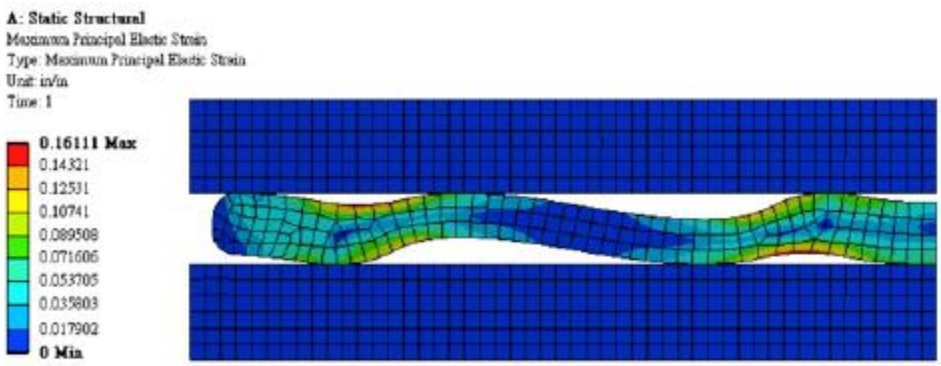
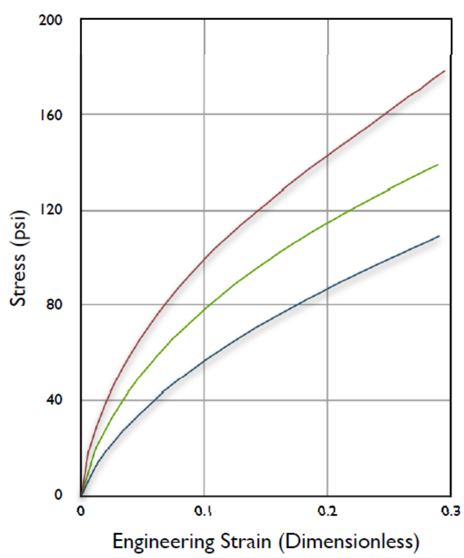
Hyperelastic Seal Analysis

The following tutorial will set up the analysis for modeling a hyperelastic seal. The analysis will be set up as a plane strain problem. The material properties for the hyperelastic material were obtained from a uniaxial tensile test, a biaxial tensile test, and a shear test. Additional key features of this analysis is the set up of a contact analysis.



The material models well be first input as tables from the data collected. Secondly, the data will be fit to the two-parameter Mooney-Rivlin hyperelastic model. By contrast, a linearly elastic material will only require the modulus of elasticity to be entered since the stress-strain relation is linear and the slope is constant.

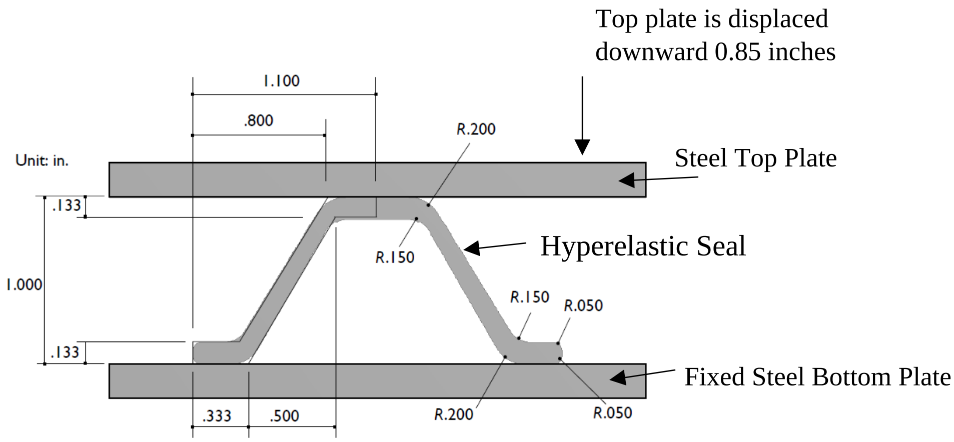
An example of the Stress-Strain plot for the tabulated data of a hyperelastic material is shown in the graph below



Biaxial Tensile Test (red)

Shear Test (green)

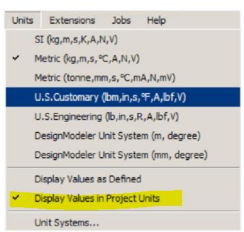
Uniaxial Tensile Test (red)



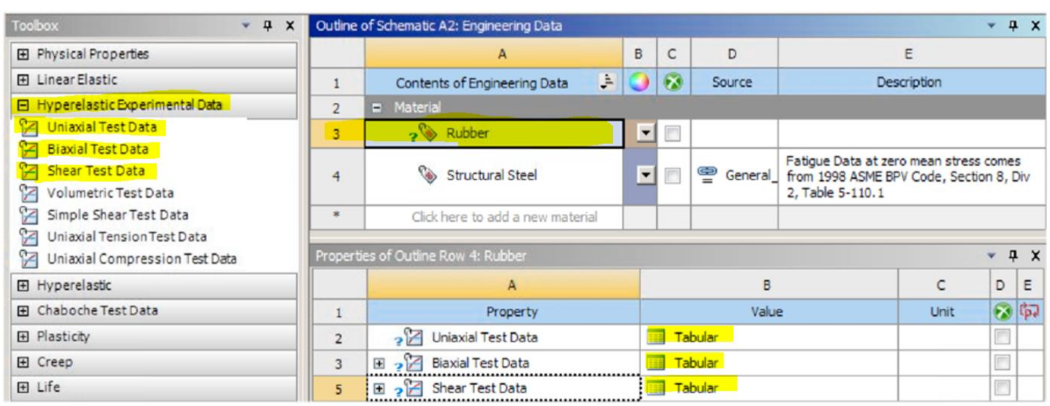
1. Prepare the geometry described in the previous tutorial.
2. Prepare the material properties for the hyperelastic material.

1. Double click on Engineering Data and select the units: US Customary. 2. Verify that there is a check mark next to “Display Values in Project Units”.

Note that in the project schematic a green checkmark will appear when that module has been completed. A green checkmark means that you can go on to the next step.

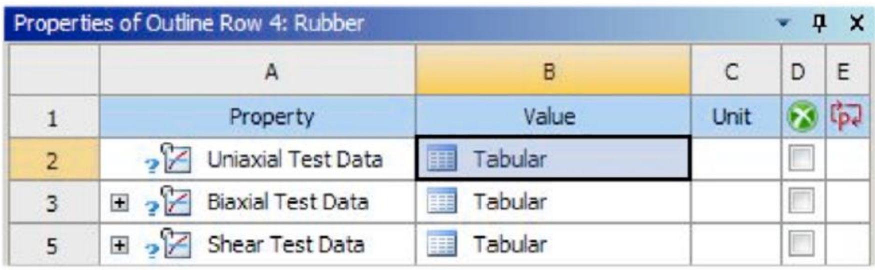


3. Click to add a new material and name it Rubber by typing in the space. 4. Expand Hyperelastic Experimental Data and double-click Uniaxial Test Data, Biaxial Test Data, and Shear Test Data.



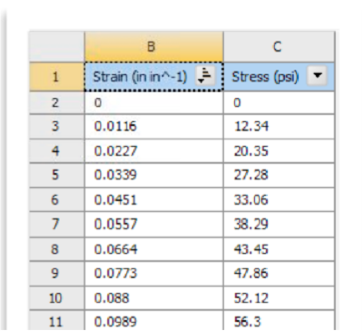
5. Click the Tabular box and enter the corresponding property test data. You can copy the data from the Excel file that was shared with you in class. Repeat for Uniaxial, Biaxial, and Shear Test data.

Note: If we were dealing with a linearly elastic, isotropic material it would not be necessary to enter the data in the form of a table. In that case the slope of the stress-strain relation is constant. Therefore, we would just enter the slope (which is E, the modulus of elasticity). However, in the case of the hyperelastic material the slope is not constant and we would either need to input the actual data in tabular form, or we can select a non-linear model that fits the data. This alternate method will be shown below.

