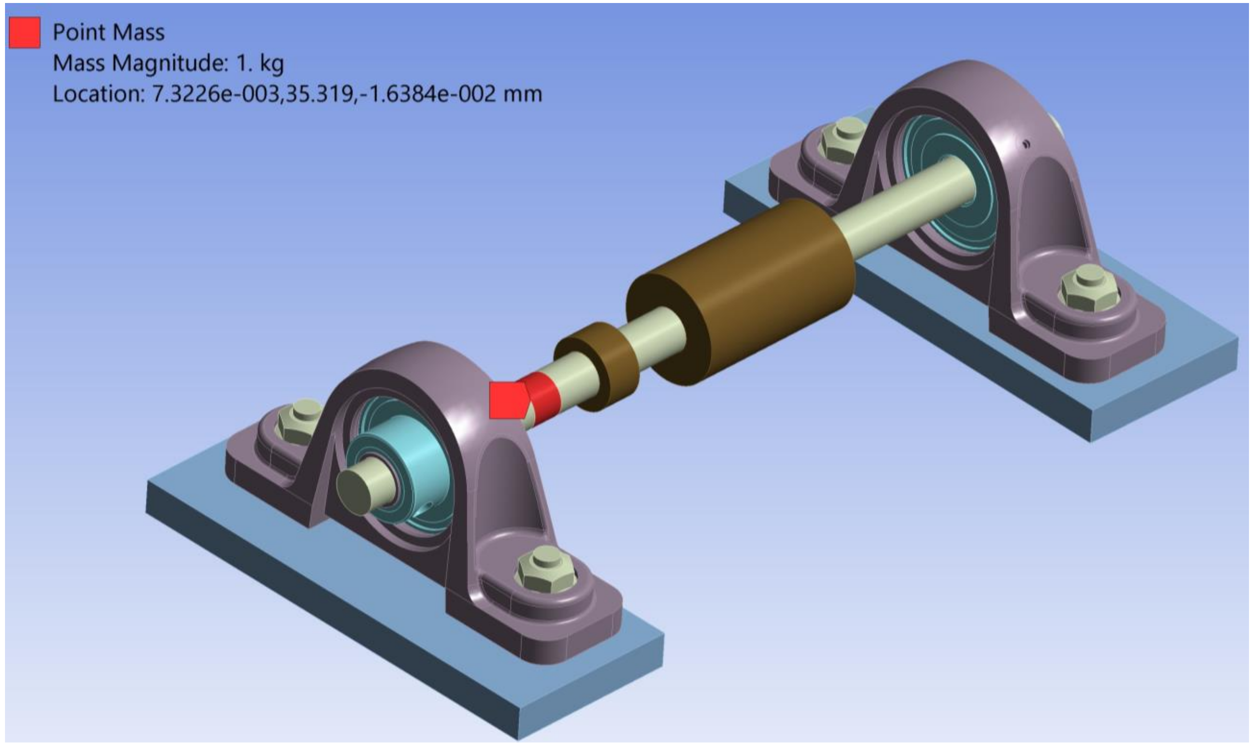
Ansys Mechanical Getting Started

Module 08 Student Workshop: Results and Validation

Release 2023 R1

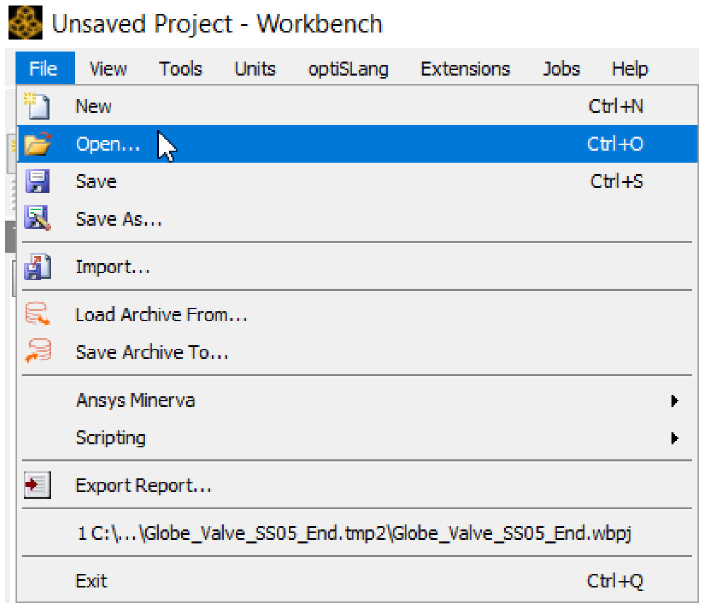
Workshop 08: Results and Validation

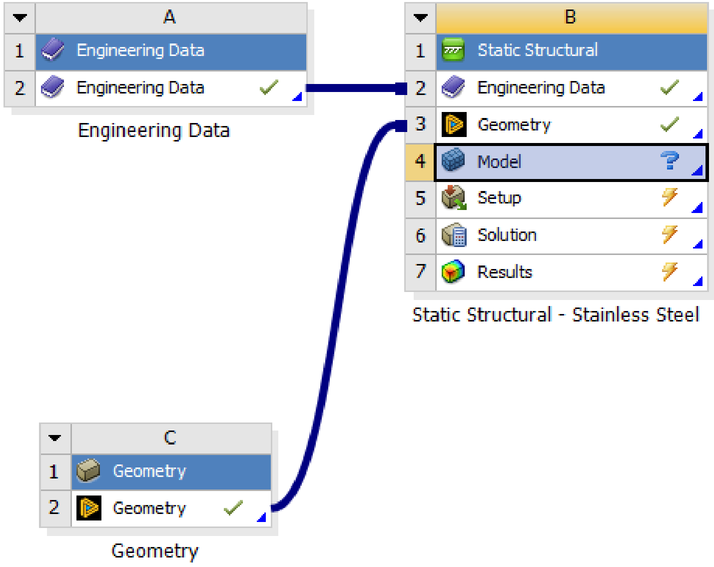
- Use this guide to work on the Journal Bearing model.



