

MEE 323 – Computer Aided Engineering II
Homework Assignment #3 – 2D Simulations

Instructions:

- Use this Word file as a template for your homework report. Add screenshots of your modeling/analysis or any required explanations below the appropriate question. Turn in the homework report (converted to PDF) on the course Gradescope before the deadline.
- Upload a copy of your ANSYS files to the appropriate assignment on the course Canvas. The uploaded ANSYS files may be used to check your work and/or ensure academic integrity.

Homework Objectives:

Learn to create and validate 2-D surface models in ANSYS Workbench. Learn how to apply symmetry boundary conditions to reduce model size and analysis time as well as cost.

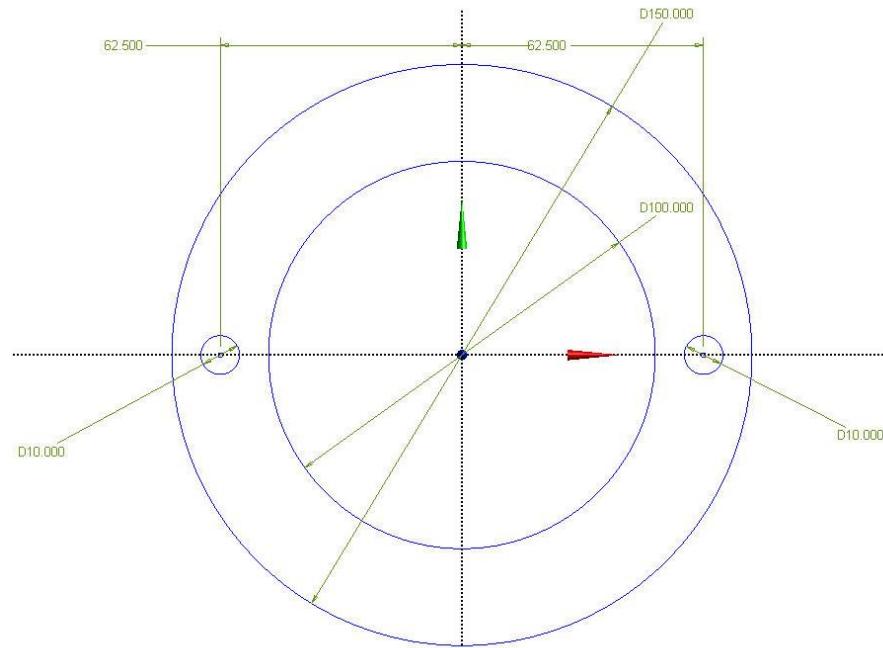
Reading Assignment:

Read **Section 3.3** of the textbook – it contains details of the 2-D plane elasticity assumptions and operational details of ANSYS Mechanical. Read and go through the tutorials of **Section 3.2** for another axisymmetric example and **Section 3.4** for another plane stress example.

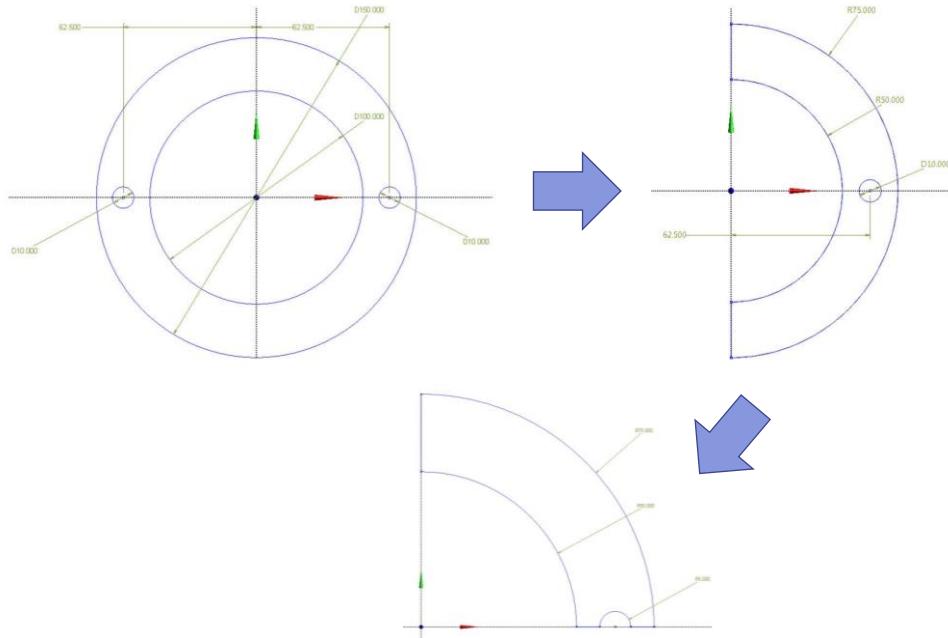
ANSYS Exercises:

Question 1:

Determine the magnitude and location of the maximum principal stress and the magnitude and location of the maximum displacement of the structural steel part shown below (all dimensions given in mm). Model the part as a 2D structure with appropriate behavior. **Use symmetry to simplify the model to a one-quarter section.**



Use the suggested symmetry simplification below (use frictionless supports if/where appropriate):



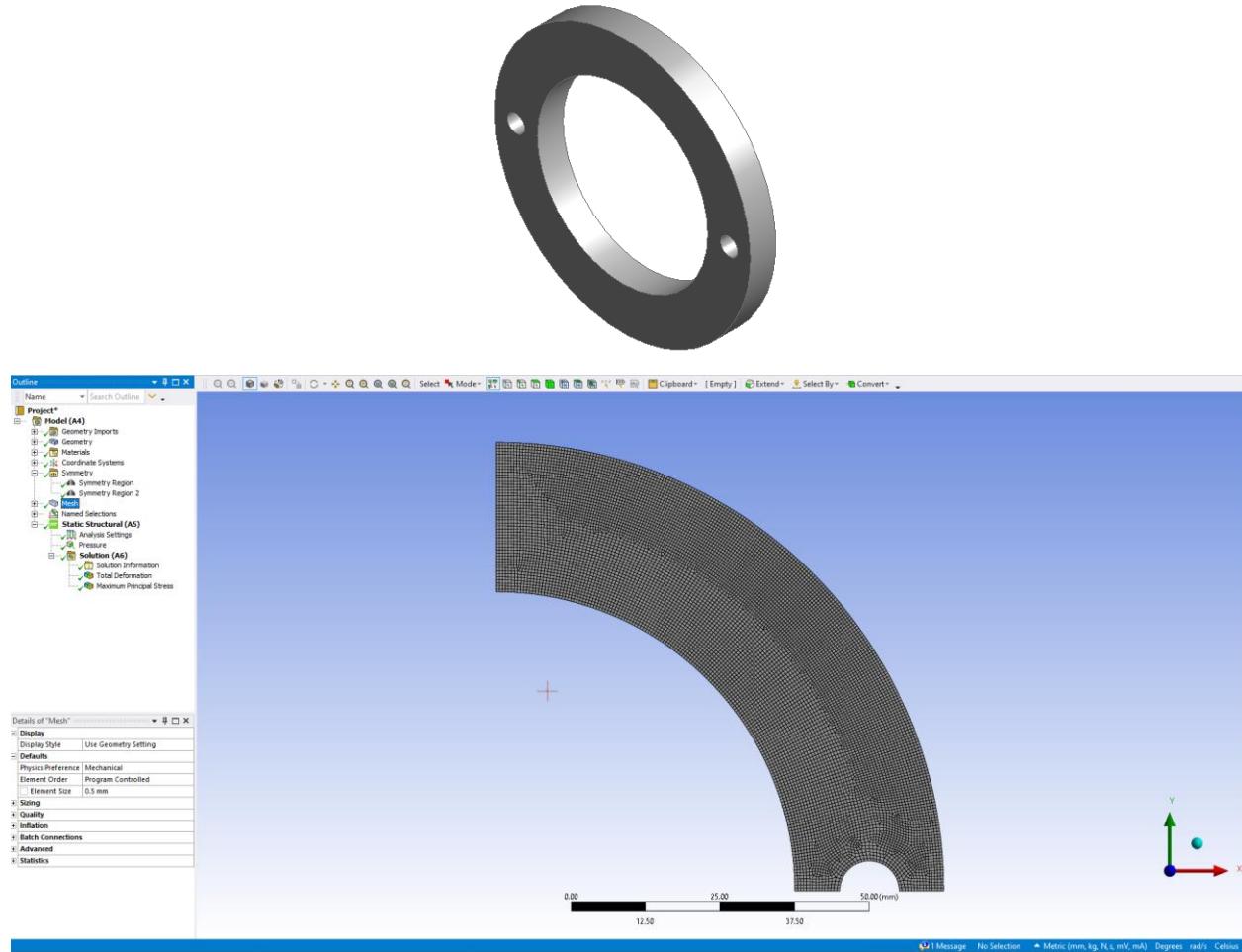
Analyze the part under the three cases described below. In all three cases, there is a pressure of 50 MPa acting on the inside walls/surfaces of the part. (**Figures shown below are just for your reference and are cut to show features. Please model all parts as 2D surface models with the appropriate 2D behavior selected. Use frictionless supports if necessary.**)