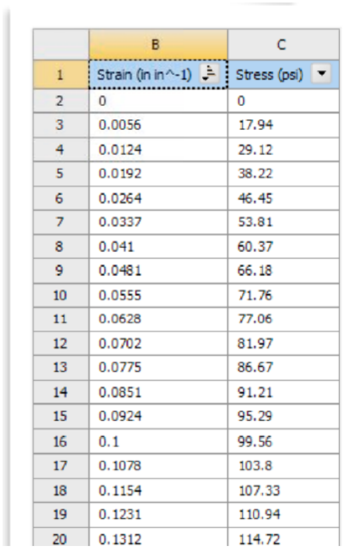

Properties of Outline Row 4: Rubber

	A	B	C	D	E
1	Property	Value	Unit		
2	Uniaxial Test Data	Tabular			
3	Biaxial Test Data	Tabular			
5	Shear Test Data	Tabular			

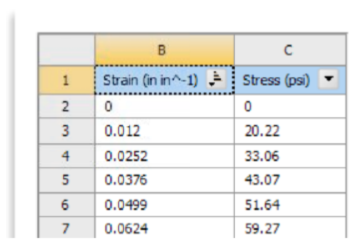
Type or copy/paste the data from the Excel sheet



	B	C
1	Strain (in in^-1)	Stress (psi)
2	0	0
3	0.0116	12.34
4	0.0227	20.35
5	0.0339	27.28
6	0.0451	33.06
7	0.0557	38.29
8	0.0664	43.45
9	0.0773	47.86
10	0.088	52.12
11	0.0989	56.3

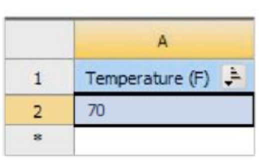


	B	C
1	Strain (in in^-1)	Stress (psi)
2	0	0
3	0.0056	17.94
4	0.0124	29.12
5	0.0192	38.22
6	0.0264	46.45
7	0.0337	53.81
8	0.041	60.37
9	0.0481	66.18
10	0.0555	71.76
11	0.0628	77.06
12	0.0702	81.97
13	0.0775	86.67
14	0.0851	91.21
15	0.0924	95.29
16	0.1	99.56
17	0.1078	103.8
18	0.1154	107.33
19	0.1231	110.94
20	0.1312	114.72

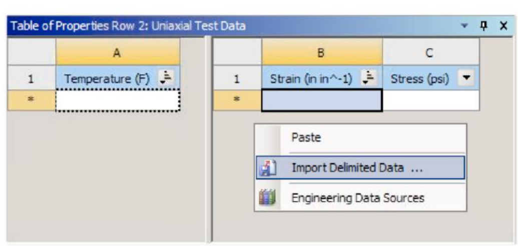


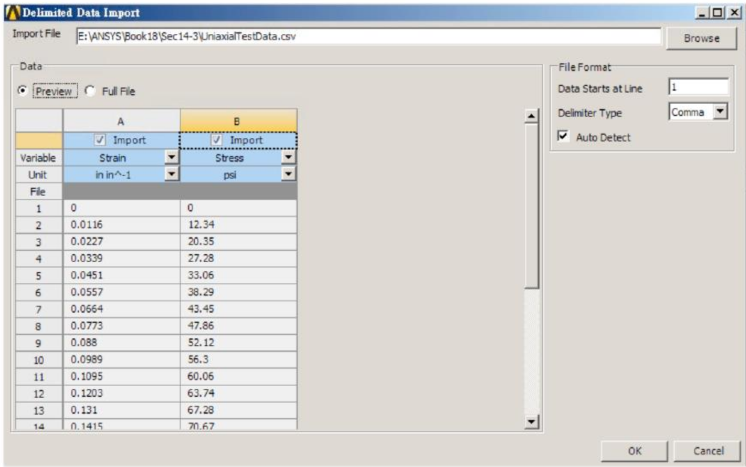
	B	C
1	Strain (in in^-1)	Stress (psi)
2	0	0
3	0.012	20.22
4	0.0252	33.06
5	0.0376	43.07
6	0.0499	51.64
7	0.0624	59.27

6. In Table of Properties, type 70 (for °F) for Temperature. If Table of Properties is not visible, turn on View/Table. Note that the data from Excel can also be imported.

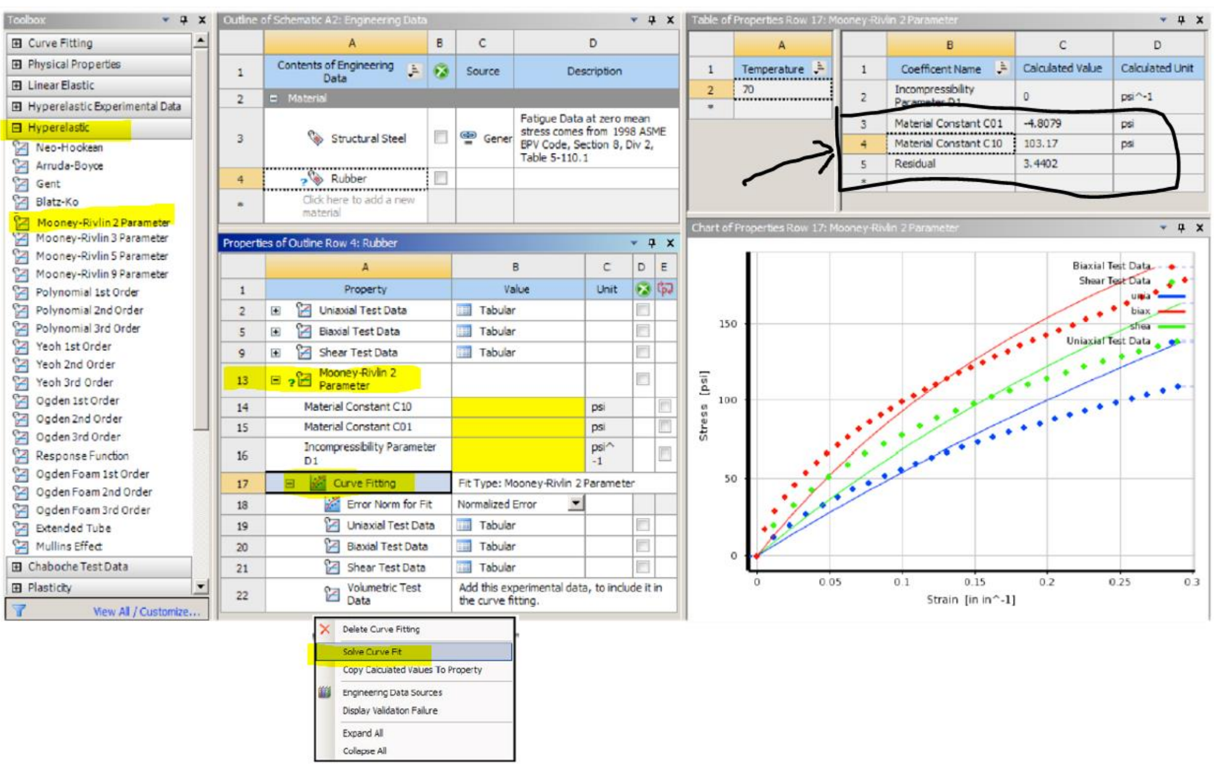


	A
1	Temperature (F)
2	70
*	



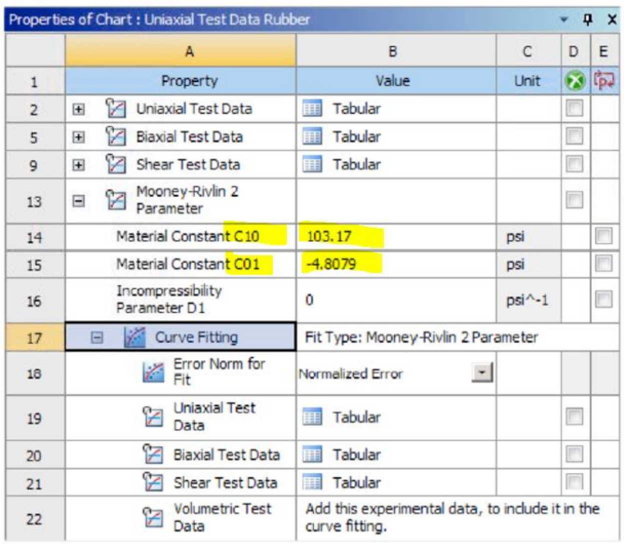


7. Expand Hyperelastic and double-click Mooney-Rivlin 2 Parameter. Then Expand the material model in the properties box. Right click **Curve Fitting** and select **Solve Curve Fit**.