Workshop 08: Results and Validation

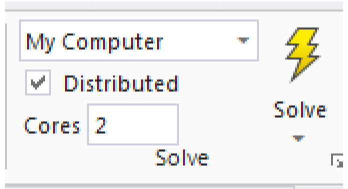
- Open archive file “Shaft_Bearings_WS08_Start.wbpz” or continue with the project as it was after Module 07 completion.
- Open Mechanical.
- Rename



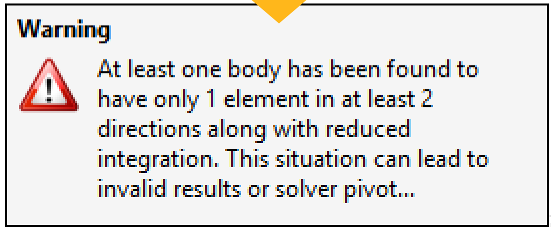


Workshop 08: Results and Validation

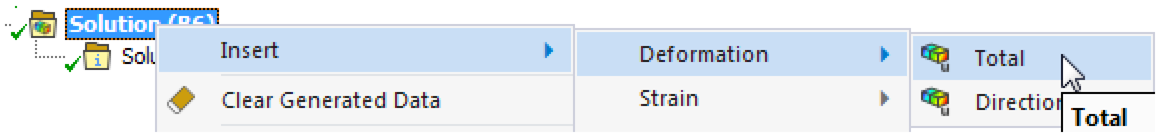
- Solve the FE model.

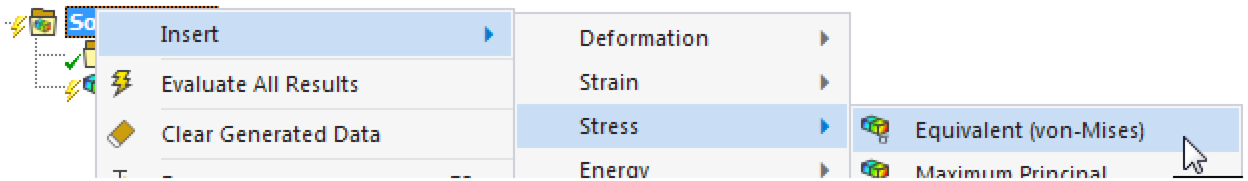


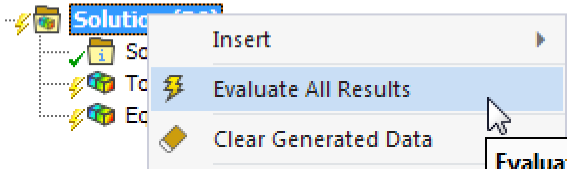
NOTE: If you receive a message like this one at the end of the solution, you may ignore it. We'll apply some additional mesh controls to address it in a later course.



- RMB – Solution → Insert → Deformation → Total
- RMB – Solution → Insert → Stress → Equivalent (von Mises)
- RMB – Solution → Evaluate All Results
- RMB – [on each result] → Rename Based on Definition







Workshop 08: Results and Validation

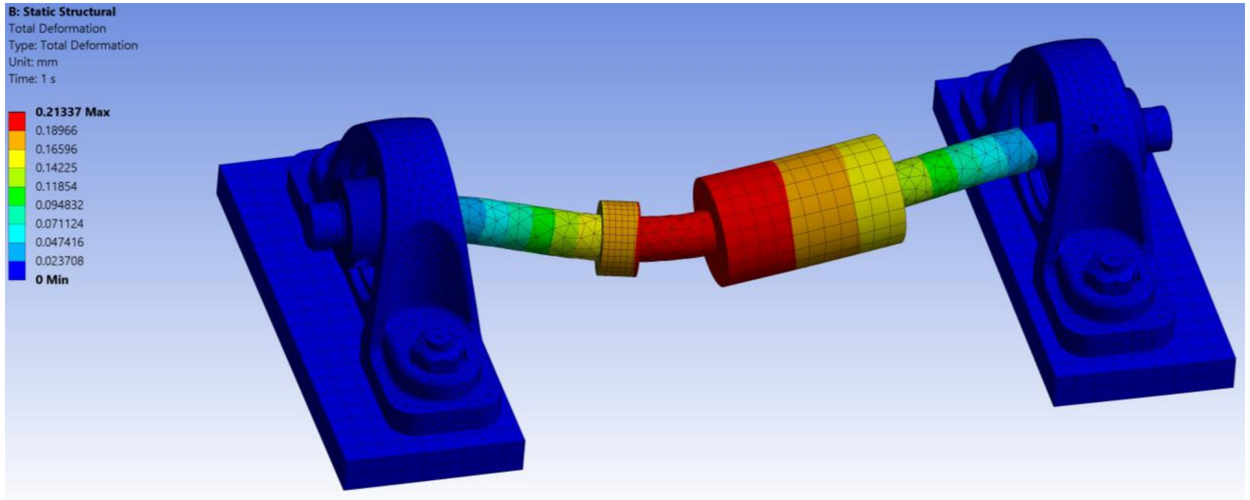
- For comparison with FE model containing housings in polycarbonate, prepare an Excel sheet or a paper sheet to note results:

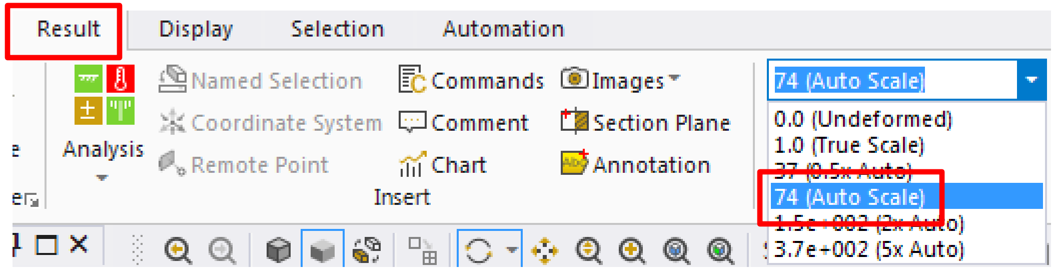
	A	B	C	D	E	F
1			Stainless Steel		Unfilled Polycarbonate	
2	Results	Units	Min	Max	Min	Max
3						
4						
5						
6						
7						
8						
9						
10						
11						

- Record Total Deformation – 1. s results and Equivalent (von-Mises) Stress - 1. s results in the table.