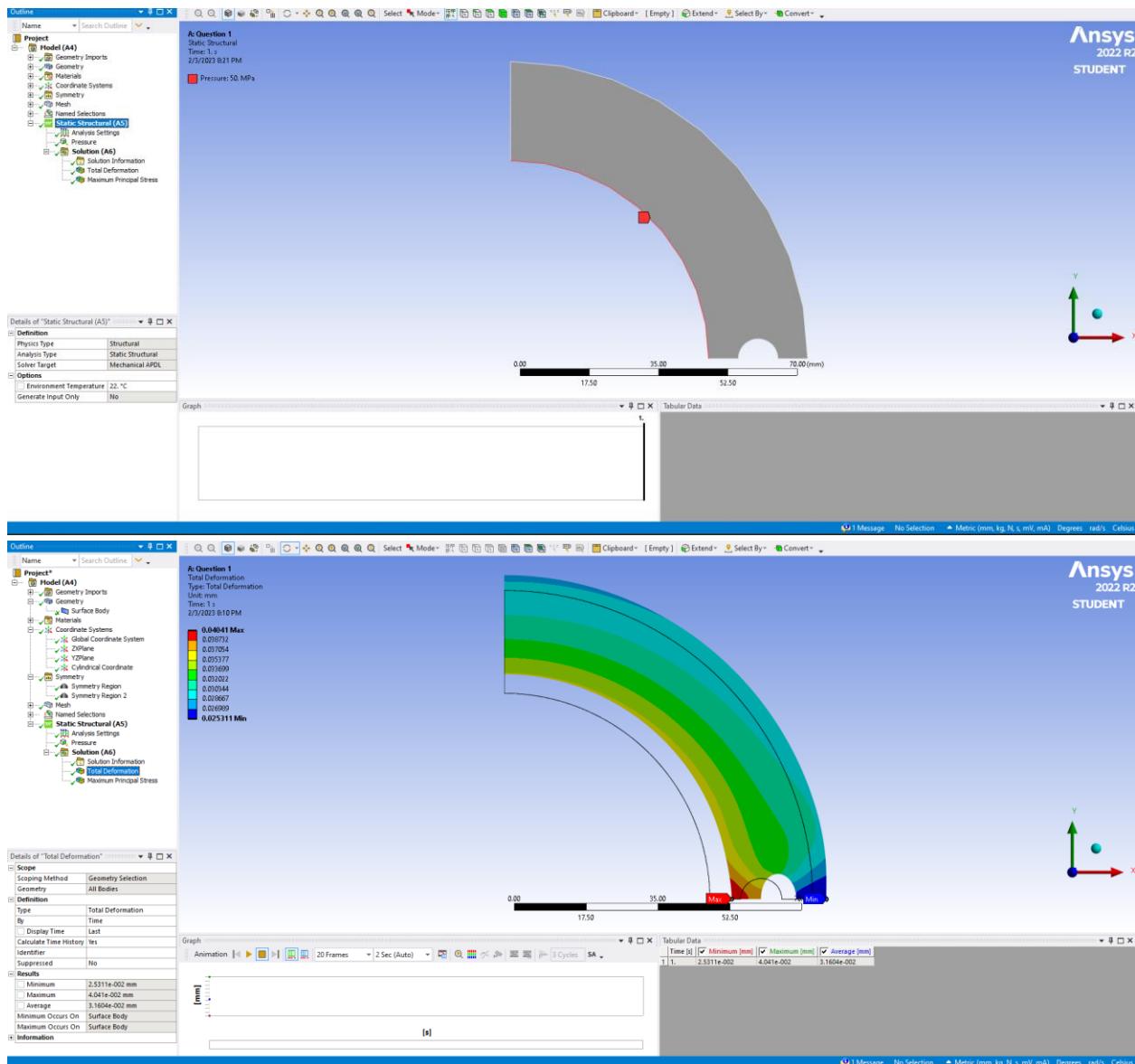
For each case,

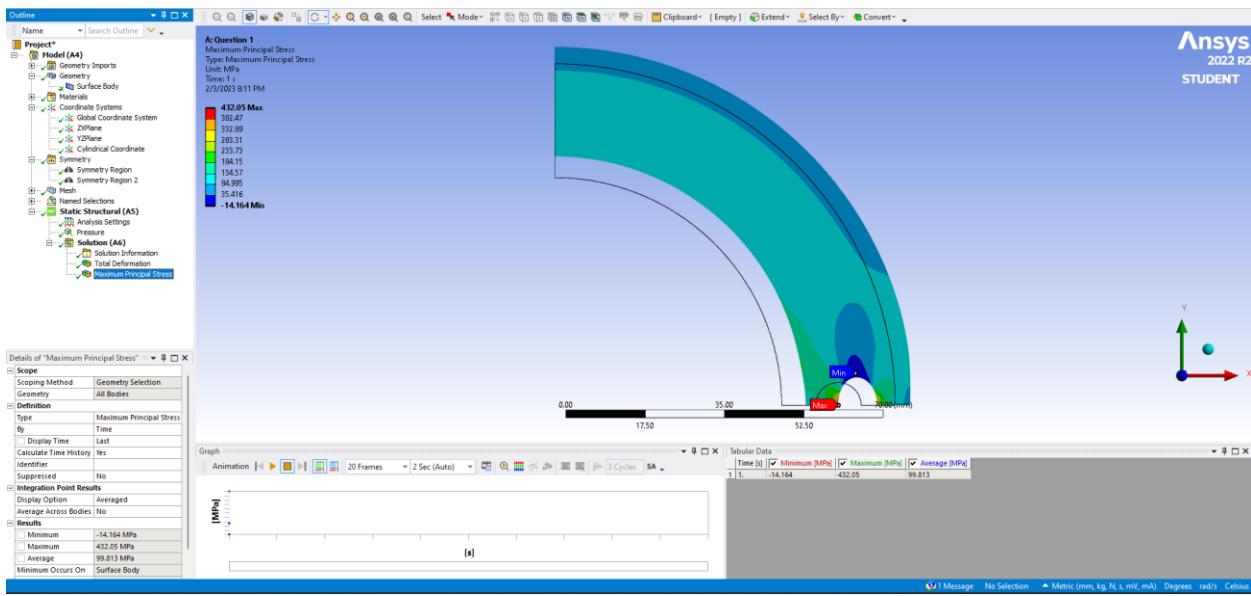
- (a) Mesh the structure with an appropriate mesh size such that the small 10 mm diameter hole would have 30-60 elements around it (this is a good rule of thumb for meshes around circular features). Show a figure of the mesh.
- (b) Show a figure of the applied loading.
- (c) Show figures of the total displacement and maximum principal stress and show the location of the maximum displacement and stress on these plots (using the "Max" probe).

Case (i): The part is a thin washer with a thickness of 15mm. What type of 2D elasticity behavior approximates this structure most appropriately?

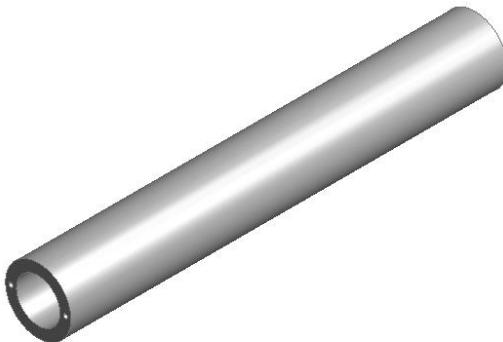
The plain stress.