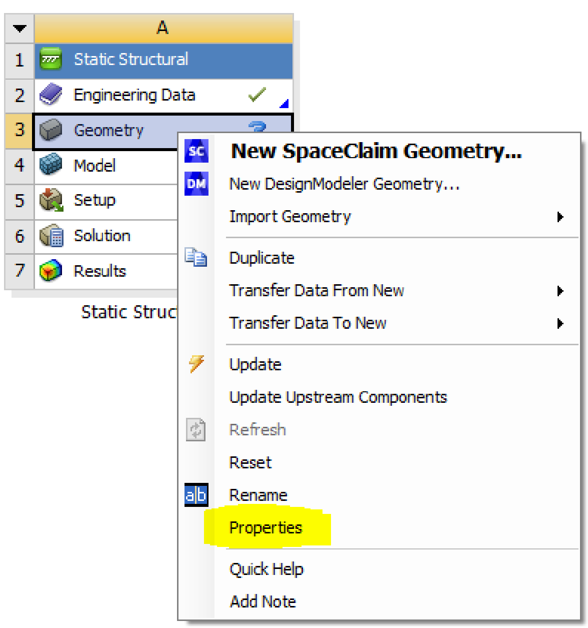


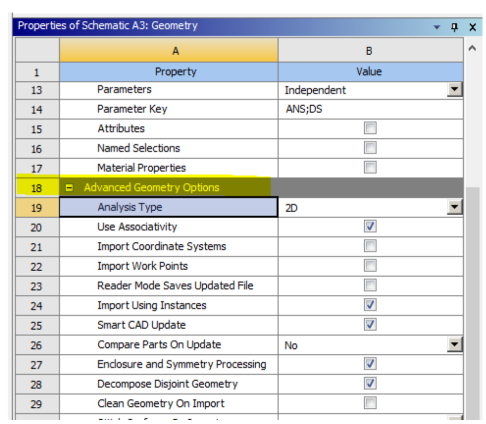
The calculated parameters, material constants C01 and C10, appear in the upper right corner above the plot. Review the constants. The fitted data and the test data are plotted for comparison.

8. Right click **Curve Fitting** again and select **Copy Calculated Values to Property** from the popup menu window. The material properties are formally entered in the rows above Curve fitting. The material parameters C10 and C01 could have been entered without any test data if you know already know the values for these parameters. For example, more common materials may have published values. However, these are usual not known and they must be calculated using the experimental data.



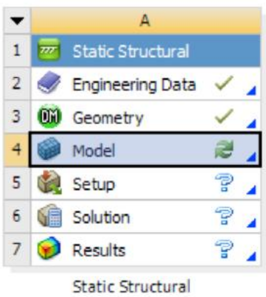
9. Return to the project schematic. Right click on Geometry and select Properties from the menu. In the pop-up menu scroll down to Advanced Geometry Options and select “2D” if not already selected in the Analysis Type row.

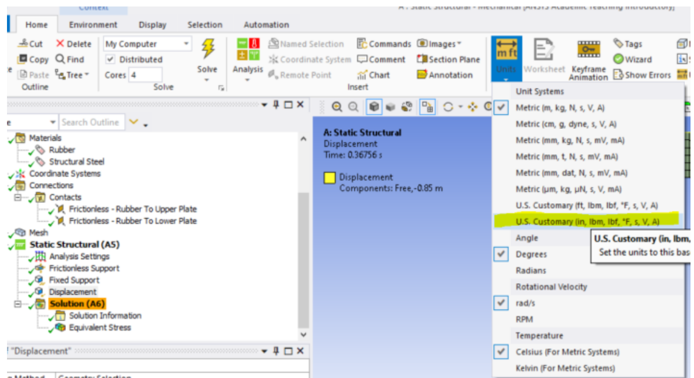




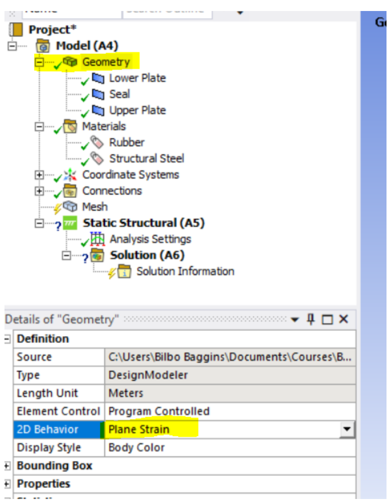
Set Up The simulation.

1. Start the mechanical setup by selecting Model from the schematic tree. Note the green check marks for Engeneering Data and Geometry before proceeding. When Mechanical opens, if prompted, select the units: in-lbm-lbf-s (inches, pound mass, pound force, seconds). Be patient, it will take a minute to open. You can verify that it is opening by looking at the alert in the lower left corner of the window.
   Verify the units.

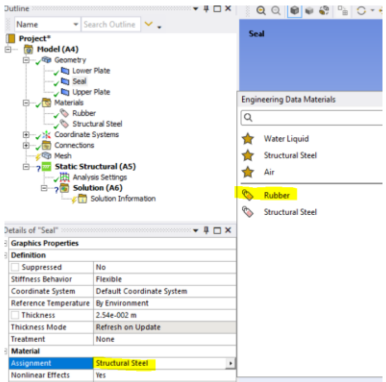




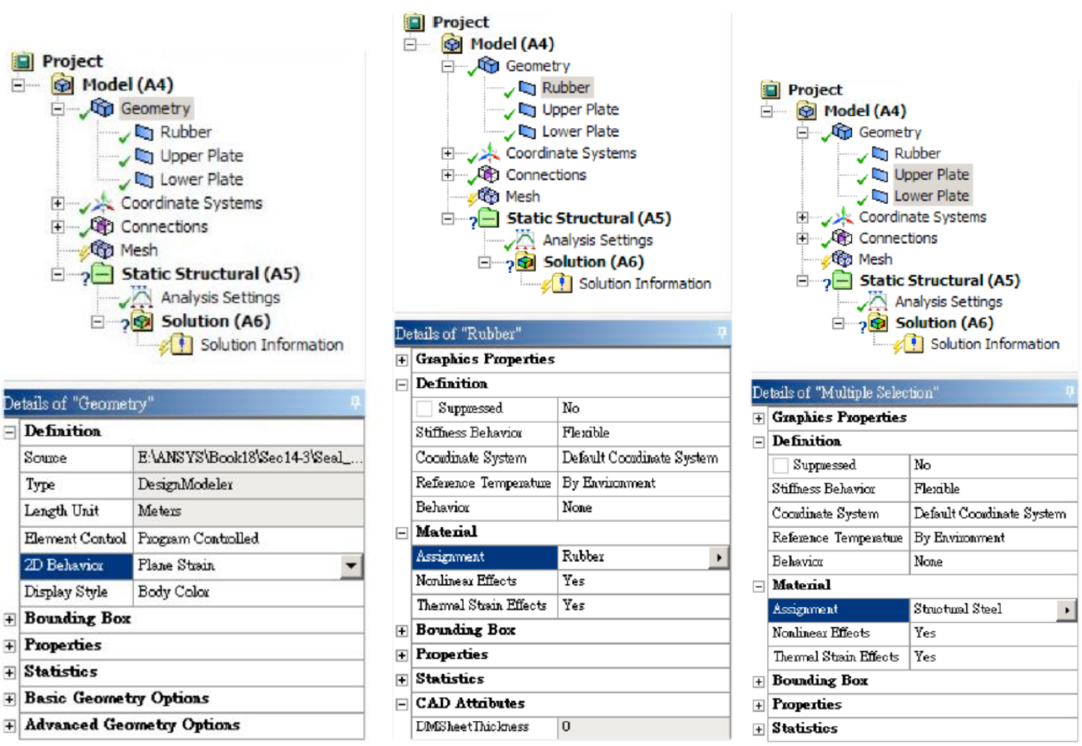
2. Select Geometry from the menu tree. For the 2D behavior select Plane Strain. (Do you recall discussing plane stress and plane strain in class?). Select 2D behavior in the left column and select Plane Strain in the pull-down menu. A 2D model is often used to model 3D geometry. This makes the setup and analysis much easier and requires far less computing time.