Workshop 08: Results and Validation

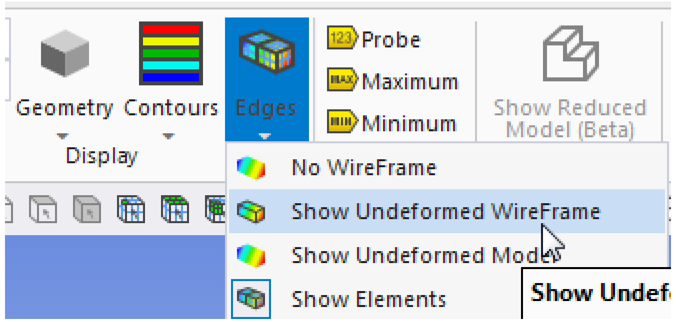
- Review and animate the Total Deformation result:
  - Select Total Deformation
  - Click the Animation “Play” button in the Animation controller below the graphics window
  - Change the results scaling in the Results tab if necessary, to have a better representation of the deformations

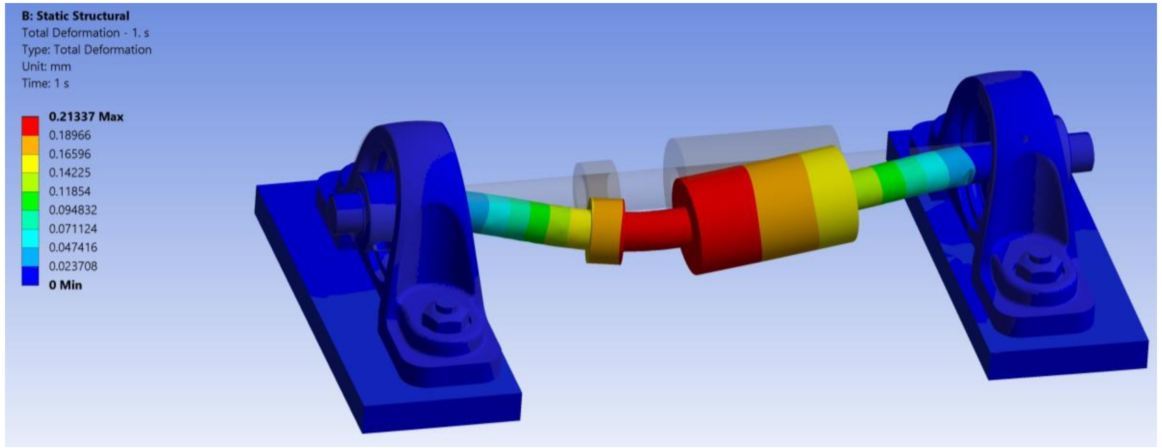




Workshop 08: Results and Validation

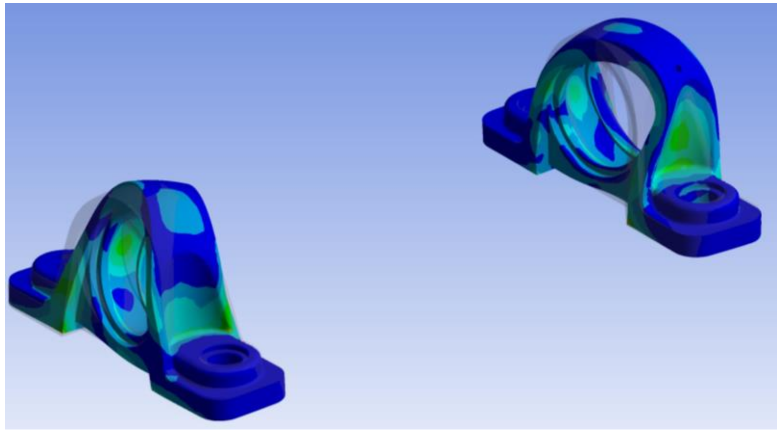
- Review results and undeformed model:
  - Change results display to Show Undeformed Model

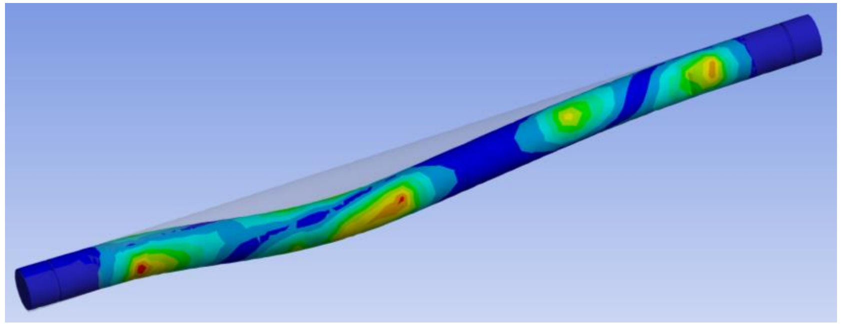




Workshop 08: Results and Validation

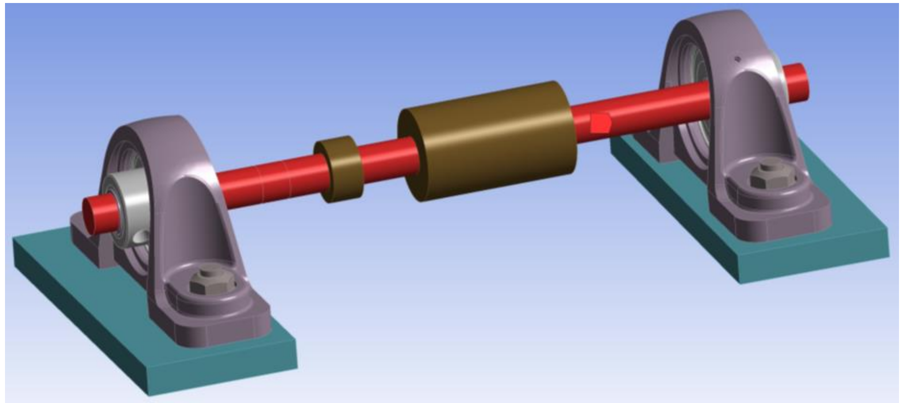
- Review Equivalent Stress results on Housings, Bearings and Shaft:
  - Select Solution
  - Select the 2 Housing bodies from graphics, RMB → Insert → Stress → Equivalent
  - Select the 2 Bearing bodies from graphics, RMB → Insert → Stress → Equivalent
  - Select Shaft from graphics, RMB → Insert → Stress → Equivalent
  - Evaluate All Results
- Record results in your table. Compare the Max values to the Yield Stress value for each material.

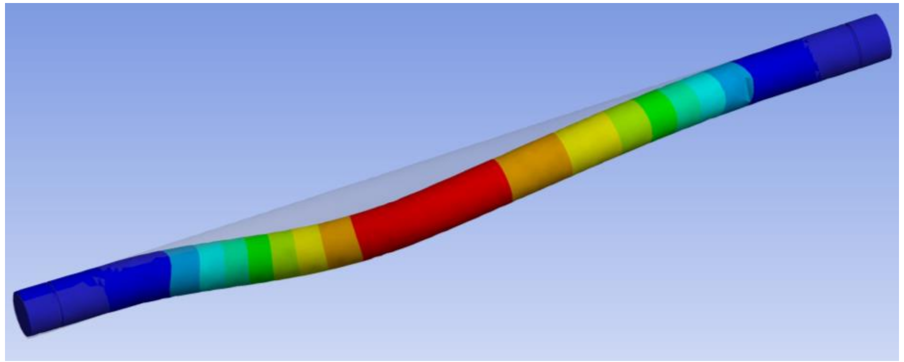




Workshop 08: Results and Validation

- Study Shaft deformation:
  - Insert a Total Deformation result scoped to the Shaft:
    - Select Solution
    - Select Shaft in the graphics window, RMB → Insert → Deformation → Total
    - Evaluate All Results
  - Record results in your table
- Shaft misalignment calculation: we are interested in X and Z deformations at each end of the shaft. The relative displacement vectors at each end will be calculated with this formula: Sqrt(UX^2+UZ^2). The misalignment will be the sum of the vectors at each end.