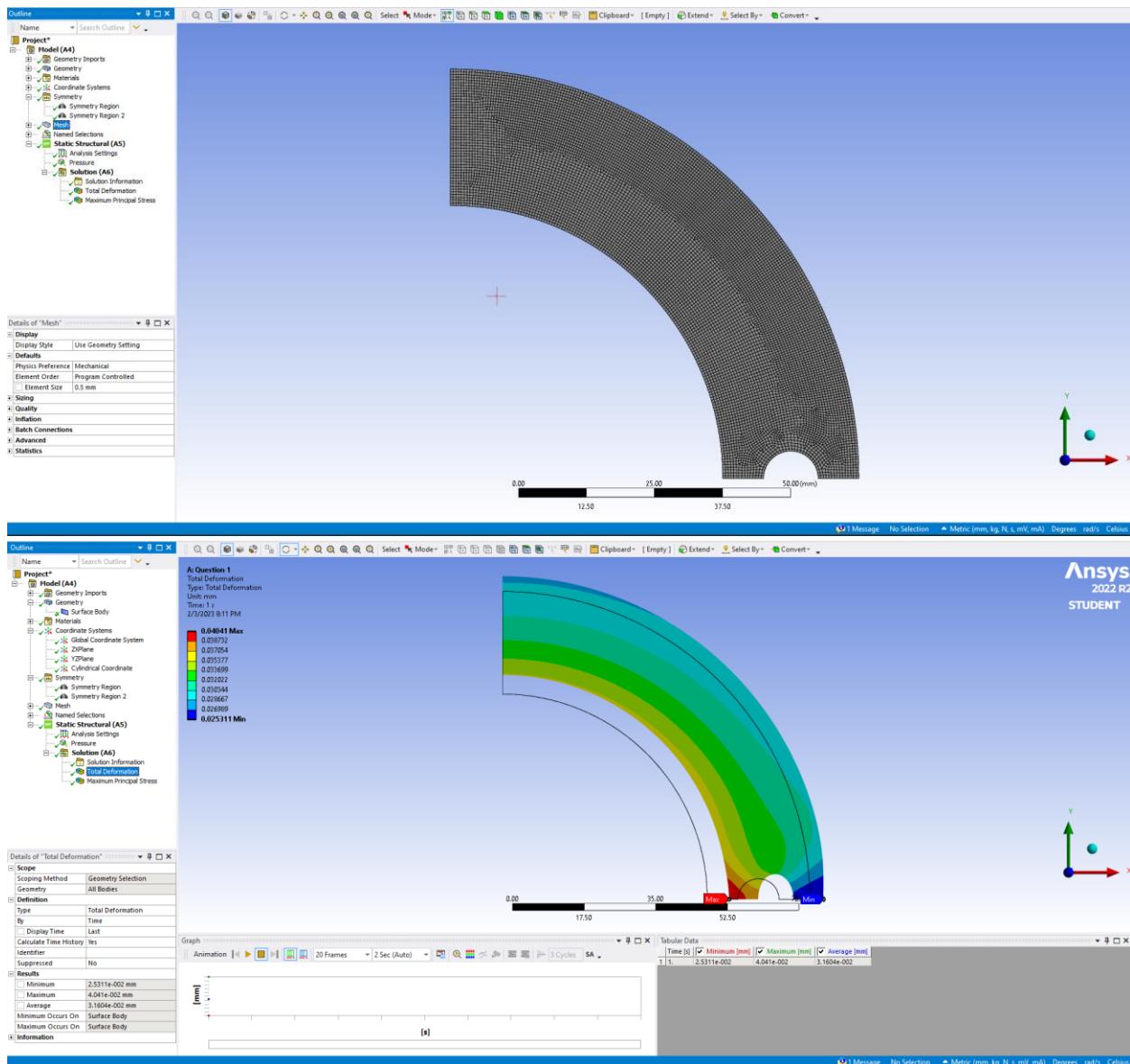


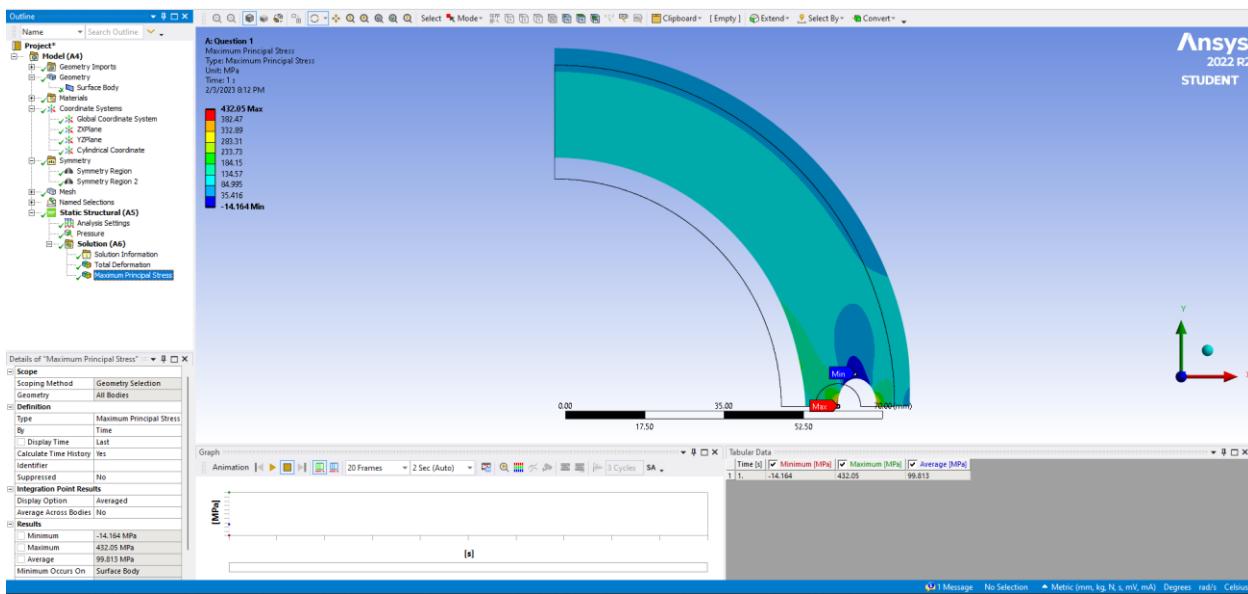




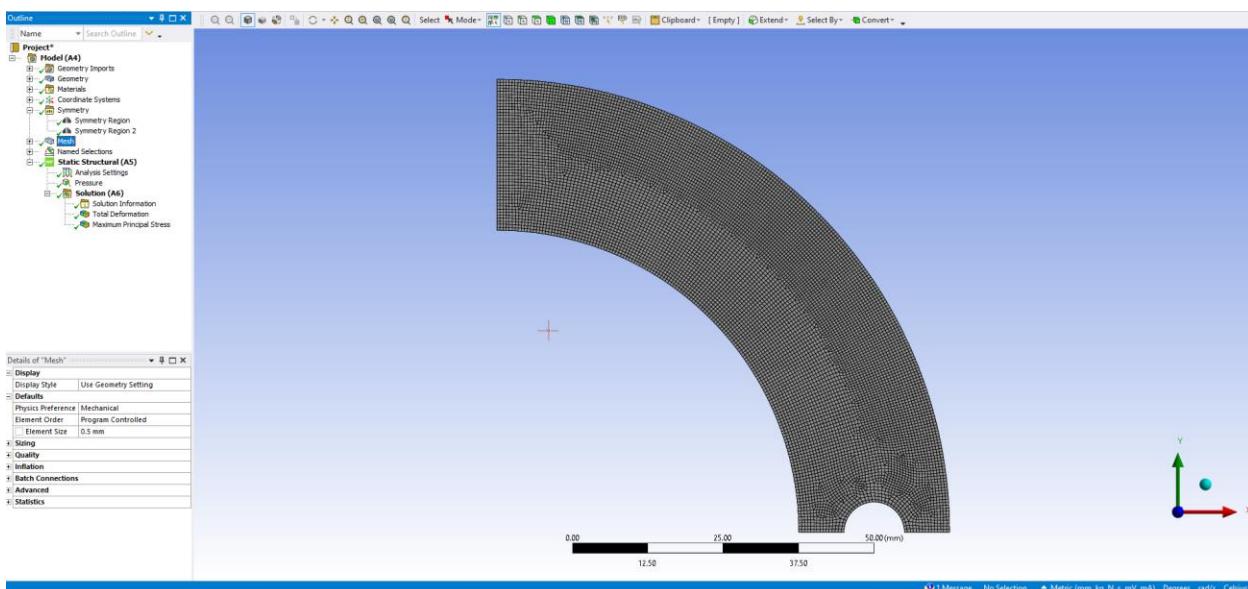
Case (ii): The part is a long underground pipeline, whose length is unknown. However, the loading on the structure (the internal pressure) is the same along the length of the pipeline. What type of 2D elasticity behavior approximates this structure most appropriately? **Plain Strain**