3. Select Rubber from the tree and change the assignment to Rubber.

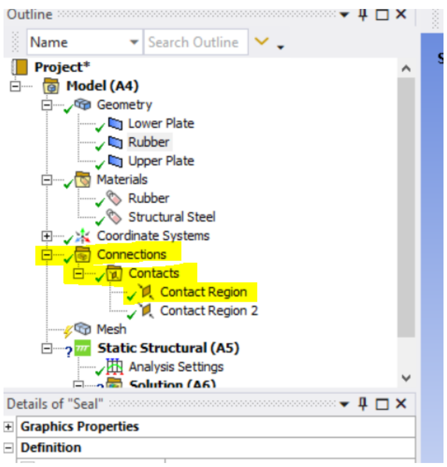


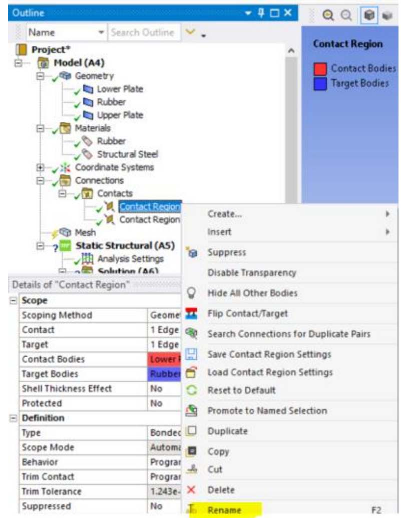
4. Hold the control key and select both the upper and lower plate and make sure the assignment is structural steel. Note that we did not set up the material properties for structural steel. We will use the default properties for structural steel in the built-in material library. If we wanted another material we could set up the properties and then select that material here in assignments.



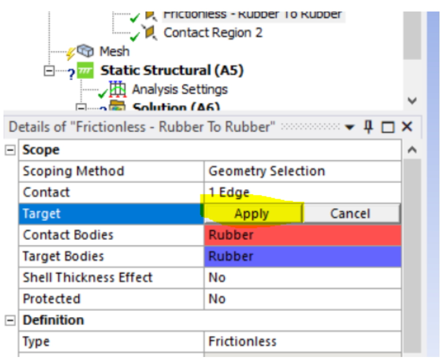
Next, we will set up the contact behavior between our seal and the upper and bottom plate. This is necessary since the seal is not connected (bonded or glued) to the plates.

5. Highlight Connections/Contacts/Contact Region. Right click the first contact region. In the pop-up menu, scroll down to the bottom and rename the contact to "Frictionless rubber to Top Plate". Rename the second contact to "Frictionless Rubber to Bottom Plate.

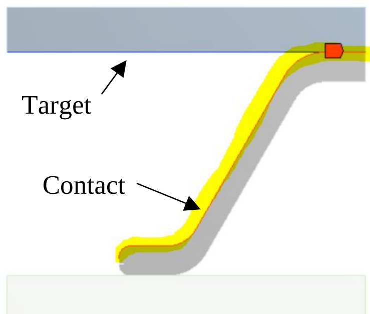




Select the contact edges. This is a total of 7 edges. Select the edges in red shown in the right figure below. Hold the control key while select. In Scope, select Contact, then "Apply". Repeat for the Target and select the line shown in blue. This is the bottom line of the upper plate in contact with the seal. Note that the red and blue boxes in the right column correspond to the contact and target bodies in the model window. In **Definition**, set the Type to **Frictionless** in the pull-down menu. For **Behavior** select **Asymmetric.**

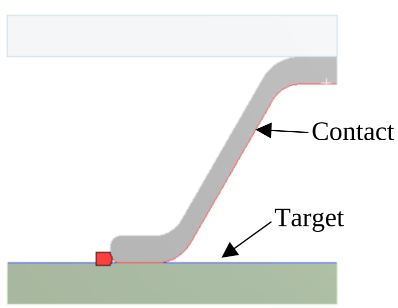


Details of "Frictionless - Rubber To Upper Plate"	
Scope	
Scoping Method	Geometry Selection
Contact	7 Edges
Target	1 Edge
Contact Bodies	Rubber
Target Bodies	Upper Plate
Shell Thickness Effect	No
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Asymmetric
Trim Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	None
Target Geometry Correction	None

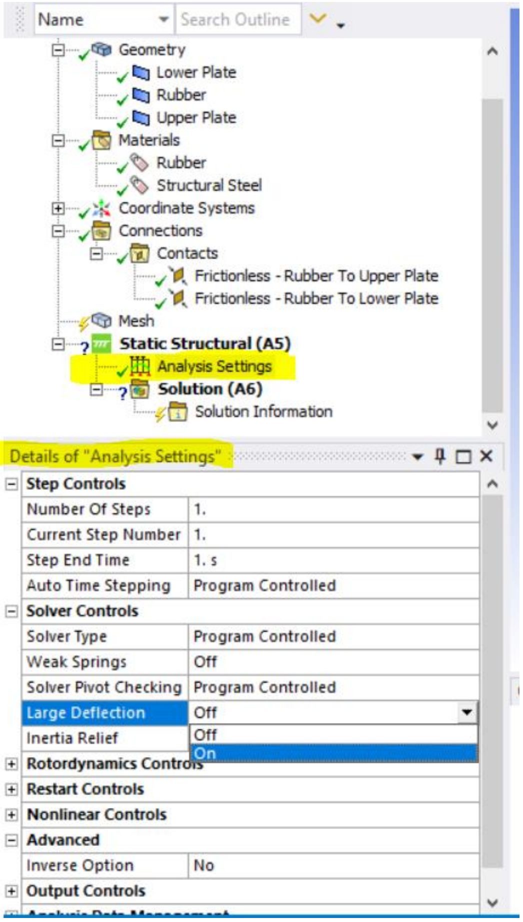


6. Repeat for the bottom plate. Select the appropriate edges as shown below. A total of 6 edges make up the Contact. Select the edges making up the red line in the figure below on right. Select Target and set the target. The target body is the upper line (in blue) of the lower plate.

