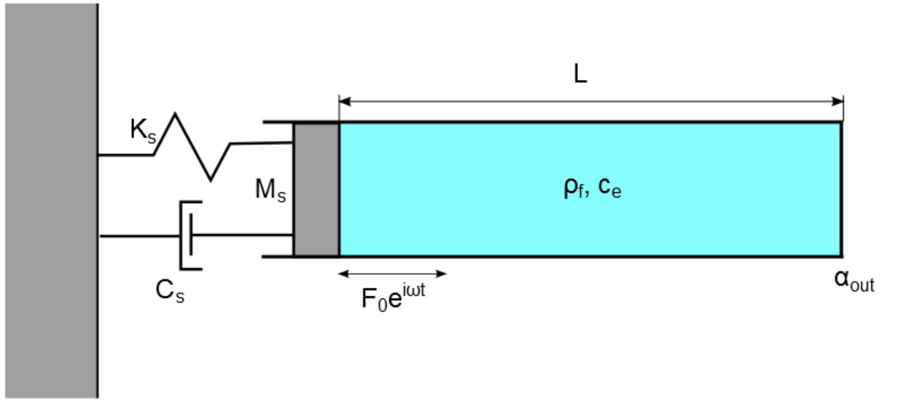
Solutions Comparison Tool – RSTMAC Enhancements

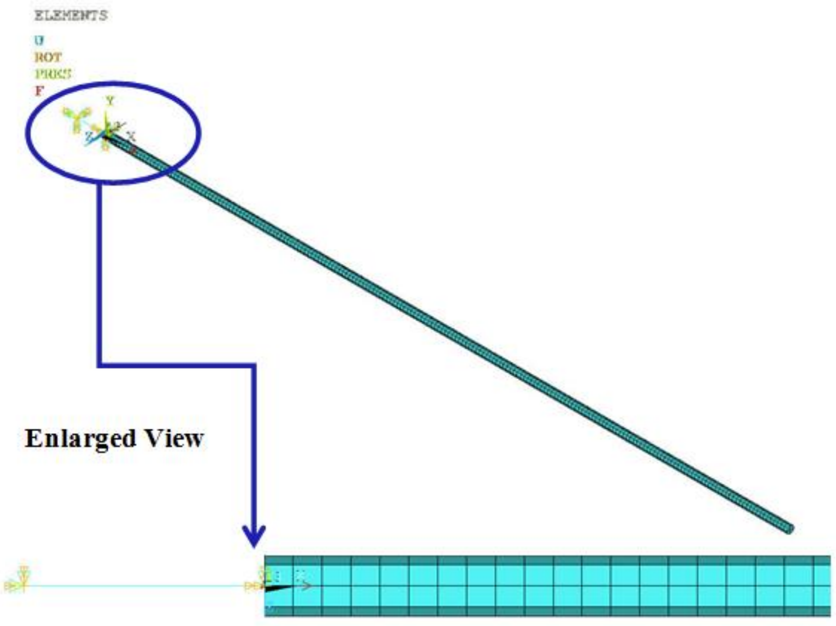
- Degree of Freedom (DOF) selection: 1 DOF or a combination of DOFs
  - New MACOPT, DOF
- Support non-structural DOFs for coupled-field analyses:
  - PRES, VOLT, CONC, MAG, TEMP, and AZ
- Support node matching based on node number
  - For morphed mesh or translated/rotated model applications

Solutions Comparison Tool – RSTMAC Example

Based on VM282

- Two different meshes of Piston-Fluid System using FLUID30
- Comparison of coupled-field modal solutions





****** MATCHED SOLUTIONS ******		
Substep in FILE1	Substep in FILE2	MAC value MACOPT,DOF ,PRES
1	1	0,999
2	2	1,000
3	3	0,998
4	4	0,998
5	5	0,998
6	6	0,998
7	7	0,998

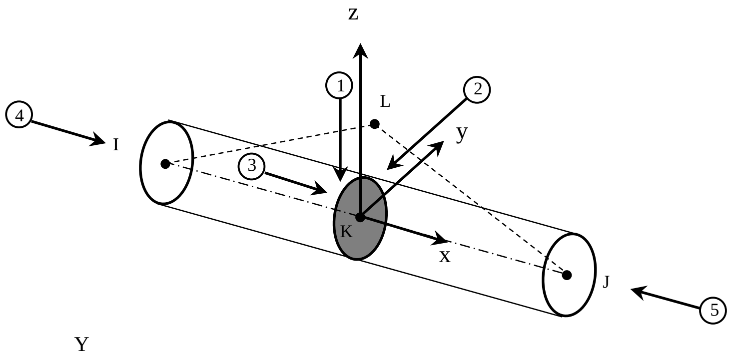
MAPDL Elements

List of New Features

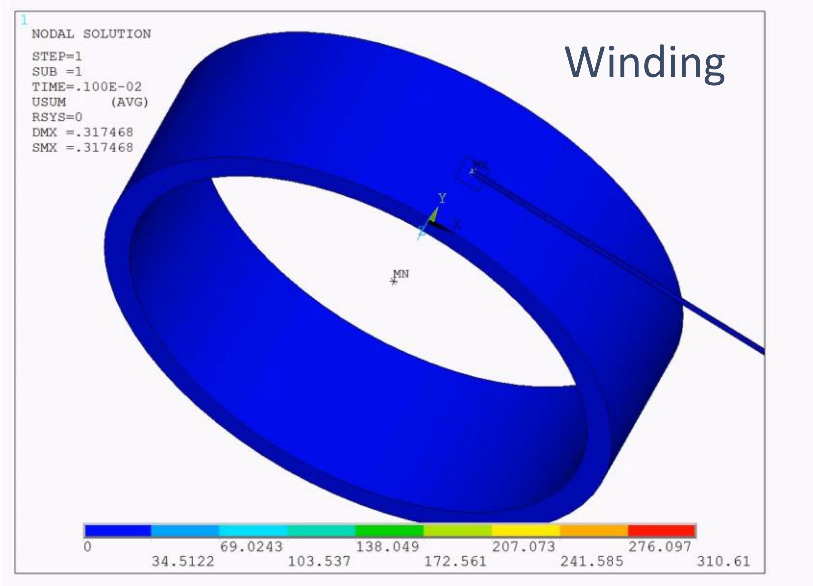
- Current technology 3D 3-Node Cable Element : CABLE280
- Support of incompressible materials with Inverse method
- Pure displacement-based formulation for SOLID285
- General distributed load for SOLID and SHELL elements
- Current technology 2D thermal element PLANE292/293
- User-Defined Material Models for 22x Coupled Field Analyses

3D 3-Node Cable Element CABLE280

- Suitable for analyzing moderate to extremely slender cable structures
- Computationally efficient with only translational DOFs
- Mixed Displacement / Force formulation for superb solution accuracy and robustness
- Wide range of applications: offshore, civil, and mechanical



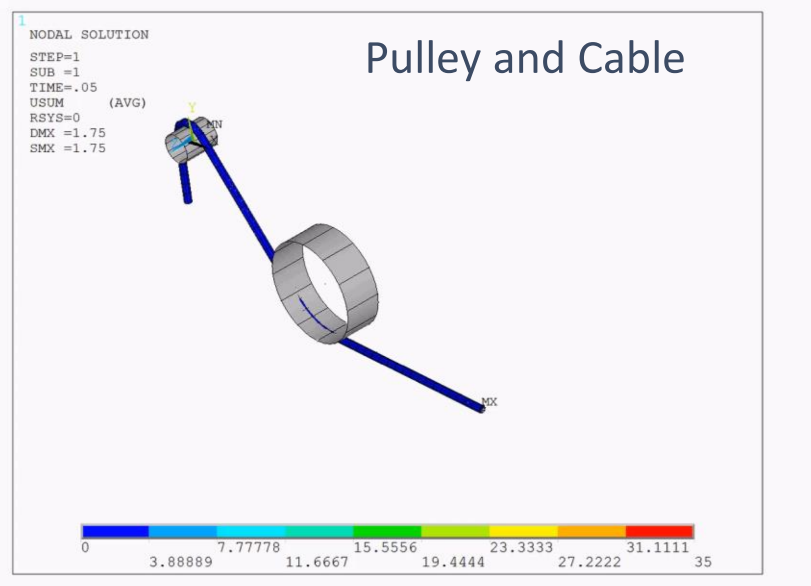
CABLE280 Geometry

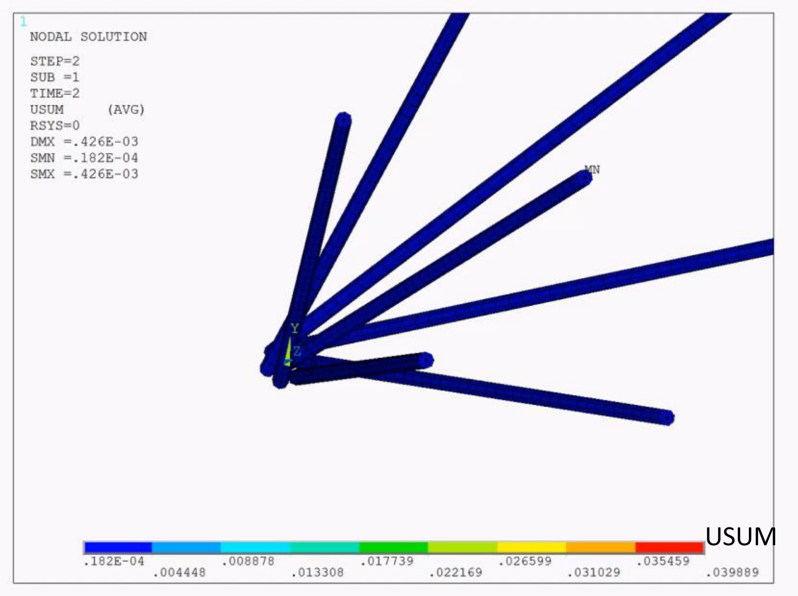


Stranding

queins.com/en/solutions/stranding/

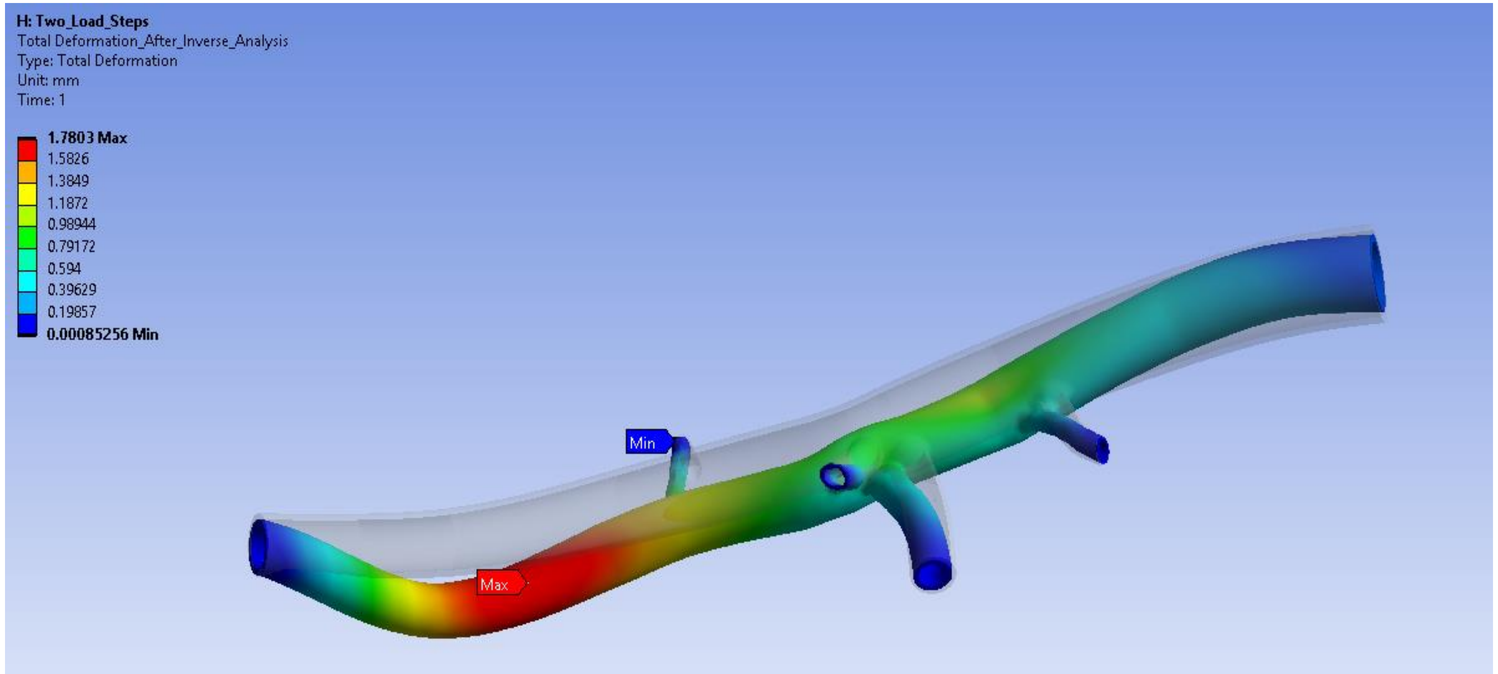
One cable is placed in the center, a second layer containing six cables is stranded around it.



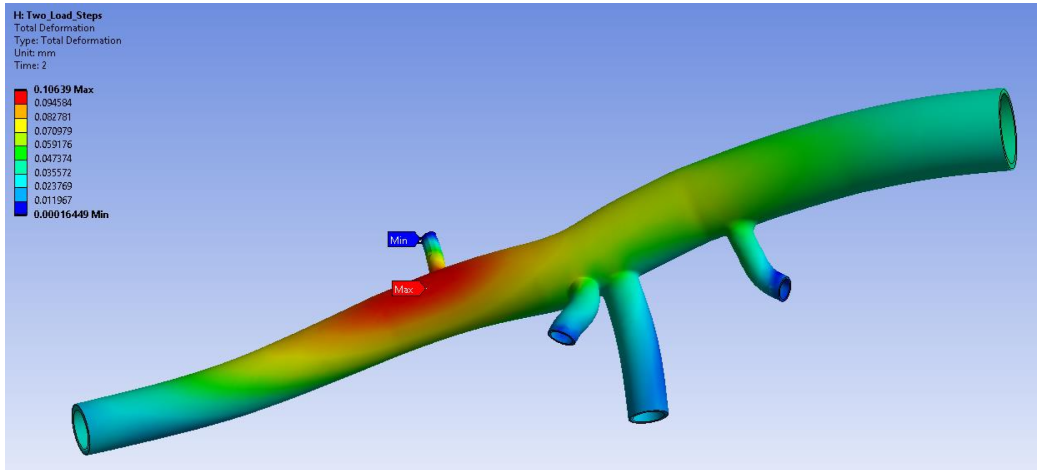


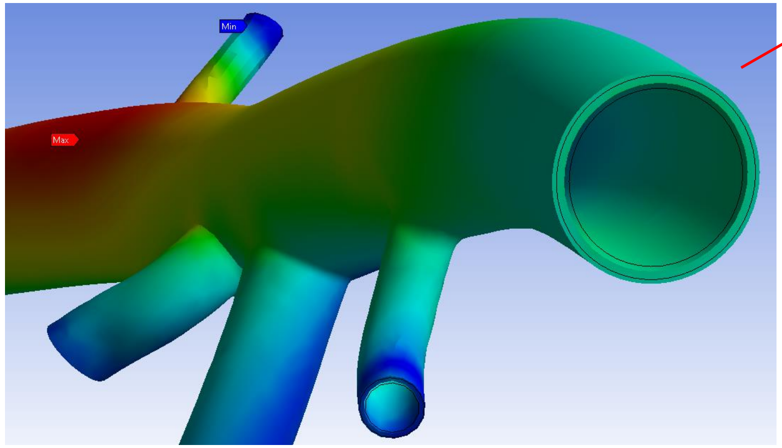
Inverse Analysis

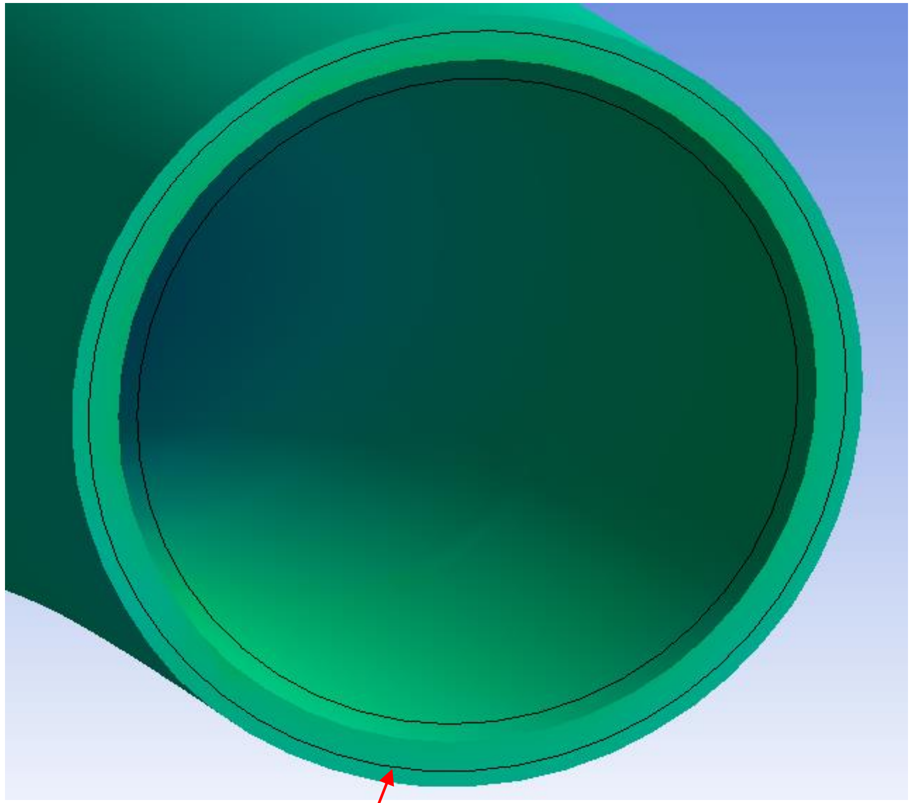
- It supports Solid185 B-bar formulation (keyopt(2)=1)
- It supports Solid186, Solid187 and Solid185(B-bar) with mixed u/P (Keyopt(6)=1,2)
- New supports large deformation with incompressibility, such as biomedical applications
- An aortic lumen under 80 mmHg blood pressure is the input geometry
  - The calculated “deformed” geometry is the zero-pressure geometry



Results when blood pressure increases to 120mmHg
…Diameter increases significantly



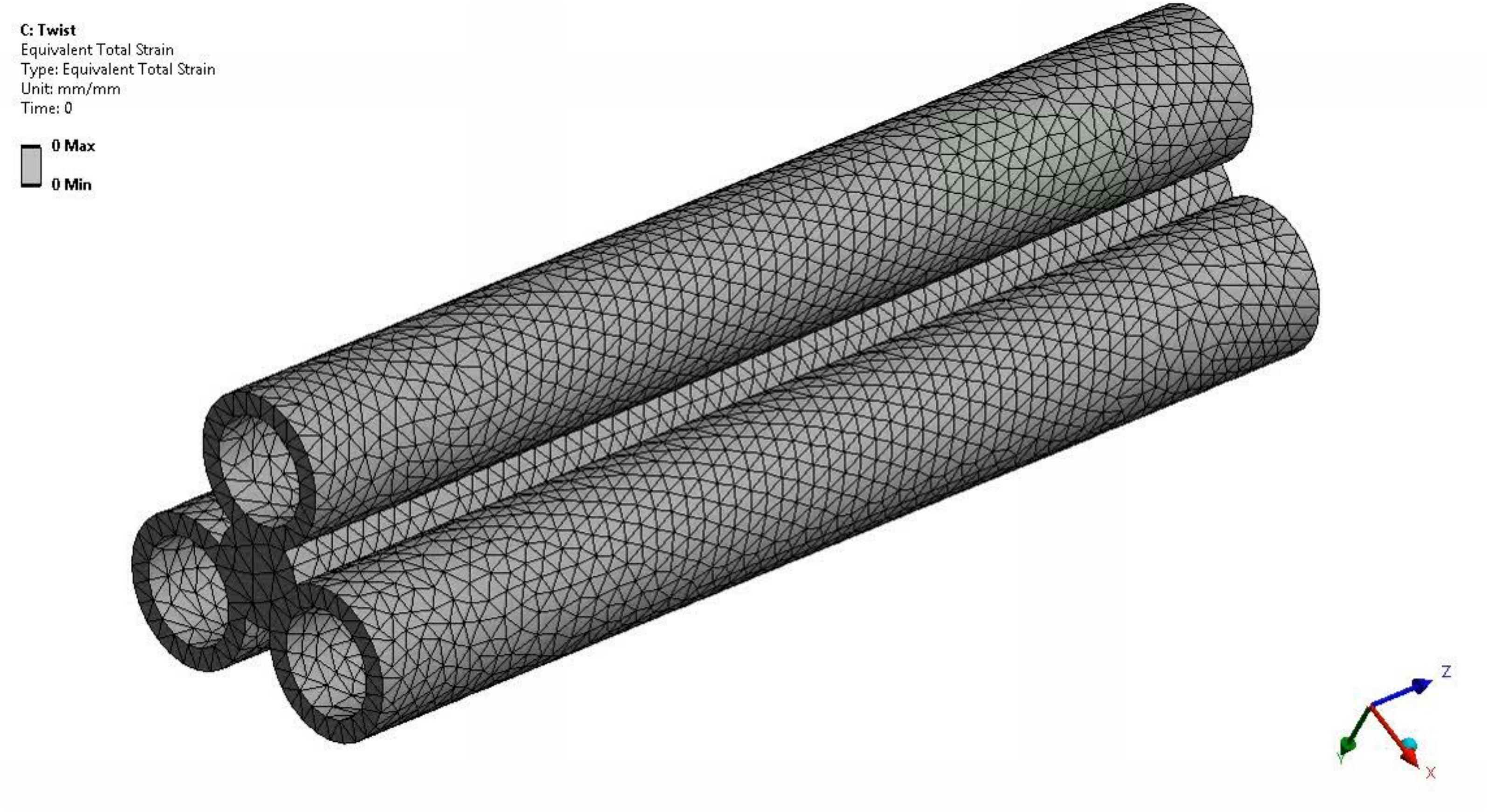




Change in diameter with further loading (120 mmHg) is shown with given cross-section at 80 mmHg loading.

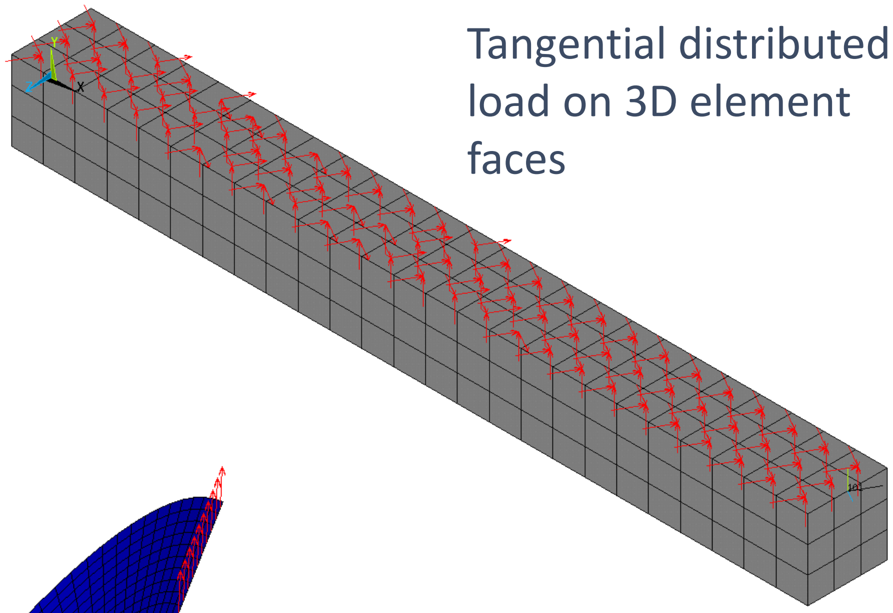
Pure Displacement Formulation for Linear TET SOLID285

- A linear tetrahedral element with displacement as DOF only: SOLID285 (KEYOPT(1)=1)
  - Efficient and robust for problems without significant bending and incompressibility

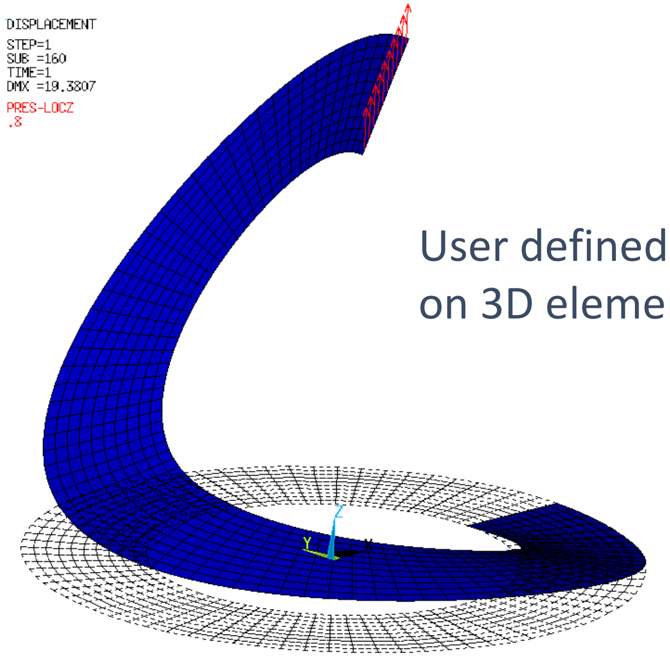


General Distributed Load for SOLIDs and SHELLs

- General load options: normal, tangential, user defined, tapered, projected, and more
- A single new PREP7 command SFCONTROL to define general load properties
- Eliminates the need for surface effects elements for most cases
- All current technology 2D/3D SOLID and SHELL elements are now supported
    - 3D/2D solid elements (185,186,187,190,285,182,183)
    - 3D/2D shell elements (181,281,208,209)
    - 3D/2D coupled-field elements (226,227,222,223)
- Enable imaginary distributed loads for the harmonic analysis

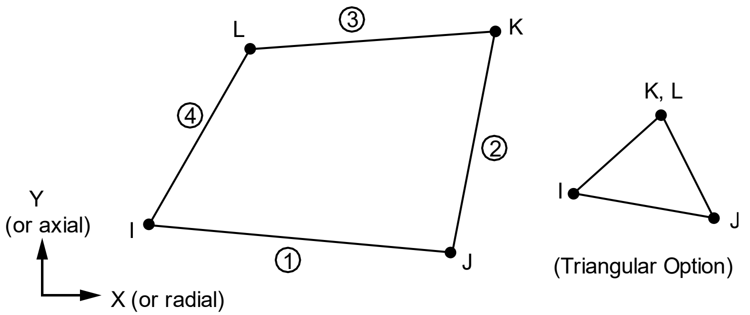


Tangential distributed load on 3D element faces



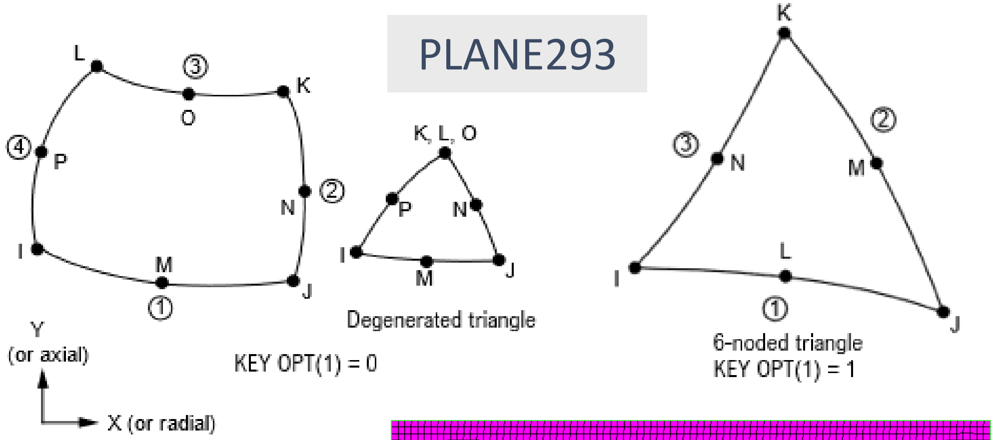
User defined line load on 3D element edges

