Ansys Mechanical Linear and Nonlinear Dynamics

WS 05.1: Linear Perturbation with Two Beams

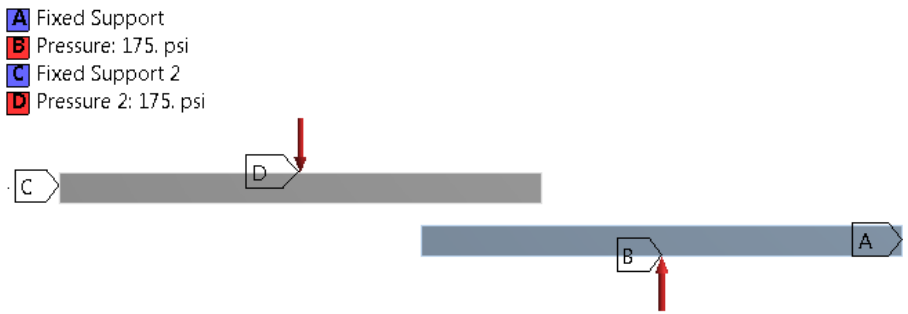
Release 2022 R2

Please note:

- These training materials were developed and tested in Ansys Release 2022 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

Workshop 05.1 - Goals

- Our goal is to determine the effect a nonlinear pre-stress has on the first 6 natural frequencies and mode shapes of opposing cantilevered beams.
- The beams are manufactured of Aluminum and are subjected to large deformation, nonlinear contact, and plasticity.

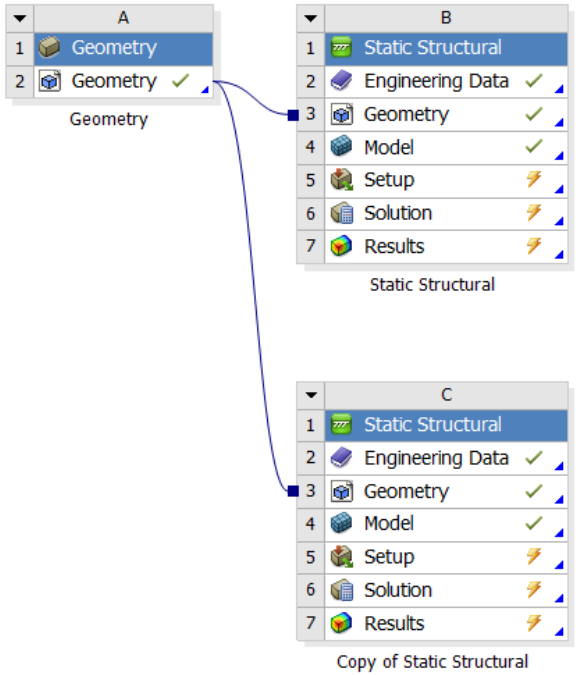


Workshop 05.1 – Project Schematic and Approach

- Begin a new Workbench session and, from the Project page, open the archive file “WS05.1_Linear_Perturbation.wbpz” and save the project to a convenient location.
- The project contains two identical Static Structural Schematics. Schematic B will be used to conduct a modal analysis of two opposing cantilevered beams, considering the geometric deformation from the static analysis WITHOUT including the pre-stress effects.
- Schematic C will be used to conduct a pre-stressed modal analysis of the same two beams, considering both the geometric deformation as well as the pre-stress effects due to large deflection, contact status and plasticity.
- Results from both scenarios will be compared.

Workshop 05.1 – Project Schematic and Approach

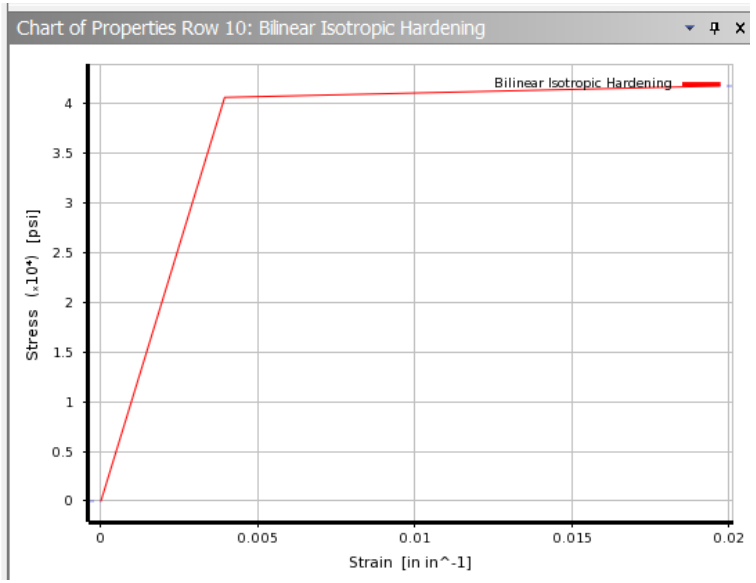
- Set the Project Units to U.S. Customary, and Display Values in Project Units.
- Edit the Engineering Data cell of Schematic B.
  - Examine the material properties for Aluminum, noting the Bilinear Plasticity



	A	B	C	D
1	Contents of Engineering Data			Source
2	Material			
3	Aluminum Alloy NL			General Materials N
*	Click here to add a new material			

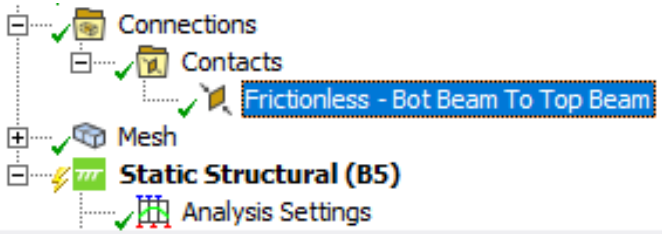
Properties of Outline Row 3: Aluminum Alloy NL

	A	B	
1	Property	Value	
2	Material Field Variables	Table	
3	Density	0.10007	lbm in^-3
4	Isotropic Elasticity		
5	Derive from	Young's Modulus a...	
6	Young's Modulus	1.0298E+07	psi
7	Poisson's Ratio	0.33	
8	Bulk Modulus	1.0096E+07	psi
9	Shear Modulus	3.8713E+06	psi
10	Bilinear Isotropic Hardening		
11	Yield Strength	40611	psi
12	Tangent Modulus	72519	psi



Workshop 05.1 – Confirm Static Structural Setup

- Return to the Project Schematic and Edit the Model cell (B4) to open the Mechanical application
- In Mechanical, set the units system as follows:
  - US Customary (in, lbm, lbf, °F, s, V, A)
- Expand Connections branch and confirm that the single contact definition is set to Frictionless behavior. This will allow the beams to deflect towards one another and eventually come into contact.