Details of "Frictionless - Rubber To Lower Plate"	
Scope	
Scoping Method	Geometry Selection
Contact	6 Edges
Target	1 Edge
Contact Bodies	Rubber
Target Bodies	Lower Plate
Shell Thickness Effect	No
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Asymmetric
Trim Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	None
Target Geometry Correction	None

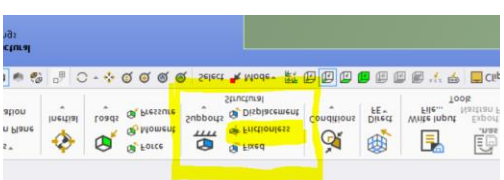


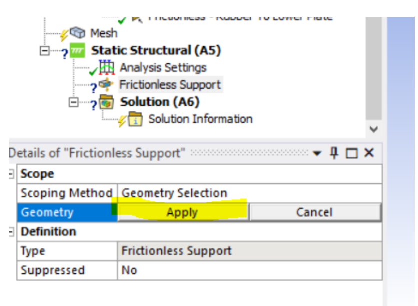
7. In Details of Analysis Settings, turn on Large Deflection

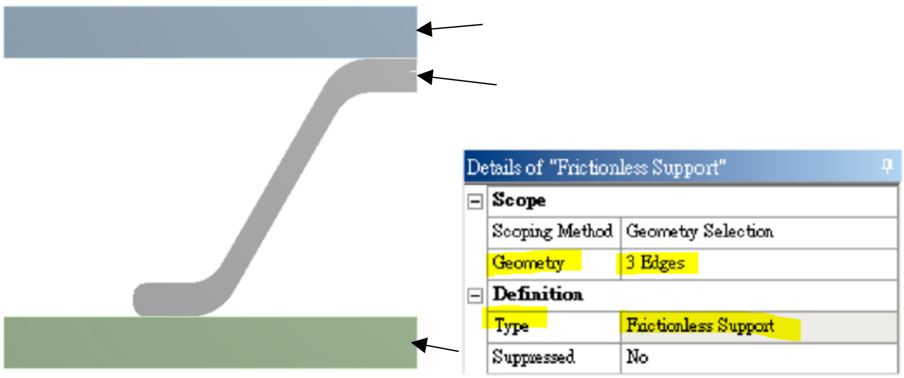


Next, we need to set up the supports.

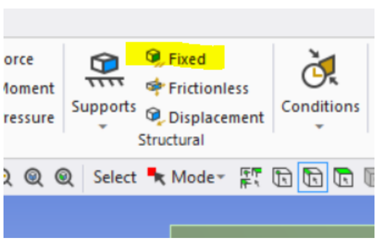
8. From the top menu bar, select Frictionless from the Supports window. Insert a Frictionless Support for the 3 edges at right. We are taking advantage of symmetry and modelling only half the seal since the left and right sides are identical. At the location of symmetry (the center) we set up a frictionless support. Select the 3 edges where the arrows point in the figure below.

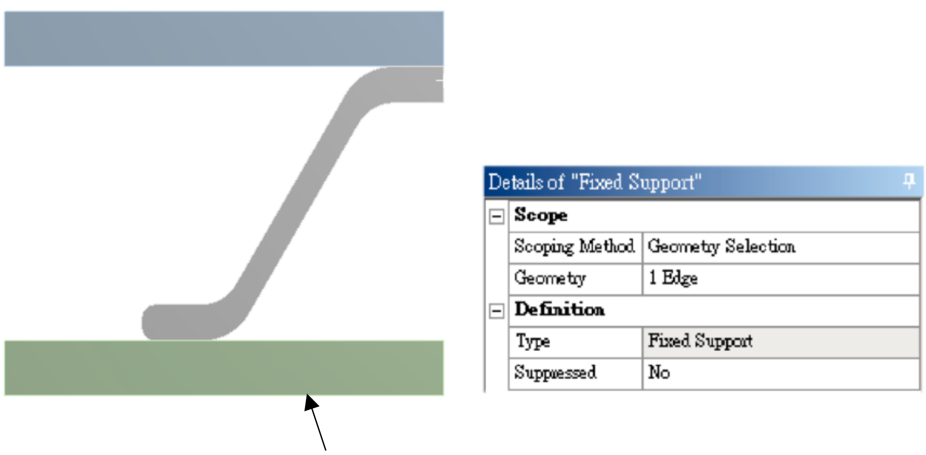




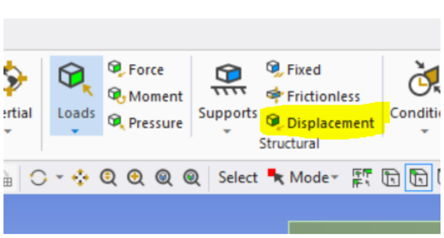


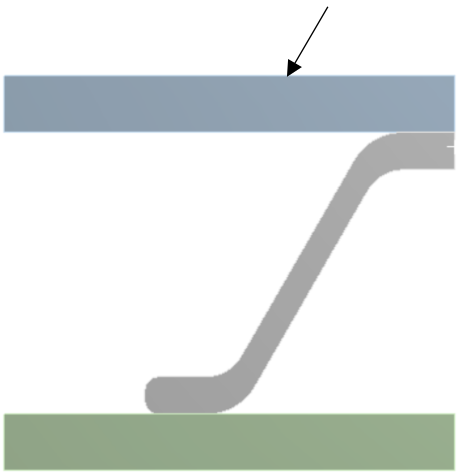
9. The bottom edge of the lower plate will be a fixed support. That is, this edge will not move and will constrain the model in the y-direction. In Scope select Geometry then select the bottle edge.





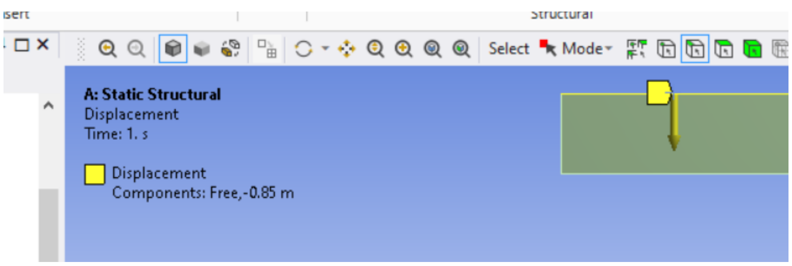
10. Finally, we will apply a displacement of 0.85 inches to the top edge. The displacement will be in the negative y-direction. Select the edge. Then select y-component in the left column and enter the value in the right column.





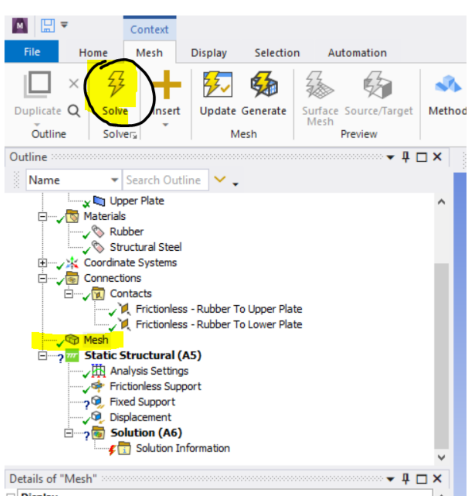
Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	-0.85 in (ramped)
Suppressed	No

Note the displacement is visible in the window



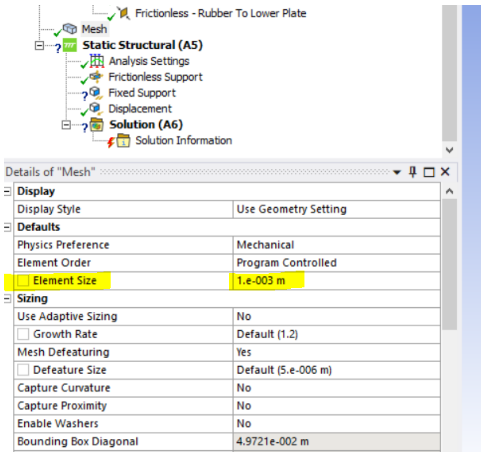
Mesh the geometry

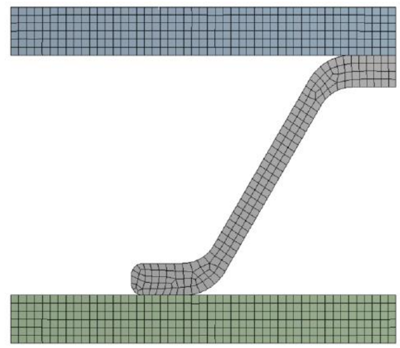
11. Select Mesh from the tree then select Solve from the top menu. Highlight Mesh from the model tree and in Details of Mesh in the lower left window select Relevance and enter 100 and in Relevance Center select Fine. Generate the mesh.