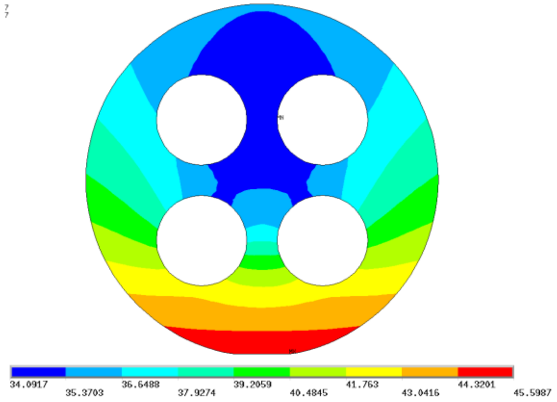
Enhanced 2D Thermal Elements PLANE292/293



PLANE292

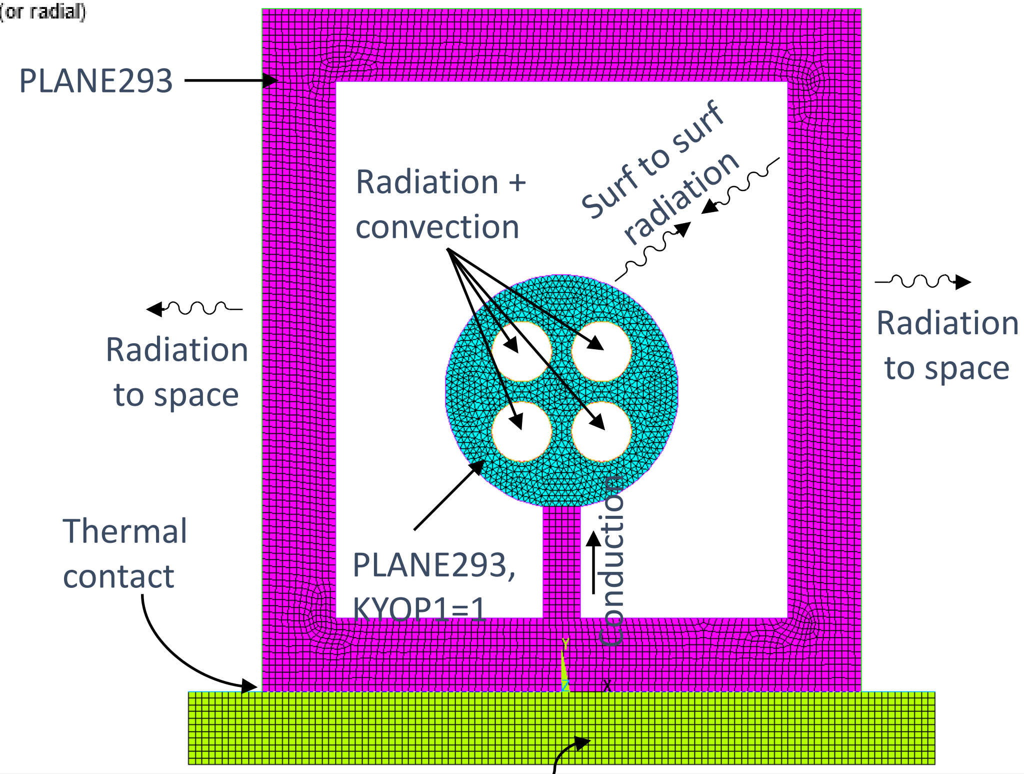
PLANE293





Temperature distribution

- Capable of simulating complex thermal problems
- Proper handling of nonlinear thermal load with consistently linearized element stiffness
- UPF support (UserMatth.F)

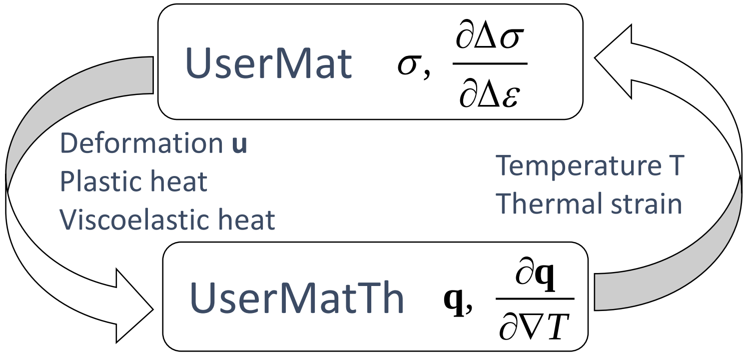


Stove with heat generation, PLANE292

User-Defined Material Models for 22x Coupled Field Analyses

- Coupled-field elements SOLID226 and SOLID227 now support the **“UserMat”** and **“UserMatTh”** subroutines for customizing structural and thermal material behaviors, respectively.
  - To define a custom structural material model (UserMat), specify user-defined structural properties via TB,USER with TBOPT = NONLINEAR, LINEAR, or MXUP.
  - To define a custom thermal model (UserMatTh), specify user-defined thermal properties using the *new* THERM option (TBOPT = THERM) with TB,USER.
- Application example:
  - Manufacturing process simulation

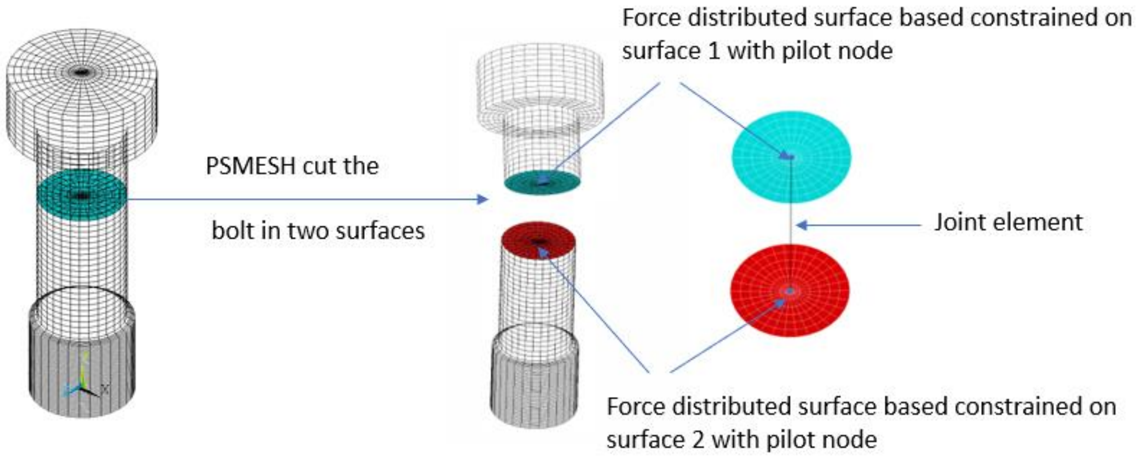
22x Coupled-Field Element

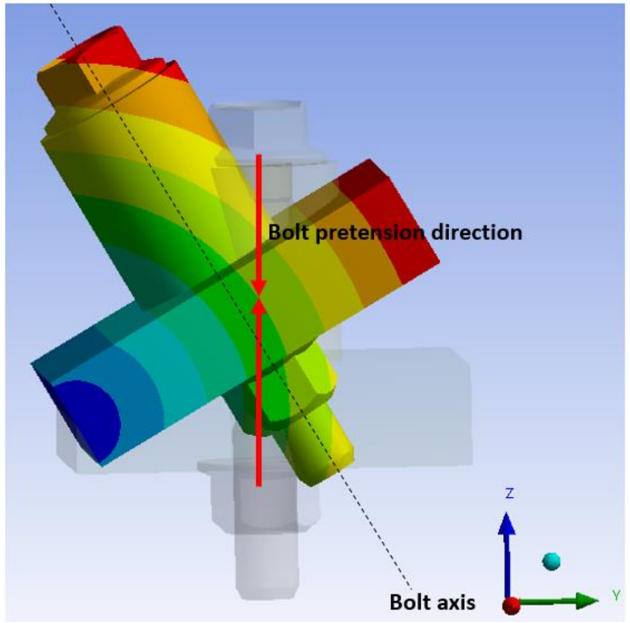


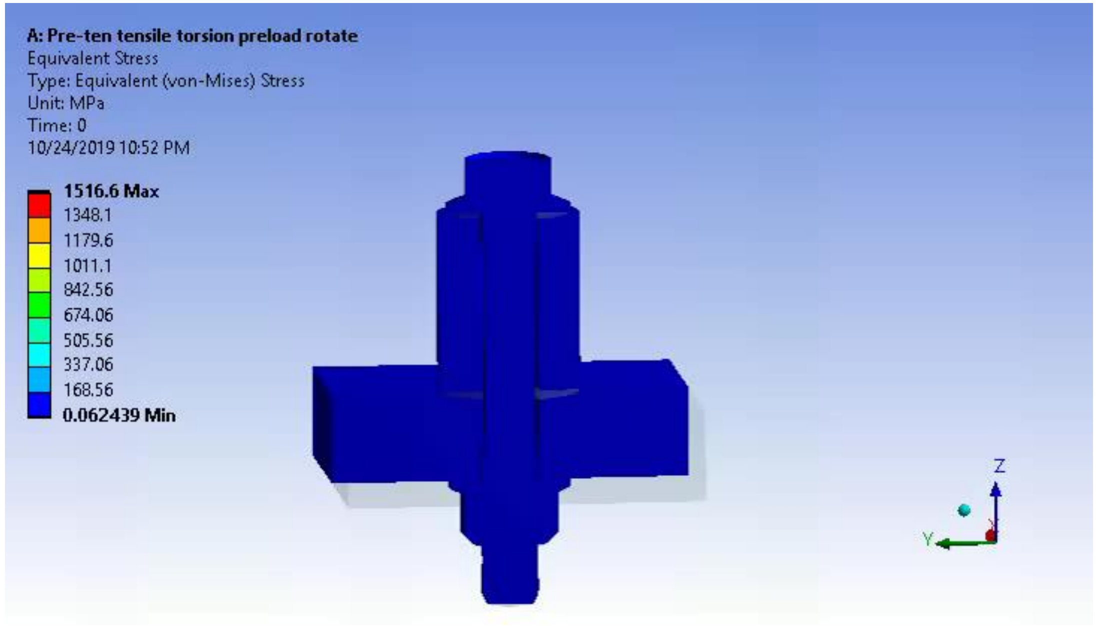
Contact Features

Defining Preload in a Fastener Undergoing Large Rotation

- **“PSMESH”** can now define **“MPC184”** joint elements for applying a preload to a bolt undergoing large rotation or large deformation
- The joint element supports large deformation and the bolt axis follows the local coordinate system defined at the joint node
- You can apply torque and rotation about the bolt axis (<u>**FJ**</u> and <u>**DJ**</u>, respectively)
- **Bolt Sleeve Model Undergoing Large Rotation:**

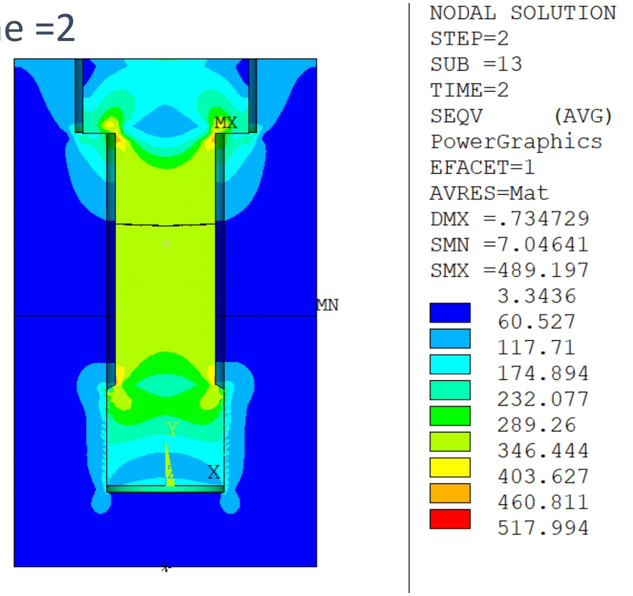




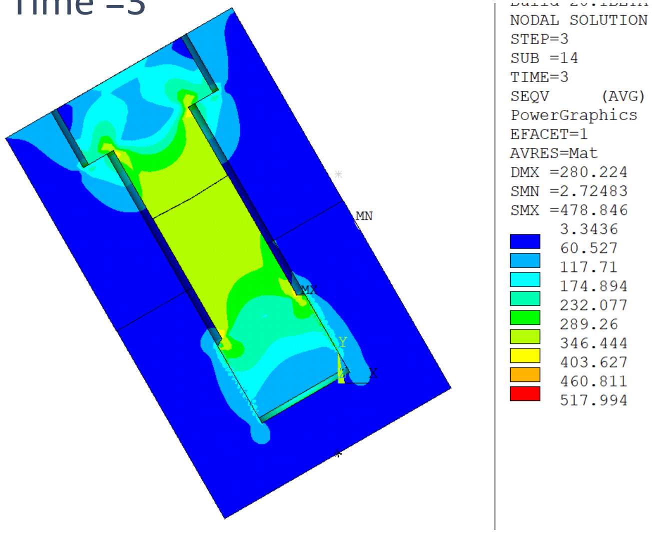


Technique with PRETE179

Time =2



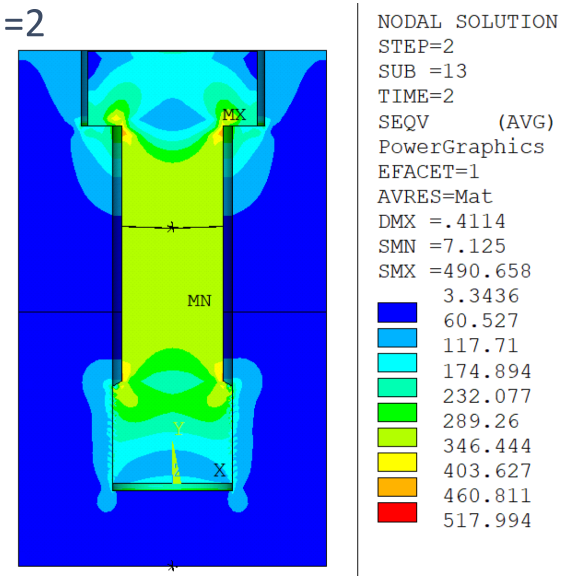
Time =3



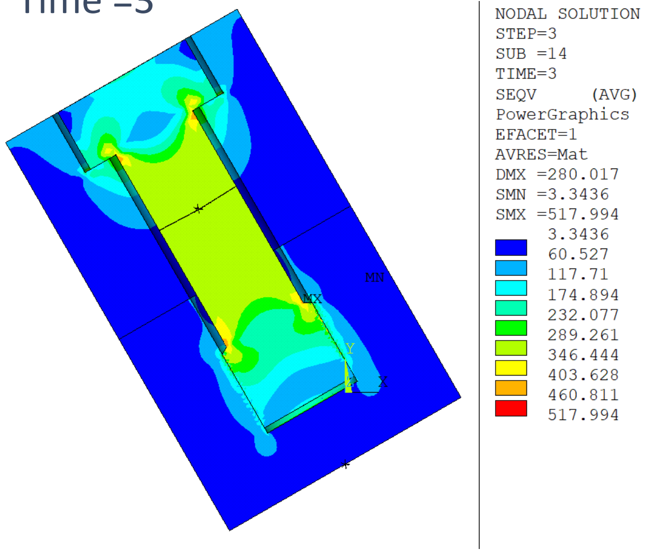
Stress changes after rigid rotation

New Technique

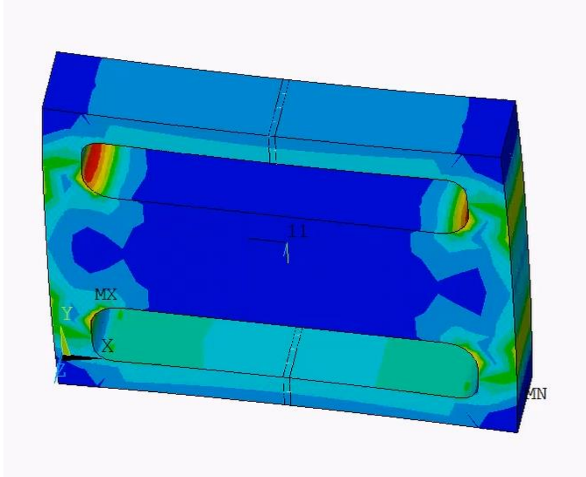
Time =2



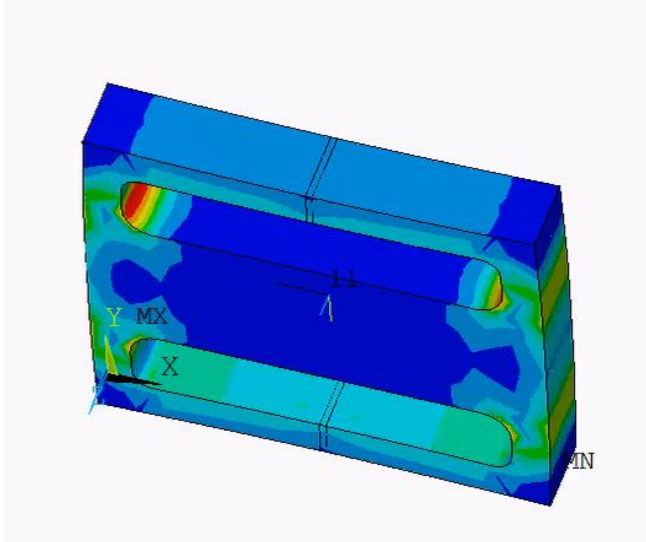
Time =3



Technique with PRETE179

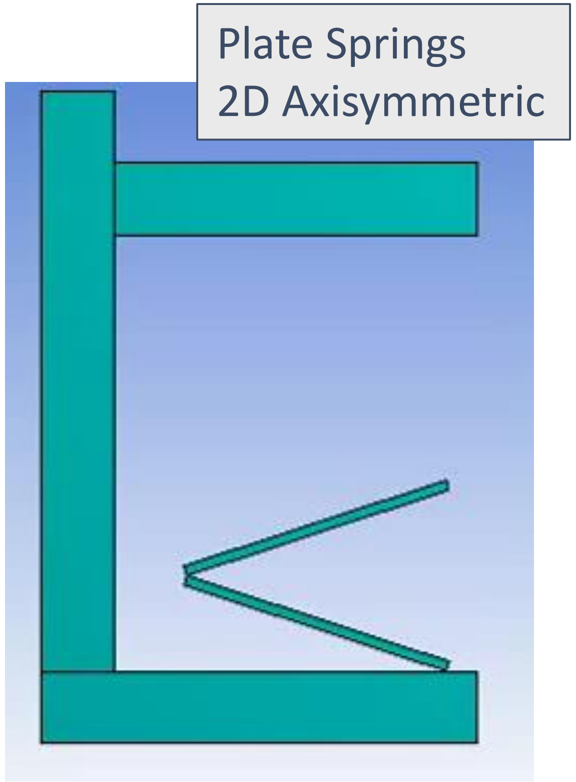


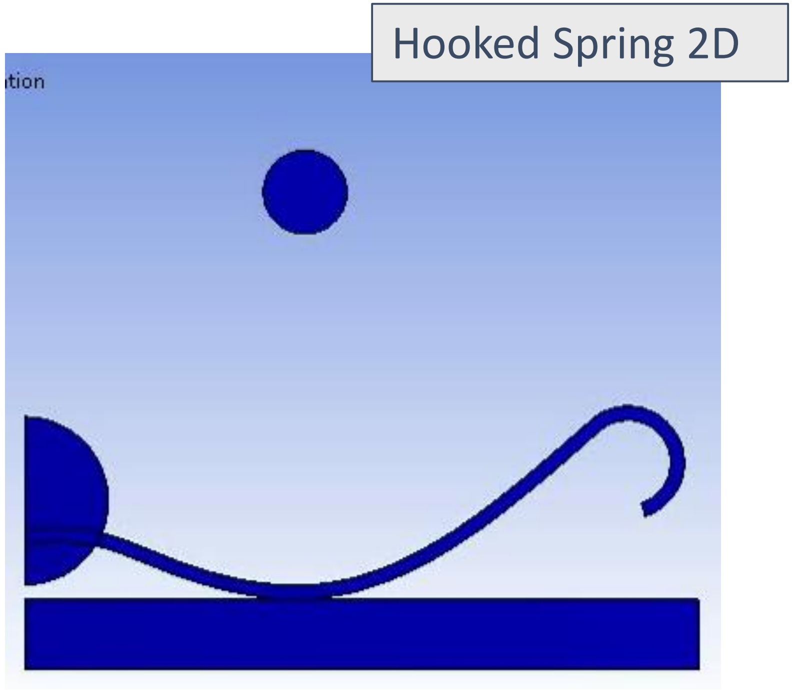
New Technique

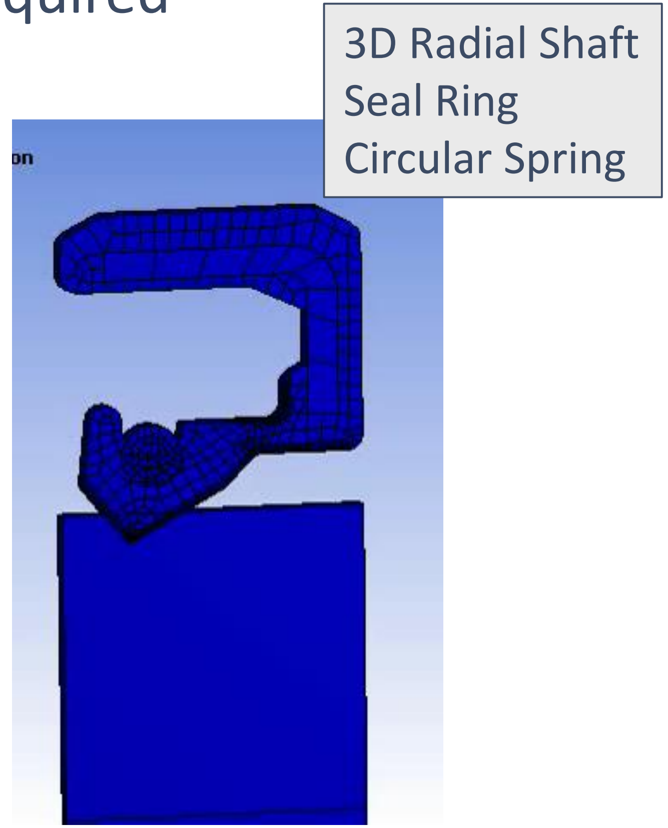


New Interference Fit Modeling Technique

- A new method for solving interference fit ramps the normal and tangential contact stiffness (FKN, FKT) as well as the friction coefficient (MU) from near-zero up to actual values. The ramping method is active during a load step or a time period that you specify. The time period can be within one load step or span across several load steps. Unlike the other interference-fit methods, initial contact engagement is not required

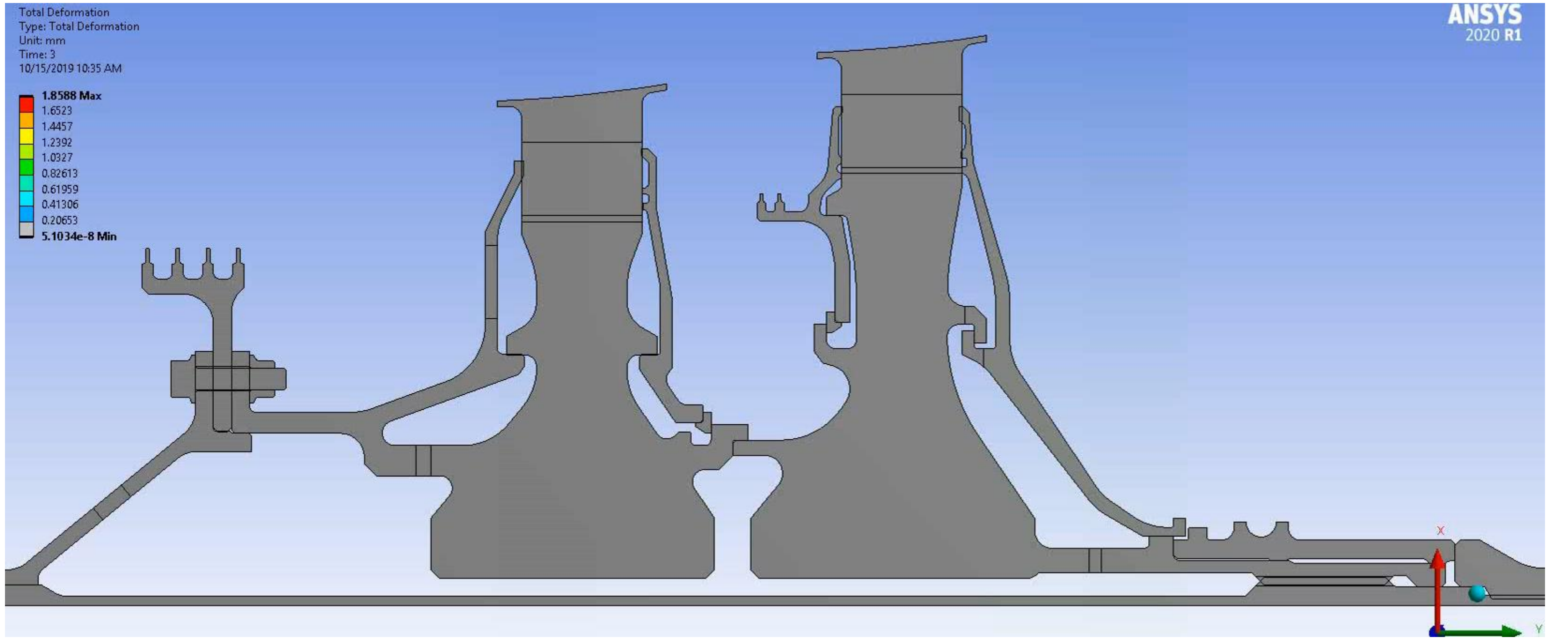


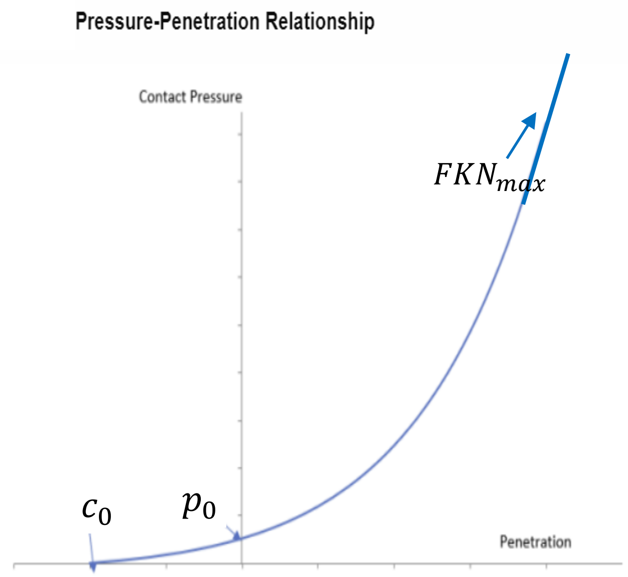




Exponential Pressure - Penetration Relationship

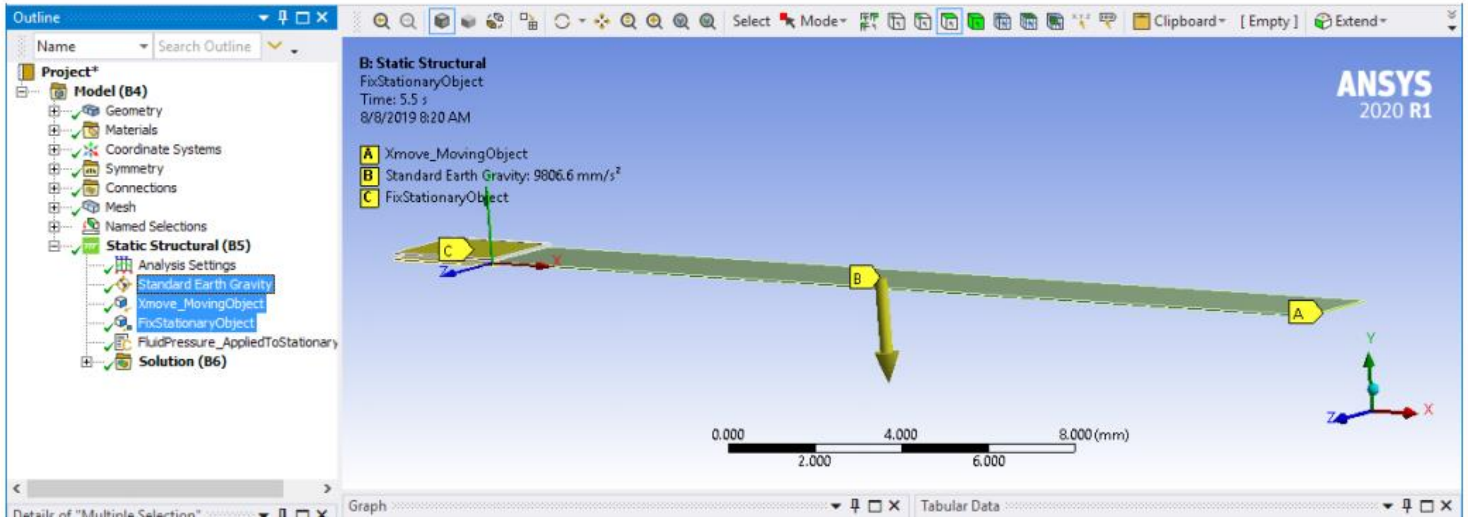
- The exponential pressure-penetration relationship (KEYOPT(6) = 3 on contact elements) can make contact behavior smoother. While the default settings of pressure at zero penetration (real constant PZER) and initial contact clearance (real constant CZER) work well for most contact models, some cases require non-default values to achieve convergences. You now have the option to define PZER and CZER as scaling factors. Previously, only absolute values could be input for the real constants
- In addition, the maximum cut-off contact stiffness FKN_{max} used in the exponential pressure-penetration relationship has been revised to prevent ill-conditioning of the global stiffness matrix