Workshop 08: Results and Validation

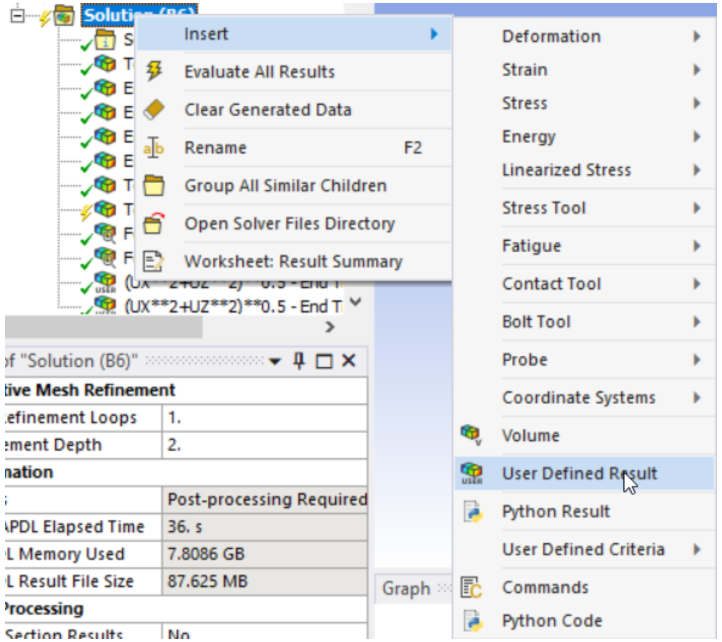
- User Defined Results will be used to display Sqrt(UX^2+UZ^2) calculation at each end of the shaft.

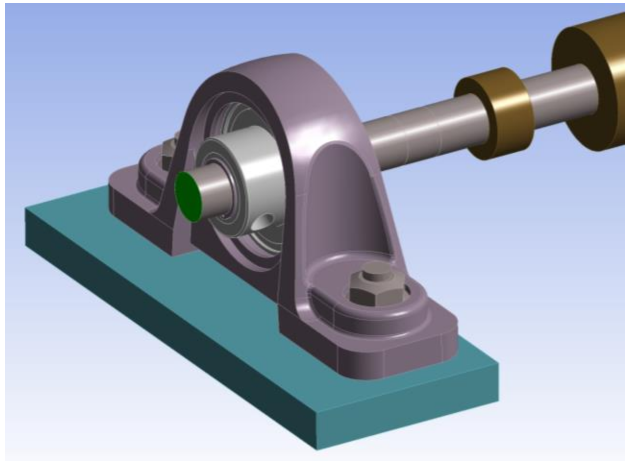
- The User Defined Results feature allows you to perform mathematical operations on results obtained following a solution.

- For more information on this capability, refer to the corresponding section in the ANSYS Mechanical Help System.

Workshop 08: Results and Validation

- Insert a User Defined Result scoped to one end face of the shaft:
  - Select Solution
  - Select a Shaft end face from graphics, RMB → Insert → User Defined Results

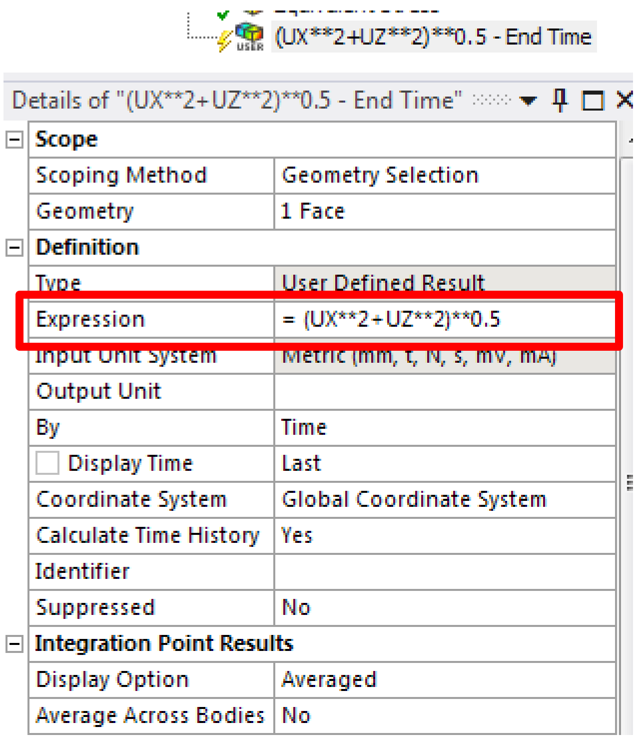




Workshop 08: Results and Validation

- Enter the Expression (UX**2+UZ**2)**0.5
- Insert another UDR scoped to the other side face of the shaft
- Enter the Expression (UX**2+UZ**2)**0.5
- Select all results, then RMB → Rename Based on Definition
- Evaluate all results
- Report both values in the table
- Calculate the relative misalignment: sum both maximum values.

Note: By summing the maximum values, we're taking a simplified approach to calculating the misalignment. If each end of the shaft moves in opposite directions, then the total misalignment will be the sum of each value. If each end moves a different amount but in the same relative direction, then the misalignment will be the difference between each value.



Ansys

Workshop 08: Results and Validation

- At this step of the Workshop, you should have the following results created in Mechanical and the following results in the table for the Stainless Steel housing configuration:

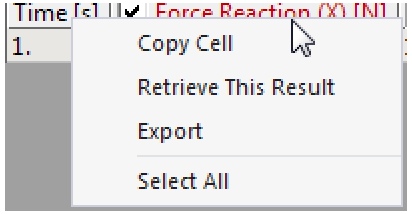
	A	B	C	D	E	F
1			Stainless Steel		Unfilled Polycarbonate	
2	Results	Units	Min	Max	Min	Max
3	Total Deformation - 1. s	mm				
4	Equivalent (von-Mises) Stress - 1. s	Mpa				
5	Equivalent (von-Mises) Stress - Multiple - 1. s (housings)	MPa				
6	Equivalent (von-Mises) Stress - Multiple - 1. s (bearings)	MPa				
7	Equivalent (von-Mises) Stress - Component4\Shaft - 1. s	MPa				
8	Total Deformation - Component4\Shaft - 1. s	mm				
9	(UX**2+UY**2)**0.5 - 1. s	mm				
10	(UX**2+UY**2)**0.5 - 1. s	mm				
11						
12	Misalignment calculation					

*Hint: Generate this table automatically by clicking on **Solution**, activating the **Worksheet** view, and choosing **Result Summary** from the 4 options. The table can then be exported via **RMB - Solution → Export Text File**.*

Workshop 08: Results and Validation

- We are now going to check static forces equilibrium: ∑Forces applied = ∑ Reactions forces:
  - Select both Fixed Supports and drag and drop them onto the Solution branch.