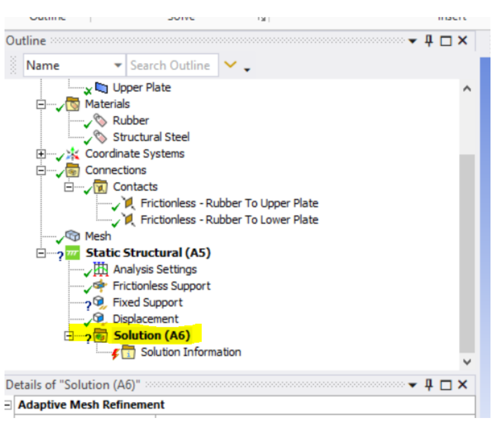
The default mesh settings produce a poor quality mesh. We will increase the number of elements by reducing the size of each element. Select Mesh again from the tree and in

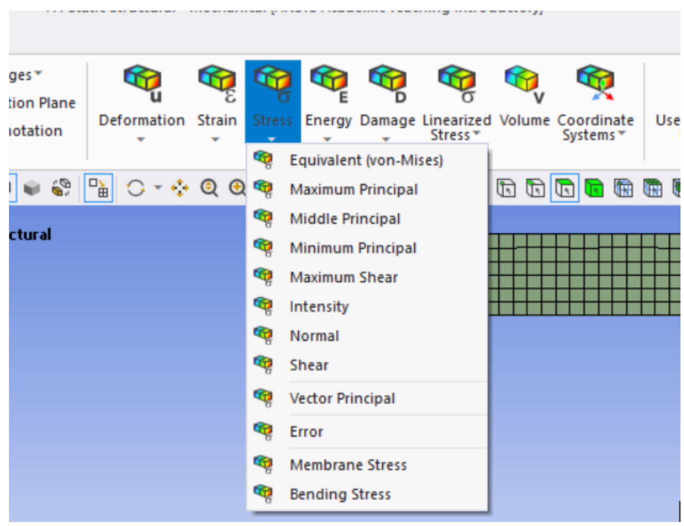
Details of mesh reduce the element size. Slightly reduce the element size and solve the mesh again. For example, change 0.0015 to 0.001. Reduce the element size so that you end up with a mesh similar to the figure on the right.



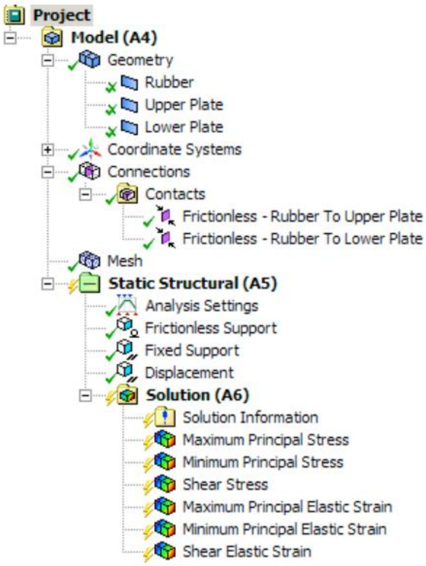


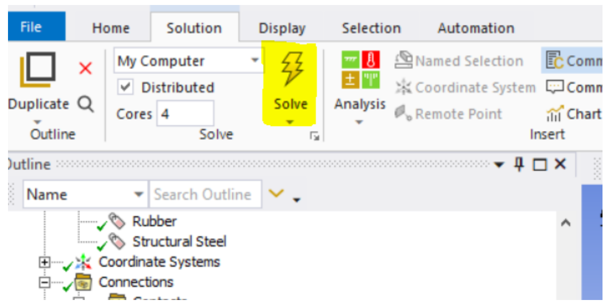
With the setup complete we are ready to solve for the solution. Select Solution from the Tree.



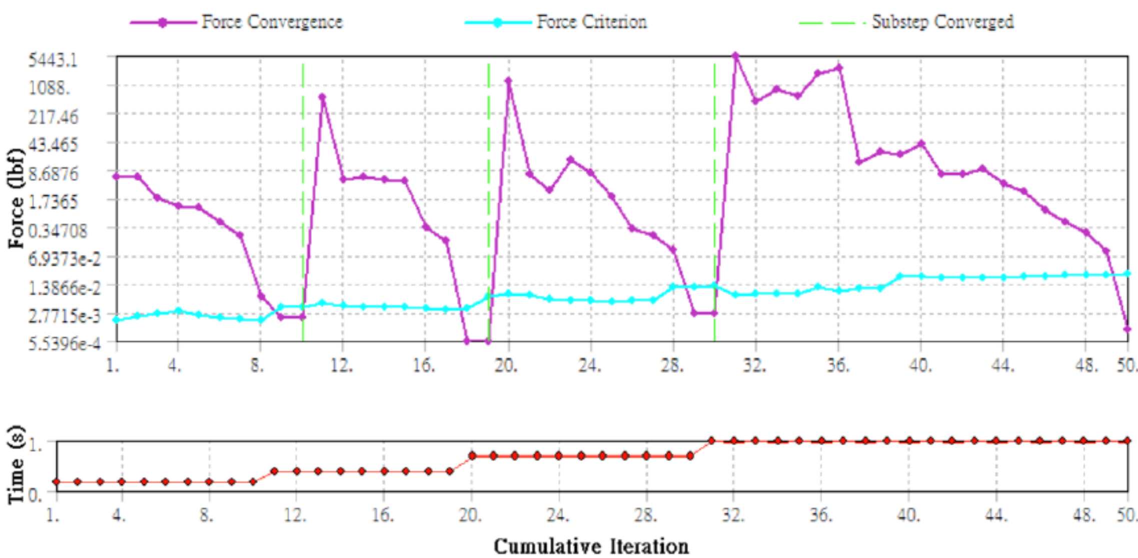


From the top menu bar select the following results: Maximum Principle Stress, Minimum Principle Stress, Shear Stress, Maximum Principle Elastic Strain, Minimum Principle Elastic Strain, and Shear Elastic Strain. The results will then be listed in the Solution tree. Finally, solve the analysis.





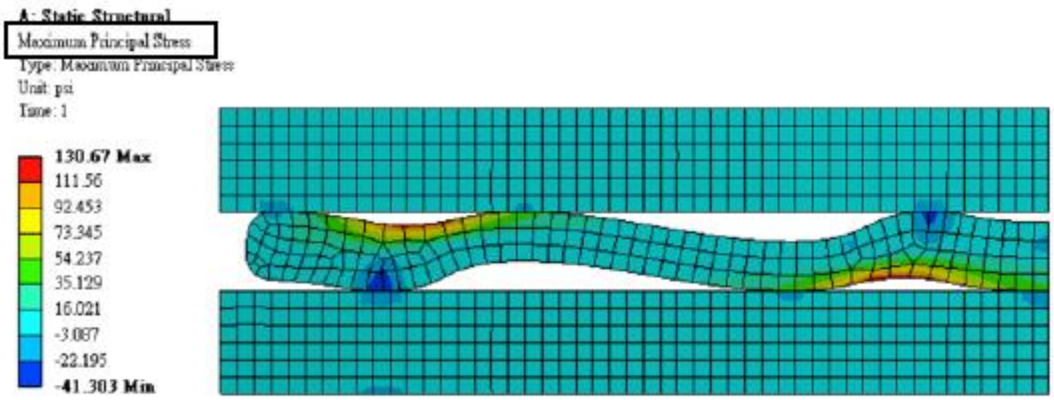
The convergence window will appear. Monitor the solution while it solves.



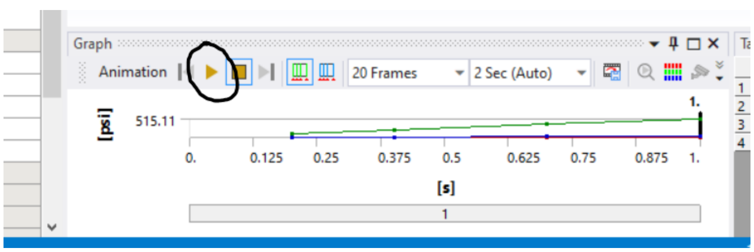
View The Results

Select the result object from the tree.