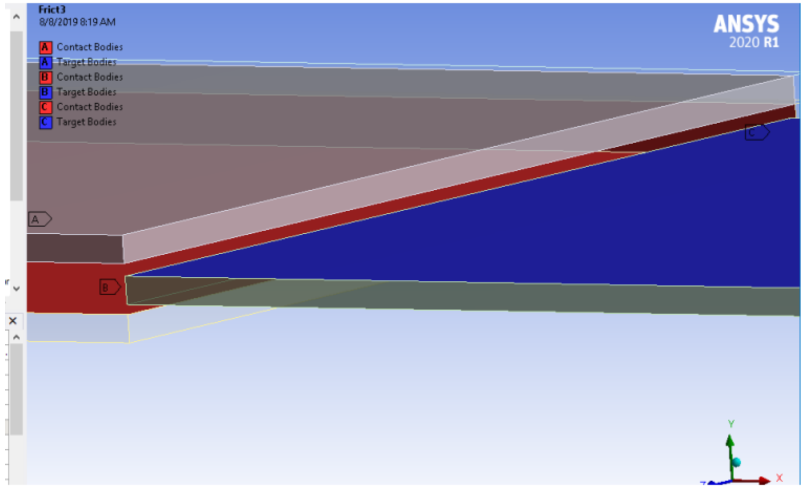


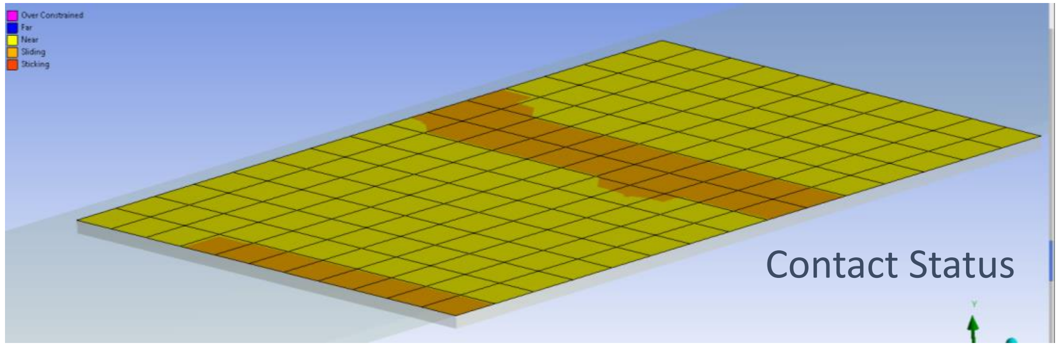


Tangential Fluid Pressure-Penetration Loads

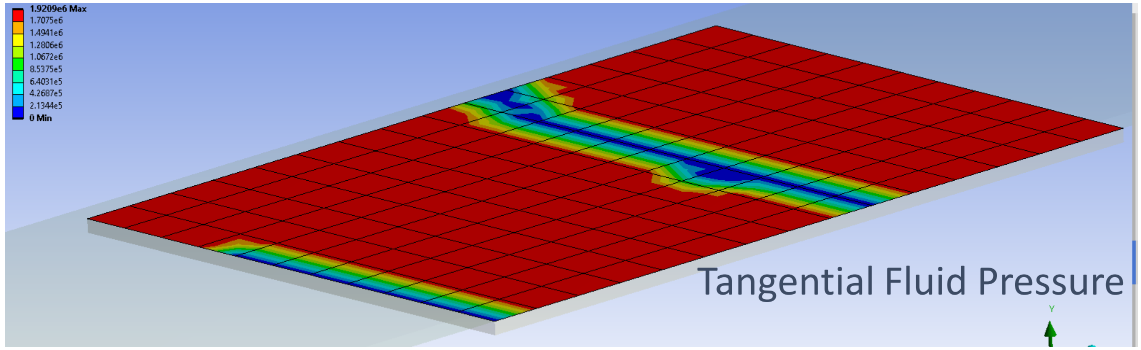
- The 2-D and 3-D surface-to-surface contact elements (CONTA172, CONTA174) and their corresponding target elements (TARGE169, TARGE170) now support tangential fluid pressure-penetration loads. In prior releases, only normal fluid pressure-penetration loads were considered
- Example: Viscous shear in thin film of fluid between plates dominates resistance to relative plate movement







Contact Status

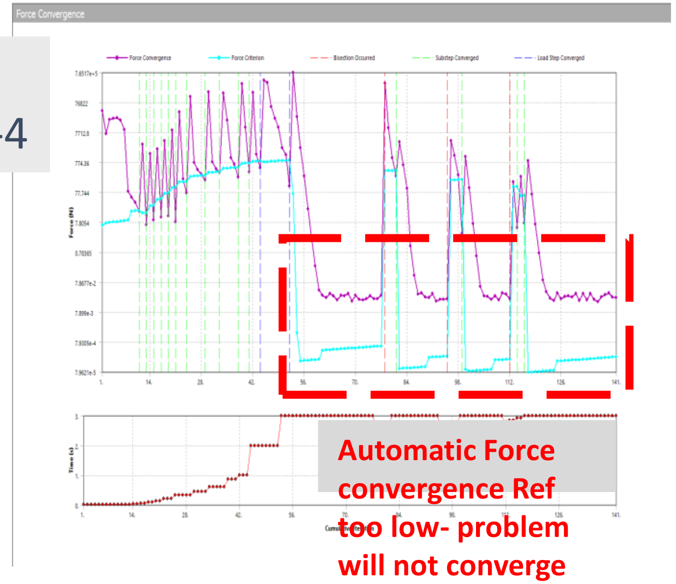


Tangential Fluid Pressure

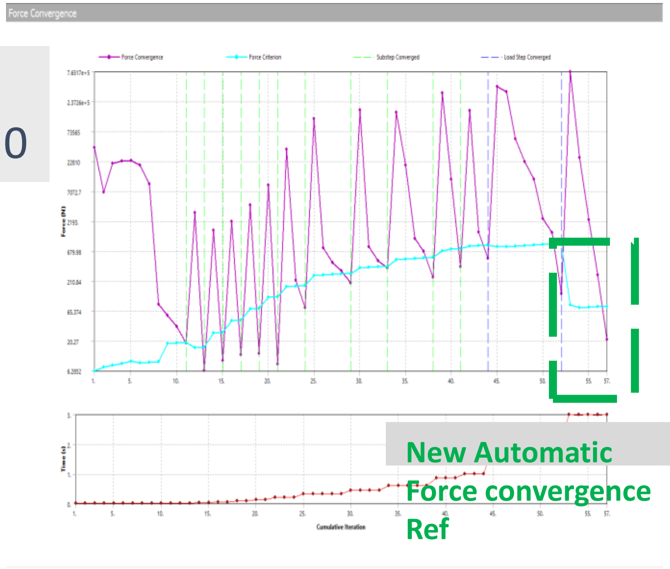
Enhanced Force, Moment, and Displacement Convergence

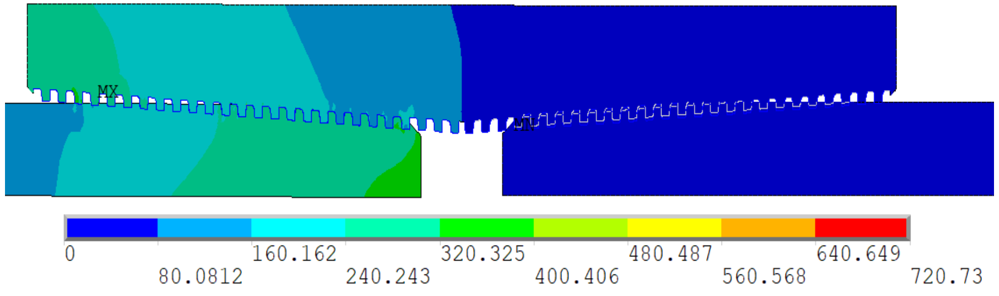
- Predictor, Force & Displacement Convergence Checks
  - Linear, quadratic predictor, Predictor off for transient. Bisection due to large displacement increment
  - Current non-linear convergence check Reference values suffer several drawback
  - Reference too low- problem will never converge
  - Reference too high- problem converges to wrong solution
  - New **“Convergence Reference”** logic aimed to provide accurate solution with minimal user intervention
  - The changes improve the robustness and accuracy of the solution, and nonlinear problems with no external loads (such as initial penetration resolution for contact and free thermal expansion) experience enhanced convergence.

Existing criterion 1.e-4



New criterion 1.0





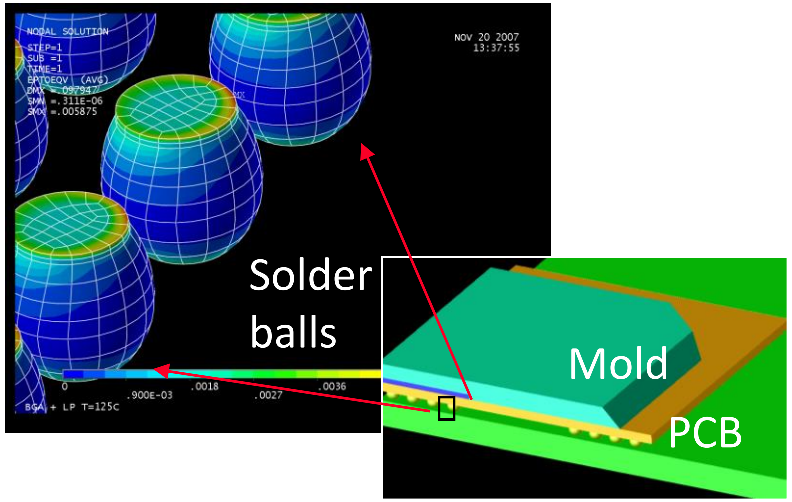
APDL Solver

Distributed ANSYS Enhancements

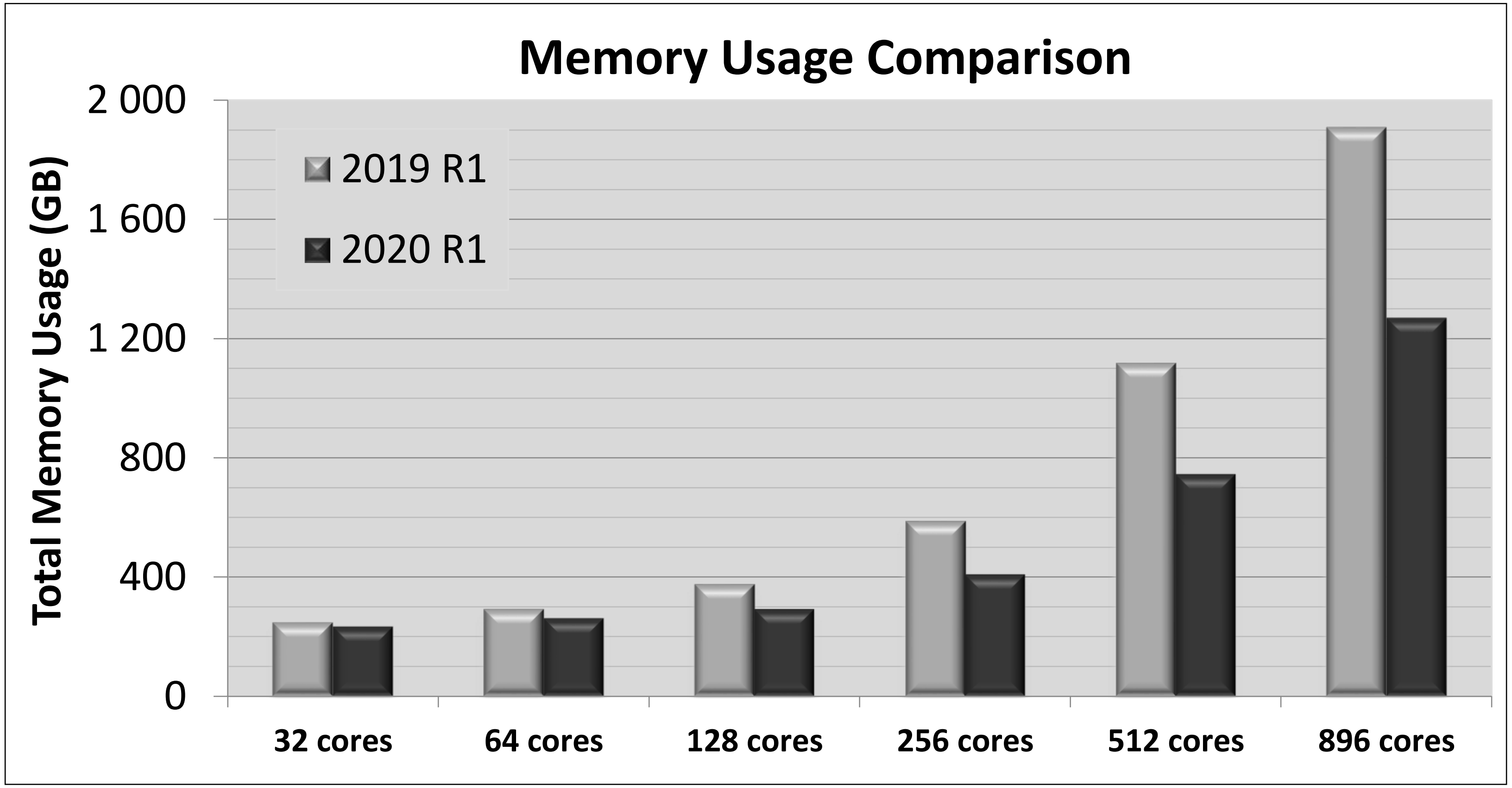
- **New features**
  - New –rcopy argument to specify remote file copy command on clusters (defaults to scp)
  - Added logic to detect SIGKILL/SIGABRT signals and provide relevant message to user
- **Improved scaling**
  - Significantly reduced memory requirements at higher core counts
  - Improved Block Lanczos scaling performance at higher core counts
  - Faster performance for fracture parameter calculations
  - Faster performance in sparse solver when running in the out-of-core memory mode on systems, which use the Microsoft Windows operating system

Distributed ANSYS Enhancements

- Significantly reduced memory usage (BGA model)

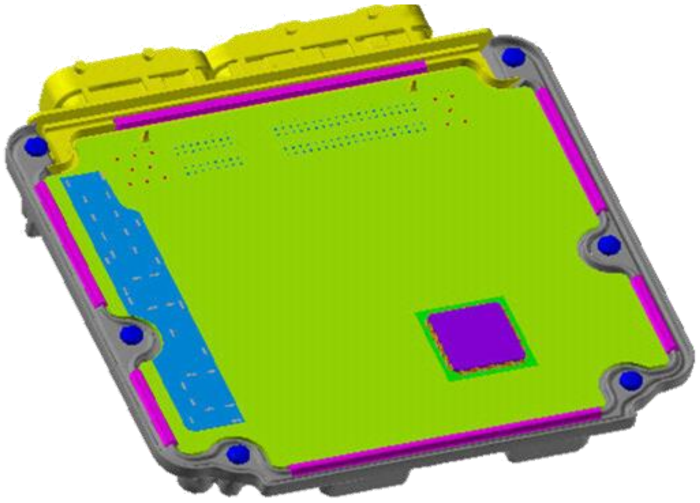


- 16 MDOF; sparse solver
- Nonlinear transient analysis involving creep and nonlinear geometric effects
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, CentOS 7.6

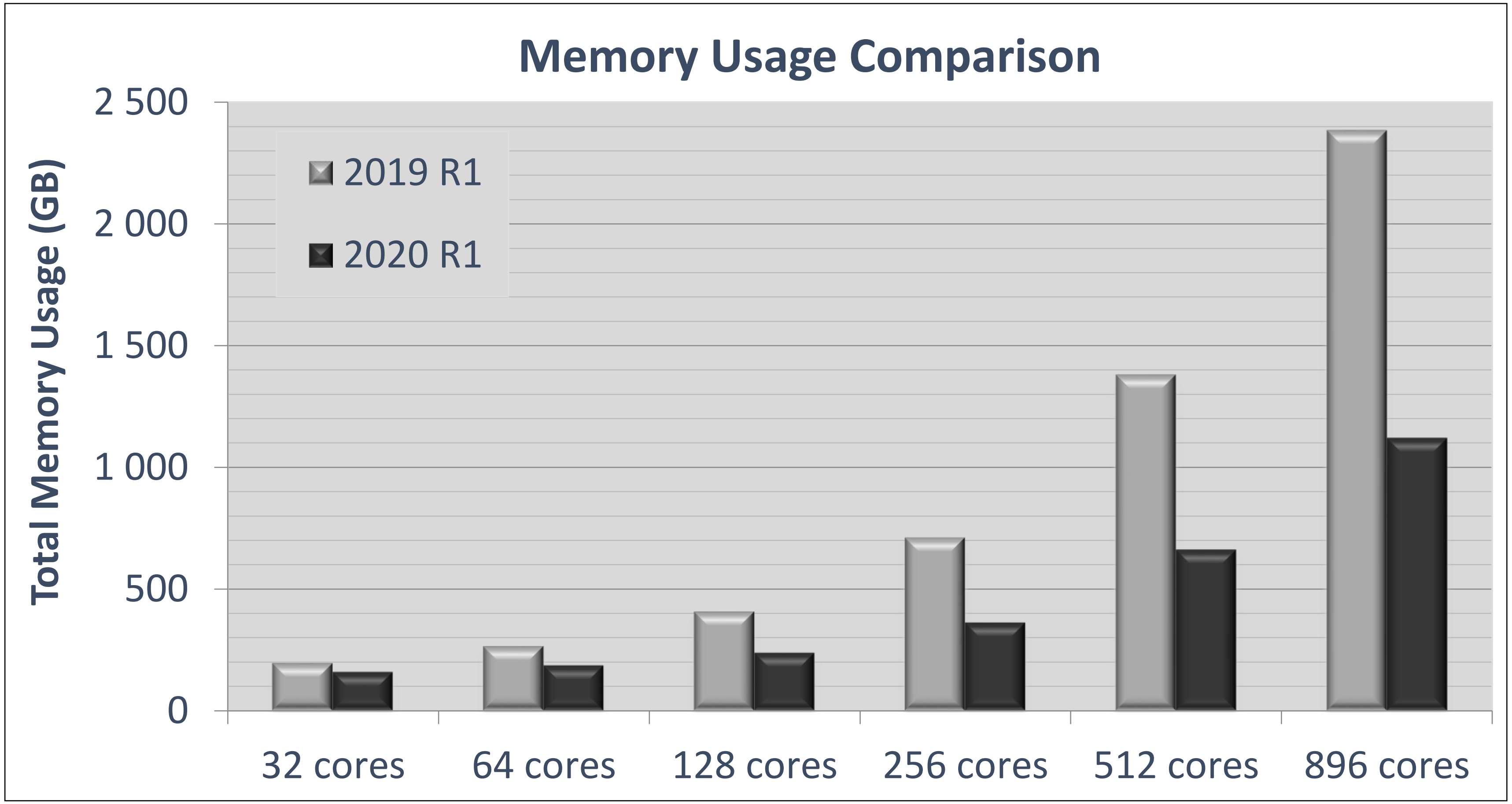


Distributed ANSYS Enhancements

- Significantly reduced memory usage (EPD model)

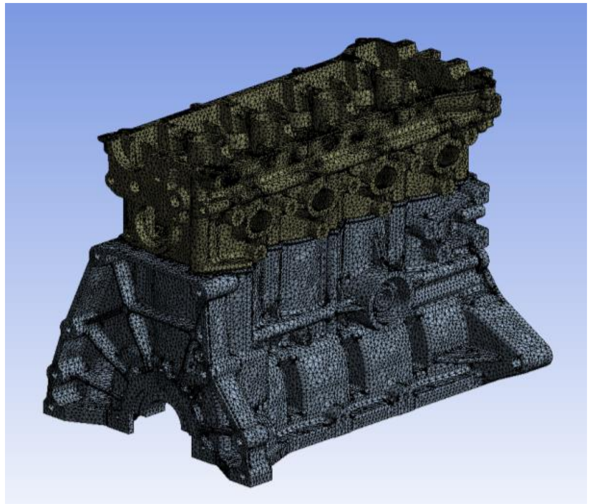


- 10 MDOF; sparse solver
- Nonlinear static analysis involving plasticity
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, CentOS 7.6

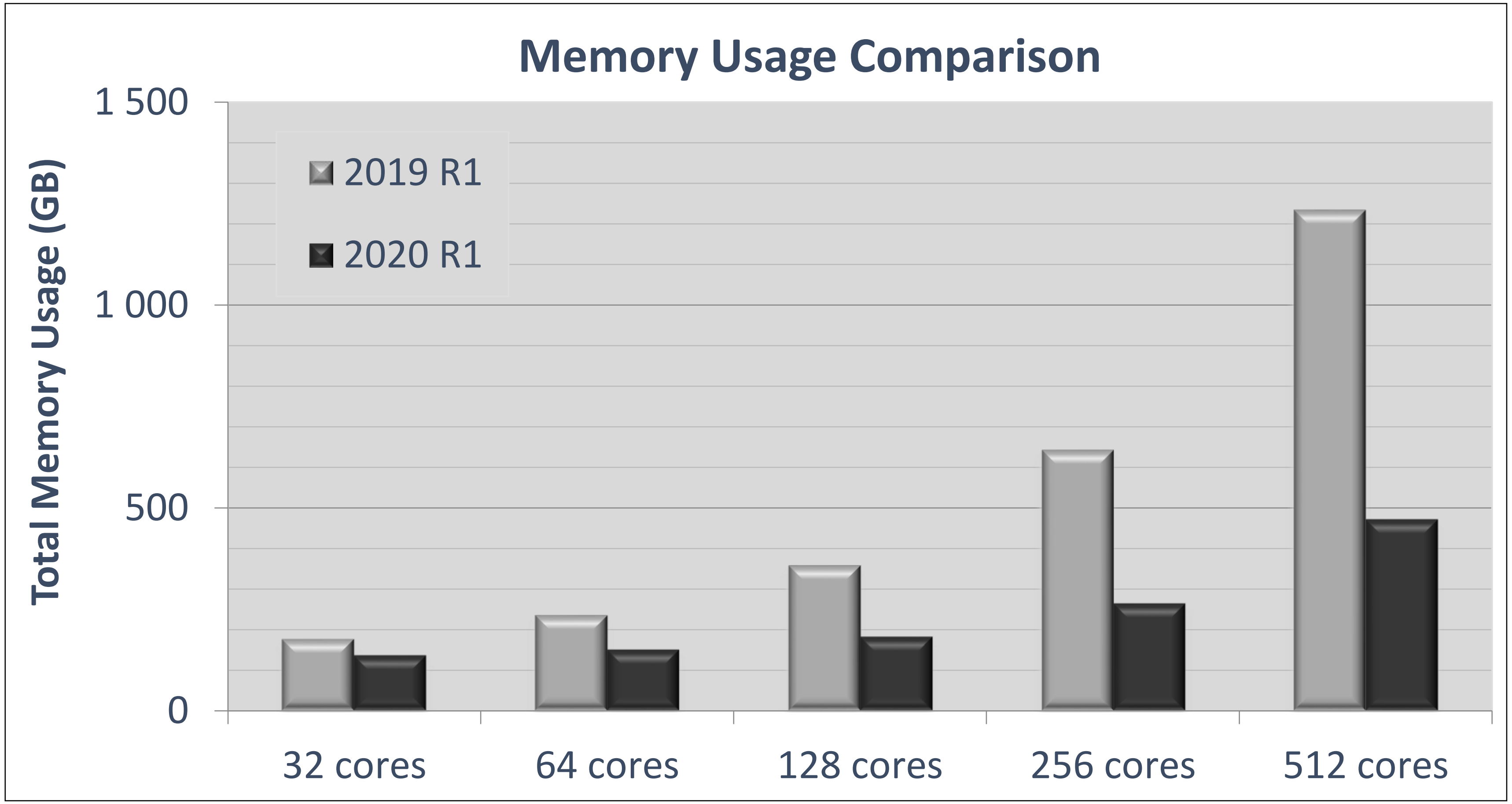


Distributed ANSYS Enhancements

- Significantly reduced memory usage (Engine model)



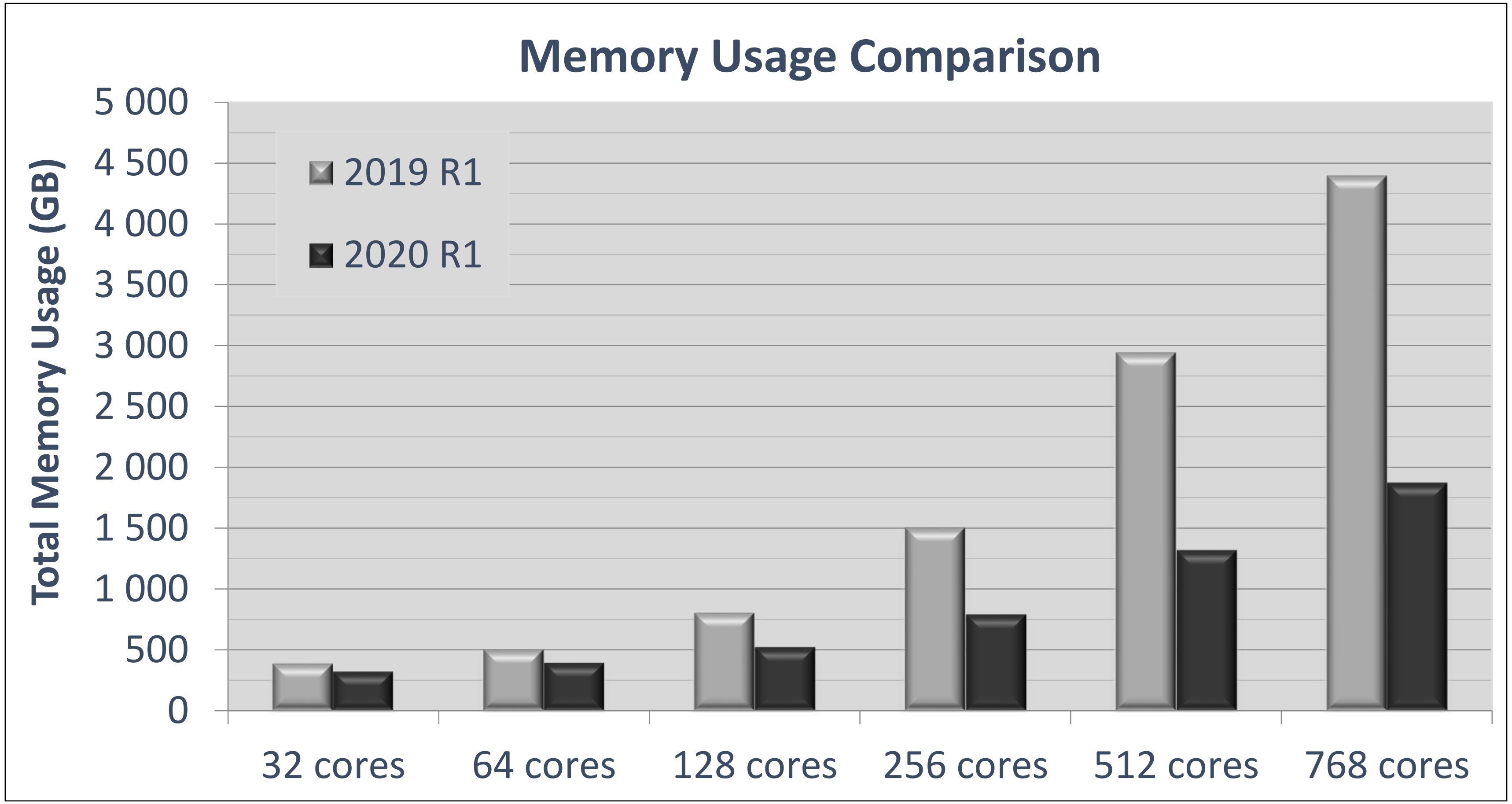
- 9.1 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity and gasket elements
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6142 processors, 384GB RAM, SSD, CentOS 7.4
- Mellanox EDR Infiniband