  - 2 Force Reaction results are created. RMB→ Evaluate all results.
  - Copy the results to your table of results.



Tabular Data

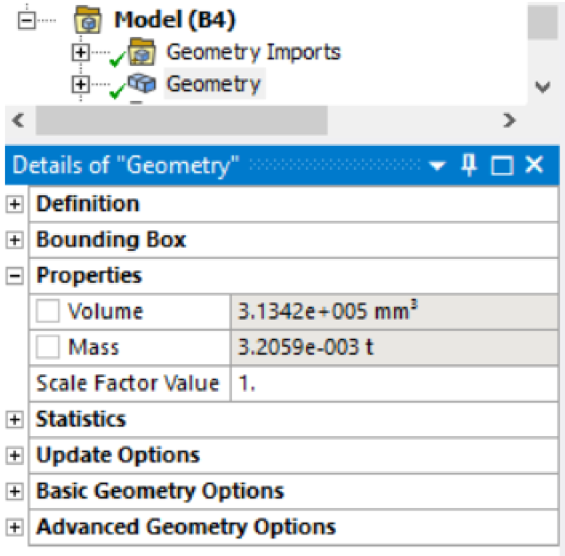
	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1.	-1331.9	148.01	12.298	1340.2

Tabular Data

	Time [s]	Force Reaction 2 (X) [N]	Force Reaction 2 (Y) [N]	Force Reaction 2 (Z) [N]	Force Reaction 2 (Total)
1	1.	-833.12	-148.01	769.12	1143.5

Workshop 08: Results and Validation

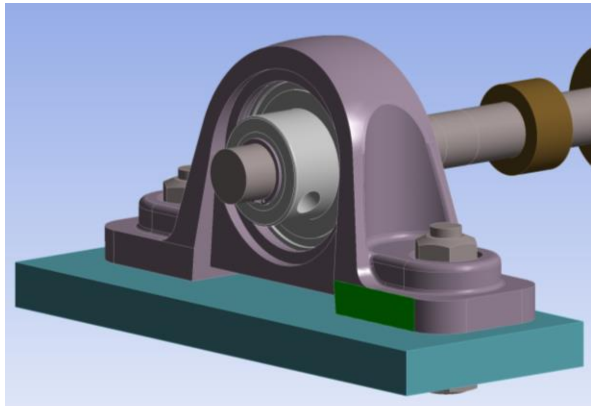
- Note the X, Y, and Z magnitudes of each applied load:
  - Standard Earth Gravity:
    F_z = mass*gravity
    $F_z = 3.21 * (-9.80) = -31.46$ N
  - Force 1: $F_z = -2000$ N
  - Force 2: In the global coordinate system, $F_X = 2165.06$ N and $F_z = 1250$ N

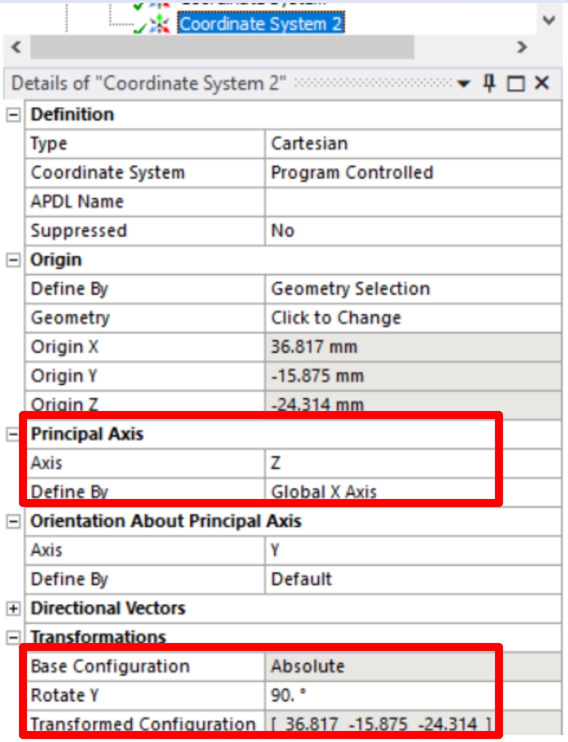


- For each component, sum the reactions forces and applied load.
- You should retrieve 0 for each. Static equilibrium is verified.

Workshop 08: Results and Validation

- We'll focus now on the housing bodies. The sides of the housings are thin walls, and we will check results in those regions. Create a Section View to review stress inside Housing:
  - Select the Coordinate Systems branch.
  - Select the face as on the image
  - RMB →Insert → Coordinate System
  - Align the Z axis with the global X-axis
  - Add a 90° rotation transformation about Y axis





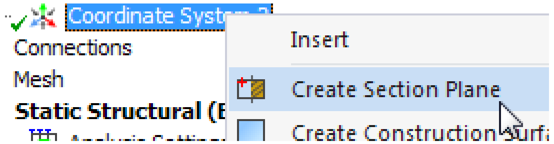
Coordinate System 2

Details of "Coordinate System 2"

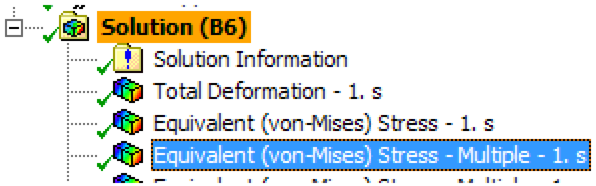
Definition	
Type	Cartesian
Coordinate System	Program Controlled
APDL Name	
Suppressed	No
Origin	
Define By	Geometry Selection
Geometry	Click to Change
Origin X	36.817 mm
Origin Y	-15.875 mm
Origin Z	-24.314 mm
Principal Axis	
Axis	Z
Define By	Global X Axis
Orientation About Principal Axis	
Axis	Y
Define By	Default
Directional Vectors	
Transformations	
Base Configuration	Absolute
Rotate Y	90. °
Transformed Configuration	[36.817 -15.875 -24.314]

Workshop 08: Results and Validation

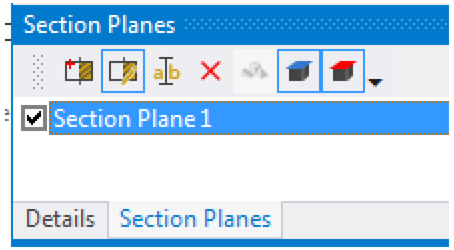
- RMB on the newly created Coordinate System → Create Section Plane.