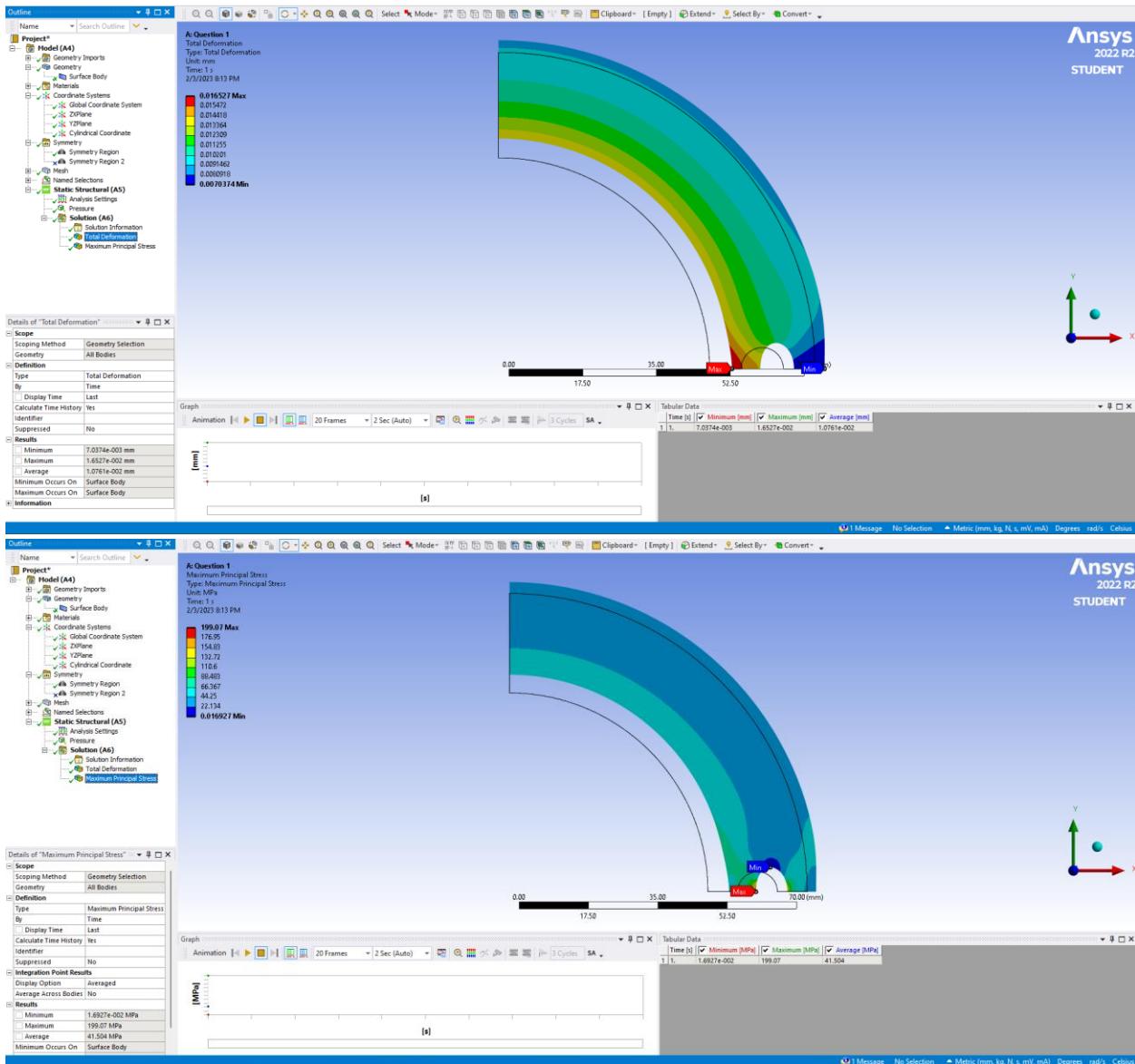






Case (iii): The part is a sphere with an internal cavity. What type of 2D elasticity behavior approximates this structure most appropriately? **Axisymmetric**

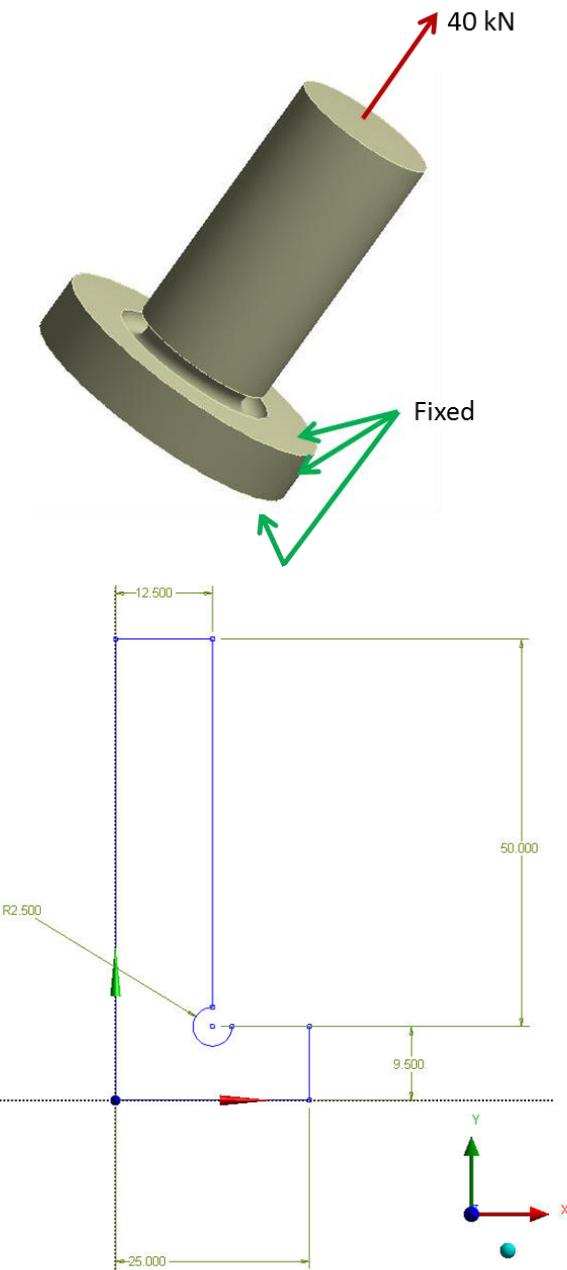




Finally, compare the maximum deformation and stress values for the three cases and comment on their comparative magnitudes. Why is the stress in the last case much smaller than the stress in the previous two cases? **The stress is distributed more because it is all around instead of just the tube or just the face.**

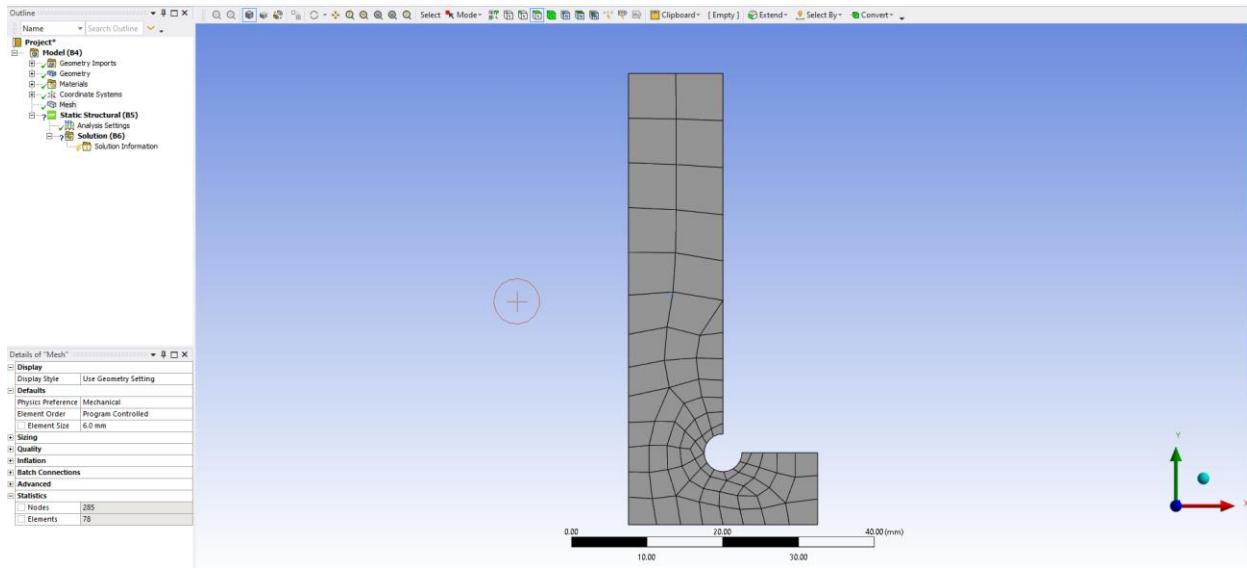
Question 2:

A structural steel anchor device is loaded by an axial force as shown below. All of the lower surfaces (excluding the groove) are fixed as indicated. Note that the center of the 5 mm diameter groove is aligned with the outer surfaces of the anchor as shown in the sketch. All dimensions are given in mm.

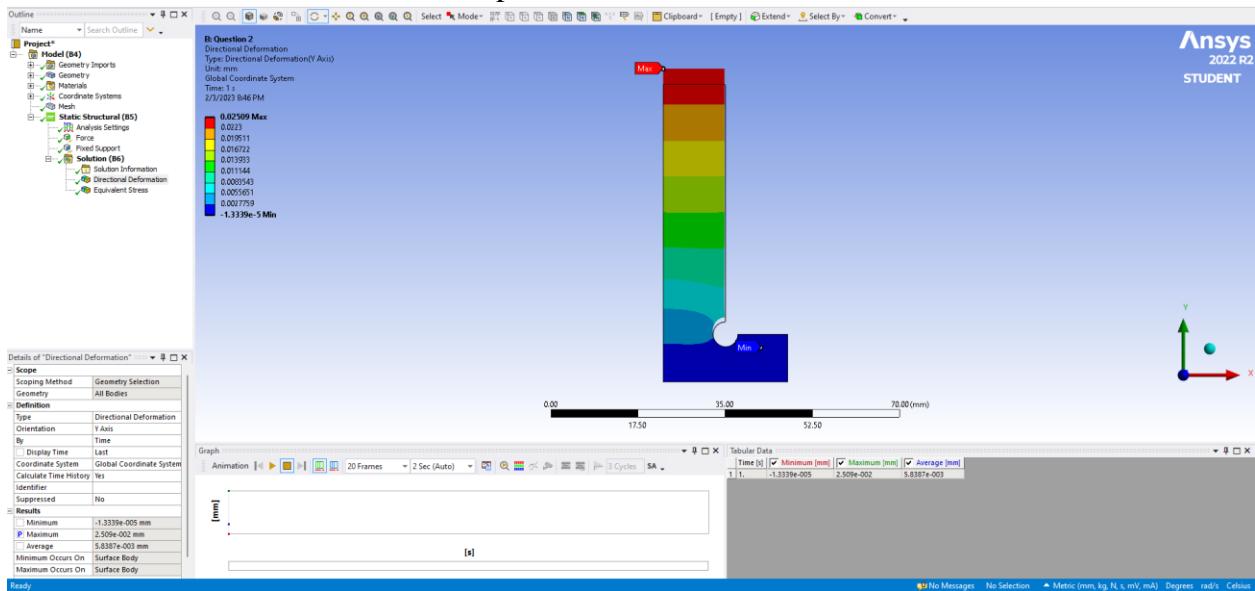


Make the model by sketching the cross section given on the X-Y plane as shown. Create a surface body and choose the appropriate 2D elasticity assumption for the simulation (i.e., plane stress, plane strain, or axisymmetry). You may choose to work from the geometry you created for the Sketching & Modeling homework for this.