Distributed ANSYS Enhancements

- Significantly reduced memory usage (ECU model)

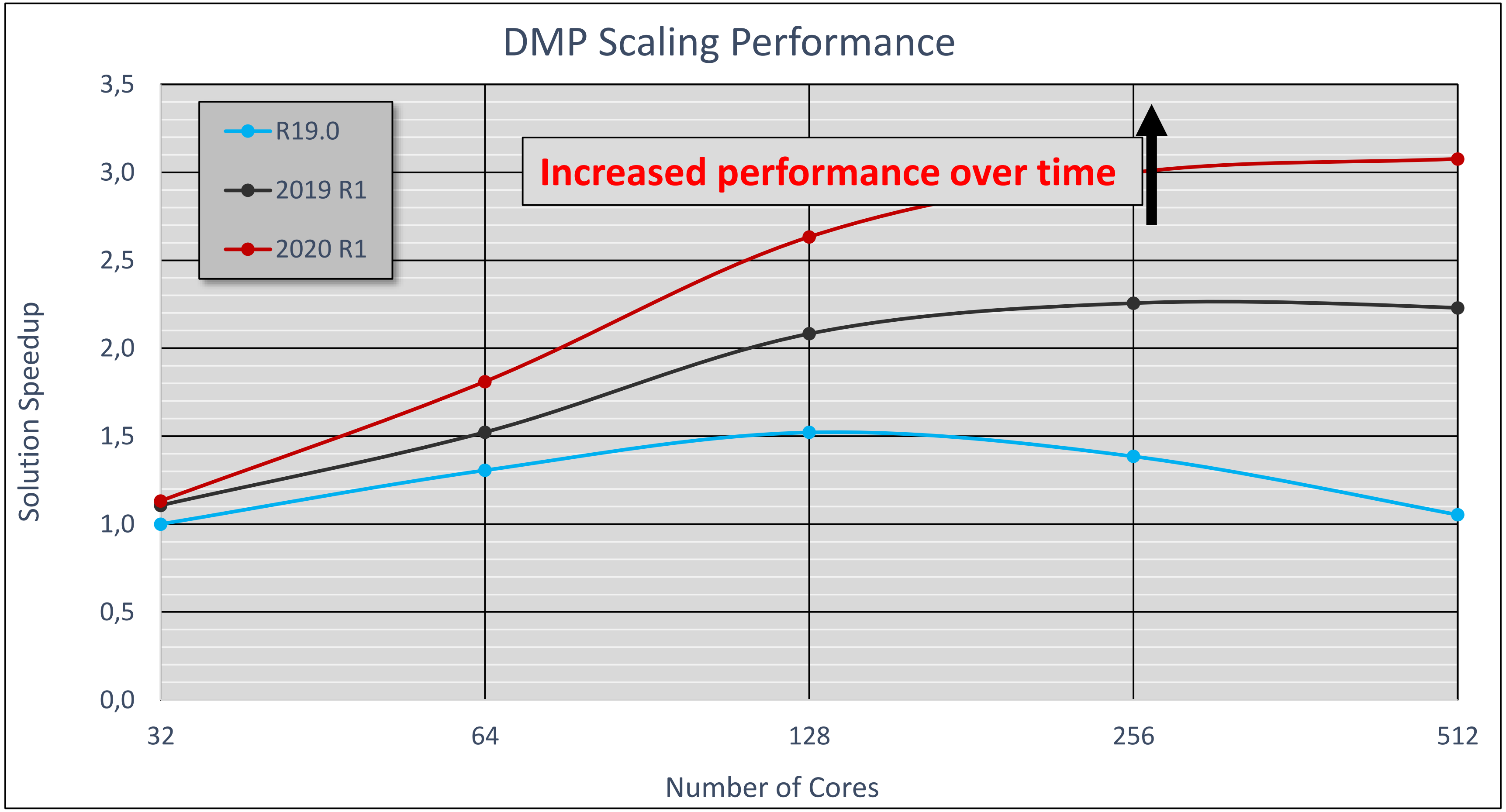
- 9.7 MDOF; Block Lanczos eigensolver
- Modal analysis requesting 100 modes; includes expansion step
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, CentOS 7.6



Distributed ANSYS Enhancements

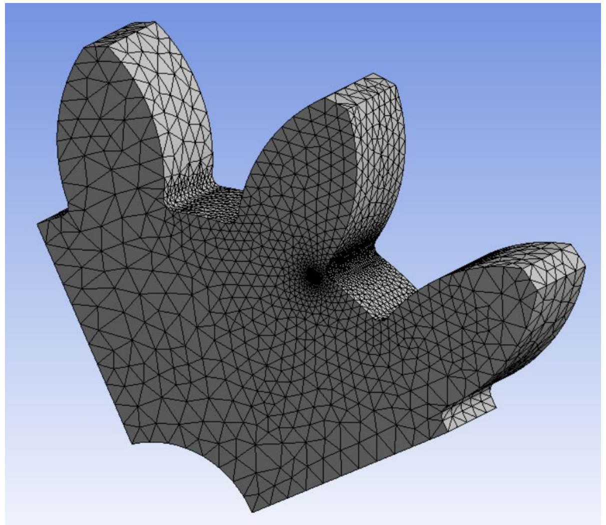
- Improved scaling for Block Lanczos eigensolver (ECU model)

- 9.7 MDOF; Block Lanczos eigensolver
- Modal analysis requesting 100 modes; includes expansion step
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors (40 cores), 384GB RAM, SSD, CentOS 7.6

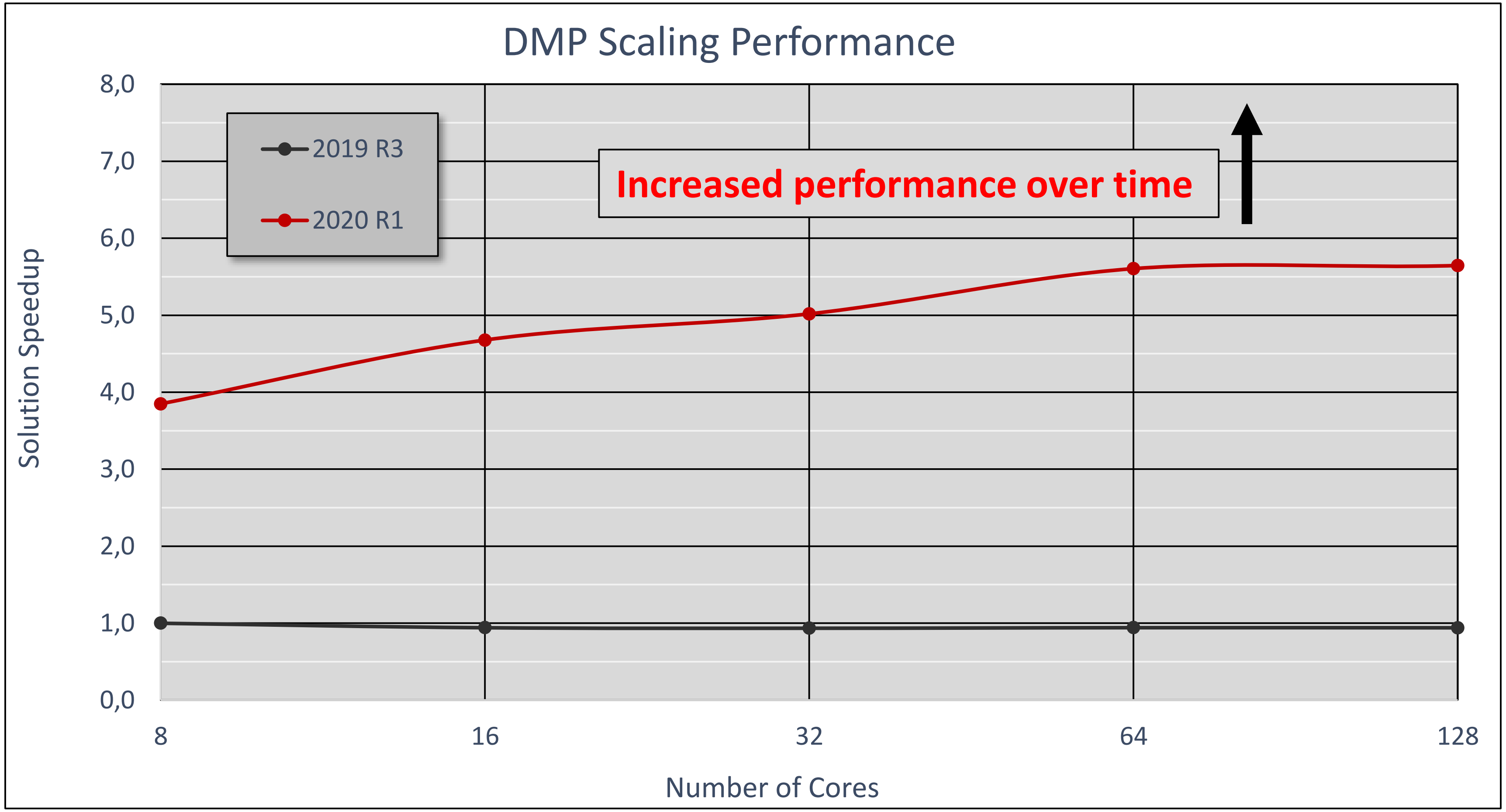


Distributed ANSYS Enhancements

- Improved performance for fracture parameter calculations (gear tooth)



- 2.2 MDOF; sparse solver
- Linear static analysis involving fracture parameter calculations
- Linux cluster; each compute node contains 2 Intel Xeon E5-2690 processors (28 cores), 128GB RAM, SSD, CentOS 6.7

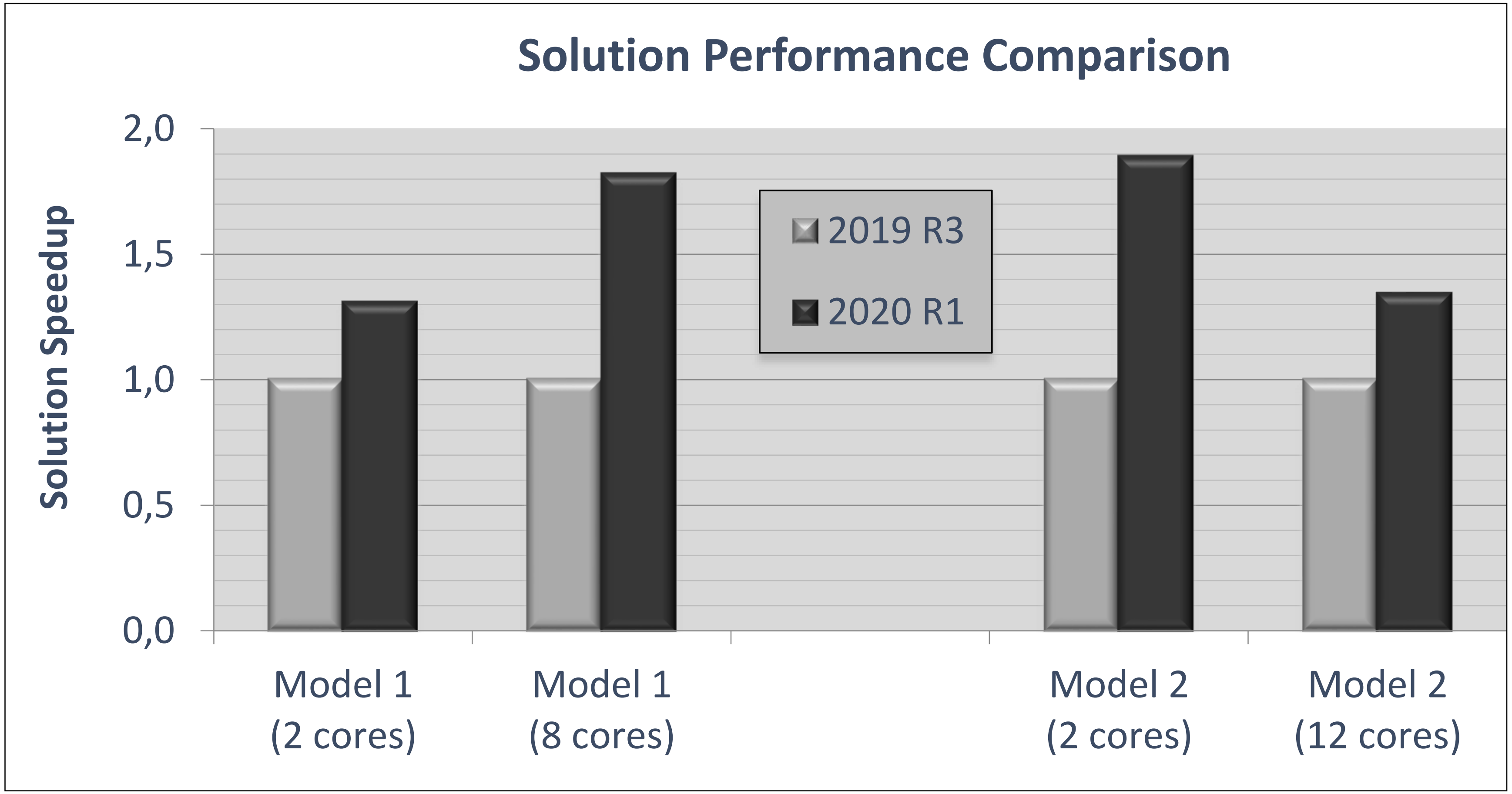


Distributed ANSYS Enhancements

- Improved sparse solver performance using Windows I/O functions

Model 1
- 4.4 MDOF; sparse solver, out-of-core memory mode
- Nonlinear static analysis
- Windows workstation containing an Intel Xeon E5-2687W processors (12 cores), 64 GB RAM, 10k RPM hard drive, Windows 10

Model 2
- 9 MDOF; sparse solver, out-of-core memory mode
- Nonlinear static analysis
- Windows workstation containing an Intel Xeon E5-2687W processors (12 cores), 64 GB RAM, 10k RPM hard drive, Windows 10



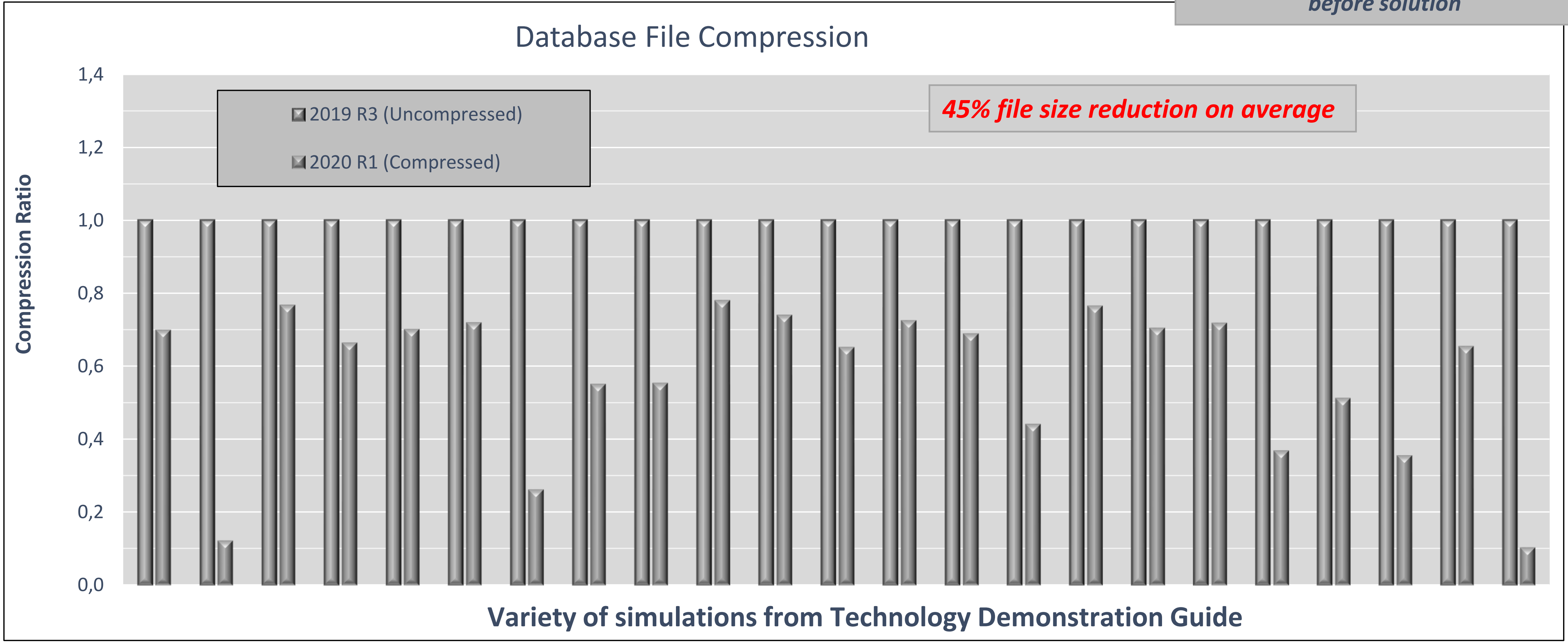
Miscellaneous Enhancements

- **Database file compression (sparsify)**
  - Activated by default via /FCOMP,DB,SPARSE
  - Can be deactivated via /FCOMP,DB,0
  - Achieves roughly 20-50% database file size reduction for most models
    - Includes .rdb database files used by the multi-frame restart procedure
    - Slightly longer times to save database files

Miscellaneous Enhancements

- Database file compression (sparsify)

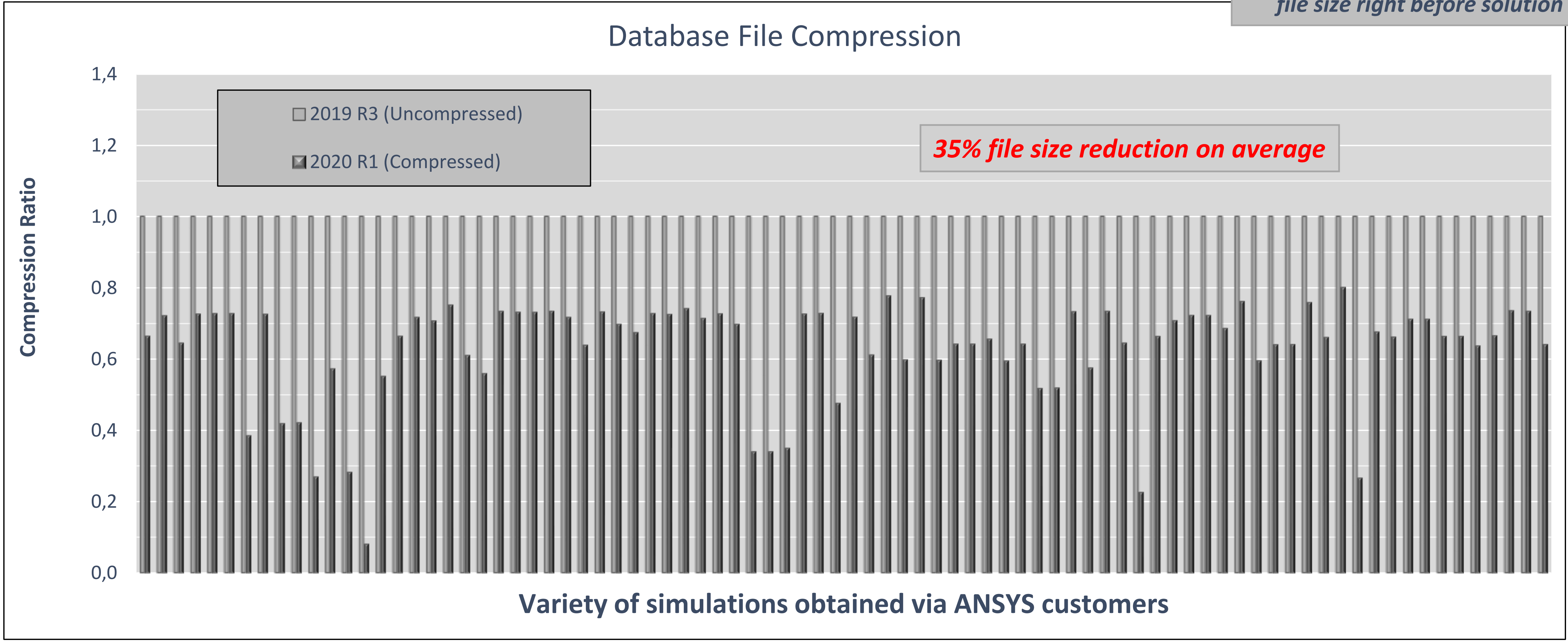
Technology Demonstration Manual models run on Linux server while measuring database file size right before solution



Miscellaneous Enhancements

- Database file compression (sparsify)

Customer models run on Linux server while measuring database file size right before solution



Miscellaneous Enhancements

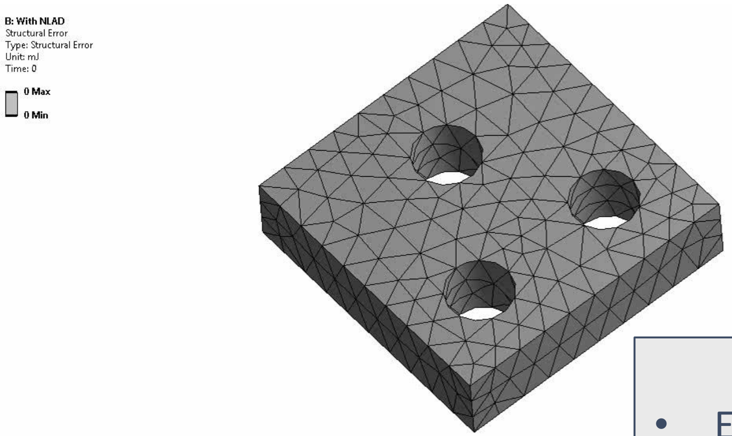
- Upgraded to the Intel 2019 Update 3 FORTRAN/C/C++ compilers
  - Includes similar updates for the Intel MKL and DAAL libraries

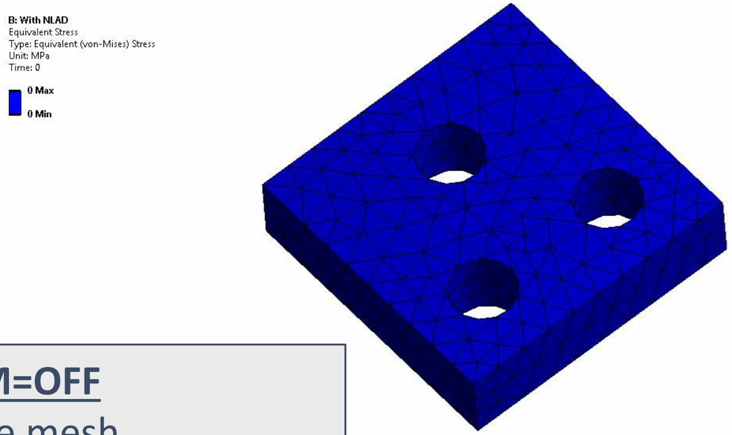
NLAD

2020 R1 Developments in NLAD

- **Current Developments in 2020 R1:**
  - NLAD with NLGEOM=OFF
  - Nonlinear Stabilization with NLAD and Rezoning
  - Mixed Remeshing
  - "KEEP" option for maintaining and updating element components in NLAD
  - Support for Tabular input of nodal temperatures and heat generation

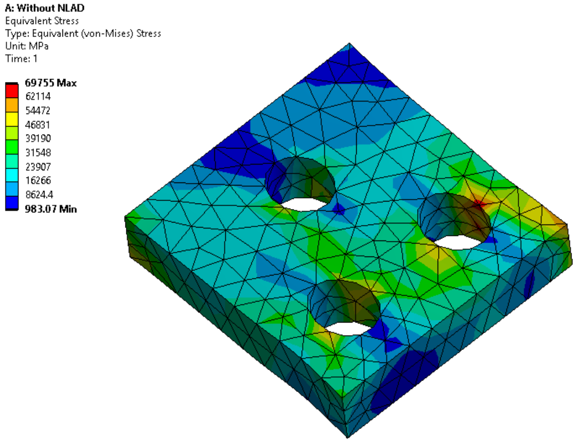
NLAD with NLGEOM=OFF



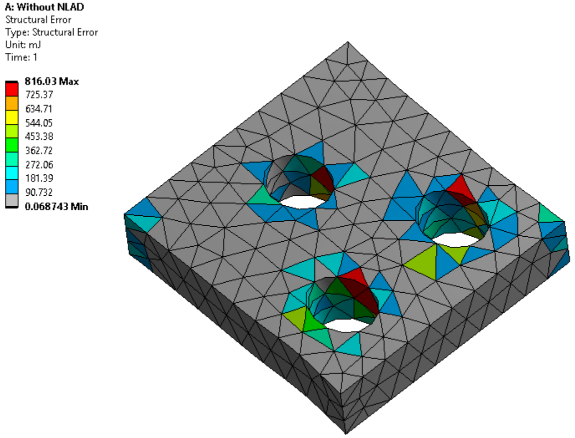


NLAD with NLGEOM=OFF

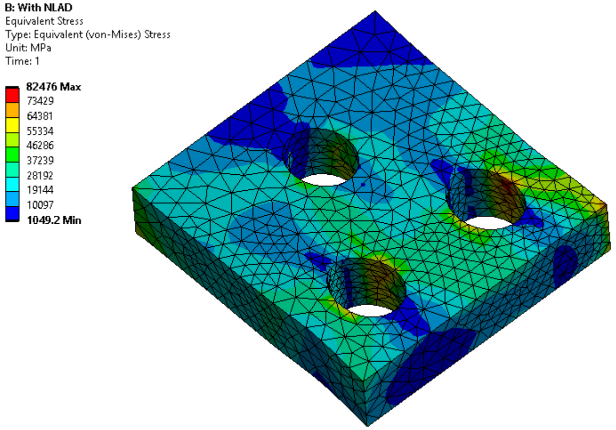
- Energy criterion used to refine mesh
- Automatic improvement of solution accuracy
- Reduction of structural errors