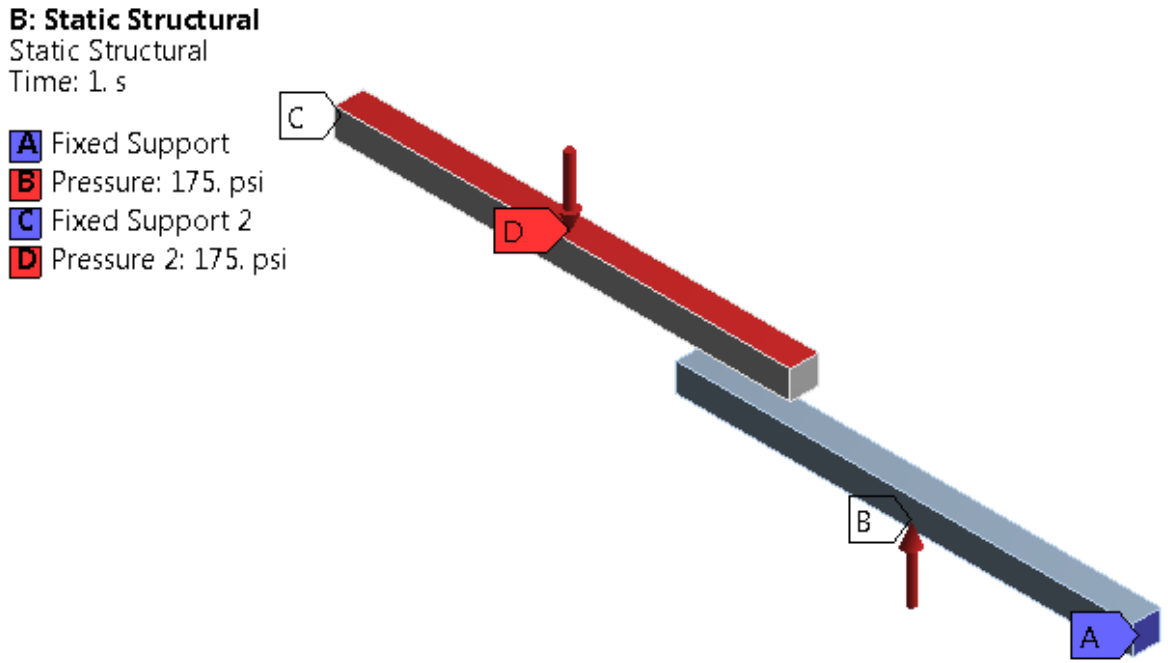


Details of "Frictionless - Bot Beam To Top Beam"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Bot Beam
Target Bodies	Top Beam
Protected	No
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Suppressed	No

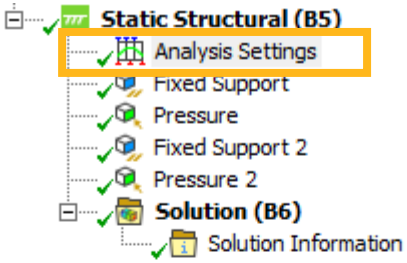
Workshop 05.1 – Confirm Static Structural Setup

- Examine the Fixed Support and Pressure load applied to each beam.



Workshop 05.1 – Confirm Static Analysis Settings and Solve

- Highlight the Analysis Settings Branch.
  - Confirm Automatic Time Stepping is enabled, with 5 initial/minimum and 10 maximum substeps
  - Confirm Large Deflection effects are turned on. This ensures greatest accuracy in the static structural solution
- Solve the Static Structural Analysis.



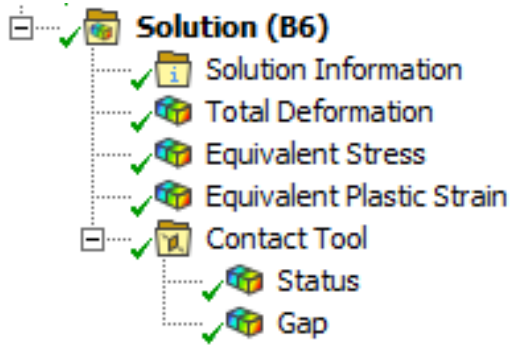
Details of "Analysis Settings"

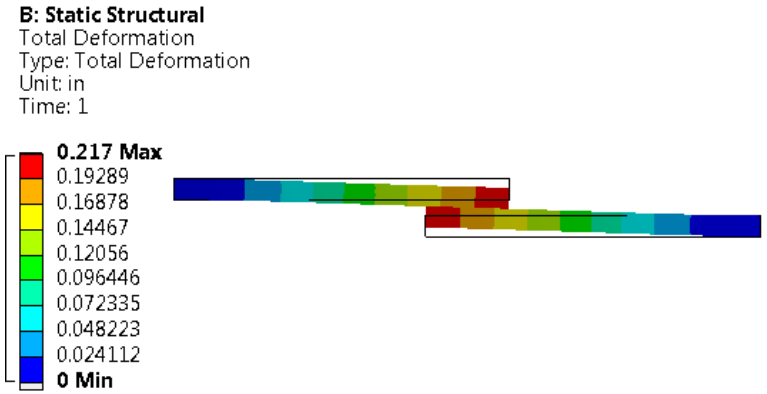
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	10.
Minimum Substeps	10.
Maximum Substeps	100.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Quasi-Static Solution	Off

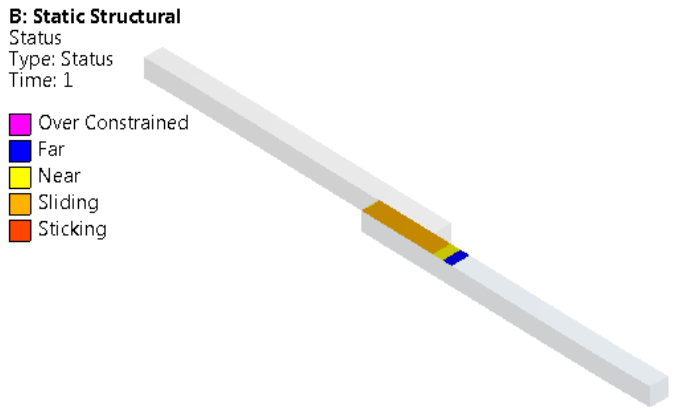
Workshop 05.1 – Post Process the Static Structural Solution

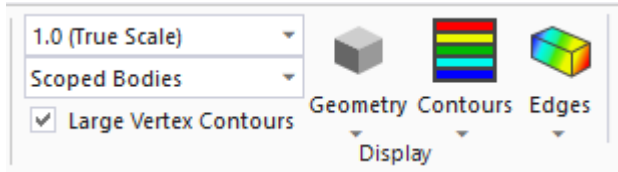
Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences

- Highlight the Solution Branch and:
  - Insert “Total Deformation”, “Equivalent Stress”, and “Equivalent Plastic Strain”
  - Insert a “Contact Tool”
  - Evaluate all these results and review each
- Set the deformed shape scale to “True Scale” and confirm that the beams have established frictionless, sliding contact.