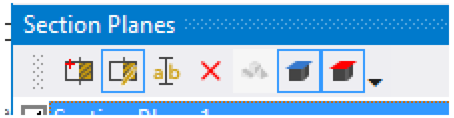
- Review stress results in the housings



- Activate the section plane view

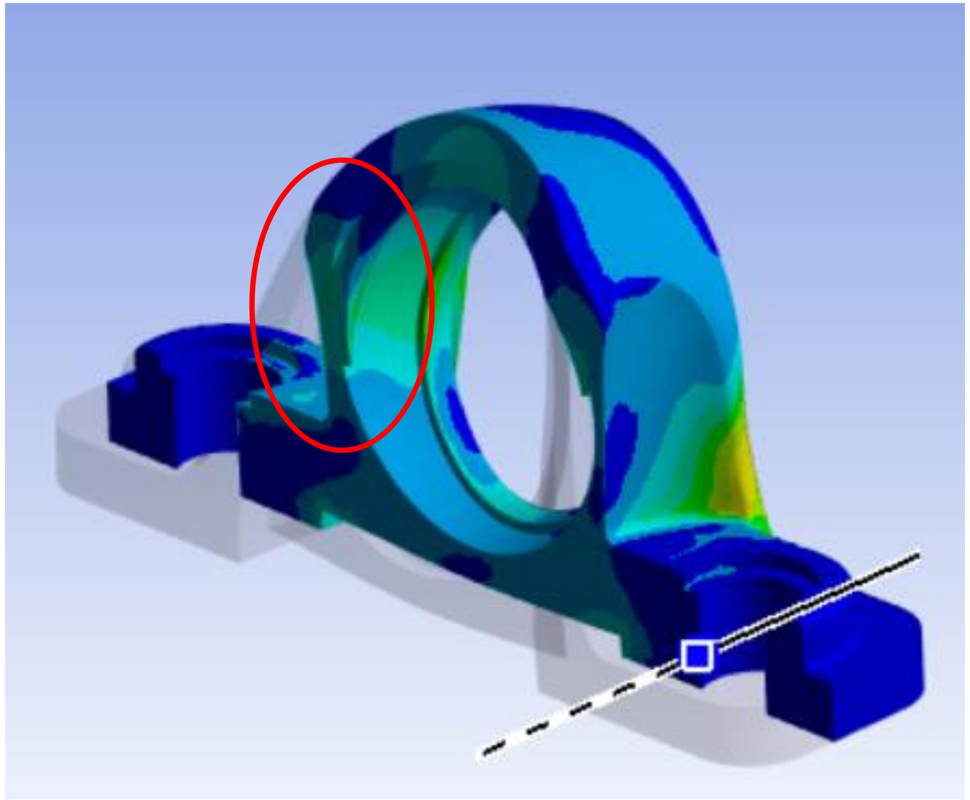


- Click on the Section Plane 1 and Edit Section Plane



Workshop 08: Results and Validation

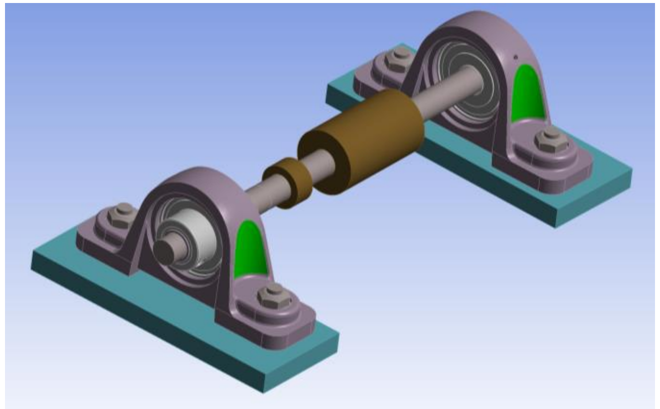
- Use the drag handle to review results inside the body. Pay particular attention to the thin wall section.

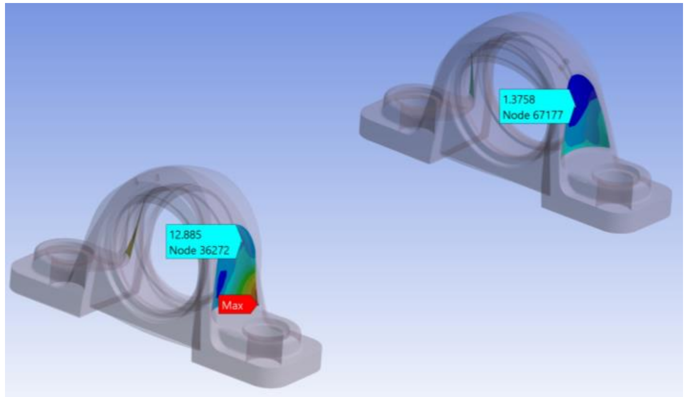


Workshop 08: Results and Validation

- Turn off the Section Plane View.
- Insert an Equivalent Stress result scoped to the side faces of the housings.

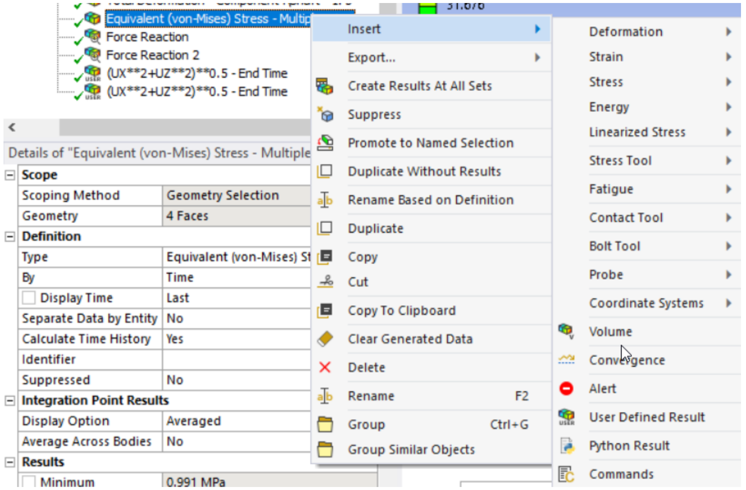
- Evaluate all results.
- Verify that the stress levels are acceptable.
- Use probes as necessary.