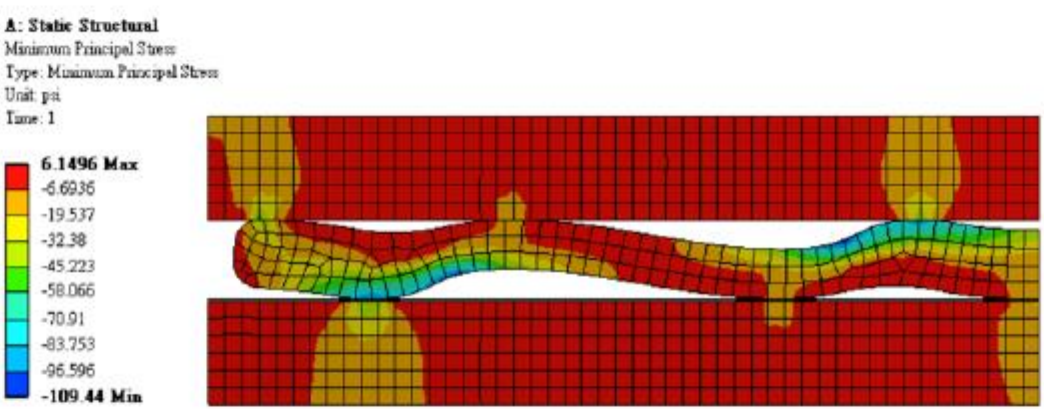
1. Maximum Principle Stress: The Maximum tensile stress is shown in red in the contour map in the figure below.



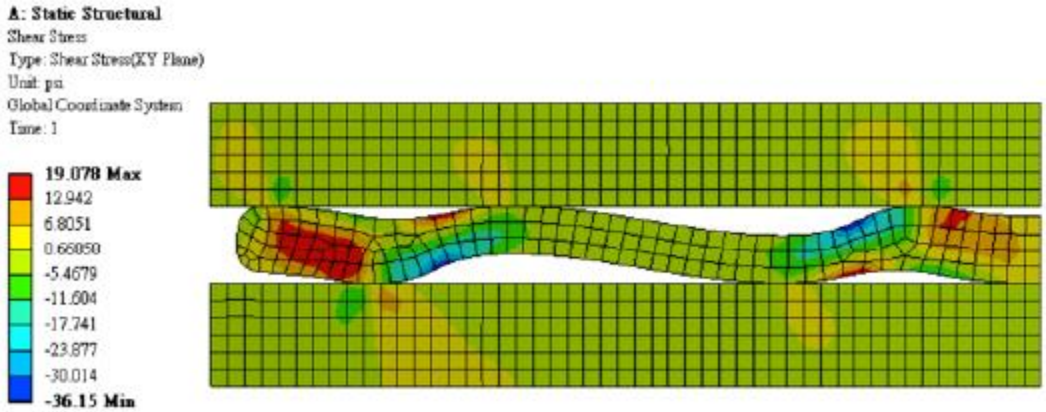
The results can be animated by selecting the Play button in Animation.



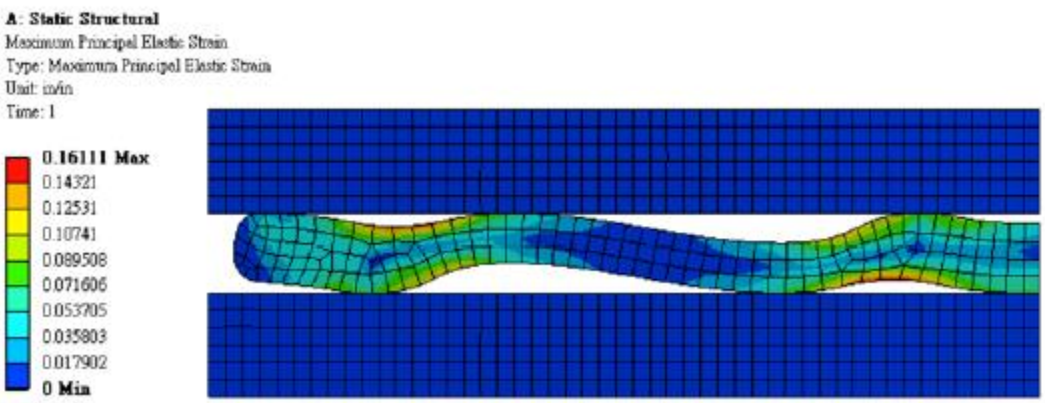
2. Minimum Principle Stress: The maximum compressive stress is shown in blue in the figure below.



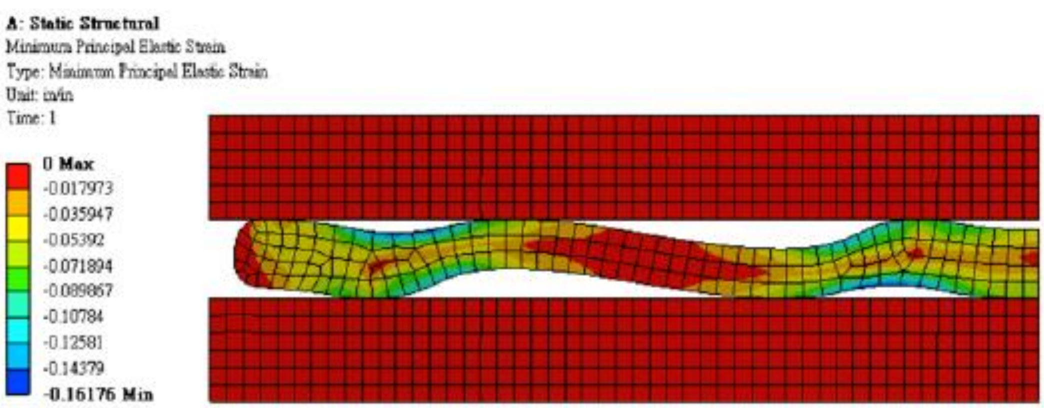
3. Maximum Shear Stress: Shown in blue.



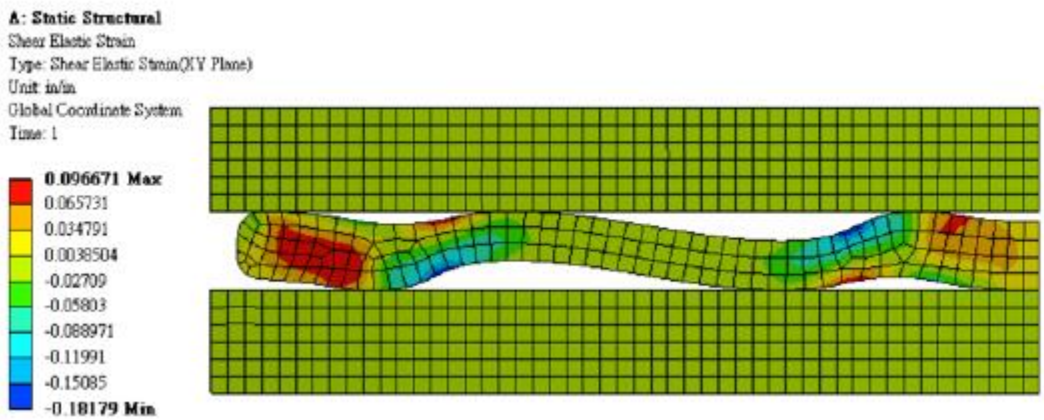
4. Maximum Principle Elastic Strain: Maximum Tensile Strain shown in red.