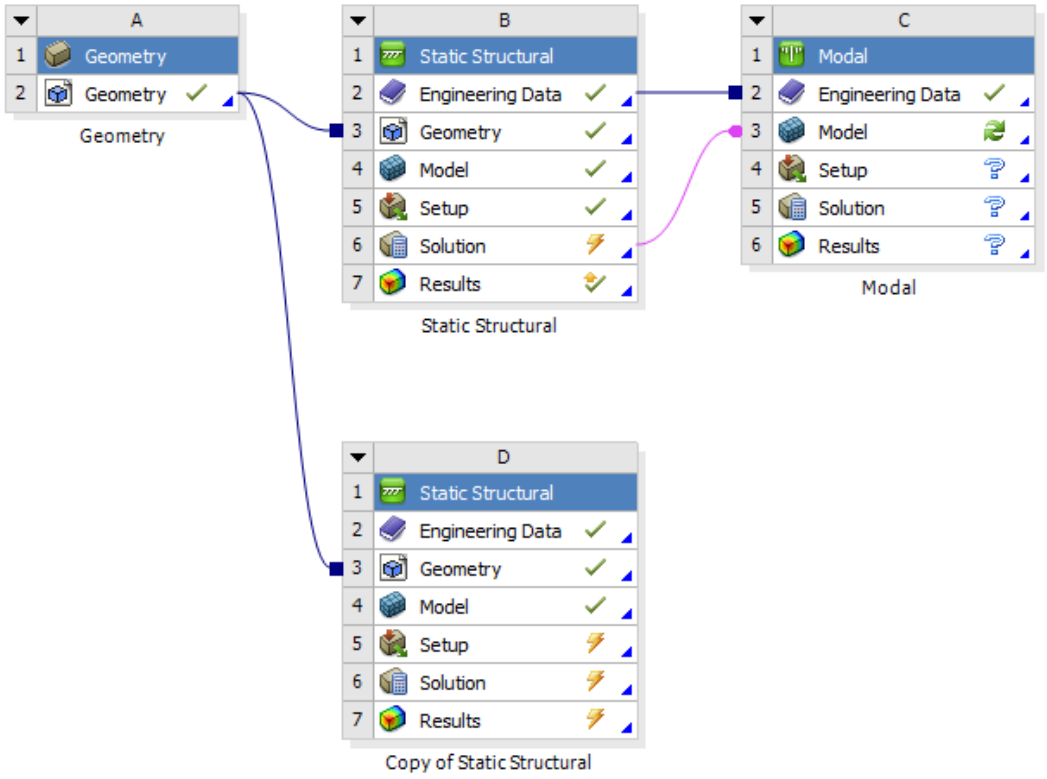






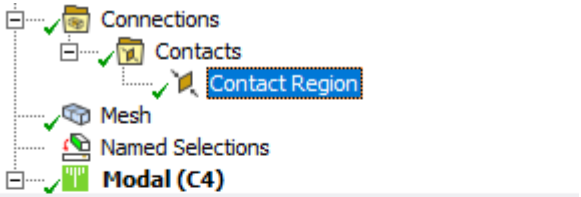
Workshop 05.1 – Set Up Modal Analysis, without Pre-stress

- Return to Project Page, drag and drop a Modal analysis to the right of Schematic B, but do not link any cells.
- Link the Engineering Data cells (B2 > C2). This will share the Aluminum material.
- Link the Solution cell to the Model Cell (B6 > C4). Note this will remove the Geometry cell in the Modal analysis. This step uses the deformed geometry from the Static Structural as the starting geometry for the model, WITHOUT including pre-stress effects.



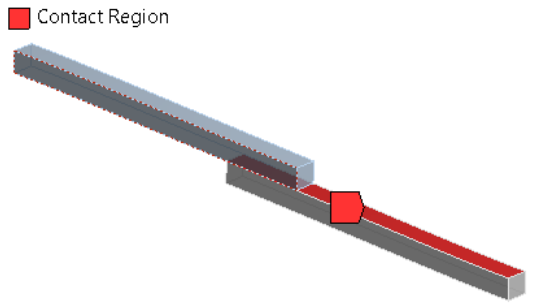
Workshop 05.1 – Set Up Modal Analysis, without Pre-stress

- Update cell B6, Refresh cell C3, then edit the Model cell (C3) in order to open the Mechanical Modal.
- Confirm the geometry represents the deformed geometry from the static structural solution.
- Confirm that a bonded contact pair has automatically been detected between the two beams.



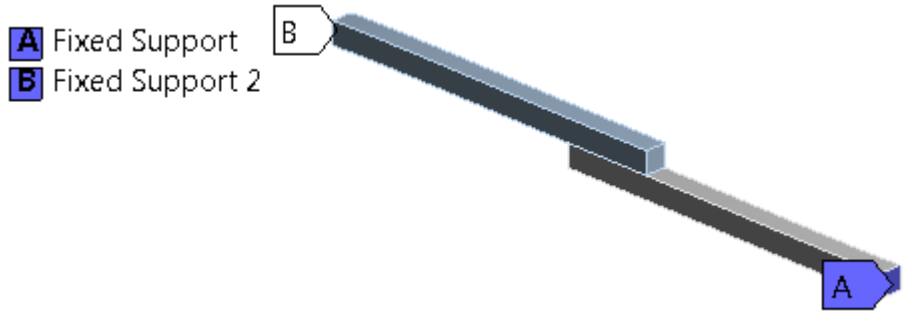
Details of "Contact Region"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Bot Beam(Static Structural)
Target Bodies	Top Beam(Static Structural)
Protected	No
Definition	
Type	Bonded



Workshop 05.1 – Set Up Modal Analysis, without Pre-stress

- Apply a fixed support on one end of each beam, identical to those in the structural analysis.