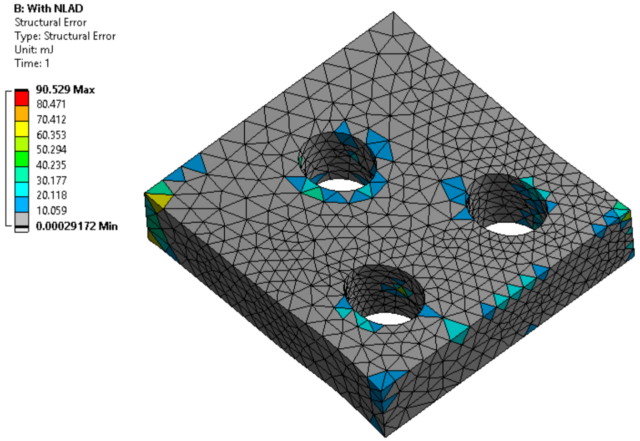
Without NLAD – Stresses



Without NLAD – Errors

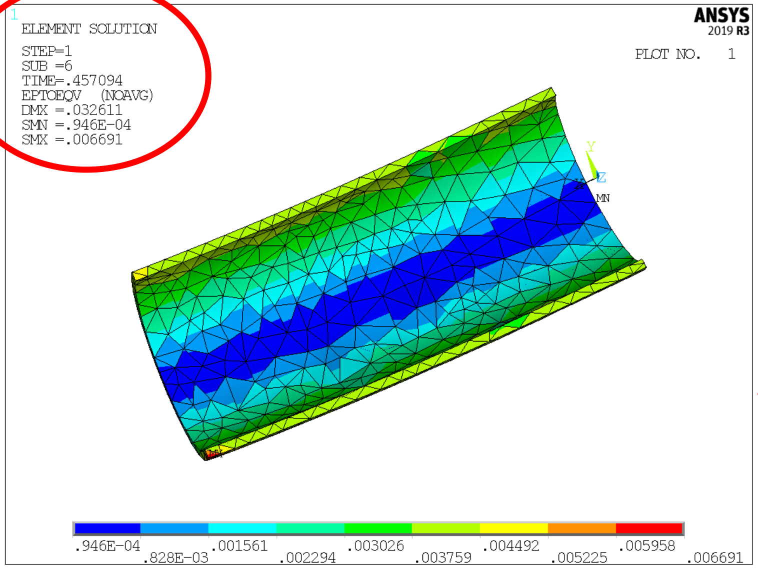


With NLAD – Stresses

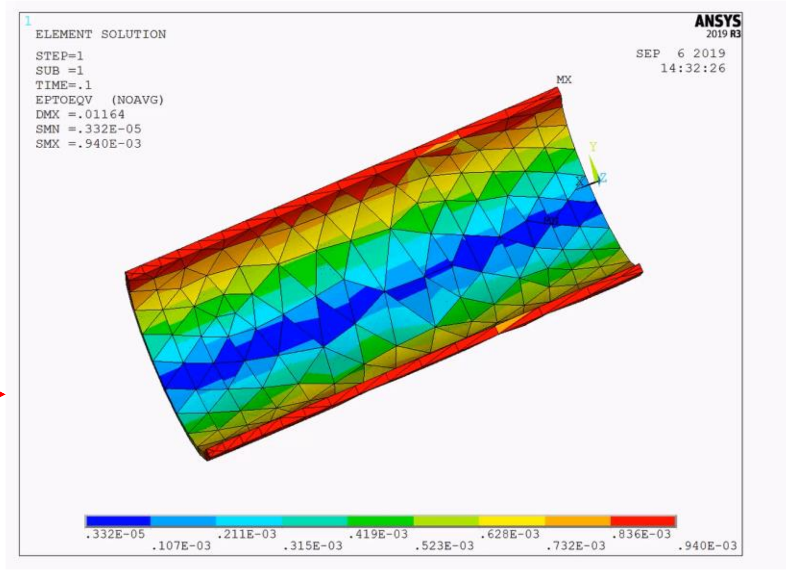


With NLAD – Errors

Nonlinear Stabilization with NLAD



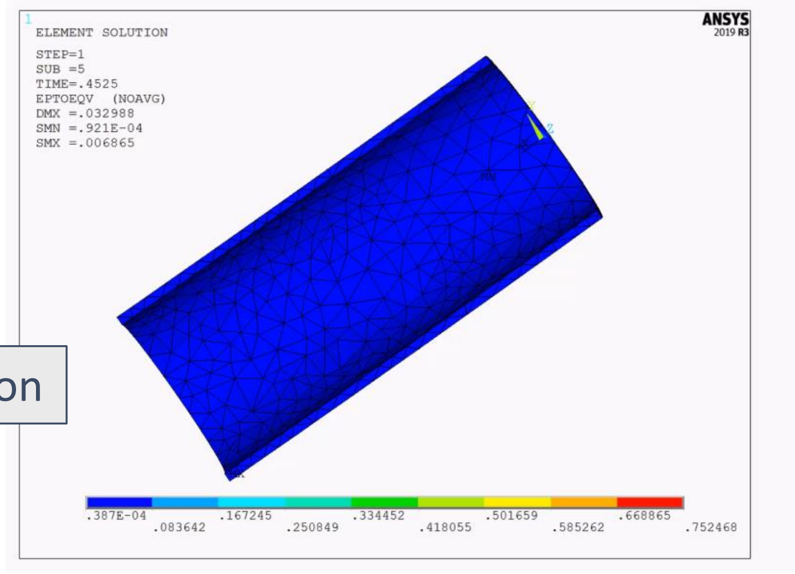
NLAD *without* stabilization:
Analysis fails due to instability



Key Features:

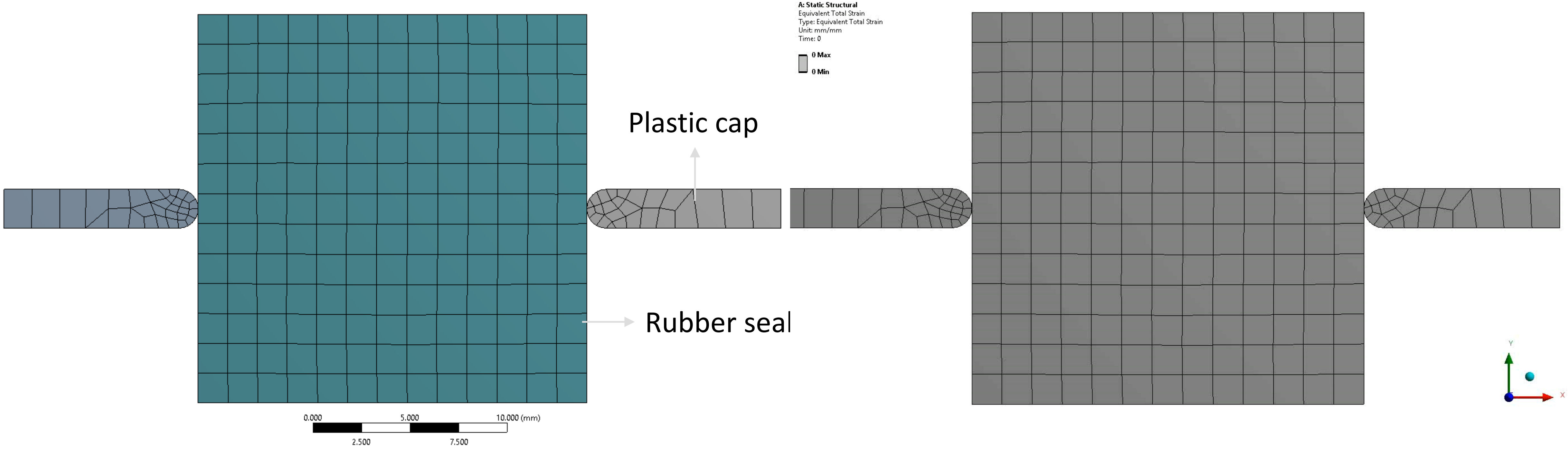
- Hollow cylindrical tube subjected to bending load
- NLAD with MESH and ENERGY criteria
- Activate global stabilization
- Stabilization included during MAPSOLVE in NLAD/Rezoning

NLAD *with* stabilization



Mixed Remeshing

- Achieve remeshing due to distortion and refinement-based criteria at the same time!




```
*** LOAD STEP     1   SUBSTEP    14  COMPLETED.    CUM ITER =     73
*** TIME =  0.303125         TIME INC =  0.506250E-01
*** AUTO STEP TIME:  NEXT TIME INC = 0.50625E-01  UNCHANGED


**** REGENERATE MESH AT SUBSTEP    14 OF LOAD STEP     1 BECAUSE OF
     NONLINEAR ADAPTIVE CRITERIA

     PREPARING DATA TO REMESH......

     REMESHING REGIONS ARE CREATED; GENERATING NEW MESH......


     REMESHING WITH BOTH DISTORTION AND REFINEMENT REGION ...
AmsMesher(ANSYS Mechanical Solver Mesher),Graph based ANSYS Meshing EXtension,v
```

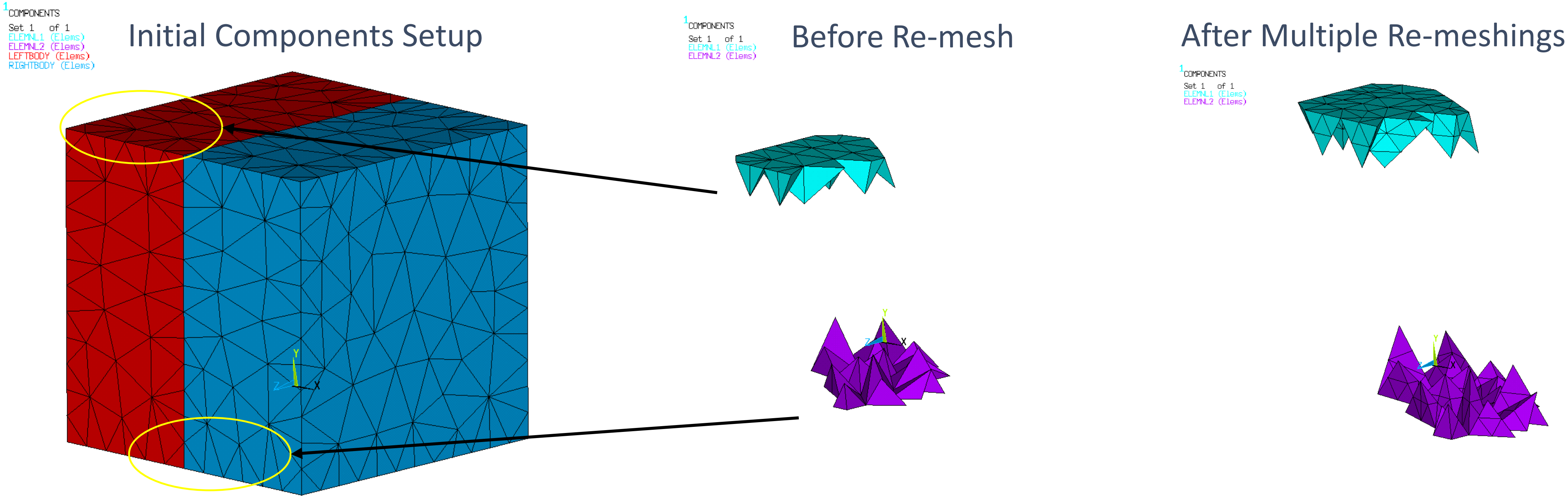
Solution output now shows which method is being tried

Key Features:

- Initially coarse mesh refined using the mixed remeshing capability
- Leads to smoother contact conditions during solution

“Keep” Option for Maintaining And Updating Element Components in NLAD

- Element components for which NLAD is **NOT** defined are now updated based on mesh changes using the **“KEEP”** option **CM, *Cname*, Entity, KEEP**



Extended Post-processing

History Variables

- New options for the **"History Variables"** by default : set to "No"
- When it is set, WB LS-DYNA sets automatically the number of history variables, it calculates the maximum depending on the material and on the type of bodies available in the model
- *Neiph* is for solid
- *Neips* is for shell
- The results are available through the worksheet

Output Controls	
Output Format	Program Controlled
Binary File Size Scale Factor	70
Stress	Yes
Strain	No
Plastic Strain	Yes
History Variables	No
Calculate Results At	Program Controlled
Stress File for flexible parts	No

```
*DATABASE_EXTENT_BINARY
$    neiph     neips    maxint    strflg    sigflg    epsflg    rltflg    engflg
         5         5         0         0         1         1         0         0
$   cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat
         0         0         0         0         0         0         0         0
$  nintsld   pkp_sen      sclp     hydro     msscl     therm    intout    nodout
         0         0         0         0         2         0
```

SHELLHIST_1	Element Nodal	Scalar
SHELLHIST_2	Element Nodal	Scalar
SHELLHIST_3	Element Nodal	Scalar
SHELLHIST_4	Element Nodal	Scalar
SHELLHIST_5	Element Nodal	Scalar
SOLIDHIST_1	Element Nodal	Scalar
SOLIDHIST_2	Element Nodal	Scalar
SOLIDHIST_3	Element Nodal	Scalar
SOLIDHIST_4	Element Nodal	Scalar
SOLIDHIST_5	Element Nodal	Scalar

Additional Results - History Variables

- The backstress for the bilinear isotropic hardening is now available through the worksheet

4	Structural Steel NL			G	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 4: Structural Steel NL

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Elasticity				
10	Bilinear Isotropic Hardening				
13	Specific Heat, C_p	434	J kg^-1...		

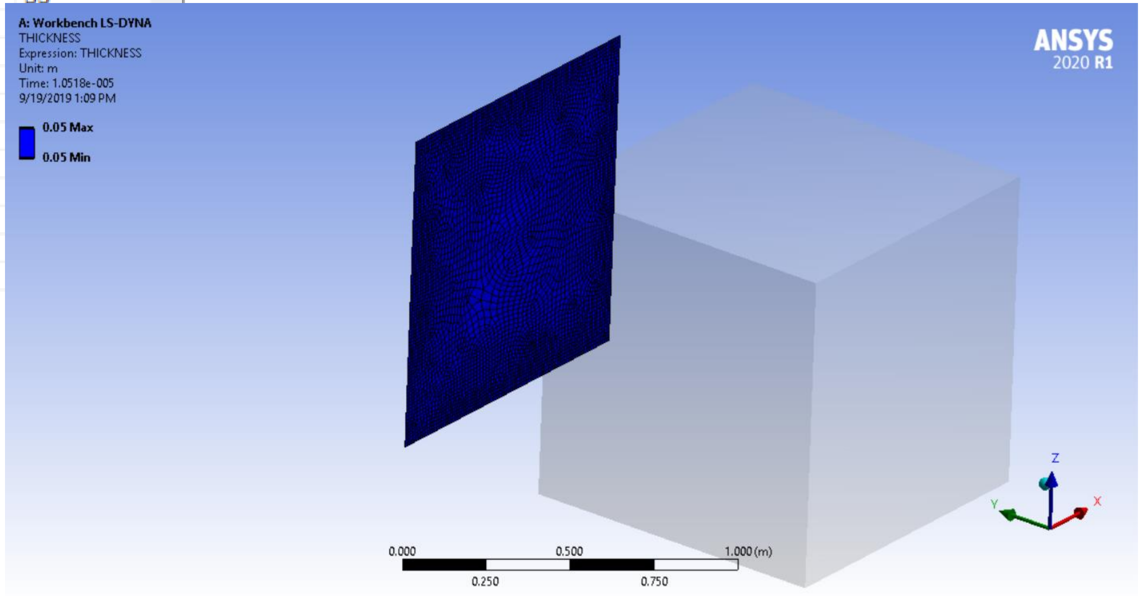
BACKSTRESS	Element Nodal	Scalar	X	E
BACKSTRESS	Element Nodal	Scalar	Y	E
BACKSTRESS	Element Nodal	Scalar	Z	E
BACKSTRESS	Element Nodal	Scalar	XY	E
BACKSTRESS	Element Nodal	Scalar	YZ	E
BACKSTRESS	Element Nodal	Scalar	XZ	E
BACKSTRESS	Element Nodal	Scalar	1	E
BACKSTRESS	Element Nodal	Scalar	2	E
BACKSTRESS	Element Nodal	Scalar	3	E
BACKSTRESS	Element Nodal	Scalar	INT	E
BACKSTRESS	Element Nodal	Scalar	EQV	E
BACKSTRESS	Element Nodal	Tensor	VECTORS	E
BACKSTRESS	Element Nodal	Scalar	MAXSHEAR	E

*MAT_003	1	back stress component xx	1	back stress component xx
	2	back stress component yy	2	back stress component yy
	3	back stress component xy	3	back stress component xy
	4	back stress component yz	4	back stress component yz
	5	back stress component zx	5	back stress component zx

Additional Results - Thickness

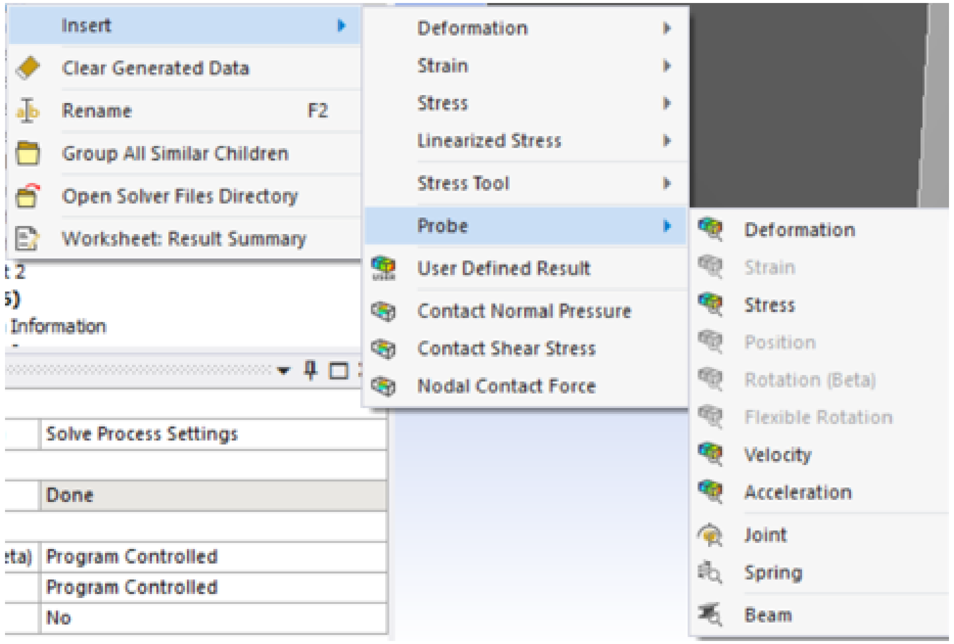
- The thickness evolution during the calculation is available as a new postprocessing item

Type	Data Type	Data Style	Component	Expression
S	Element Nodal	Scalar	XY	SXY
S	Element Nodal	Scalar	YZ	SYZ
S	Element Nodal	Scalar	XZ	SXZ
S	Element Nodal	Scalar	1	S1
S	Element Nodal	Scalar	2	S2
S	Element Nodal	Scalar	3	S3
S	Element Nodal	Scalar	INT	
S	Element Nodal	Scalar	EQV	
S	Element Nodal	Tensor	VECTORS	
S	Element Nodal	Scalar	MAXSHEAR	
EROSION	Elemental	Scalar		
EPS	Element Nodal	Scalar		
THICKNESS	Element Nodal	Scalar		
BACKSTRESS	Element Nodal	Scalar	X	
BACKSTRESS	Element Nodal	Scalar	Y	
BACKSTRESS	Element Nodal	Scalar	Z	



Additional Results - Probes

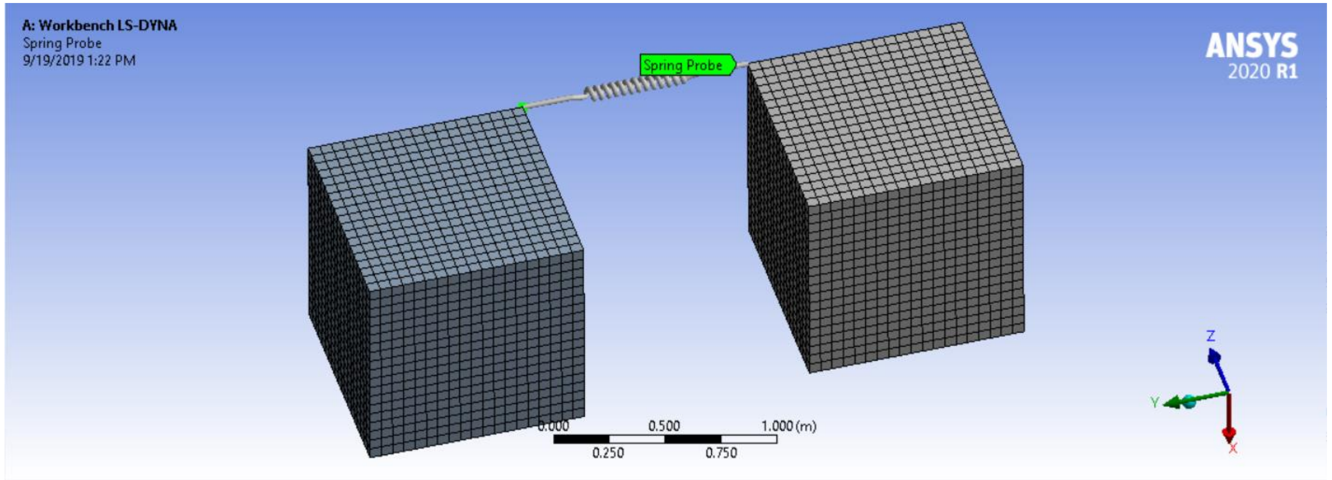
- The following three probe types are now available within the Mechanical environment
  - Joint Probe
  - Spring Probe
  - Beam

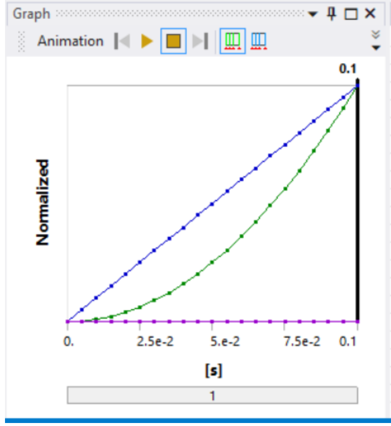


Spring Probe

Details of "Spring Probe"

Definition	
Type	Spring Probe
Boundary Condition	Spring
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
Elastic Force	4.9022 N
Damping Force	0. N
Elongation	4.9039e-002 m
Velocity	0.97972 m/s
Velocity 1	0.
Velocity 2	0.
Damping Force 1	0.
Damping Force 2	0.

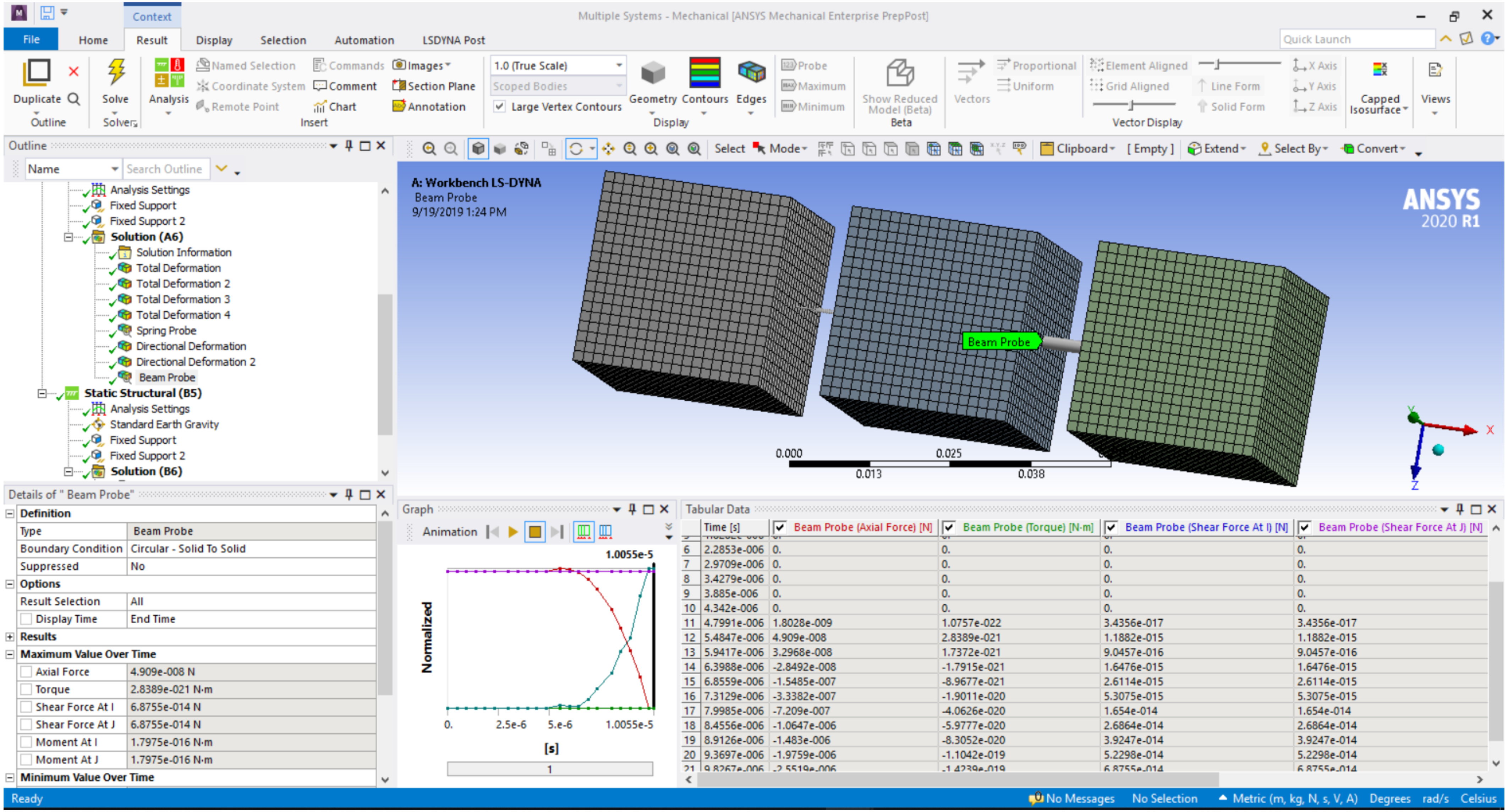




Tabular Data

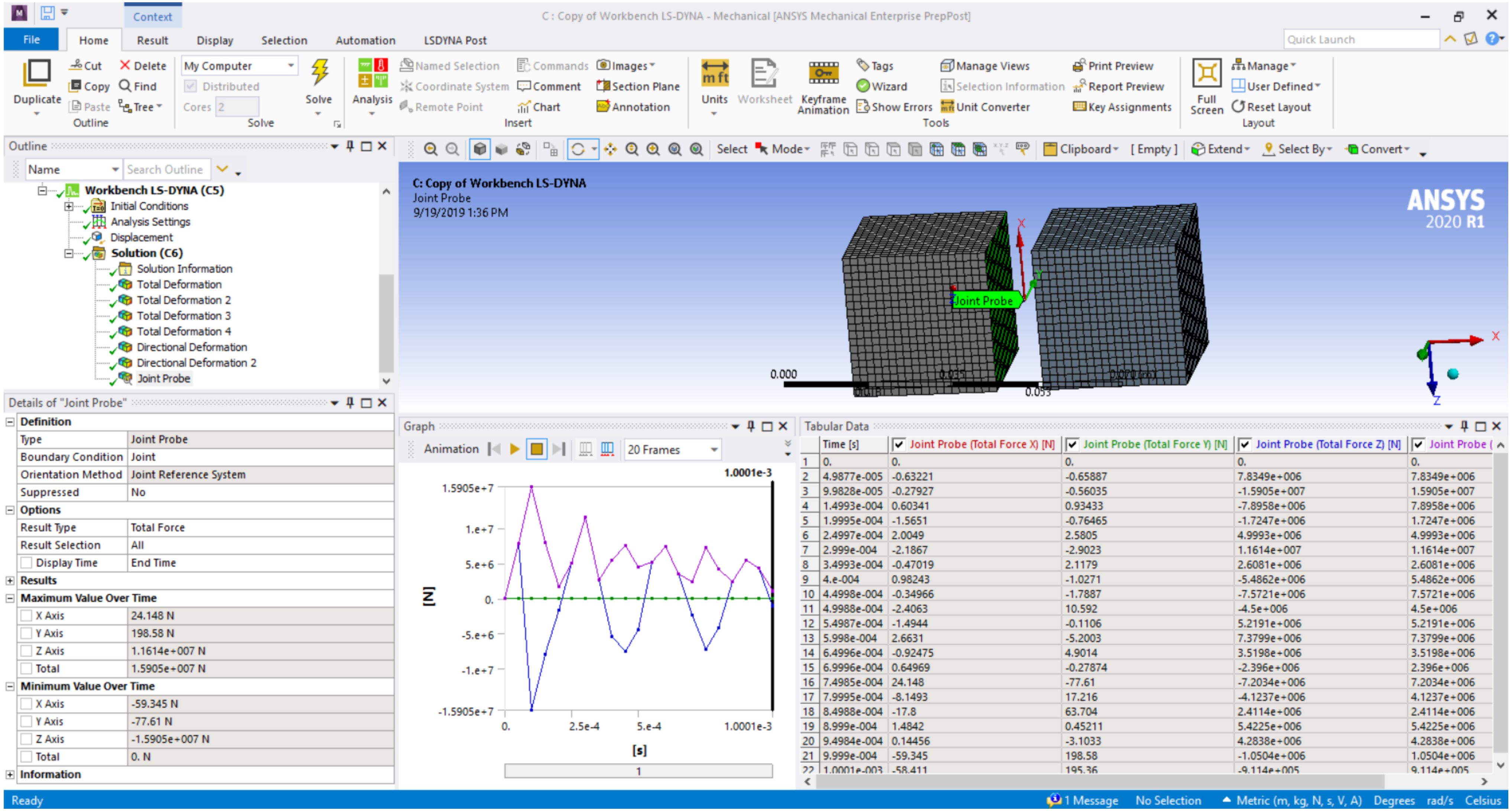
	Time [s]	Spring Probe (Force) [N]	Spring Probe (Elongation) [m]	Spring Probe (Velocity) [m/s]	Spring Probe (Damping Force) [N]
1	0.	0.	0.	0.	0.
2	4.9972e-003	1.224e-002	1.2225e-004	4.8072e-002	0.
3	9.9943e-003	4.8983e-002	4.9005e-004	9.9188e-002	0.
4	1.4999e-002	0.11029	1.1036e-003	0.14571	0.
5	1.9996e-002	0.196	1.9611e-003	0.19686	0.
6	2.4993e-002	0.30626	3.0639e-003	0.24496	0.
7	2.9998e-002	0.4412	4.4132e-003	0.29373	0.
8	3.4995e-002	0.60043	6.0061e-003	0.34411	0.
9	3.9992e-002	0.78416	7.8437e-003	0.39098	0.
10	4.4997e-002	0.99269	9.9294e-003	0.44211	0.
11	4.9994e-002	1.2255	1.2257e-002	0.48967	0.
12	5.4999e-002	1.4831	1.4834e-002	0.53934	0.
13	5.9996e-002	1.7649	1.7652e-002	0.58895	0.
14	6.4994e-002	2.0711	2.0714e-002	0.63637	0.
15	6.9998e-002	2.4023	2.4027e-002	0.68751	0.
16	7.4995e-002	2.7574	2.758e-002	0.73463	0.
17	7.9993e-002	3.1371	3.1377e-002	0.78466	0.
18	8.4997e-002	3.5418	3.5427e-002	0.83385	0.
19	8.9995e-002	3.9704	3.9714e-002	0.88162	0.
20	9.4999e-002	4.4244	4.4254e-002	0.93266	0.
21	9.9996e-002	4.9022	4.9031e-002	0.97968	0.