5. Minimum Principle Elastic Strain: The maximum compressive strain shown in blue.

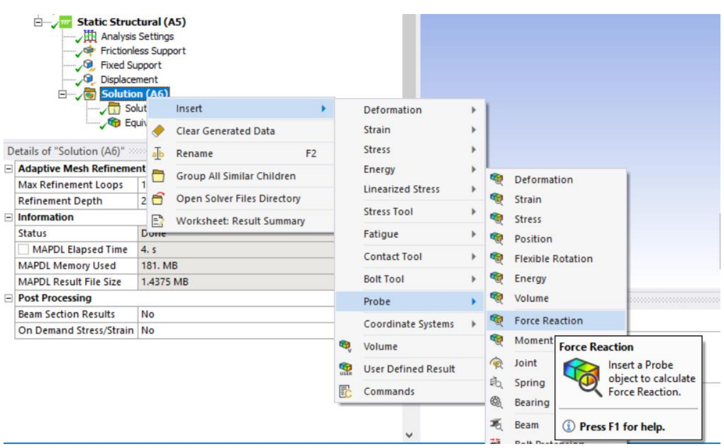


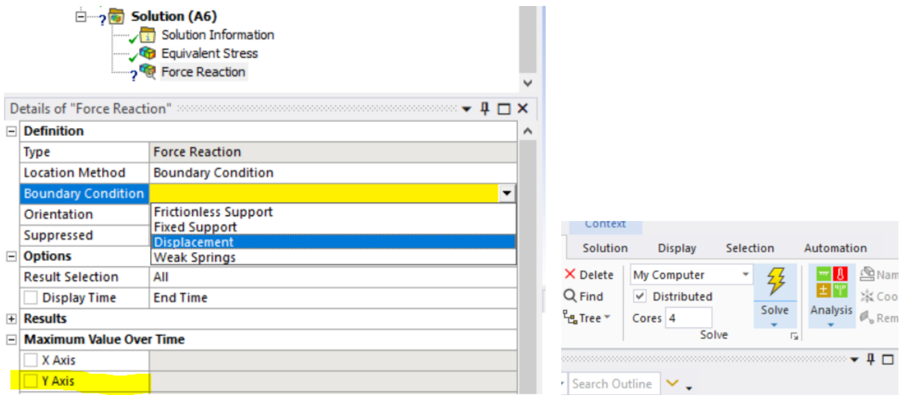
6. Shear Elastic Strain: The maximum shear strain shown in blue.



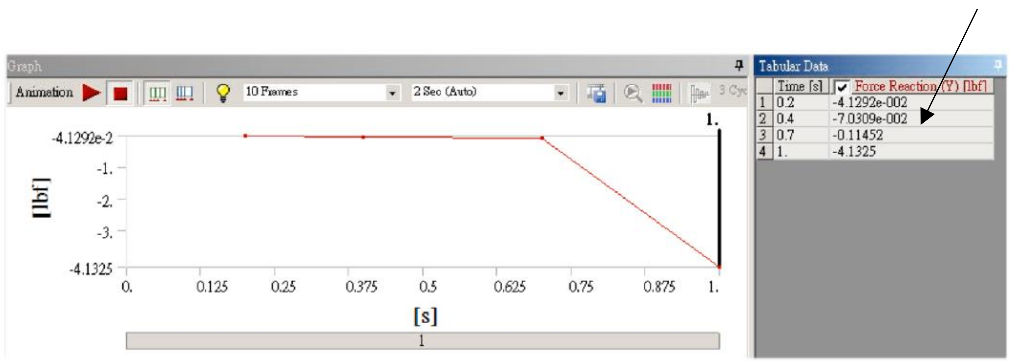
7. Animate the results. Select play animation.

8. Right click on **Solution** in the tree. Insert a **Probe/Force Reaction**. In Details of Force Reaction Change the boundary condition to Displacement and select Y-Axis. Then solve.



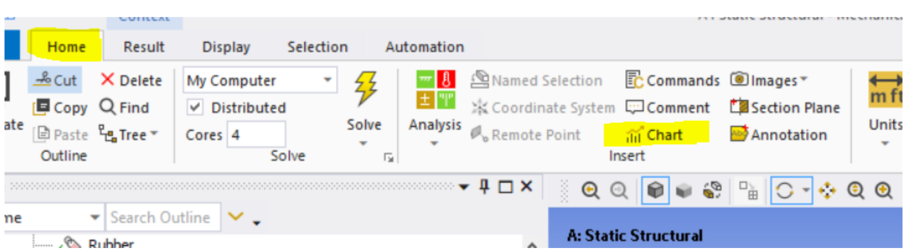


Note that for a plane strain problem, Workbench assumes a unit depth (in this case, one inch). Therefore the force unit should be read as lbf/in (pound force per inch). For example, if the dept was 2 inches, multiply the values in the Tabular Data on the right times 2 to get the actual value.

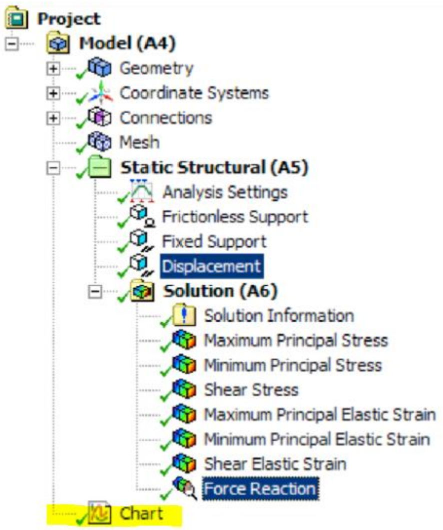


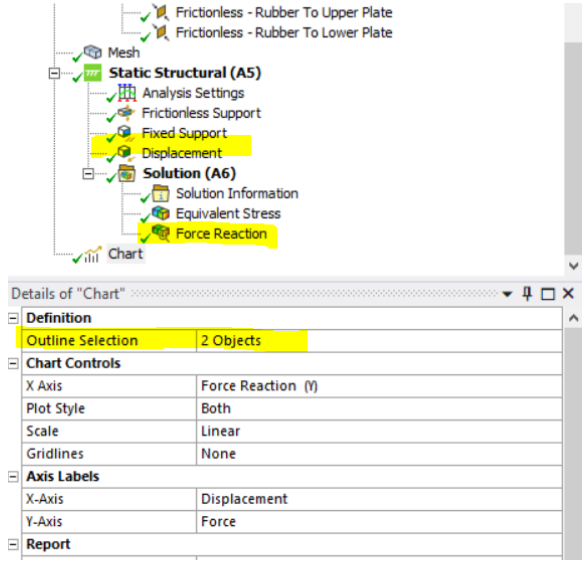
Create a Force vs Displacement Chart

1. With the Home Tab selected, select the Chart button. Select New Chart and Table icon on the toolbar. Note that Chart is added to the project tree at bottom of the list.



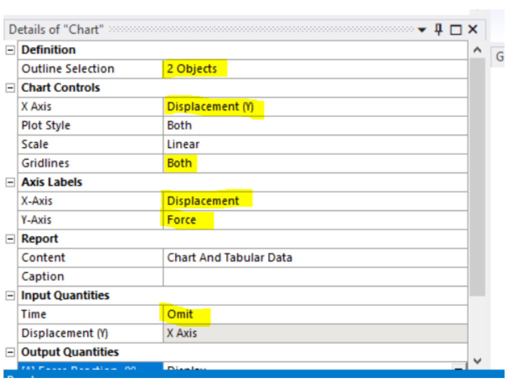
2. Click on Chart in the project tree. Set Details of Chart as shown below.





Definition	
Outline Selection	2 Objects
Chart Controls	
X Axis	Force Reaction (Y)
Plot Style	Both
Scale	Linear
Gridlines	None
Axis Labels	
X-Axis	Displacement
Y-Axis	Force
Report	

Holding the control key, select Displacement from Static Structural and Force Reaction from Solution. Then next to Outline Selection select Apply. The left column will change to “2 Objects”. Set the highlighted boxes below as follows:



Definition	
Outline Selection	2 Objects
Chart Controls	
X Axis	Displacement (Y)
Plot Style	Both
Scale	Linear
Gridlines	Both
Axis Labels	
X-Axis	Displacement
Y-Axis	Force
Report	
Content	Chart And Tabular Data
Caption	
Input Quantities	
Time	Omit
Displacement (Y)	X Axis
Output Quantities	

A force-versus-displacement curve is generated. Note that as before the force unit is in lbf/in for a 2D problem.