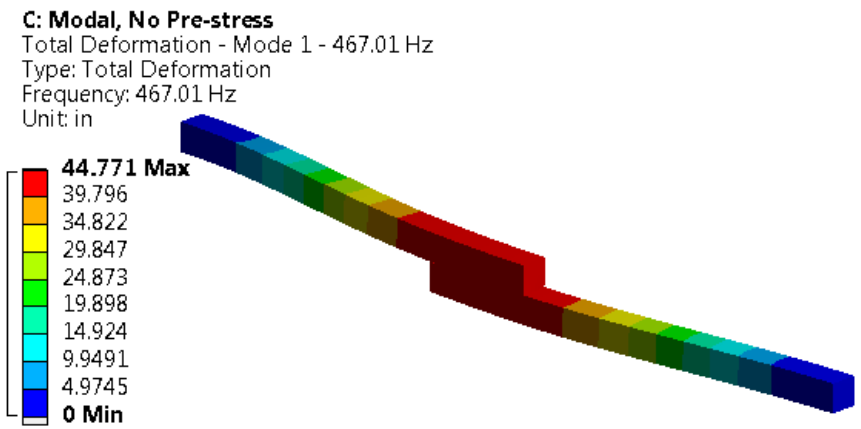
- Highlight the Solution branch and Solve for the first 6 (default) mode shapes.
- When the solution finishes, click on the Solution branch, RMB in the timeline:
  - Select all
  - Create mode shape results
  - Evaluate results

Workshop 05.1 – Set Up Modal Analysis, without Pre-stress

- Make a note of the 6 natural frequencies. Your frequencies may differ slightly from those shown here. We'll compare them with those obtained from a Modal Analysis with Linear Perturbation.

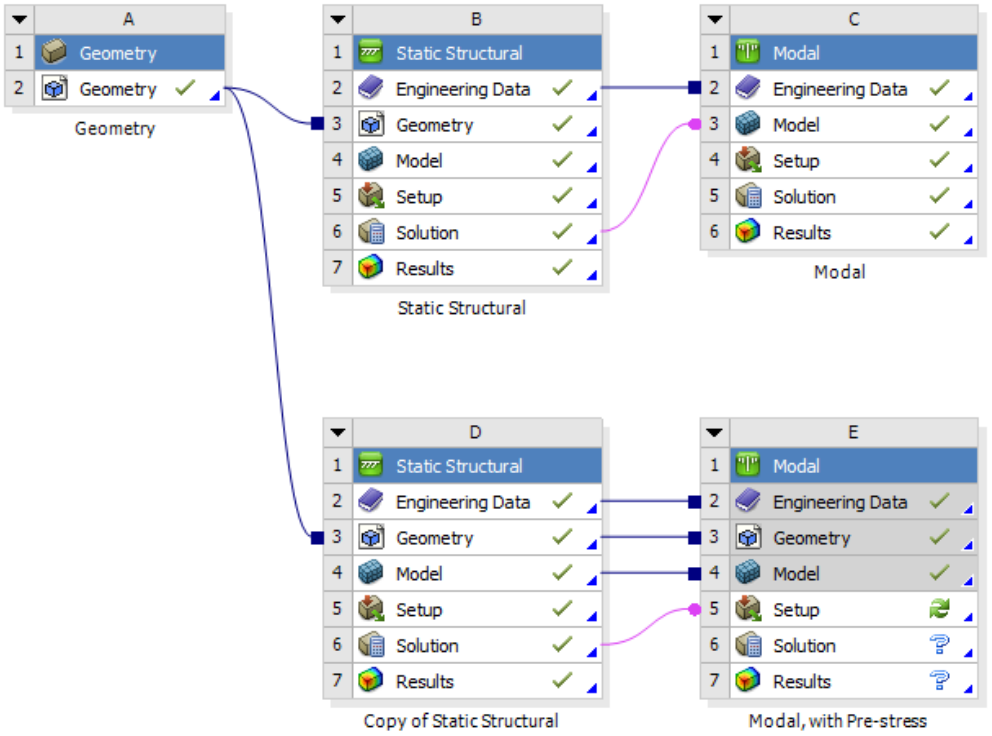
Tabular Data

	Mode	Frequency [Hz]
1	1.	467.01
2	2.	491.68
3	3.	1372.3
4	4.	1631.6
5	5.	2667.1
6	6.	2685.4



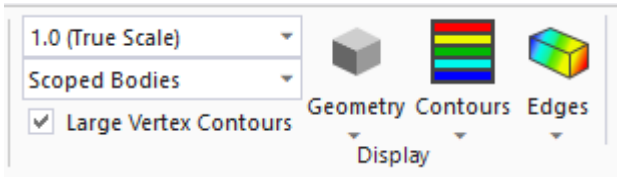
Workshop 05.1 – Set Up Modal Analysis, with Pre-stress

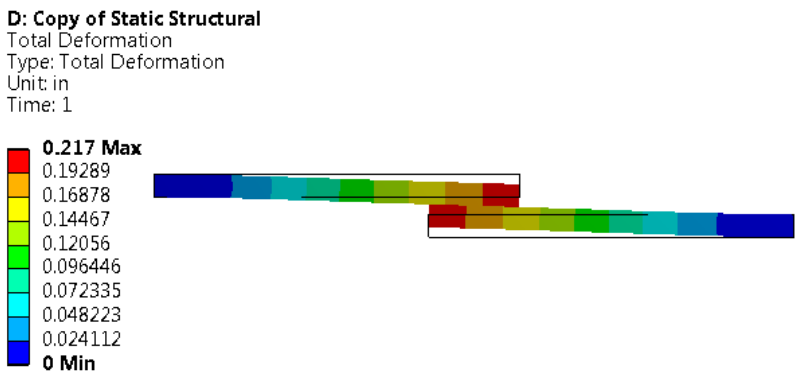
- Return to the Project and click “Update Project”
- This will solve the Static Structural Schematic D, in preparation for the Pre-stressed Modal analysis using Linear Perturbation
- Drag a Modal Analysis System and drop it onto Cell D6. Rename this Schematic to “Modal , with Pre-stress”.