- (a) Mesh the model using a mesh size of 6 mm. Show a figure of the mesh and list the number of nodes and elements in the mesh.

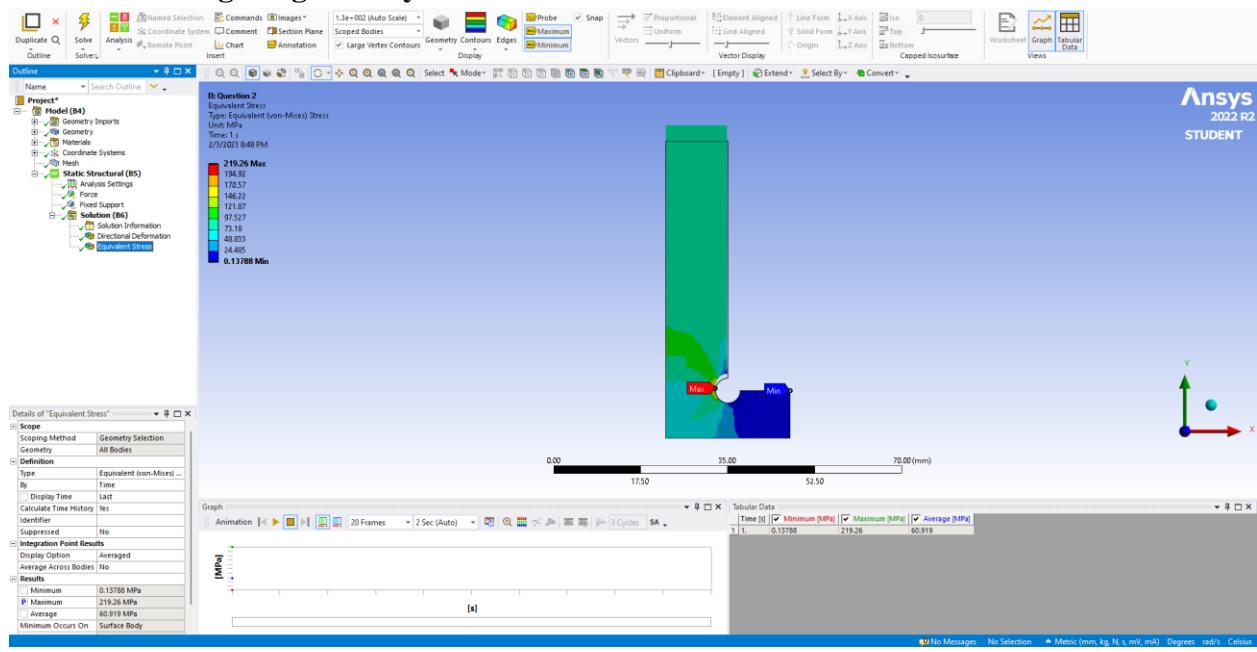


- (b) Apply the loads and boundary conditions. Find the deformation of the structure along the direction of the loading. Show a figure of the deformation with the undeformed edge with the maximum and minimum deformation probes selected.



- (c) Find the equivalent (von Mises) stress and show a figure of the stress distribution. Add a probe to show the location of the maximum von Mises stress. Explain in a sentence why the stress

occurs at this location. **The stress occur in this location because of the stress concentration due to the change of geometry.**



- (d) Repeat the analysis for mesh sizes of 4 mm and 2 mm. Tabulate the values of maximum deformation and maximum equivalent stress vs. the mesh size (including data from the original 6 mm mesh). Plot a graph of the mesh size vs. maximum deformation and another graph of mesh size vs. maximum equivalent stress. What do you observe? Calculate a percentage difference of the maximum deformation and stress between the 6 mm and 2 mm mesh sizes. What do you observe? **The percentage difference for maximum deformation is 0 percent and 2 percent for stress. We observe this because as we decrease the mesh sizes the values will be more accurate.**

Toolbox

Outline of All Parameters

A	B	C	D
ID	Parameter Name	Value	Unit
1			
2	Input Parameters		
3	3 Static Structural (B1)		
4	P1 Mesh Element Size	6	mm
*	New input parameter	New name	New expression
6	Output Parameters		
7	7 Static Structural (B1)		
8	P2 Mesh Nodes	285	
9	P3 Mesh Elements	78	
10	P4 Directional Deformation Maximum	0.02509	mm
11	P5 Equivalent Stress Maximum	219.26	Mpa
*	New output parameter	New name	New expression
13	Charts		
14	Parameters Chart		

Table of Design Points

A	B	C	D	E		
Name	P1 - Mesh Element Size	P2 - Mesh Nodes	P3 - Mesh Elements	P4 - Directional Deformation Maximum	P5 - Equivalent Stress Maximum	
Units	mm					
3	DP 0 (Current)	6	285	78	0.02509	219.26
4	DP 1	4	350	97	0.02509	240.8
5	DP 2	2	795	234	0.02509	223.73

Properties of Outline A (4): 0

A	B
Property	Value
2	Parameter Chart: General
3	Exclude Current Design point
4	X-Axis (Bottom) P1 - Mesh Element Size
5	X-Axis (Top)
6	Y-Axis (Left) P4 - Directional Deformation Maximum
7	Y-Axis (Right)

Parameter Chart 0

Table of Design Points

A	B	C	D	E		
Name	P1 - Mesh Element Size	P2 - Mesh Nodes	P3 - Mesh Elements	P4 - Directional Deformation Maximum	P5 - Equivalent Stress Maximum	
Units	mm					
3	DP 0 (Current)	6	285	78	0.02509	219.26
4	DP 1	4	350	97	0.02509	240.8
5	DP 2	2	795	234	0.02509	223.73

Properties of Outline A (4): 0

A	B
Property	Value
2	Parameter Chart: General
3	Exclude Current Design point
4	X-Axis (Bottom) P1 - Mesh Element Size
5	X-Axis (Top)
6	Y-Axis (Left) P5 - Equivalent Stress Maximum
7	Y-Axis (Right)

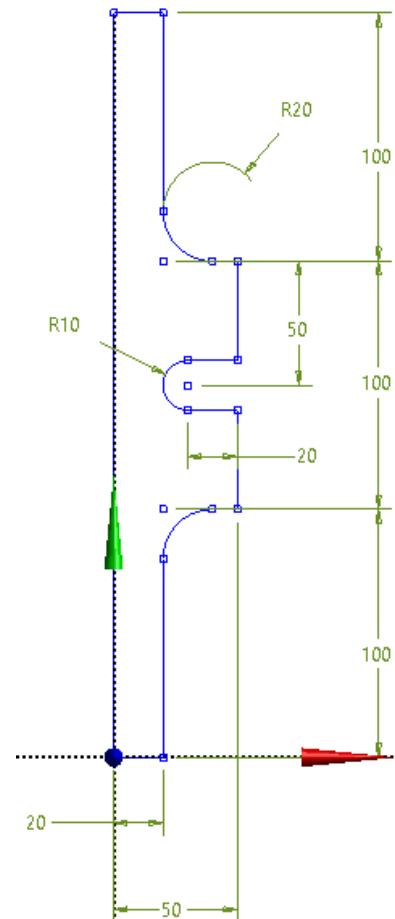
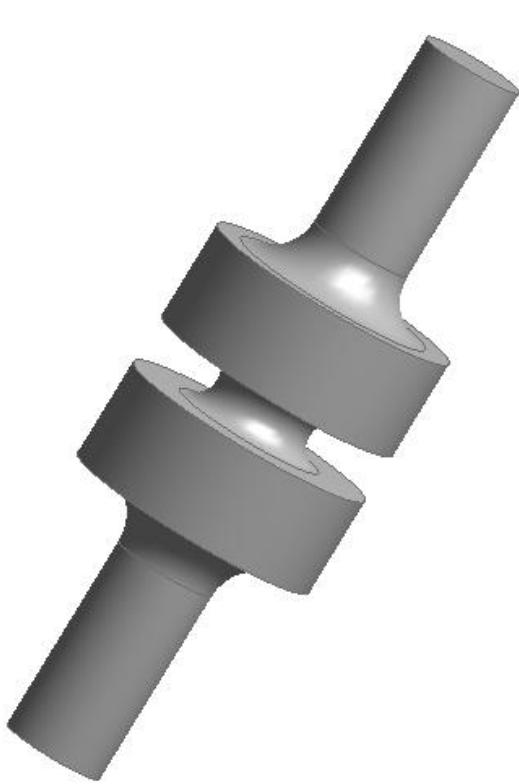
Parameter Chart 0

Table of Design Points

A	B	C	D	E		
Name	P1 - Mesh Element Size	P2 - Mesh Nodes	P3 - Mesh Elements	P4 - Directional Deformation Maximum	P5 - Equivalent Stress Maximum	
Units	mm					
3	DP 0 (Current)	6	285	78	0.02509	219.26
4	DP 1	4	350	97	0.02509	240.8
5	DP 2	2	795	234	0.02509	223.73

Question 3:

A structural steel hole punch device with a side groove is shown in the figure. All dimensions are in mm. One end of the hole punch is fixed and a load of 21 MPa is applied on the other end.