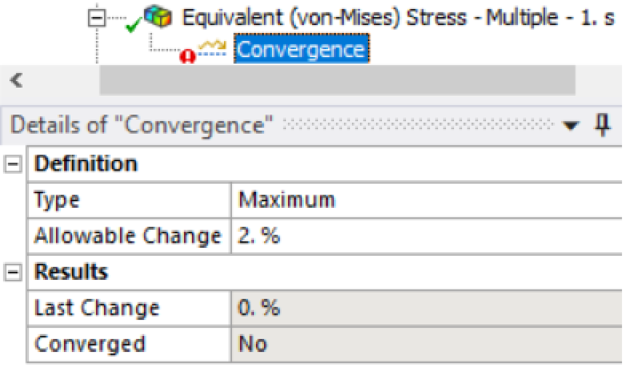


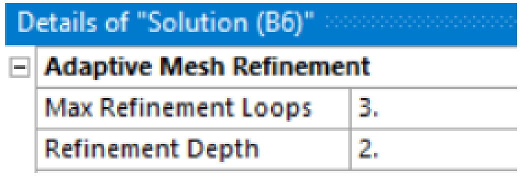


Workshop 08: Results and Validation

- Evaluate Stress Convergence on the newest result:
  - RMB→ Insert → Convergence
  - Define 2% Allowable Change and set Max Refinement Loops to 3
  - Solve the analysis.
  - Review the stress results to see how they have changed.

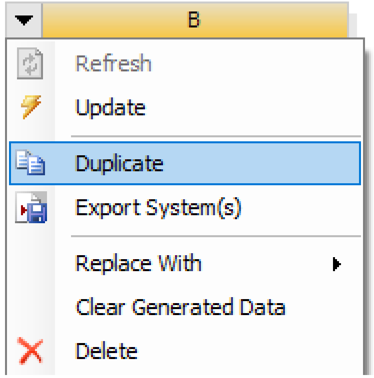


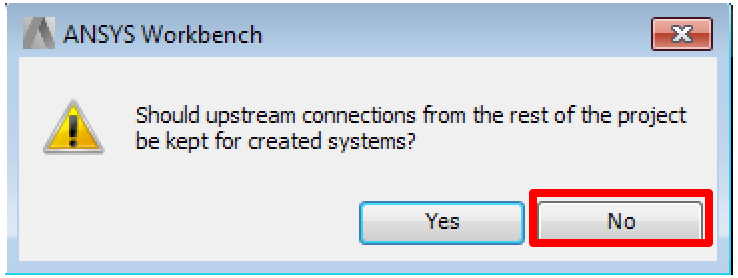


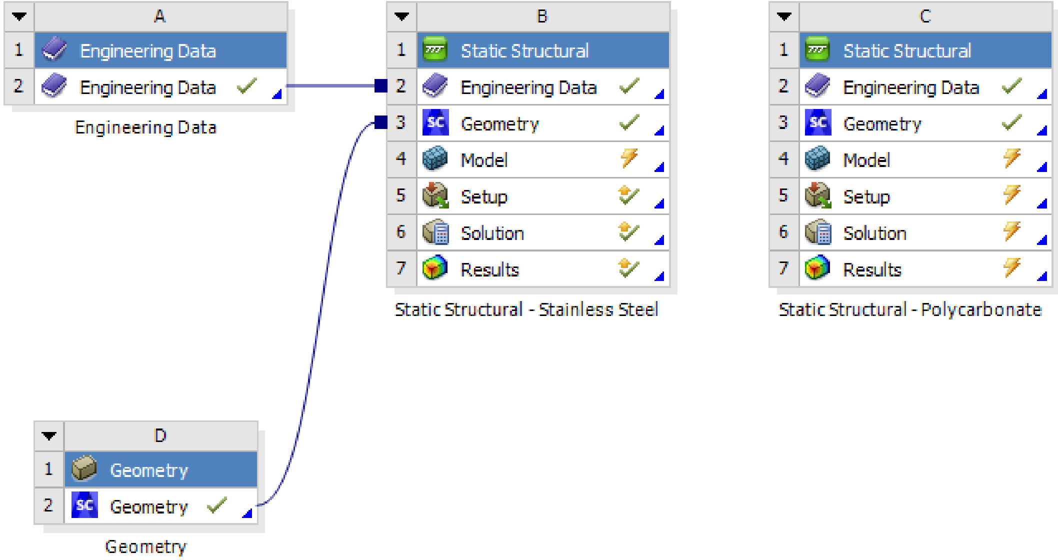


Workshop 08: Results and Validation

- We'll now consider the case with the housings in polycarbonate material:
  - Close Mechanical and return to the Workbench project page.
  - Duplicate System B: click on the top left arrow and pick Duplicate from the drop-down menu.
  - Choose No in the subsequent dialog.
  - Rename the newly created system to Static Structural – Polycarbonate.

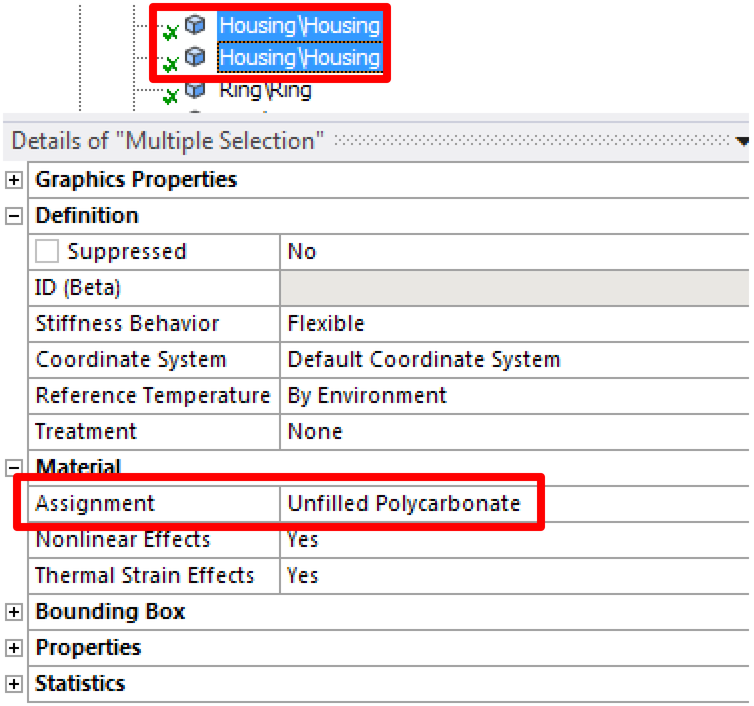


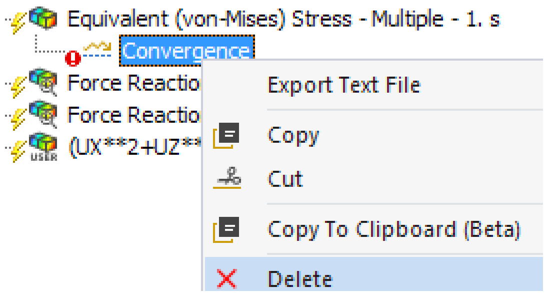




Workshop 08: Results and Validation

- Double click on Model Cell C4 to open Mechanical
- Change the material assignments for both Housing parts to Unfilled Polycarbonate
- Delete the convergence tool
- Solve the Analysis