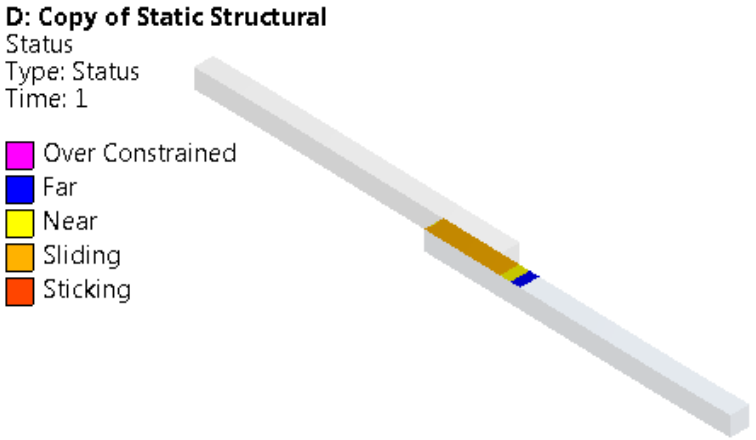


Workshop 05.1 – Set Up Modal Analysis, with Pre-stress

- Refresh Cell E5, then open Mechanical from cell E5.
- Confirm results from Static Structural Solution once again (they will be the same as those from slide 8.

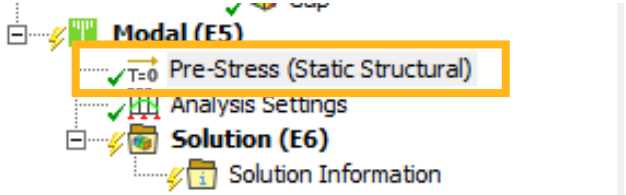






Workshop 05.1 – Establish Pre-stress Conditions

- Click on “Pre-Stress (Static Structural)” and in the details:
  - Set “Pre-stress Define By” to Load Step
  - Set “Pre-stress Loadstep” to Last
  - Set “Contact Status” to Force Bonded
- By setting Contact Status to Force Bonded, we will treat the original frictionless contact as permanently bonded.
- Notice at this point that the geometry appears as the original geometry, prior to the static deflection. Recall that in Phase II of the Perturbation process, the geometry used in the Modal analysis is updated from the deformed geometry of the static analysis. This occurs as a background process and will therefore not be visible in Mechanical.

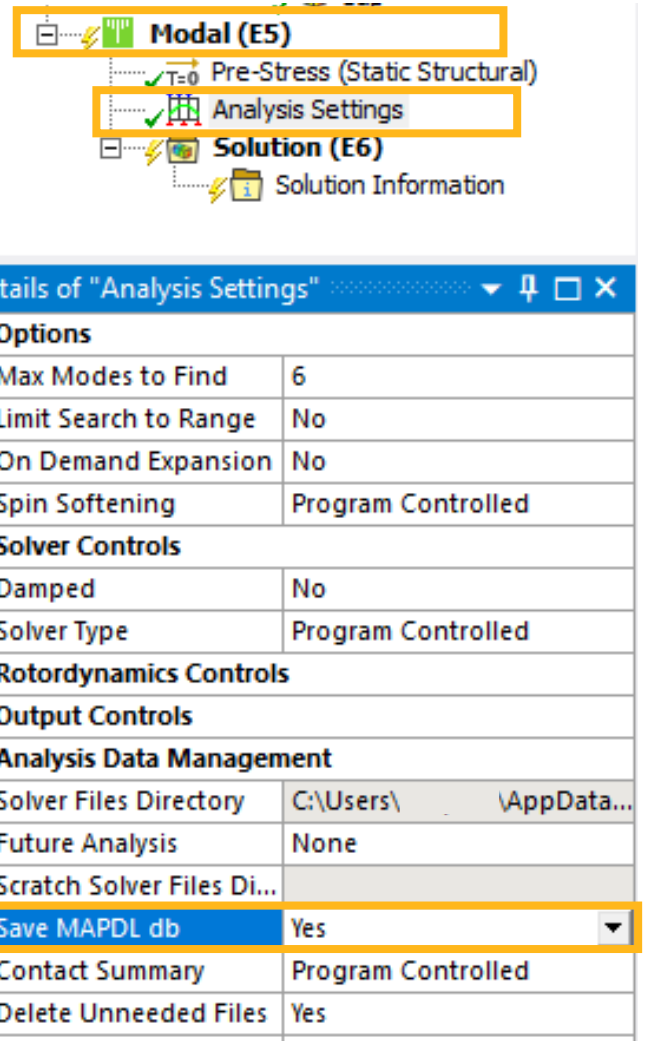


Details of "Pre-Stress (Static Structural)"

Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Load Step
Pre-Stress Loadstep	Last
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Force Bonded
Newton-Raphson Option	Program Controlled

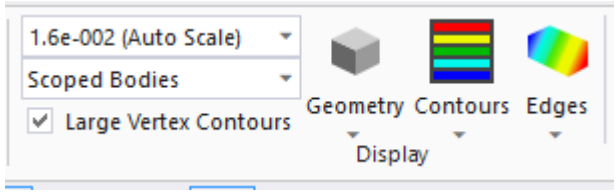
Workshop 05.1 – Establish Pre-stress Conditions

- Notice at this point that the Modal Analysis is ready for solution. Displacement constraints are not needed, as they are managed (inherited) from the Static Structural analysis.
- Prior to solving, we will request an MAPDL database file be saved, as we will use it for later post-processing purposes.
  - Highlight Analysis Settings
  - Under “Analysis Data Management” set Save MAPDL db to Yes
- Solve



Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- Generate results for the 6 modes calculated
- Review the mode shapes. You may have to set the Deformed Shape scaling to “Auto Scale”

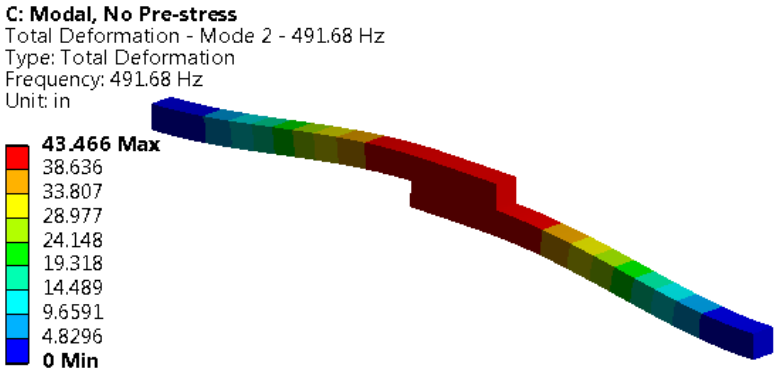


- Compare the frequencies against those obtained without pre-stress effects:

Without Pre-stress

Tabular Data

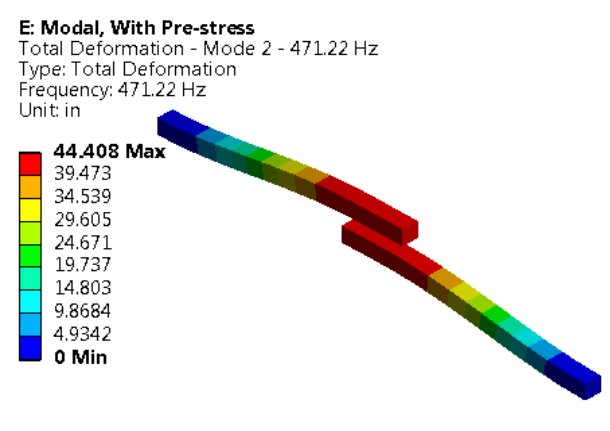
	Mode	Frequency [Hz]
1	1.	467.01
2	2.	491.68
3	3.	1372.3
4	4.	1631.6
5	5.	2667.1
6	6.	2685.4



With Pre-stress

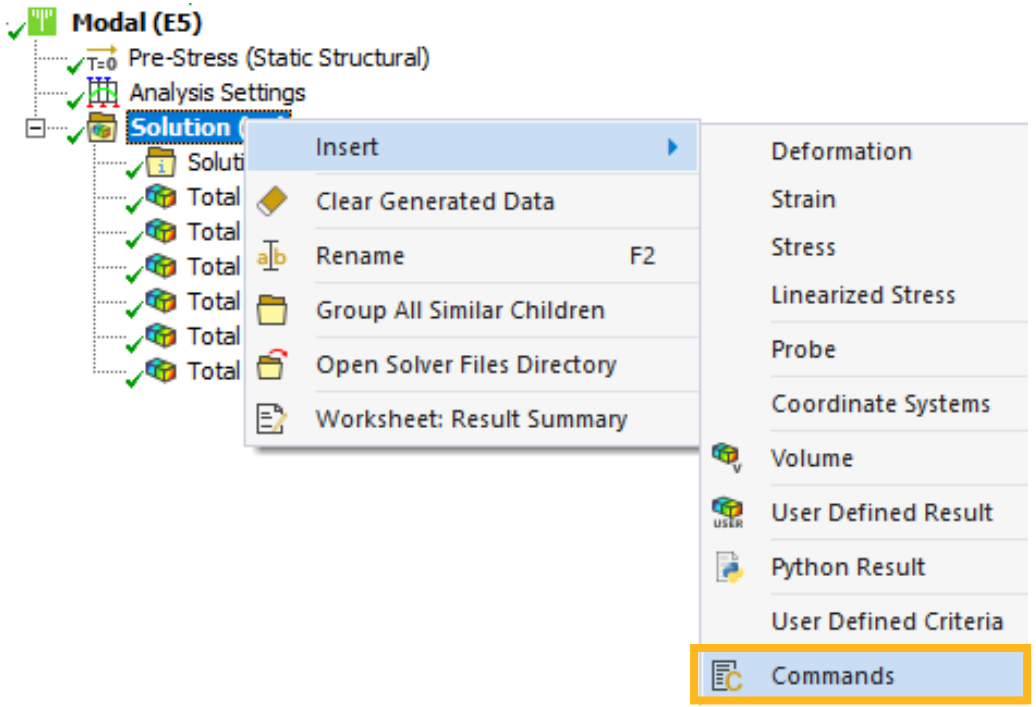
Tabular Data

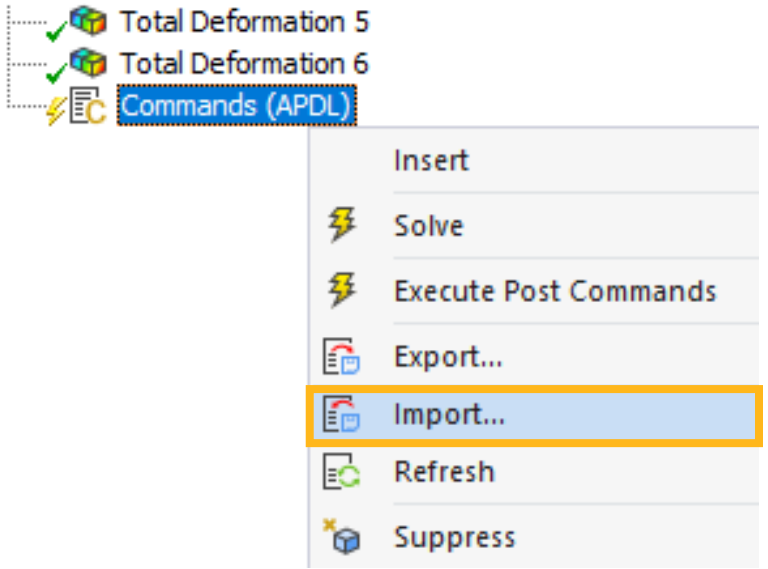
	Mode	Frequency [Hz]
1	1.	457.63
2	2.	471.22
3	3.	1368.4
4	4.	1627.1
5	5.	2588.9
6	6.	2657.4



Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- In order to confirm that the geometry was properly updated, highlight the solution branch and insert Commands.
- RMB on Commands and choose Import; click yes to the message asking if you want to continue.





- Browse to the file called “post commands.txt” supplied with this workshop.

Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- The resulting window is shown here. Review the commands, along with their description. The file instructs Ansys to recall the MAPDL database file saved as part of the modal solution. The contents of this database include the updated geometry from the static structural analysis once the beams have contacted one another.

Commands

```
1  !   Commands inserted into this file will be executed immediately after the Ansys /POST1 command.
2
3  !   Active UNIT system in Workbench when this object was created:  U.S. Customary (in, lbm, lbf, s, V, A) temperature units of F
4
5  FINI            !EXIT THE POST PROCESSOR
6  RESUME,FILE,DB      !RESUME DATABASE FROM PERTURBED MODAL ANALYSIS
7  /POST1          !ENTER THE POST PROCESSOR
8
9  !!!! CHANGE APDL BACKGROUND FROM BLACK TO WHITE
10 /RGB,INDEX,100,100,100, 0
11 /RGB,INDEX, 80, 80, 80,13
12 /RGB,INDEX, 60, 60, 60,14
13 /RGB,INDEX, 0, 0, 0,15
14 !!!!
15
16 /VIEW,1,0,0,1   !SET VIEW ORIENTATION TO Z AXIS
17 /AUTO,1     !FIT THE MODEL TO THE VIEW
18
19 /SHOW,PNG       !DIRECT PLOTS TO PNG FORMAT
20 EPLOT           !PLOT ELEMENTS (UPDATED GEOMETRY FROM BASE ANALYSIS)
21
22 /VIEW,1,1,1,1   !SET VIEW ORIENTATION TO ISOMETRIC
23 /AUTO,1
24
25 !LOOP THROUGH EACH MODE SHAPE AND GENERATE DEFORMATION PLOTS
26 *DO,i,1,6
27 SET,1,i      !READ EACH MODE SHAPE INTO MEMORY
28 /SHOW,PNG
29 PLNSOL,U,SUM     !PLOT TOTAL DEFORMATION
30 *ENDDO       !END OF *DO LOOP
31
32
```

Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

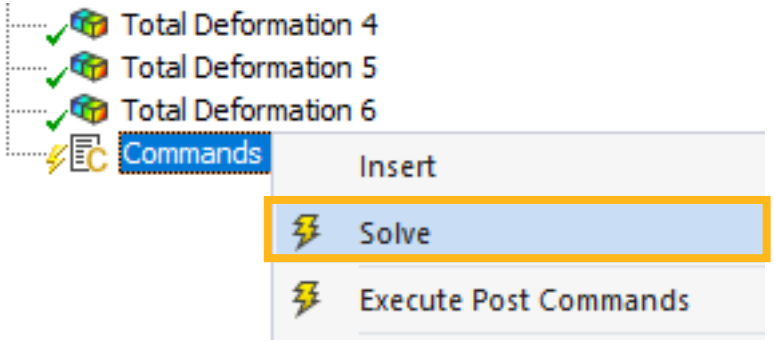
- The file then instructs Ansys to plot the elements of the deformed geometry.
- Finally, we loop through all 6 mode shapes, capturing plots of each.

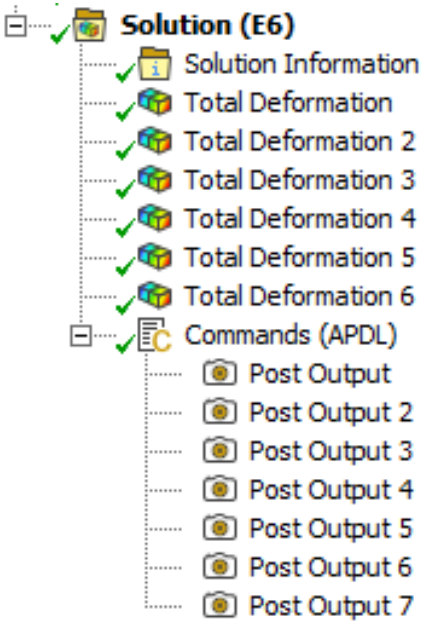
Commands

```
1  !   Commands inserted into this file will be executed immediately after the Ansys /POST1 command.
2
3  !   Active UNIT system in Workbench when this object was created:  U.S. Customary (in, lbm, lbf, s, V, A) temperature units of F
4
5  FINI            !EXIT THE POST PROCESSOR
6  RESUME,FILE,DB      !RESUME DATABASE FROM PERTURBED MODAL ANALYSIS
7  /POST1          !ENTER THE POST PROCESSOR
8
9  !!!! CHANGE APDL BACKGROUND FROM BLACK TO WHITE
10 /RGB,INDEX,100,100,100, 0
11 /RGB,INDEX, 80, 80, 80,13
12 /RGB,INDEX, 60, 60, 60,14
13 /RGB,INDEX, 0, 0, 0,15
14 !!!!
15
16 /VIEW,1,0,0,1   !SET VIEW ORIENTATION TO Z AXIS
17 /AUTO,1     !FIT THE MODEL TO THE VIEW
18
19 /SHOW,PNG       !DIRECT PLOTS TO PNG FORMAT
20 EPLOT           !PLOT ELEMENTS (UPDATED GEOMETRY FROM BASE ANALYSIS)
21
22 /VIEW,1,1,1,1   !SET VIEW ORIENTATION TO ISOMETRIC
23 /AUTO,1
24
25 !LOOP THROUGH EACH MODE SHAPE AND GENERATE DEFORMATION PLOTS
26 *DO,i,1,6
27 SET,1,i      !READ EACH MODE SHAPE INTO MEMORY
28 /SHOW,PNG
29 PLNSOL,U,SUM    !PLOT TOTAL DEFORMATION
30 *ENDDO       !END OF *DO LOOP
31
32
```

Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- RMB on Commands and select Solve. Ansys will execute the commands in the background and return seven static image captures under the Commands branch.



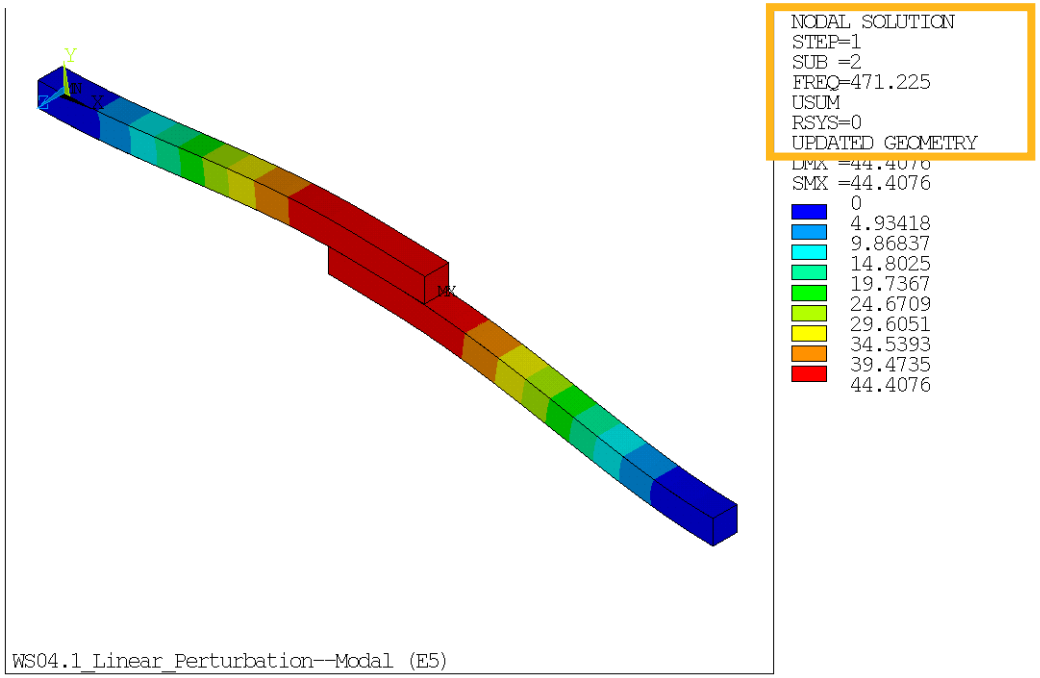


Clicking each “Post Output” item will display the corresponding graphic in the Mechanical graphics window, confirming the updated geometry without the presence of a gap between the beams. This behavior is a result of having large deflection turned “on” in the static structural pre-stress solution.

Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

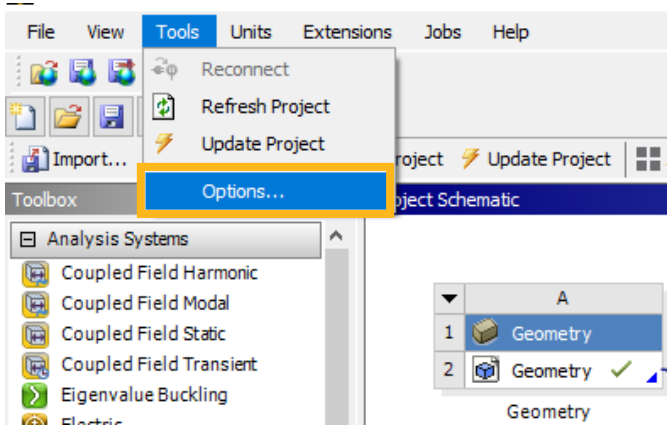
- Pre-stressed geometry:

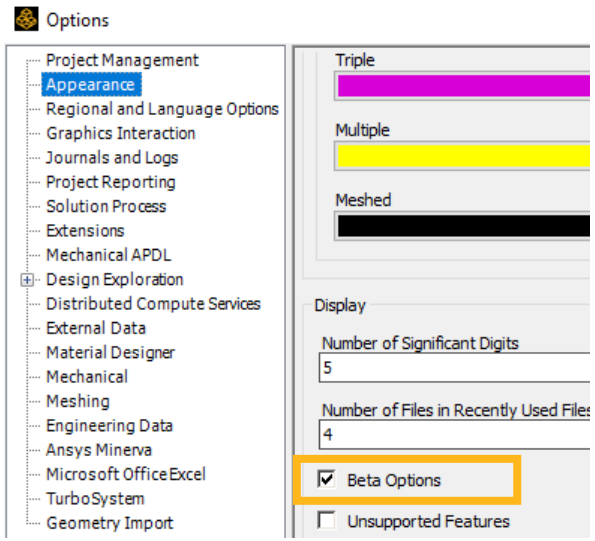
- Representative mode shapes (mode 2 shown):

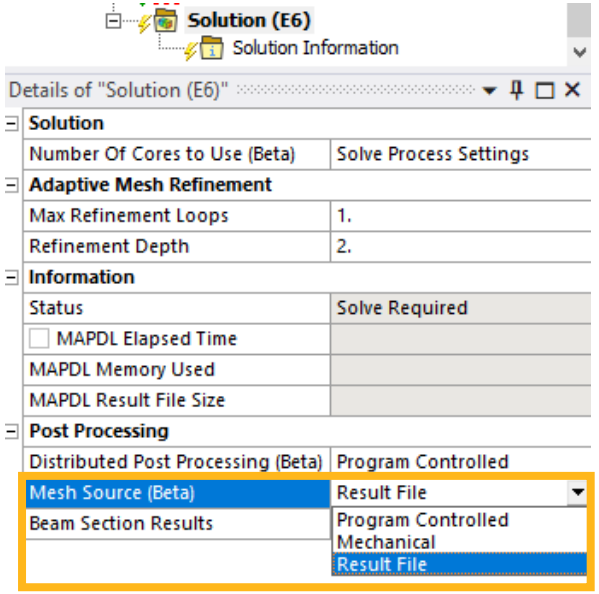


Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- Another way to view the modal shapes based upon the deformed geometry is by using the Mesh Source (Beta) feature.
- On the Workbench Project page, select Tools > Options and then select the Appearance option. Scroll down and activate the Beta Options check box. Click OK and return to Mechanical
- In the Details of the Solution object notice the newly added Mesh Source (Beta) feature. Set it to Result File and generate results.

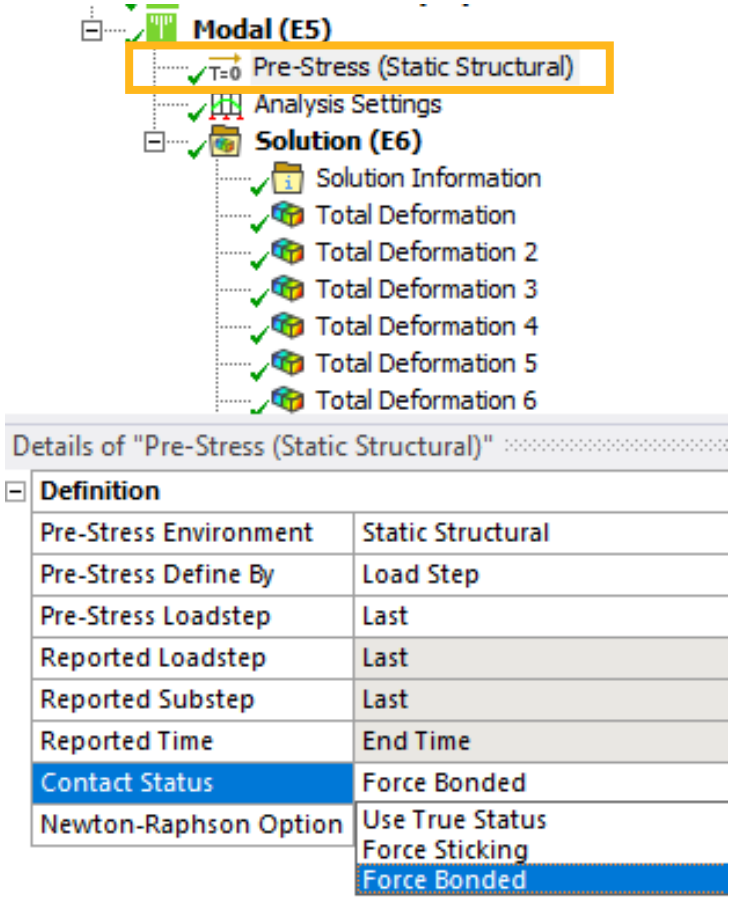






Workshop 05.1 – Post Process the Pre-stressed Modal Analysis

- Go Further! – Experiment with the various options available for the “Contact Status” definition under “Pre-Stress (Static Structural)”, solving each and noting the change in behavior of the model.



End of presentation