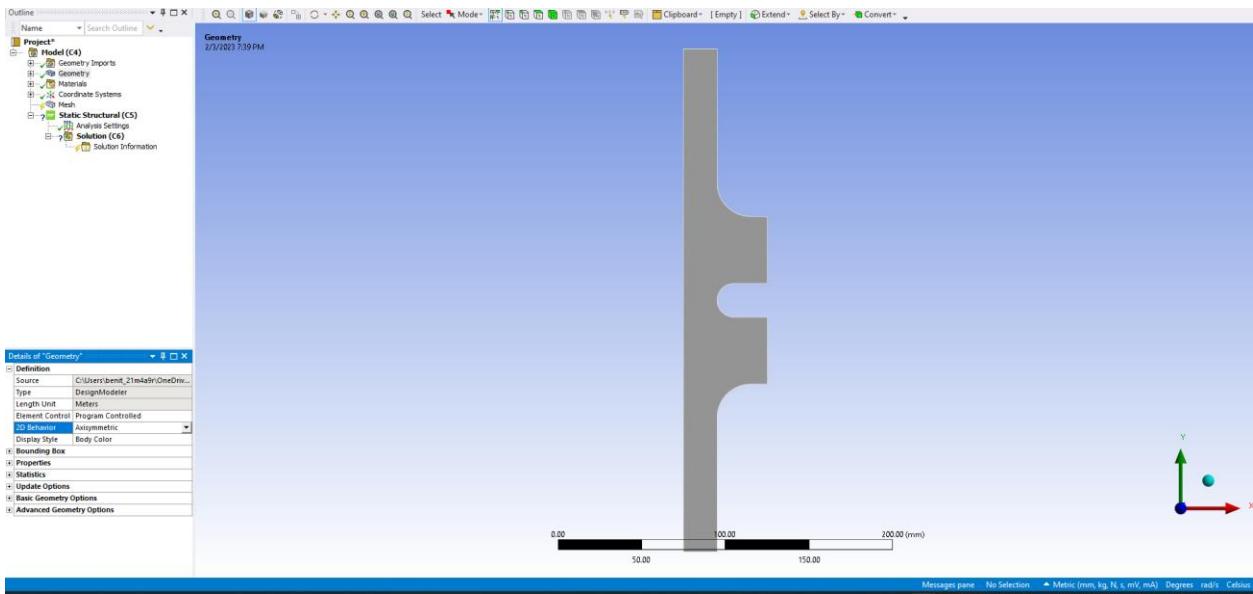


3-D Geometry

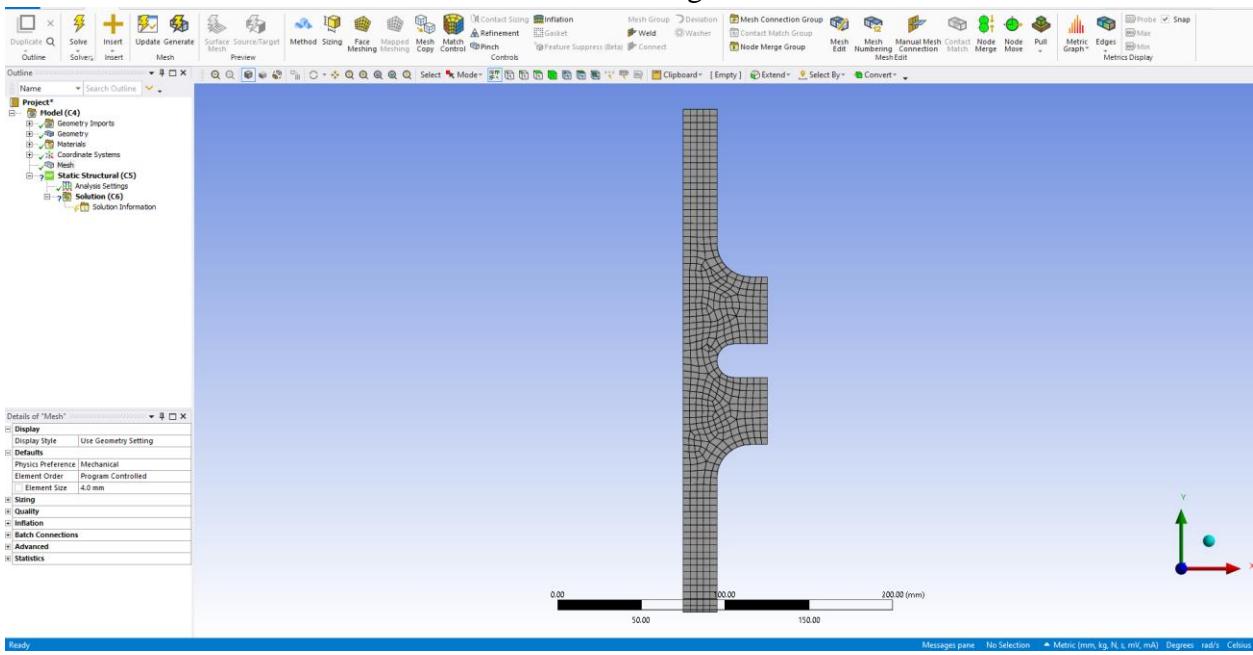
2-D Geometry & Sketch

Analyze the structure in 2D assuming the appropriate 2D elasticity assumption (plane stress, plane strain, or axisymmetry).

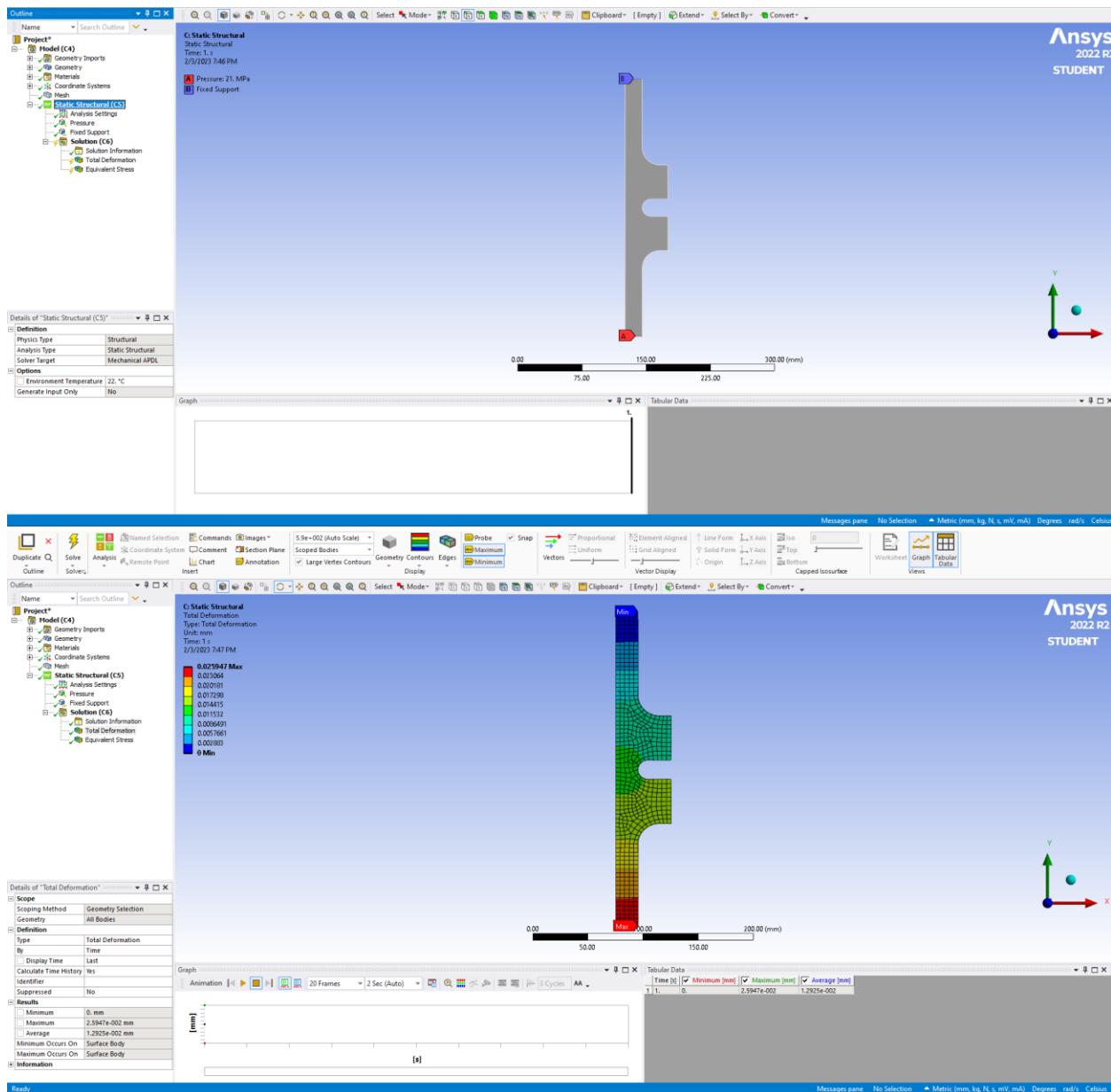
- (a) Create a 2-D surface body by creating a sketch on the XY plane. Show a figure of the surface body.