Beam Probe



Multiple Systems - Mechanical [ANSYS Mechanical Enterprise PrepPost]
Context
File
Home
Result
Display
Selection
Automation
LSDYNA Post
Quick Launch
Outline
Name
Search Outline
Analysis Settings
Fixed Support
Fixed Support 2
Solution (A6)
Solution Information
Total Deformation
Total Deformation 2
Total Deformation 3
Total Deformation 4
Spring Probe
Directional Deformation
Directional Deformation 2
Beam Probe
Static Structural (B5)
Analysis Settings
Standard Earth Gravity
Fixed Support
Fixed Support 2
Solution (B6)
A: Workbench LS-DYNA
Beam Probe
9/19/2019 1:24 PM
ANSYS
2020 R1
Beam Probe
0.000
0.013
0.025
0.038
Details of "Beam Probe"
Definition
Type: Beam Probe
Boundary Condition: Circular - Solid To Solid
Suppressed: No
Options
Result Selection: All
Display Time: End Time
Results
Maximum Value Over Time
Axial Force: 4.909e-008 N
Torque: 2.8389e-021 N·m
Shear Force At I: 6.8755e-014 N
Shear Force At J: 6.8755e-014 N
Moment At I: 1.7975e-016 N·m
Moment At J: 1.7975e-016 N·m
Minimum Value Over Time
Graph
Animation
1.0055e-5
Normalized
0.
2.5e-6
5.e-6
1.0055e-5
[s]
Tabular Data
Time [s]
Beam Probe (Axial Force) [N]
Beam Probe (Torque) [N·m]
Beam Probe (Shear Force At I) [N]
Beam Probe (Shear Force At J) [N]
6 2.2853e-006 0. 0. 0. 0.
7 2.9709e-006 0. 0. 0. 0.
8 3.4279e-006 0. 0. 0. 0.
9 3.885e-006 0. 0. 0. 0.
10 4.342e-006 0. 0. 0. 0.
11 4.7991e-006 1.8028e-009 1.0757e-022 3.4356e-017 3.4356e-017
12 5.4847e-006 4.909e-008 2.8389e-021 1.1882e-015 1.1882e-015
13 5.9417e-006 3.2968e-008 1.7372e-021 9.0457e-016 9.0457e-016
14 6.3988e-006 -2.8492e-008 -1.7915e-021 1.6476e-015 1.6476e-015
15 6.8559e-006 -1.5485e-007 -8.9677e-021 2.6114e-015 2.6114e-015
16 7.3129e-006 -3.3382e-007 -1.9011e-020 5.3075e-015 5.3075e-015
17 7.9985e-006 -7.209e-007 -4.0626e-020 1.654e-014 1.654e-014
18 8.4556e-006 -1.0647e-006 -5.9777e-020 2.6864e-014 2.6864e-014
19 8.9126e-006 -1.483e-006 -8.3052e-020 3.9247e-014 3.9247e-014
20 9.3697e-006 -1.9759e-006 -1.1042e-019 5.2298e-014 5.2298e-014
Ready
No Messages
No Selection
Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Joint Probe

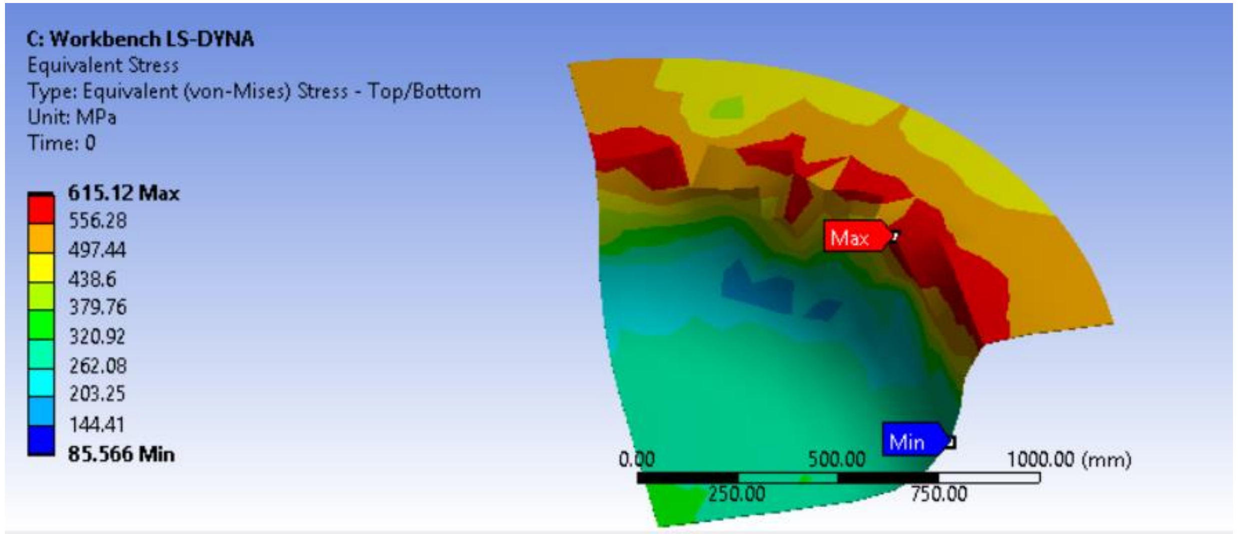


C : Copy of Workbench LS-DYNA - Mechanical [ANSYS Mechanical Enterprise PrepPost]
File Home Result Display Selection Automation LSDYNA Post
Outline
Workbench LS-DYNA (C5)
Initial Conditions
Analysis Settings
Displacement
Solution (C6)
Solution Information
Total Deformation
Total Deformation 2
Total Deformation 3
Total Deformation 4
Directional Deformation
Directional Deformation 2
Joint Probe
Details of "Joint Probe"
Definition
Type Joint Probe
Boundary Condition Joint
Orientation Method Joint Reference System
Suppressed No
Options
Result Type Total Force
Result Selection All
Display Time End Time
Results
Maximum Value Over Time
X Axis 24.148 N
Y Axis 198.58 N
Z Axis 1.1614e+007 N
Total 1.5905e+007 N
Minimum Value Over Time
X Axis -59.345 N
Y Axis -77.61 N
Z Axis -1.5905e+007 N
Total 0. N
Information
C: Copy of Workbench LS-DYNA
Joint Probe
9/19/2019 1:36 PM
ANSYS
2020 R1
Joint Probe
Graph
Animation
20 Frames
1.0001e-3
[N]
[s]
Tabular Data
Time [s] | Joint Probe (Total Force X) [N] | Joint Probe (Total Force Y) [N] | Joint Probe (Total Force Z) [N] | Joint Probe (
1 0. 0. 0. 0. 0.
2 4.9877e-005 -0.63221 -0.65887 7.8349e+006 7.8349e+006
3 9.9828e-005 -0.27927 -0.56035 -1.5905e+007 1.5905e+007
4 1.4993e-004 0.60341 0.93433 -7.8958e+006 7.8958e+006
5 1.9995e-004 -1.5651 -0.76465 -1.7247e+006 1.7247e+006
6 2.4997e-004 2.0049 2.5805 4.9993e+006 4.9993e+006
7 2.999e-004 -2.1867 -2.9023 1.1614e+007 1.1614e+007
8 3.4993e-004 -0.47019 2.1179 2.6081e+006 2.6081e+006
9 4.e-004 0.98243 -1.0271 -5.4862e+006 5.4862e+006
10 4.4998e-004 -0.34966 -1.7887 -7.5721e+006 7.5721e+006
11 4.9988e-004 -2.4063 10.592 -4.5e+006 4.5e+006
12 5.4987e-004 -1.4944 -0.1106 5.2191e+006 5.2191e+006
13 5.998e-004 2.6631 -5.2003 7.3799e+006 7.3799e+006
14 6.4996e-004 -0.92475 4.9014 3.5198e+006 3.5198e+006
15 6.9996e-004 0.64969 -0.27874 -2.396e+006 2.396e+006
16 7.4985e-004 24.148 -77.61 -7.2034e+006 7.2034e+006
17 7.9995e-004 -8.1493 17.216 -4.1237e+006 4.1237e+006
18 8.4988e-004 -17.8 63.704 2.4114e+006 2.4114e+006
19 8.999e-004 1.4842 0.45211 5.4225e+006 5.4225e+006
20 9.4984e-004 0.14456 -3.1033 4.2838e+006 4.2838e+006
21 9.999e-004 -59.345 198.58 -1.0504e+006 1.0504e+006
22 1.0001e-003 -58.411 195.36 -9.114e+005 9.114e+005
Ready
1 Message No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

LS-DYNA

Imported Stresses with LS-DYNA Implicit

- Springback can now be solved with the LS-DYNA Implicit solver. It is provided as an alternate to the MAPDL Solver for the implicit part of the calculation. It enables a greater compatibility in terms of materials laws and formulation between the explicit calculation and the implicit calculation

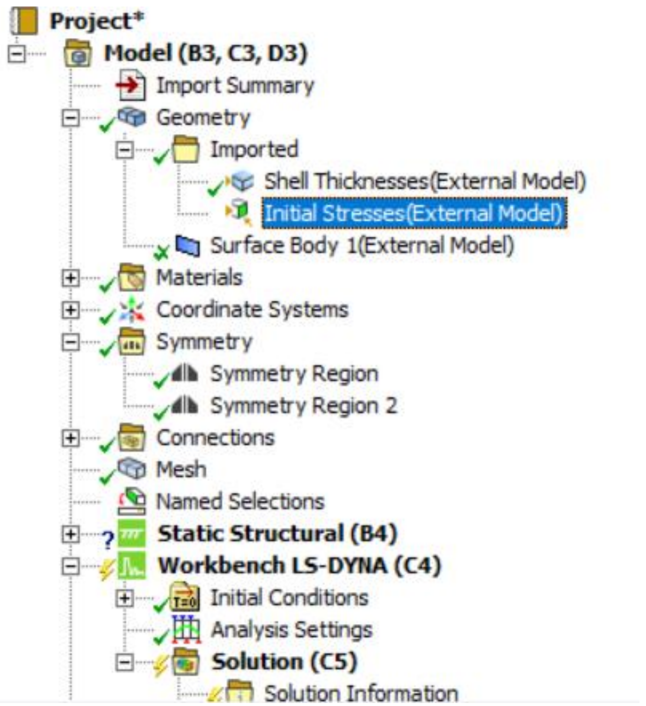


***Stress at the beginning of the calculation*

Imported Stresses with LS-DYNA Implicit

- This property is hidden by default (when they are no imported stresses in the tree or when they are deactivated and set to "yes")

- When it is set to "No", the **"Implicit Controls Menu"** appears



Step Controls	
End Time	1
Time Step Safety Factor	0.9
Maximum Number Of Cycles	10000000
Automatic Mass Scaling	No
CPU and Memory Management	
Memory Allocation	Program Controlled
Number Of CPUS	1
Processing Type	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Solver Precision	Program Controlled
Unit System	[illegible]
Explicit Solution Only	No

Implicit Controls	
Type	Implicit
Initial Time Step	0.1
Line Search	0
Displacement Convergence	0
Stabilization	On
---Scale Factor	0.001
Start Time	0
End Time	0

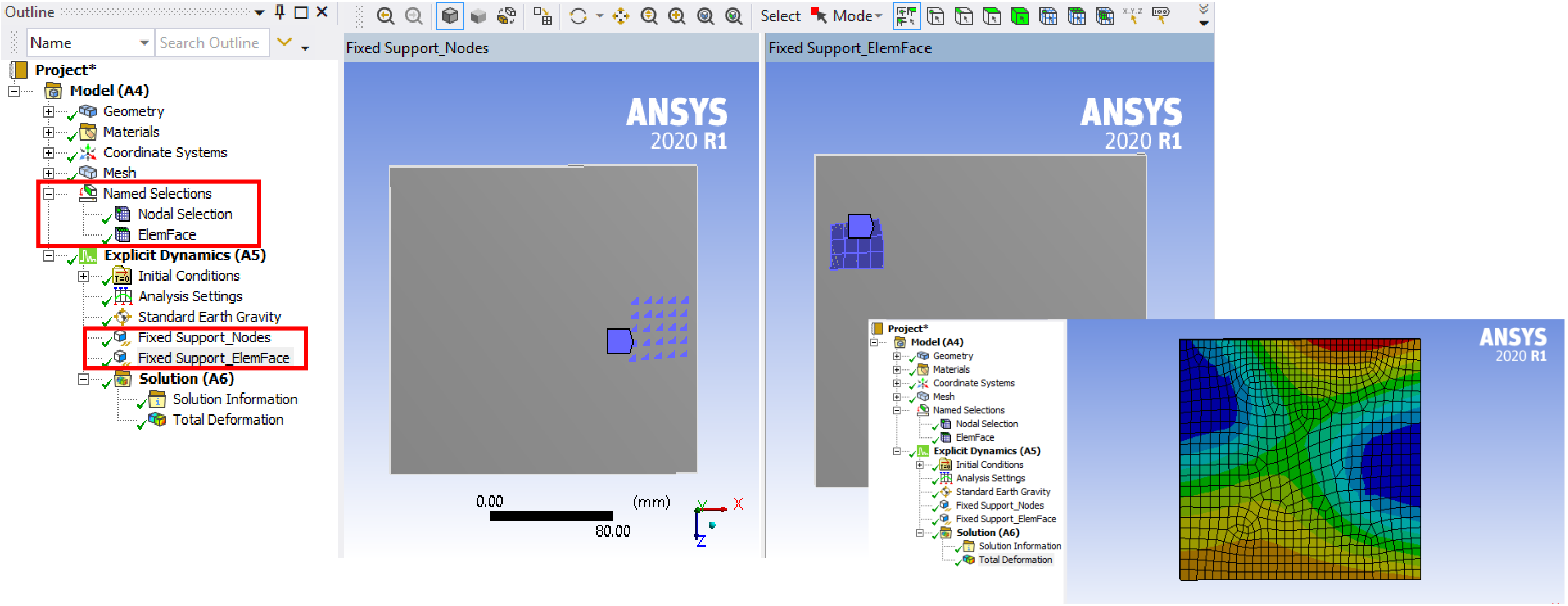
Explicit Dynamics

Explicit Dynamics Summary

- Scoping to a mesh-based selection
  - Both Node and Element Face mesh-based selections can be used in applicable boundary conditions
- Drop Test Wizard
  - Rotate Geometry object in the Drop Test Wizard has been replaced with the native Part Transform object
- Reaction Probes
  - Force and Moment reaction probes are now supported

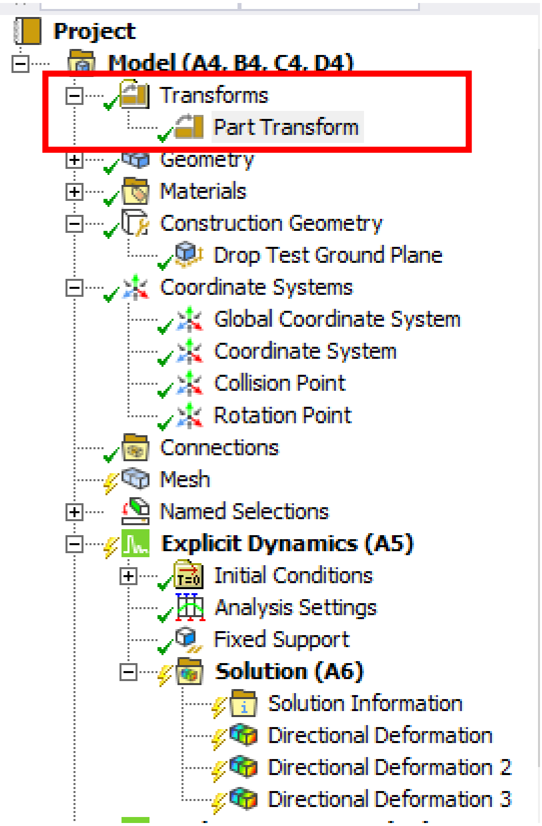
Scoping to a Mesh-Based Selection

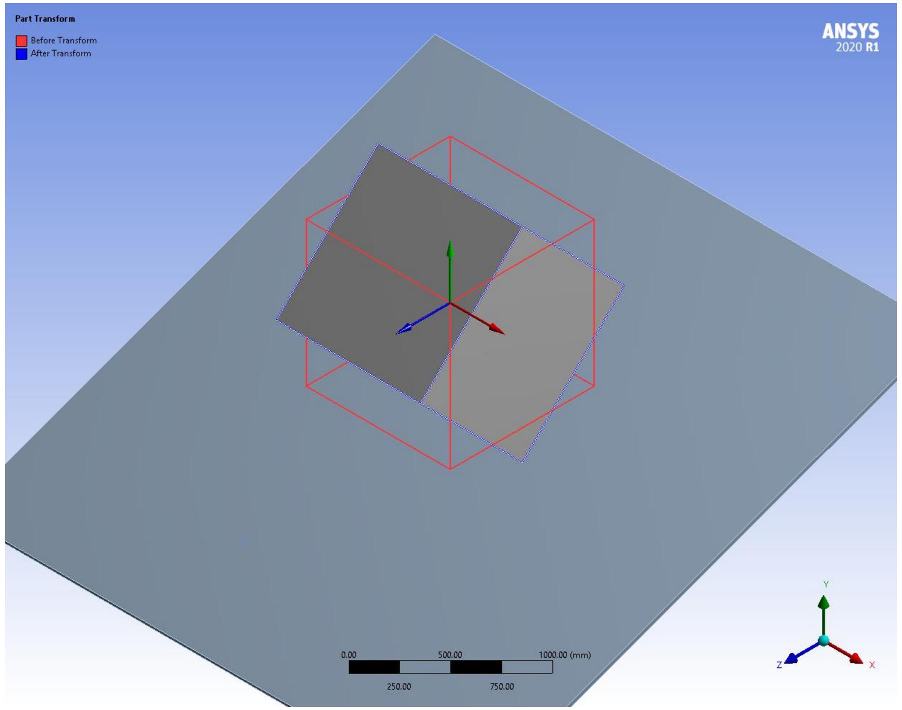
- Both *Node* and *Element Face* mesh-based selections can be used in **“Fixed Support”** boundary conditions

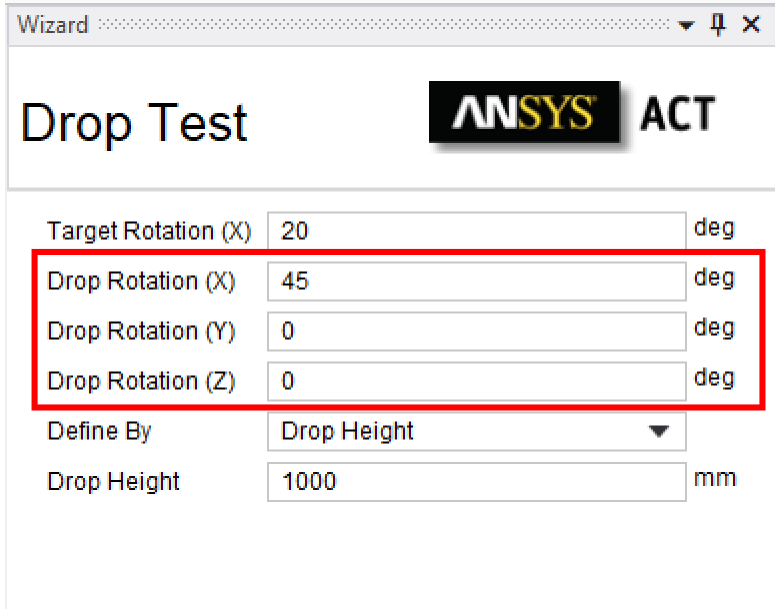


Drop Test Wizard

- *Rotate Geometry* object in the **"Drop Test Wizard"** has been replaced with the native *Part Transform* object for better compatibility
  - Upward compatible with older projects (*Rotate Geometry* object will be migrated to *Part Transform)*

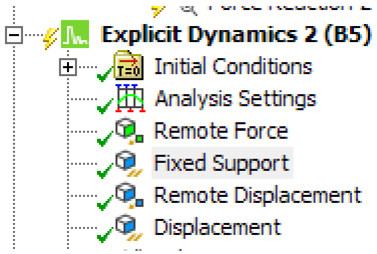




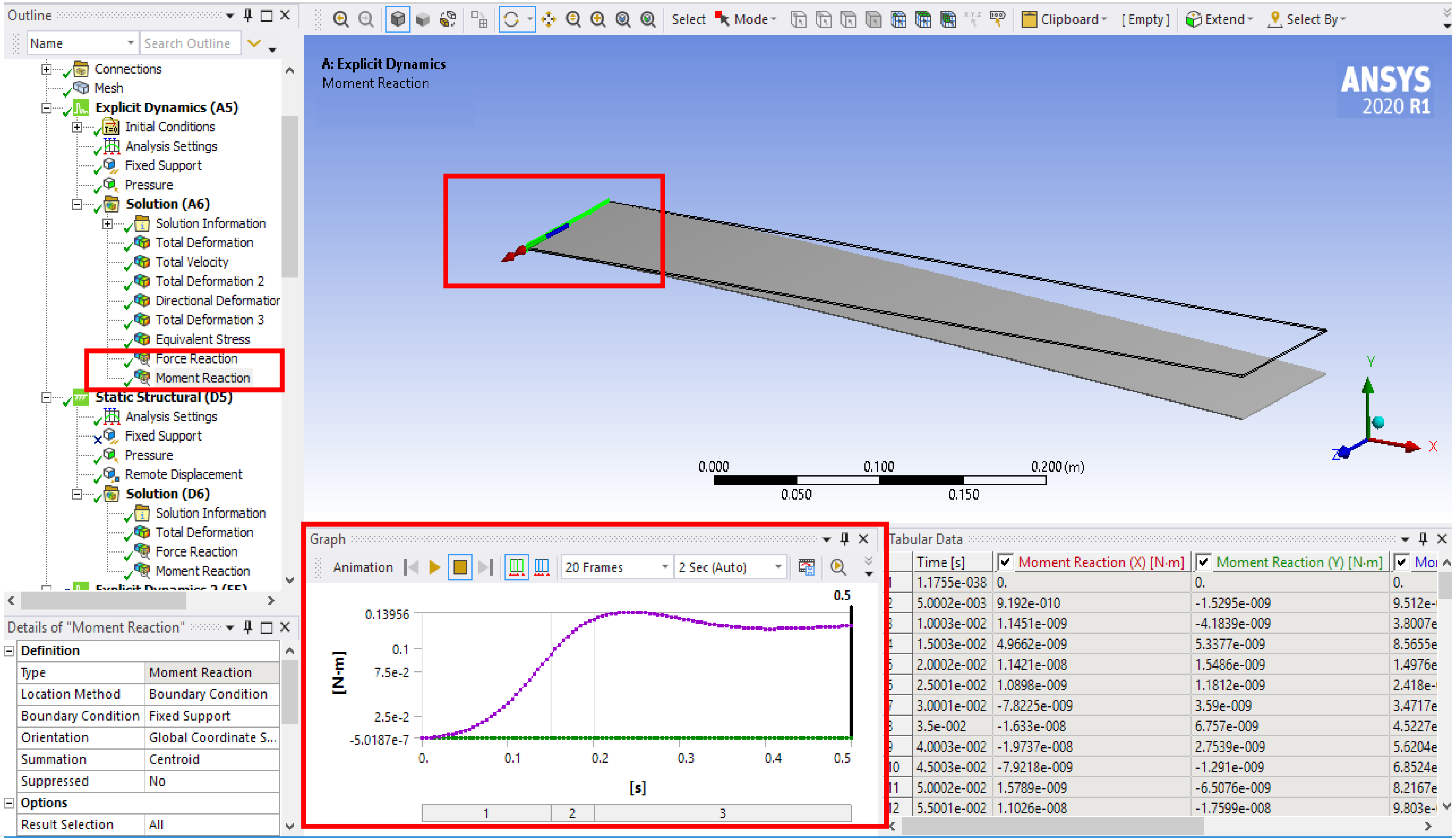


Reaction Probes

- Force and Moment reaction probes are now supported for the following boundary conditions
  - Fixed support
  - Displacement
  - Remote Displacement