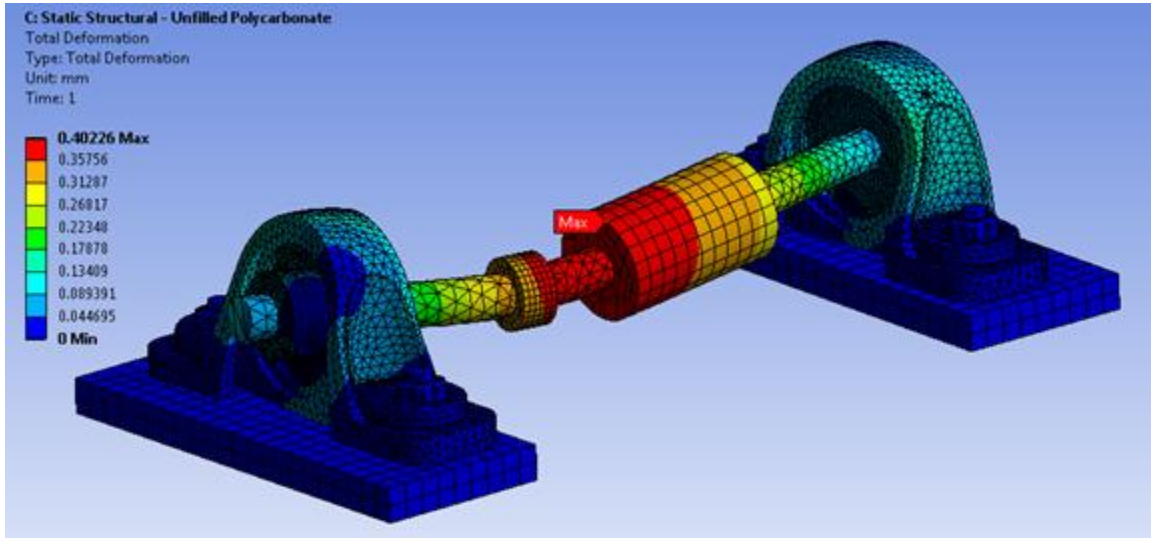




Ansys

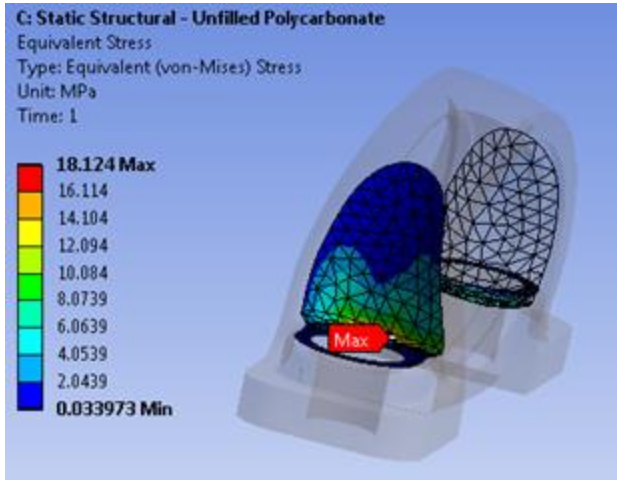
Workshop 08: Results and Validation

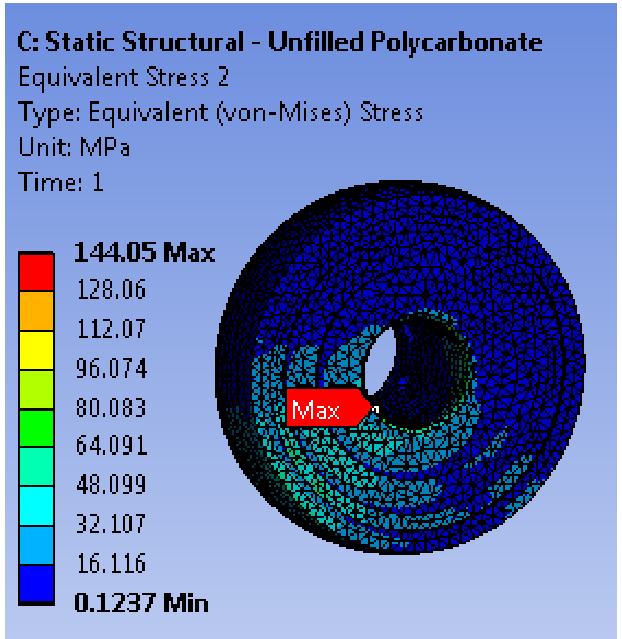
- Review deformation and stress results
- Finish filling in the results table and make a comparison with the Stainless Steel housing results.



Workshop 08: Results and Validation

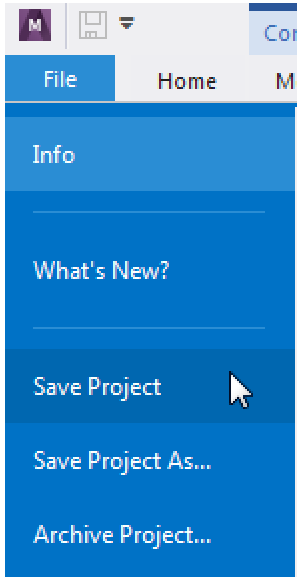
- Draw conclusions. For the Unfilled Polycarbonate case:
  - The location of maximum stress in the housings has changed.
  - Stress results need to be refined in the thin-wall regions of the housings.
  - Stress results need also to be refined in the bearings.
  - Stress are higher and beyond yield strength in the shaft.
  - Misalignment between the ends of the shaft has increased by more than 5%.
  - The unfilled polycarbonate material does not satisfy the specified design constraints. Further work is needed to refine the model and validate this conclusion.





Workshop 08: Results and Validation

- Save Project for later use if desired.



Ansys