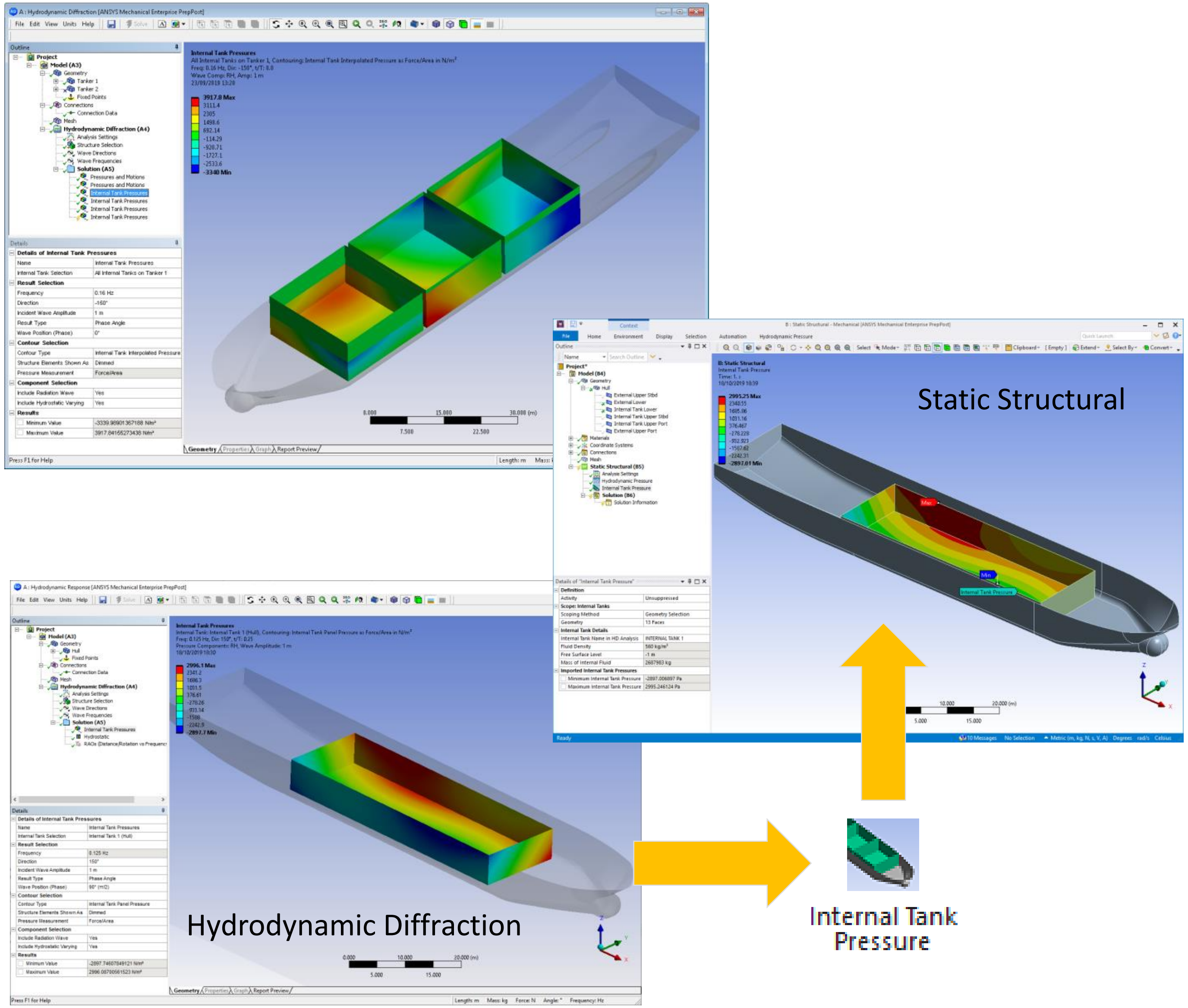
- Direction and size of force and moment can be animated
- Resolution based on result output frequency
- On demand after solution



AQWA

Transfer of Internal Tank Pressures to Static Structural System

- Display of **“Internal Tank Pressures”** in Hydrodynamic Diffraction system:
  - Select Wave Frequency/Direction/Amplitude
  - Display by Phase Angle/Maximum/Minimum values
  - Interpolated Pressures or Panel Pressures
  - Radiated and/or Hydrostatic-Varying pressure components
- New **“Internal Tank Pressure”** object in AqwaLoadMapping ACT extension
- Internal Tank definitions and pressures read into Mechanical (Static Structural) from AQWA backing files (.RES/.TPC)
- Internal Tank elements represented by `SURF154`, pressures written to ds.dat as `SFE...PRES...`
- ACT extension accounts for any difference in axis systems, unit systems, and position of Aqwa combined CoG vs Mechanical structural CoG



Calculation of Hydrodynamic Pressure in Time Domain

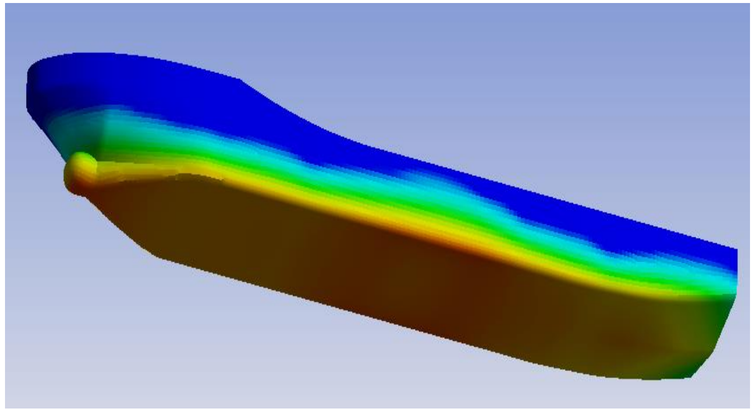
- Computing time domain pressures on the external hull and internal tanks
- Running irregular or regular waves
- Valid for single or hydrodynamic interaction structures with internal tanks
- Use of parallel processing
- Selecting and visualizing pressure at specified times
- Outputting pressure distribution at specified time in .CSV file

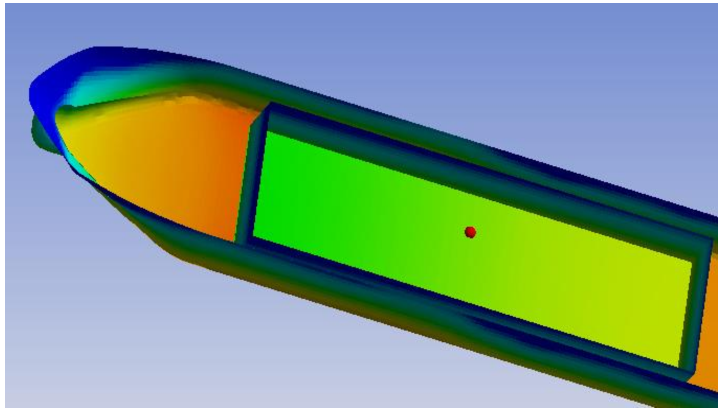
Time Response Pressure Output	
Output for Structure	Hull
Output Start Time	20 s
Output Time Step	0.5 s
Output Finish Time	120 s

Result Selection	
Display Pressures At	Multiple Time Steps (Animation)
Output Start Time	20 s
Output Finish Time	120 s
Number of Steps	201

Export Results	
Export Panel Pressures to CSV File	Select CSV File...
Export at Time	80 s

Pressure calculation/output definition

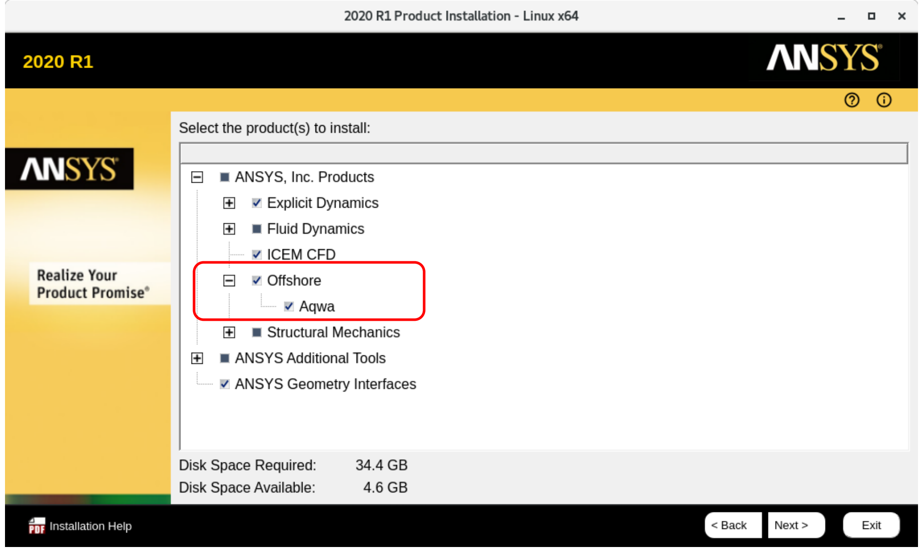


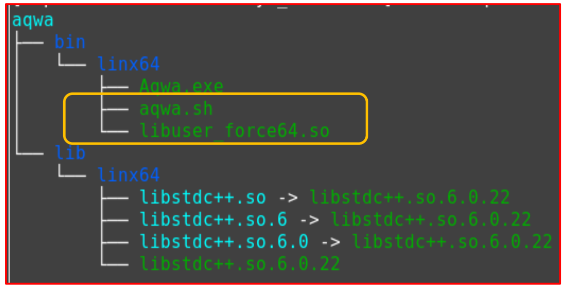


Nonlinear hydrodynamic pressure on external hull and internal tank

AQWA Availability on Linux Operating Systems

- Available functionalities:
  - Aqwa-Librium / Aqwa-Line / Aqwa-Fer
  - Aqwa-Drift / Aqwa-Naut
  - External users force as a dynamic library (.so)
- Installation
  - GUI: under the **Offshore** item
  - Silent with command line: ./INSTALL -silent –aqwa
- Installation location
  - **<root install>/ansys_inc/v201/aqwa**
  - bin: executable, wrapper, example of external user force lib
- Aqwa runs using the script **./aqwa.sh**
  - Sets the library path and the user stack limits
  - Input DAT file (upper case only)
  - Runs as a command line (flag compatible with Windows call option)
  - Binary results files compatible between Linux & Windows

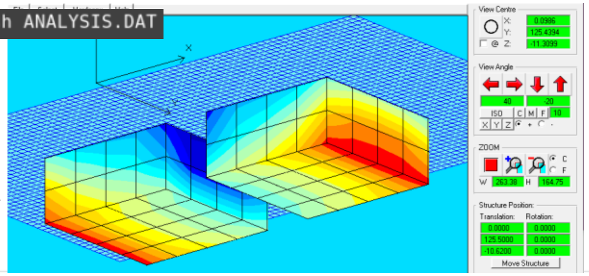





```
ansys_inc/v201/aqwa/bin/linx64/aqwa.sh ANALYSIS.DAT
```

Run on Linux

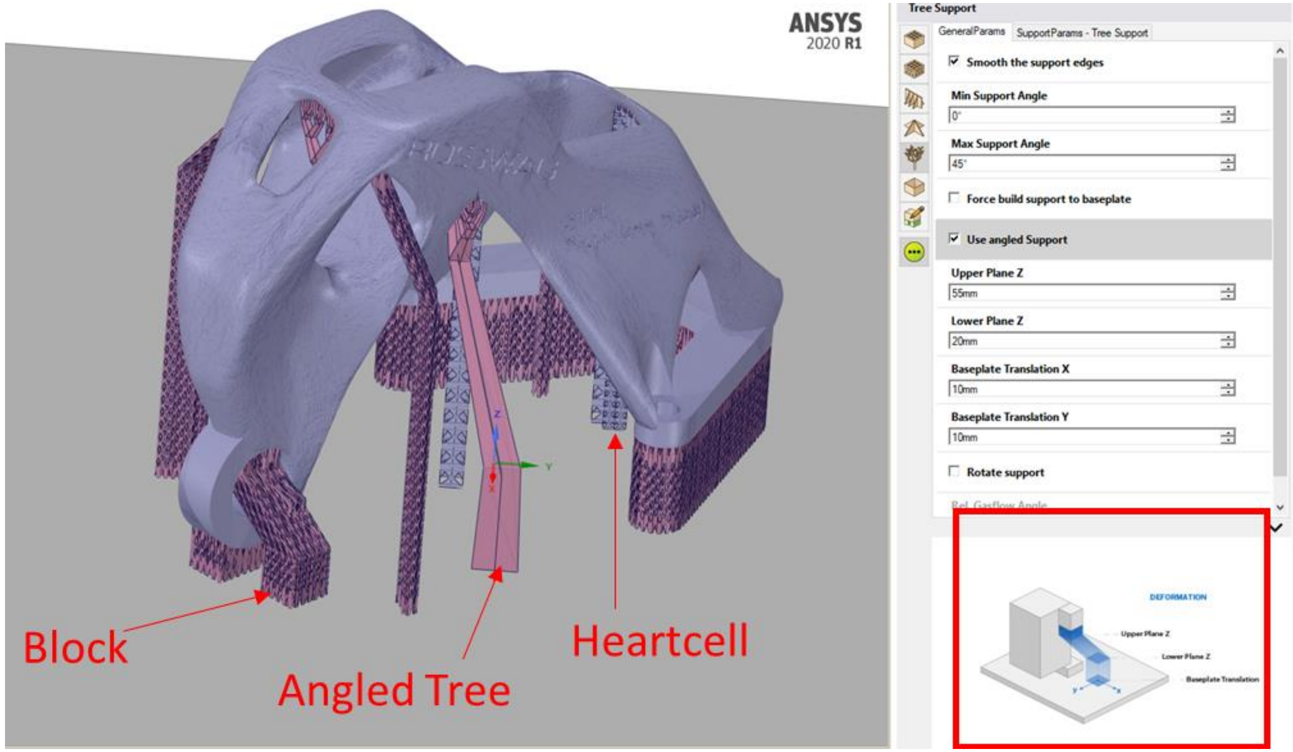
Visualize on Windows



Additive Manufacturing

Additive Prep

- Tree supports
- Write build files
  - SLM
  - CLI – EOS
- Modify power, speed, and focus parameters for different vector types
- Modify scan order
- Cost estimator
- Licensing option

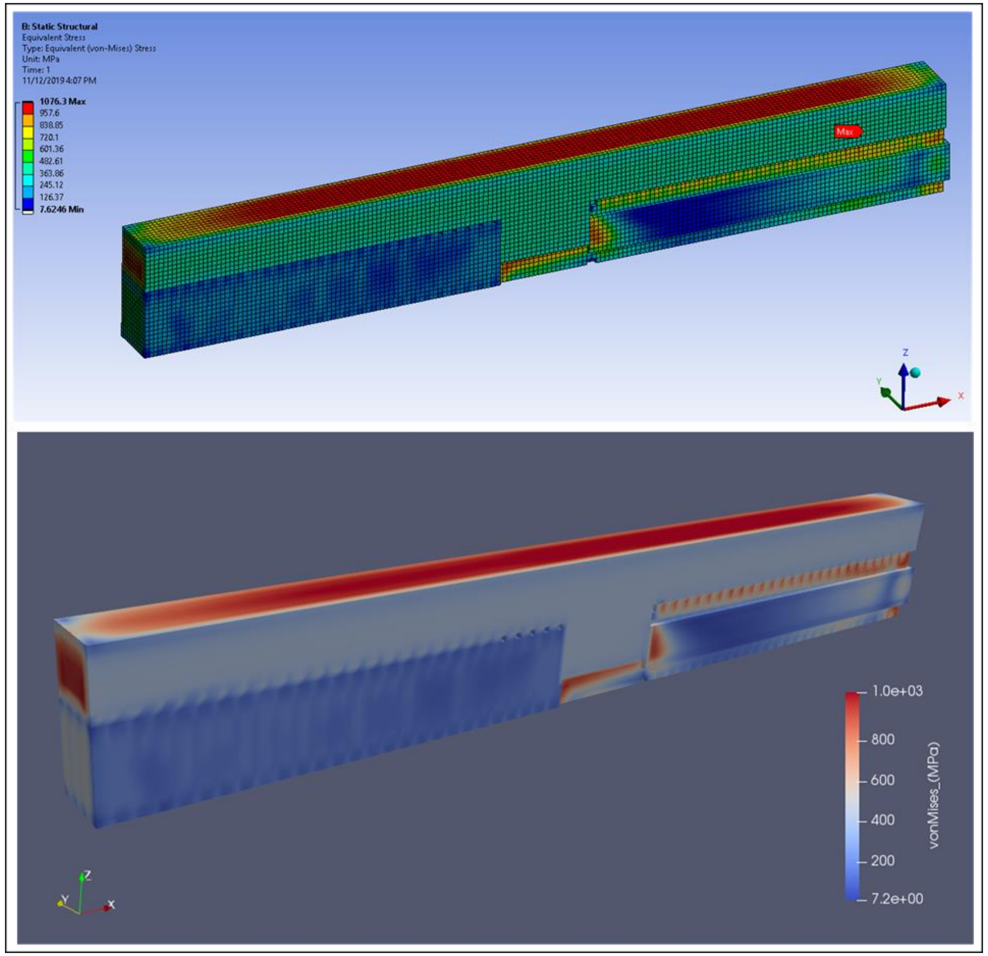


0_PASB_engraved

Id	Type	Posit...	Focu...	Powe...	Spee...	Index
11	Hatch	Volu...	0	190	750	1
24	Hatch	Up S...	0	180	1000	1
4	Border	Dow...	0	100	1200	1
10	Hatch	Dow...	0	100	1000	1
5	Border	Shell2	0	80	840	1
6	Border	Shell2	0	80	1680	1

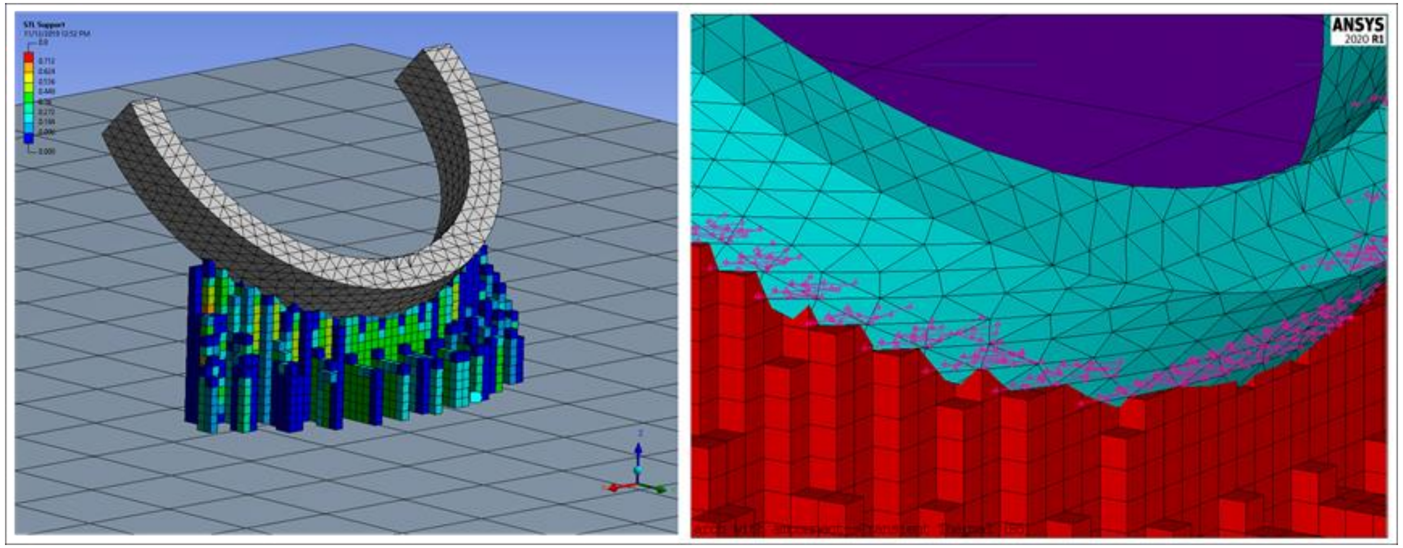
ANSYS Additive (Print/Science)

- EOS build files available (Beta)
- AlSi10Mg for all simulation types
  - Microstructure
  - Single Bead
  - Porosity
  - Thermal History
  - AP all modes
- User controlled laser beam diameter
- Log icons to assist in finding log information
- J2 Plasticity threading
- J2 Plasticity beta removed
- Build size increased to 1m^3
- Disable support optimization
- Meltpool Reference Width
- Additive Print to Workbench Additive Workflow
  - Cutoff



Workbench Additive

- Inherent Strain Method
- SSF Available for Calibration
- AMCONNECT Macro for connecting Layered Tet part and voxel support
- Wizard Updates
  - Inherent Strain
  - Materials Automatically Loaded
  - Sequential Progression for Cutoff
- Print to Workbench for Cutoff
- Smaller Result File
- Conformal Voxel Part and Support (Beta)
- Improvements to Blade Interference (Beta)



Workbench Additive

Inherent Strain Option Added

- You may use an inherent strain approach with Workbench Additive
  - Isotropic or orthotropic stains may be input
  - These are the same strains as would be used with Additive Print
    - The same calibration process can be used to generate the values
  - Restarting is supported, so you may perform the build first and restart to investigate cutoff sequences and/or heat treatment

Supports from Additive Prep

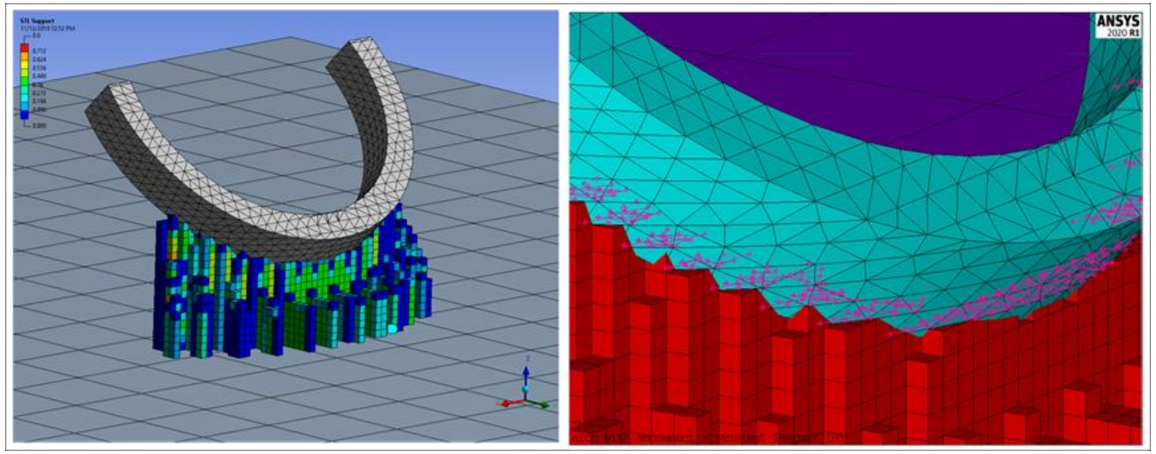
- STL supports generated in Additive Prep can be represented as voxel (Cartesian) meshes
  - Knockdown factors are also generated and used as in Additive Print
    - These factors account for support build material contained within each voxel
    - Thermal and structural material properties are adjusted by these factors, including the plasticity and creep behaviors

Supports from Additive Prep

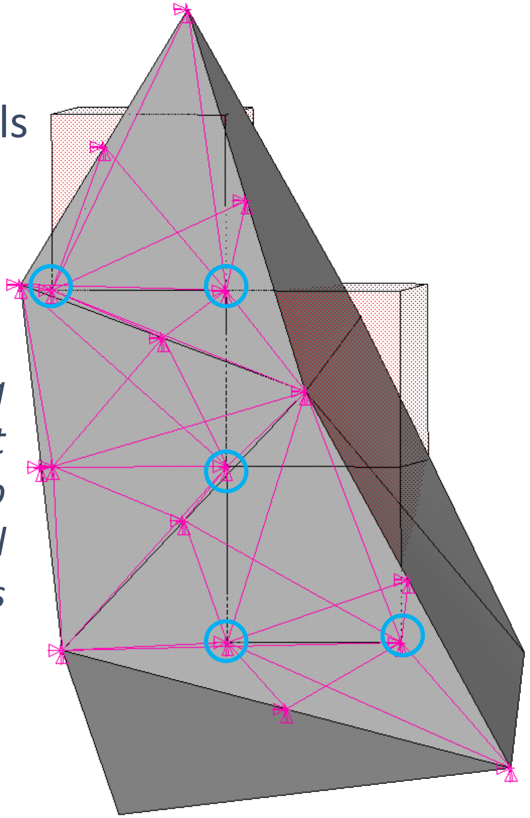
- An MAPDL command macro may be used to connect voxel supports to parts meshed with layered tetrahedral
  - The command generates constraint equations tying the support nodes to the part elements
- To use, insert a Command Object under the analysis object (to both the Transient Thermal and Static Structural objects) and add:

```
amconnect,partID,supportID
```

  - Where `partID` is the mat ID of the part and `supportID` is the mat ID of the support voxels



Left: Tet-meshed part with voxel-meshed support
Right: Close-up of the CEs connecting the two



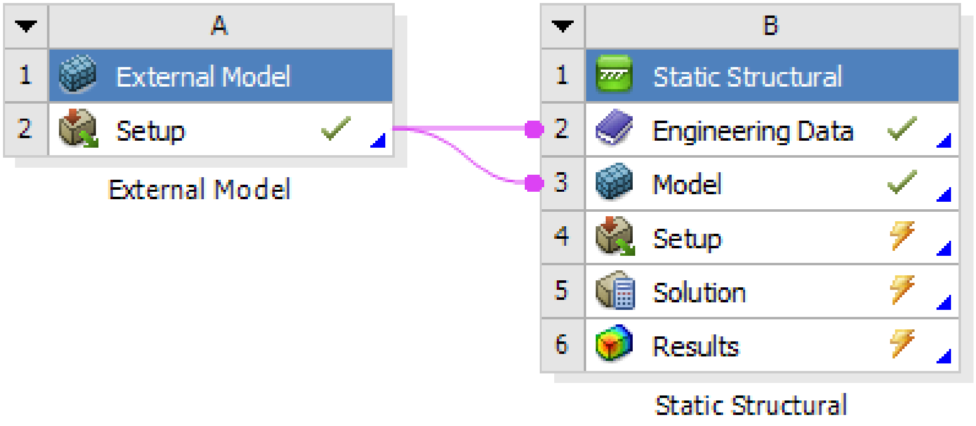
Bottom of a layer showing CEs generated to connect support voxel nodes ◯ to layered tetrahedral elements

Voxel Mesh Both Part and Support (Beta)

- Both the support and part can be meshed with a voxel mesh
  - Also generates knockdown factors for both
  - Quick workflow, avoiding layered tetrahedral meshing and connecting of the support and part
  - Uses same voxelization as Additive Print

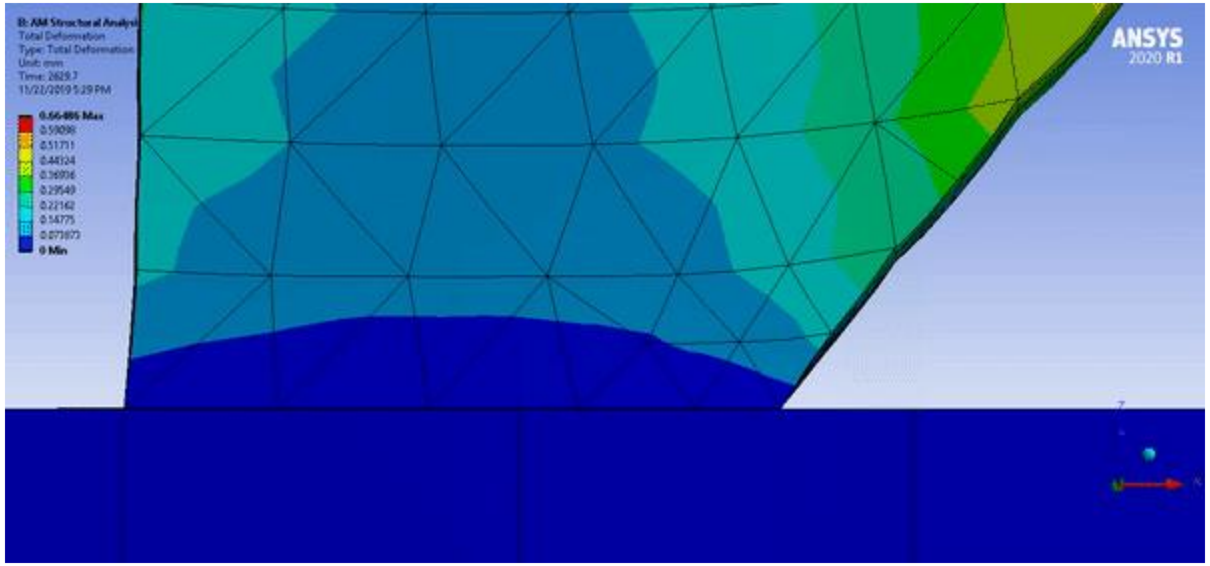
Additive Print to Workbench Additive Workflow

- Simulate build in Additive Print, simulate cutoff and heat treatment in Workbench Additive
  - Use powerful and easy-to-use Additive Print to perform the build simulation
  - Use robust and feature-rich Workbench Additive to perform the downstream cutoffs and/or heat treatment
    - Ability to perform a prescribed cutoff sequence
    - Ability to cutoff supports
    - Extensive heat treatment material library
  - Automated wizard available to automate the transfer of data and the cutoff steps



Additive Wizard Updates

- Inherent strain available as an option
- Progressive base/support cutoff available as an option
- Automatic population of AM Materials in engineering data



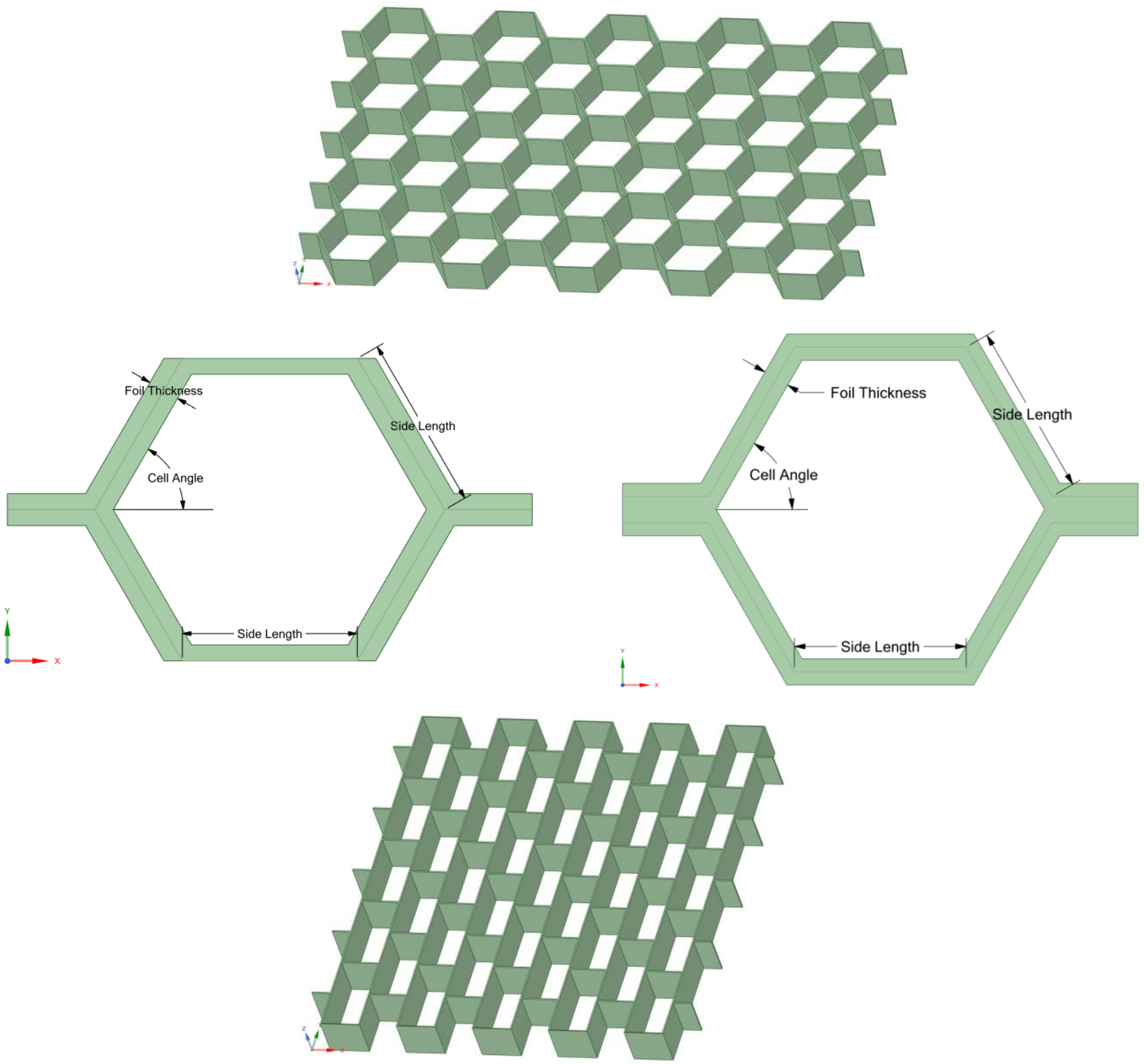
Miscellaneous Enhancements/Corrections

- Result file size has been optimized to output only quantities for additive
- For STL supports scoped to a STL file, the file is now maintained in the project
- Restarts are now supported for USER steps and inherent strain simulations
  - For example, can perform a bolt pretension user step first then restart to perform the build
- In a thermo-mechanical build simulation, you can apply a scaling factor to the thermal strains generated in order to reduce (or increase) the obtained distortions
  - The as-supplied material properties available for Workbench Additive are average values obtained from literature, and scaling those values is sometimes required to match actual distortions
- Block supports now knockdown the thermal density to obtain more realistic heat dissipation
  - Will now match the equivalent methodology using knockdown factors
- For the first layer, Tmelt is not applied to the bottom layer nodes
  - Eliminates spurious hot spots in the build plate and excessive plate deformation

Material Designer

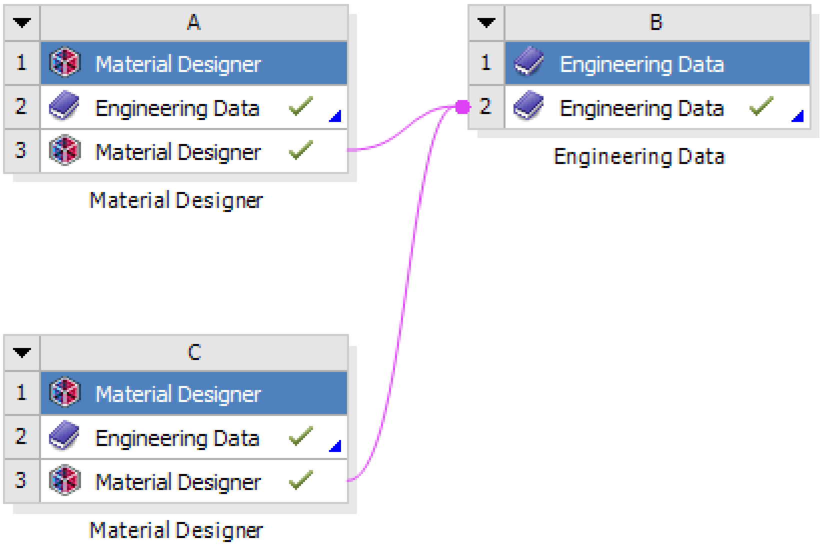
Honeycomb Structures

- Material Designer now supports honeycomb structures as an additional predefined RVE type
- Extruded or expanded honeycombs
- The cell angle can be varied. Thus, you can also model over expanded honeycombs



Combining Several Materials from Material Designer (Beta)

- You can now transfer the materials from several Material Designer Systems to a single Engineering Data cell

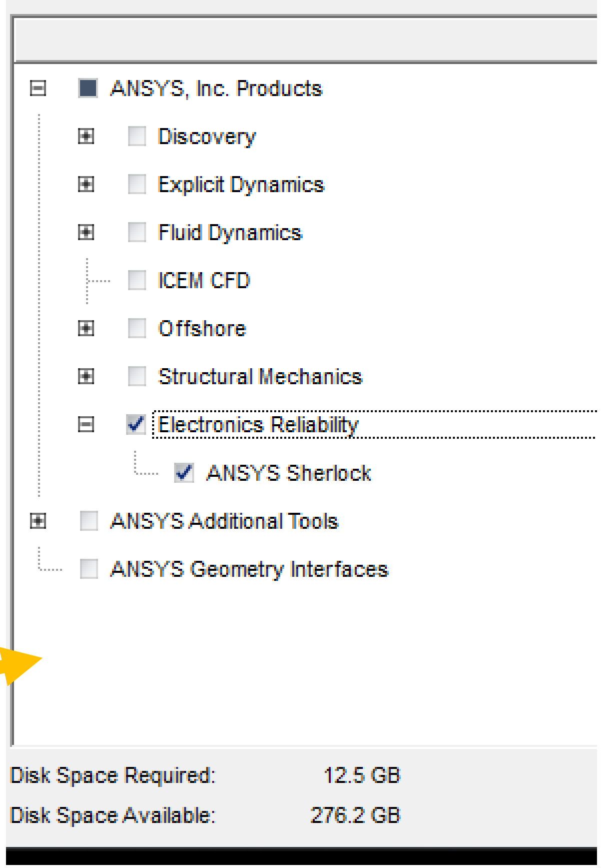


- Allows you to combine several materials from Material Designer in a macro-scale analysis
- To use it, you first need to activate **“Workbench Beta Options”** and then enable **"Unlimited MatML connections"** under *Tools > Options > Engineering Data*

Sherlock

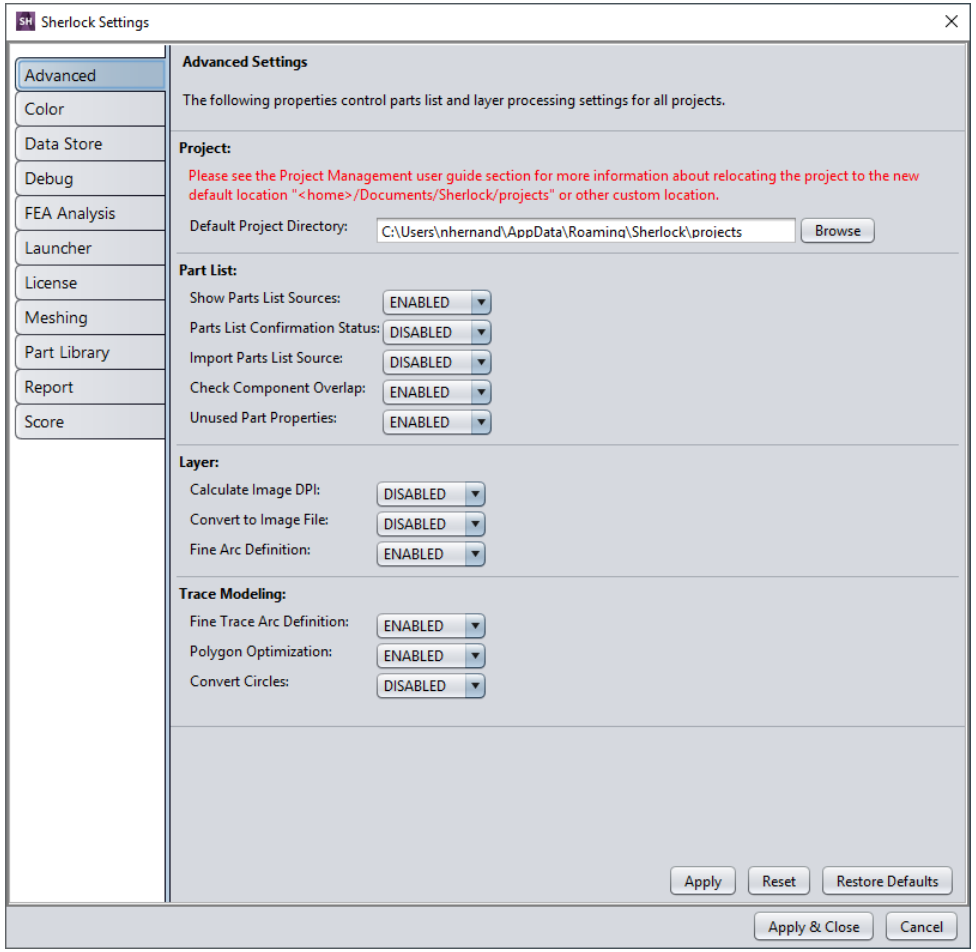
ANSYS Integration Elements

- **FEA Solver:** Sherlock has been updated to use ANSYS MAPDL as the default FEA engine. Calculix is no longer available or used for FEA analysis other than for Thermal Mechanical analysis
- **Results Processing:** Processing of ANSYS FEA results has been updated to process ANSYS result (.rst) files directly for both the results and the model import
- **Installation:** Sherlock is included in the Structures Unified Installer



ANSYS Sherlock Enhancements

- **Application Settings:** All settings have been consolidated into one form for ease of access
- **Part Libraries:** The Sherlock Part Library no longer requires Internet access and is included in the Sherlock distribution. The ability to configure multiple part libraries is also available
- **Parts List:** More capabilities to copy and add parts from various sources
- **Project Management:** Added functionality to view, search, manage categories of projects
- **Failure Rate Module:** This feature has been removed since it is no longer necessary

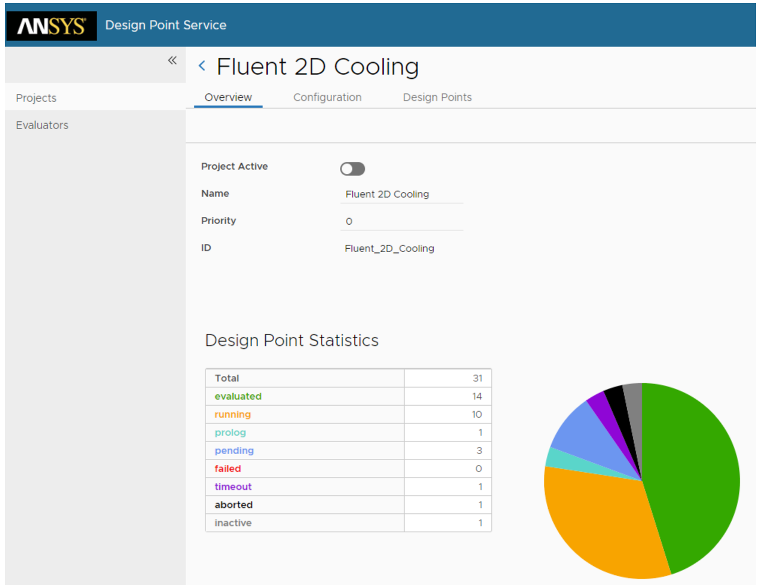


DCS
Distributed Compute Services

Distributed Compute Services - Introduction

- ANSYS introduces a host of new services that enable the distributed evaluation and management of simulation workflows:
  They are called **DCS**, the **Distributed Compute Services**.

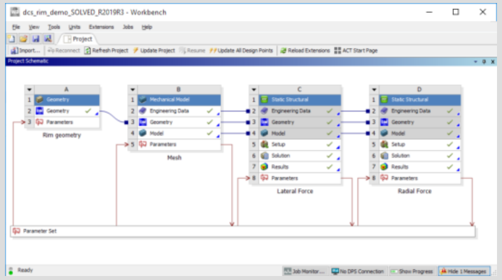
- As a part of this, **Design Point Service (DPS)** allows the robust and distributed evaluation of tens of thousands of Design Points, starting from a standard ANSYS Workbench simulation project



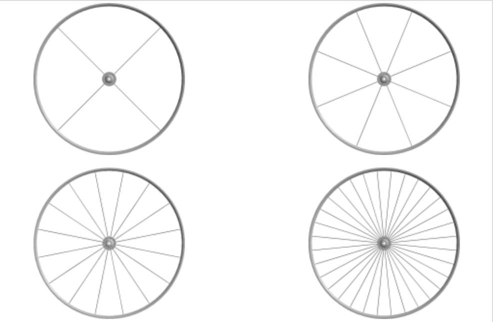
Open, Modular, Service-Oriented Architecture

Clients
Arbitrary clients can interact with the Design Point Service.

Parametric Workbench Model



Design Points



Other Design Exploration systems (DX & others)

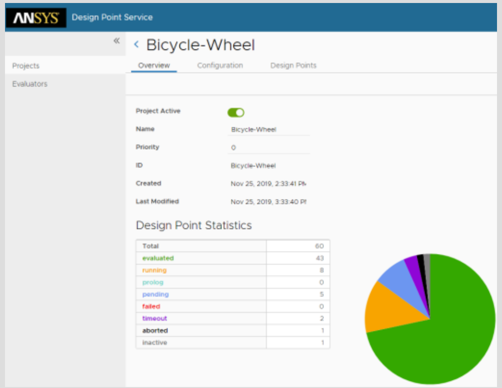
Other optimizers

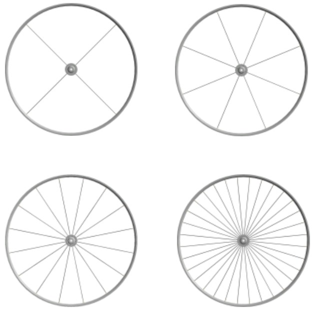
Transfer of simulation project, model parameters & Design Points

Import of Evaluated Design Points

DC Services
Manage simulation workflows. Handle data transfers and file storage.

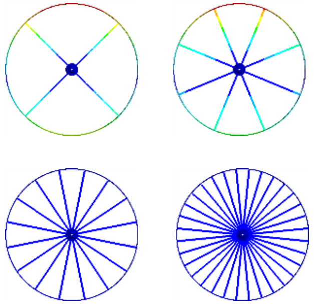
Design Point Service





Compute task pulled by Evaluator

Output parameters & Design Point files are passed back



DC Evaluators
Can run on arbitrary compute resources. Query the Design Point Service for tasks to evaluate based on their capabilities.

DC Evaluator 1
Linux workstation

DC Evaluator 2
Windows workstation

DC Evaluator 3
Linux cluster

DC Evaluator 4
Cloud resource

DC Evaluator 'N'
Arbitrary compute resource

Key Points

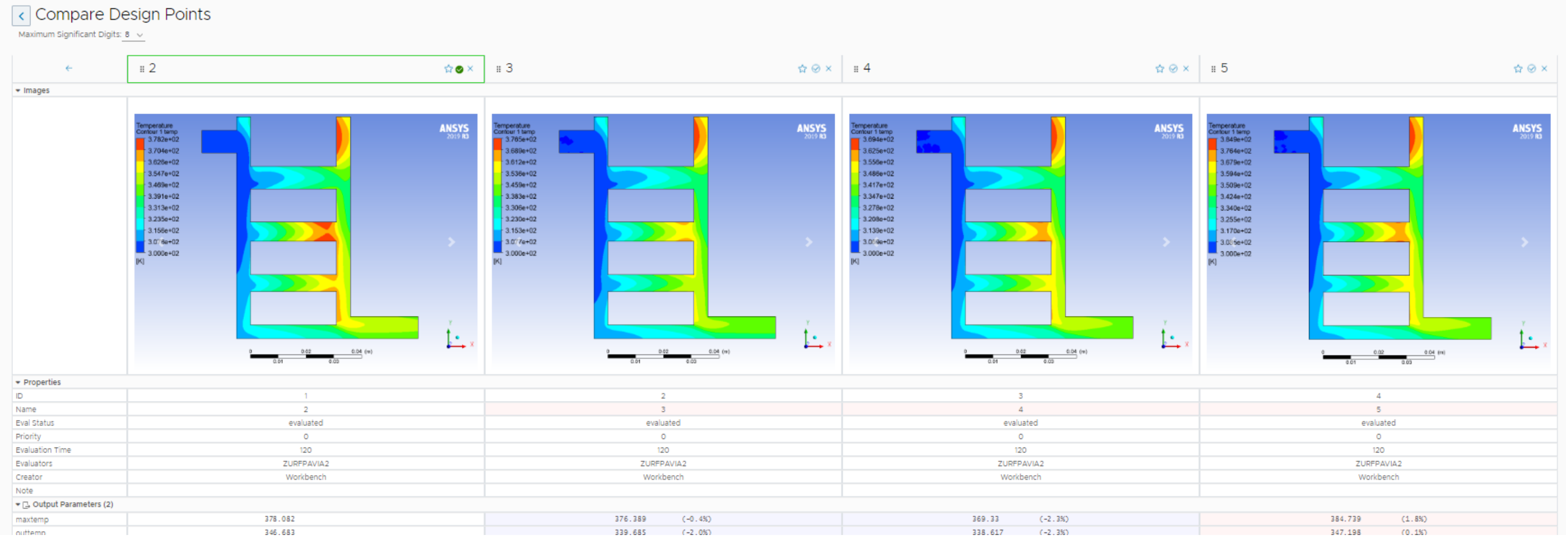
- Handle **large number of Design Points** (10’000s) robustly

- **Distributed evaluations**
  Seamless support for many usage / compute scenarios
  - On single user desktop
  - On group of workstations
  - On clusters/HPC resources, connecting to queuing systems
  - On private or public cloud resources
  - On heterogeneous systems with different capabilities

- Evaluate geometry updates on **Windows only** and solve model on **Linux** or Windows

- **Open and extensible**
  - Supporting arbitrary batch capable simulation workflows
  - Connect arbitrary design exploration systems

- **Minimal network footprint** - HTTPS only



Testimonial - BMC

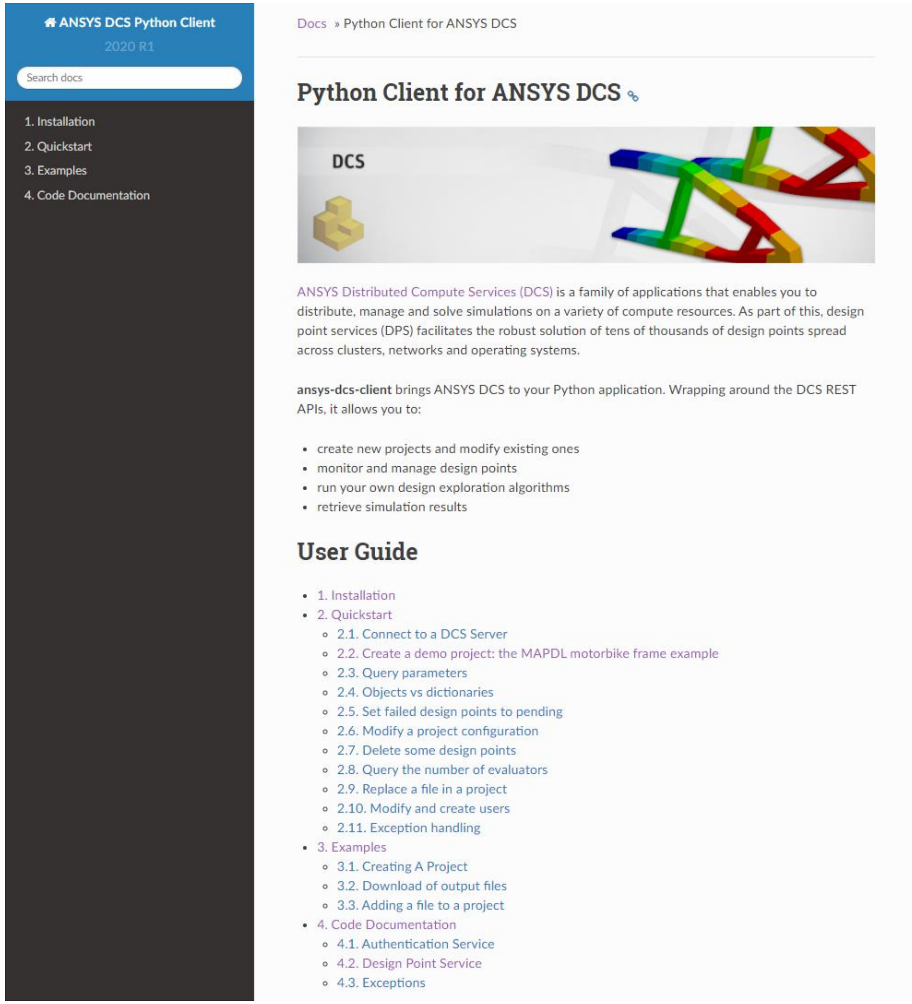
- ***“The results of this project were imperative for the success of the brand.”***
  Stefan Christ, Head of R&D at BMC Switzerland AG

- The bicycle manufacturer BMC benefitted from DCS Technology recently. Their flagship road bike, the SLR01, was optimized with the help of DCS and went on to be very popular on the market

DCS – What's New

- Improved scalability for evaluating large numbers of design points on many parallel evaluators
- New DCS Python Client

  Providing an easy-to-use and powerful scripting interface to interact with all DCS components, such as projects, configurations, design points, and more
- Extended workflow definitions
  - Modify parameter definitions
  - Define number of attempts for running a process step
  - Define success criteria
- Tighter integration between DesignXplorer and DPS
- DPS Web App enhancements
  - More unified look and feel, evaluation times, last modified dates, and the ability to open text and image files in the browser
- Run DCS as system service on Linux

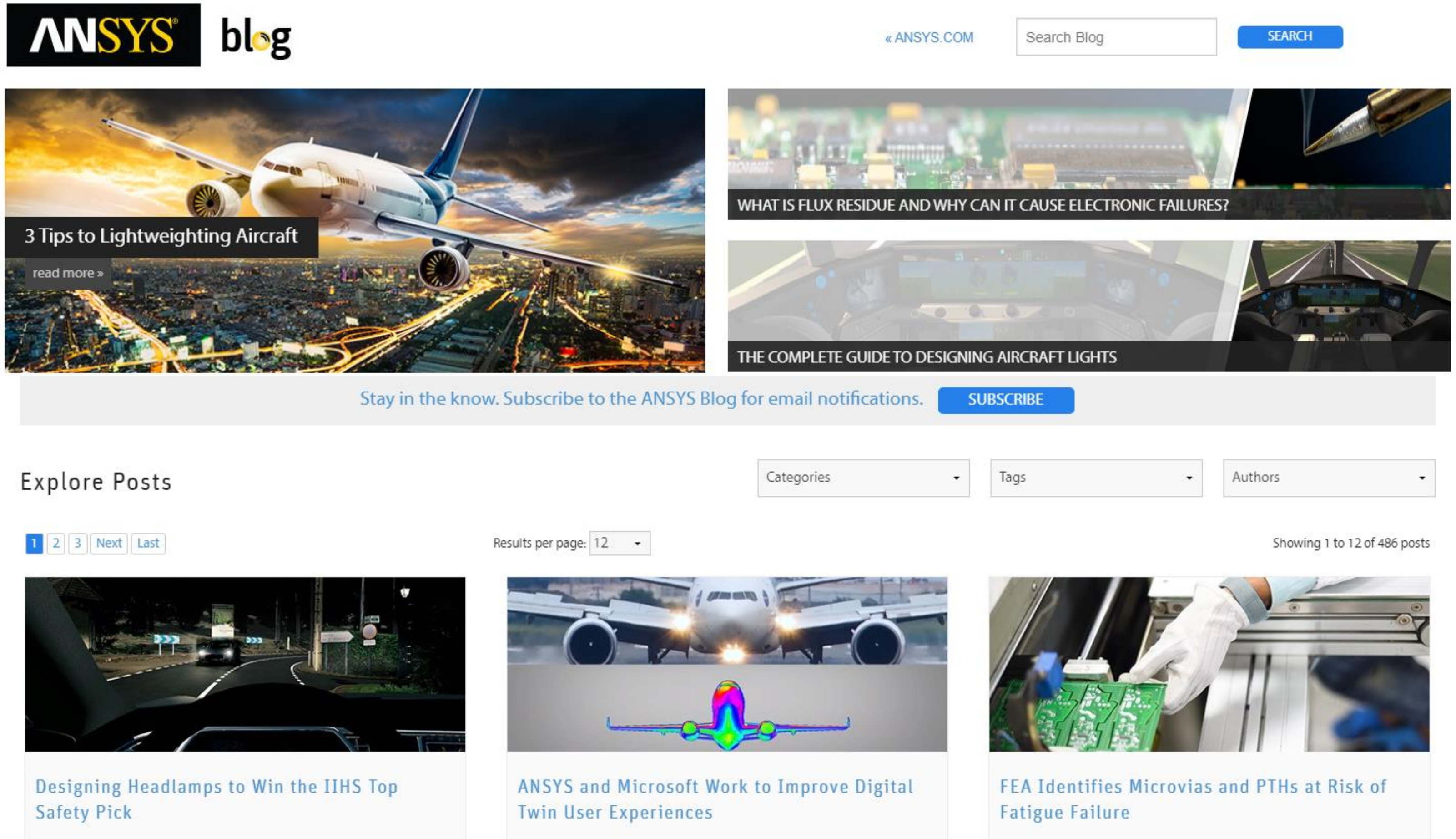


http://storage.ansys.com/mbu-assets/dcs/v201/dcs_python/index.html

Join the ANSYS conversation!

Read. Comment.
Join the conversation!

The new and improved ANSYS blog is live at:
ansys.com/blog



Sign up for an ANSYS Event near you!

ansys.com/about-ansys/events

Filter on “Webinar” under Events to sign up. A Structures 2020 R1 update webinar will be available in late February.

HOME / ABOUT ANSYS / EVENTS

ANSYS Events

ANSYS hosts major conferences around the world to gather our customers — and potential customers — to discuss and demonstrate the latest developments in engineering simulation technology. Our engineering experts will be on hand to help you develop solutions to your toughest simulation and product development challenges. You can also learn from technical presentations given by colleagues in your industry. In addition, we hold seminars dedicated to single industrial sectors, and webinars for easy online participation and learning. Check for upcoming events in your region or industry!

Refine Search

By Region

Webinar