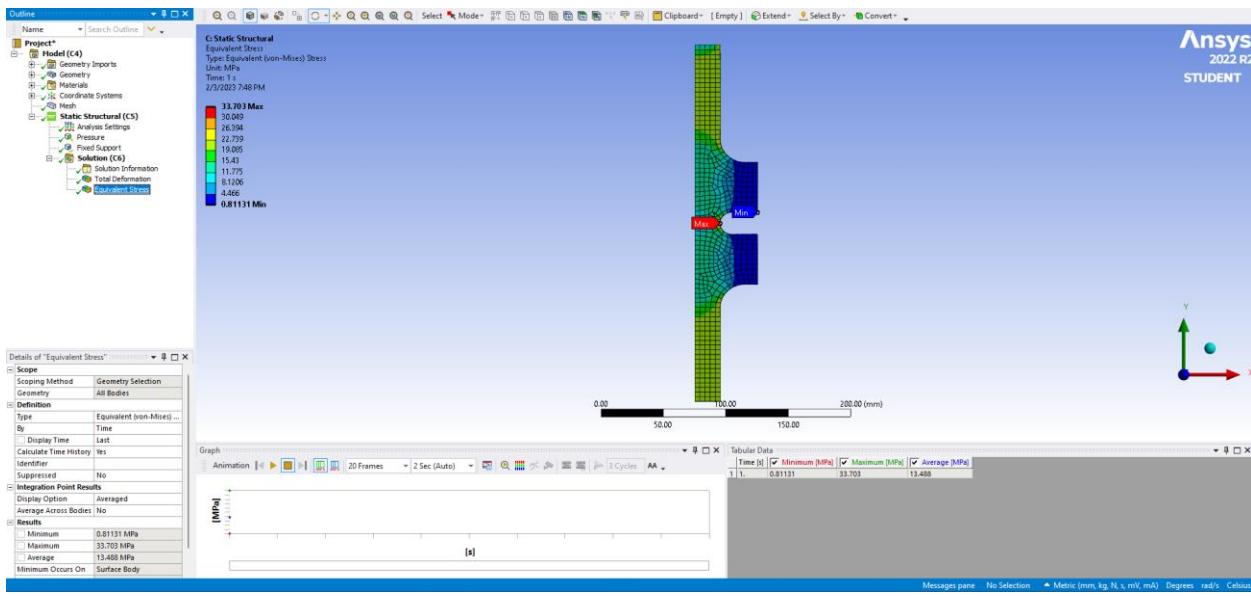


(b) Mesh the model with a mesh size of 4 mm. Show a figure of the mesh.

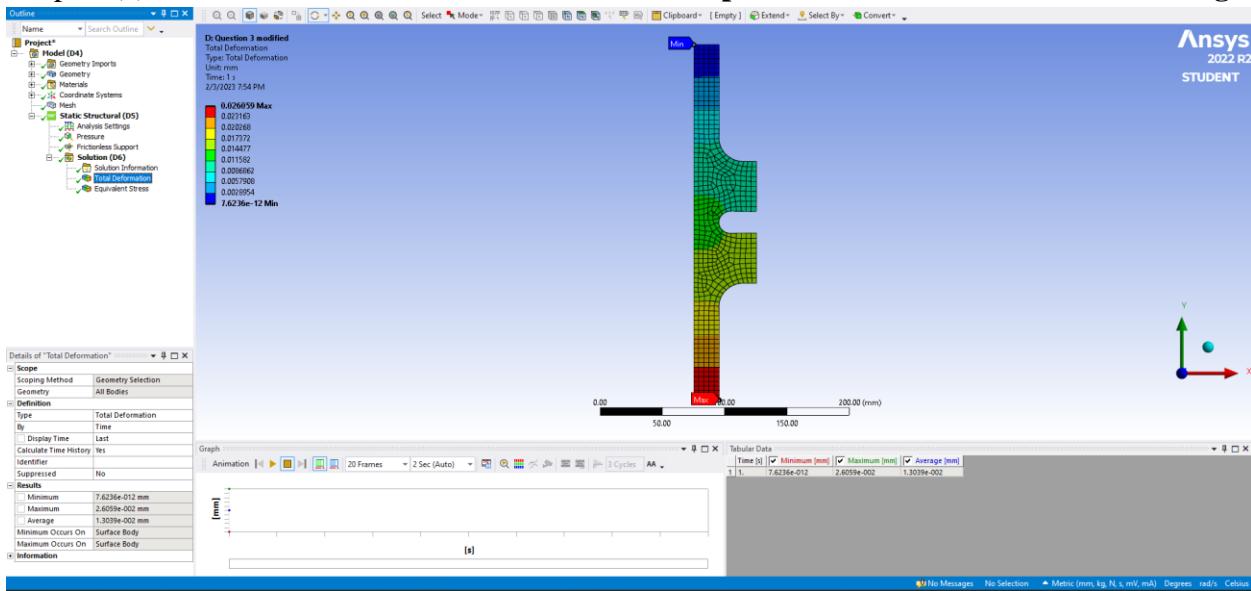


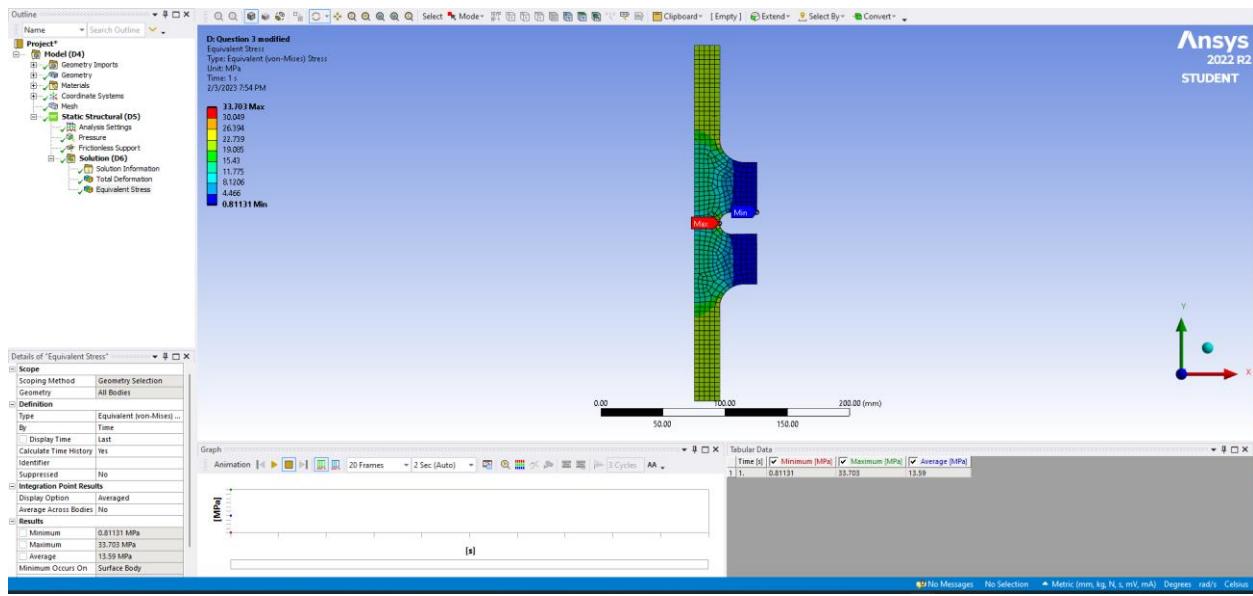
(c) Apply the boundary conditions and loads and determine the maximum total deformation, and maximum von Mises stress. Show figures of the loads & boundary conditions, the total deformation, and the von Mises stress. Where do you expect the maximum stress to occur on this structure and where does the maximum stress occur on this structure? Explain any differences seen. **I expect the maximum stress to occur at the support. The maximum stress occurs. The place where the maximum occurs on this structure is at the middle.**





- (d) Modify the support condition – replace the fixed support with a frictionless support. Duplicate your analysis in ANSYS for this part – do not just make modifications in the existing analysis of part (c). Where does the maximum stress occur now? Is this different from the solution of part (c)? **The maximum stress occurs at the same place and the solution does not change.**





MEE 323 – Computer Aided Engineering II
Homework Assignment #4 – Stress Concentration & Convergence

Instructions:

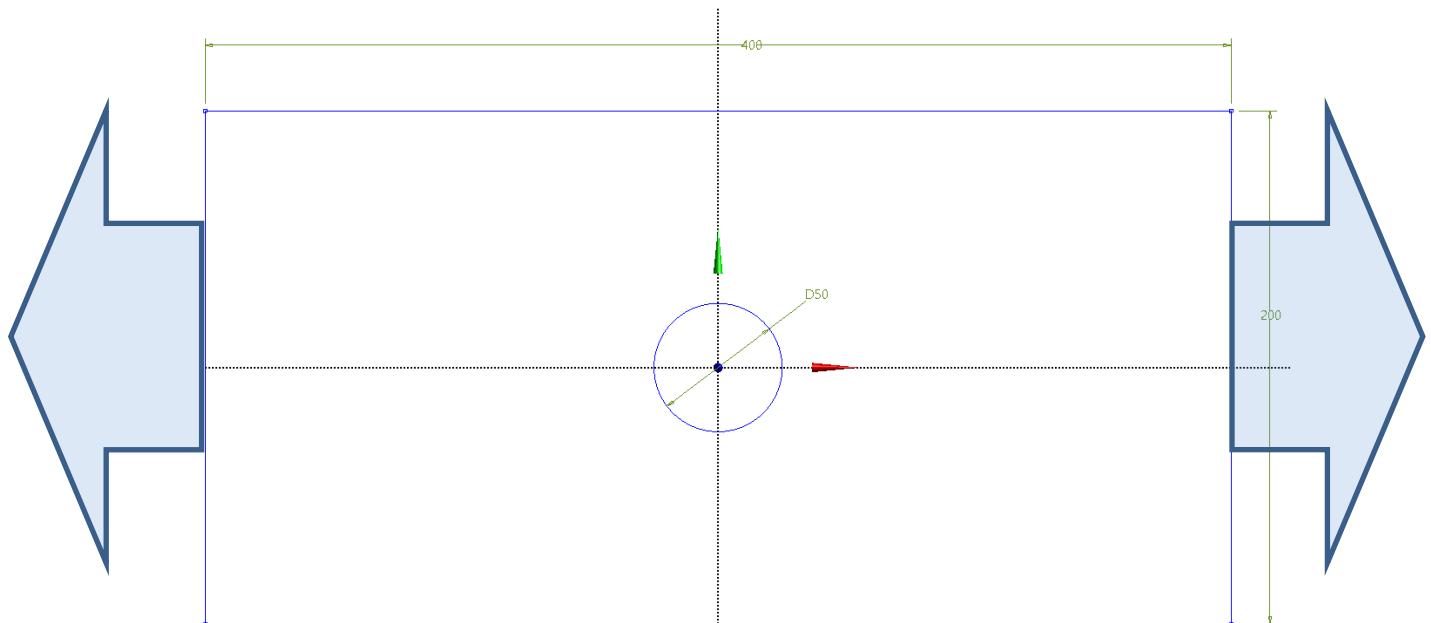
- Use this Word file as a template for your homework report. Add screenshots of your modeling/analysis or any required explanations below the appropriate question. Turn in the homework report (converted to PDF) on the course Gradescope before the deadline.
- Upload a copy of your ANSYS files to the appropriate assignment on the course Canvas. The uploaded ANSYS files may be used to check your work and/or ensure academic integrity.

Homework Objectives:

Learn to create and validate 2-D surface models in ANSYS Workbench. Learn how to apply symmetry boundary conditions to reduce model size and analysis time & cost. Learn about stress concentration, structural error, shape functions, and FE convergence.

Question 1:

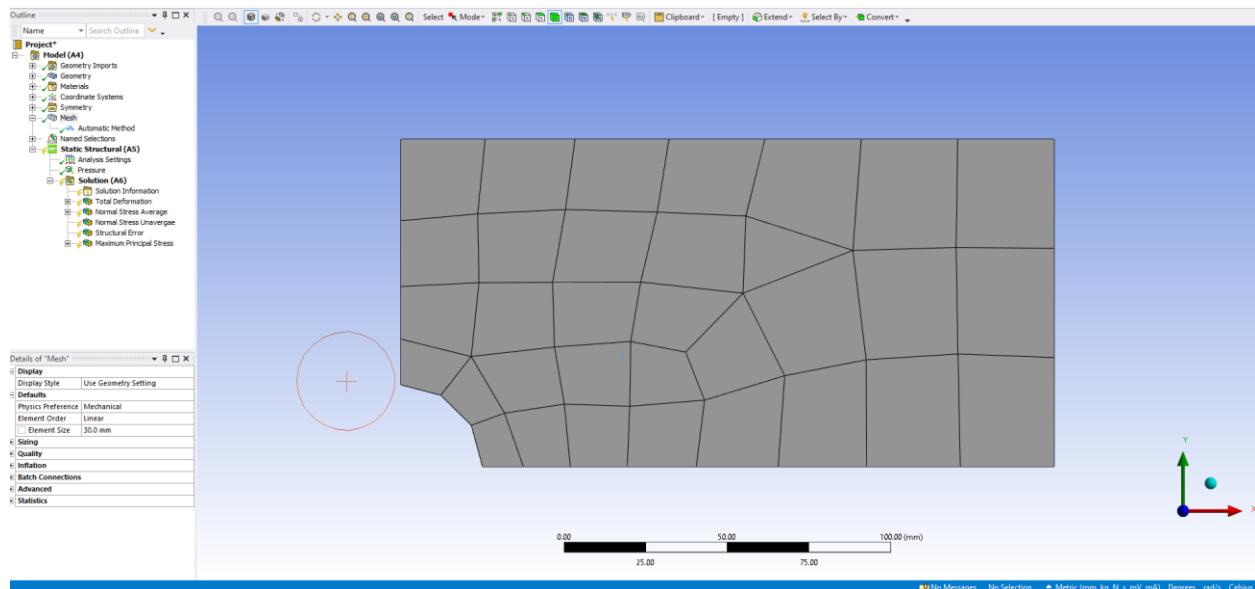
Consider a thin structural steel plate (thickness of 10 mm) with a circular hole in the center as shown below. A pressure load of 50 MPa is applied to the left and right end faces (represented by blue arrows). Reduce the model using two planes of symmetry and analyze one-quarter of the plate. Assume plane stress condition and model the plate as a 2-D surface model in ANSYS Workbench. Use 30 mm mesh size.



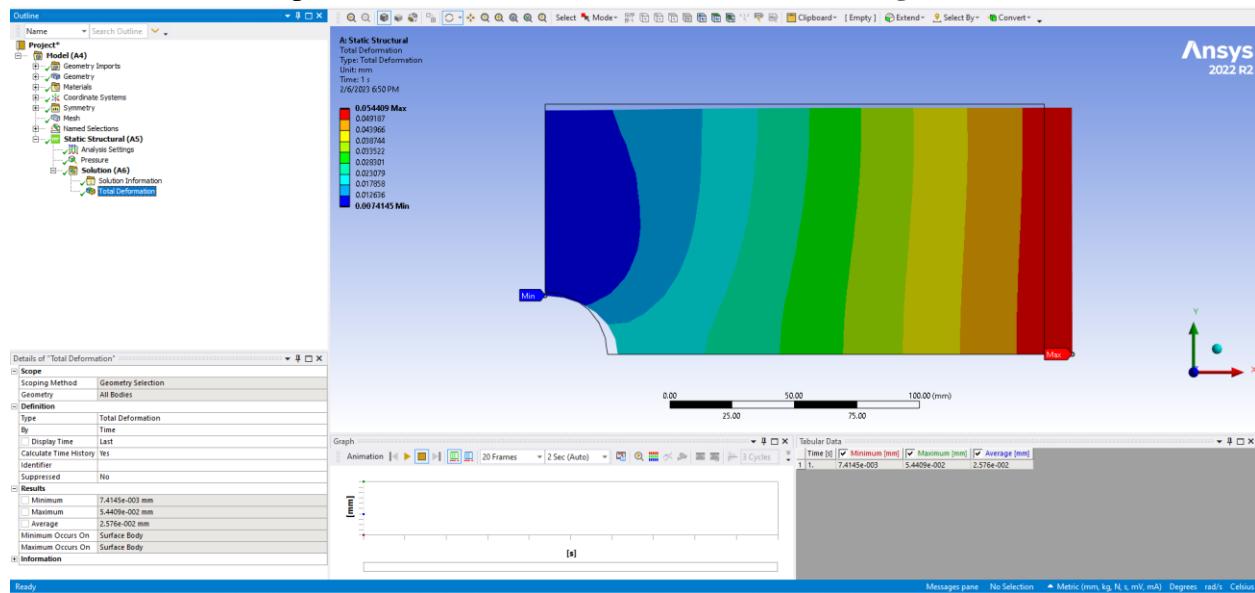
- (1) Set up mesh controls as demonstrated in class and carry out the analysis with a quadrilateral element mesh with linear element order (lower order shape function).

- (a) Show the figure of the mesh. Point out any relevant features. Does this mesh agree with our “rules of thumb” for meshing? Hint: the “rule of thumb” for number of elements around a circular hole – we want 30-60 elements around a circular hole.

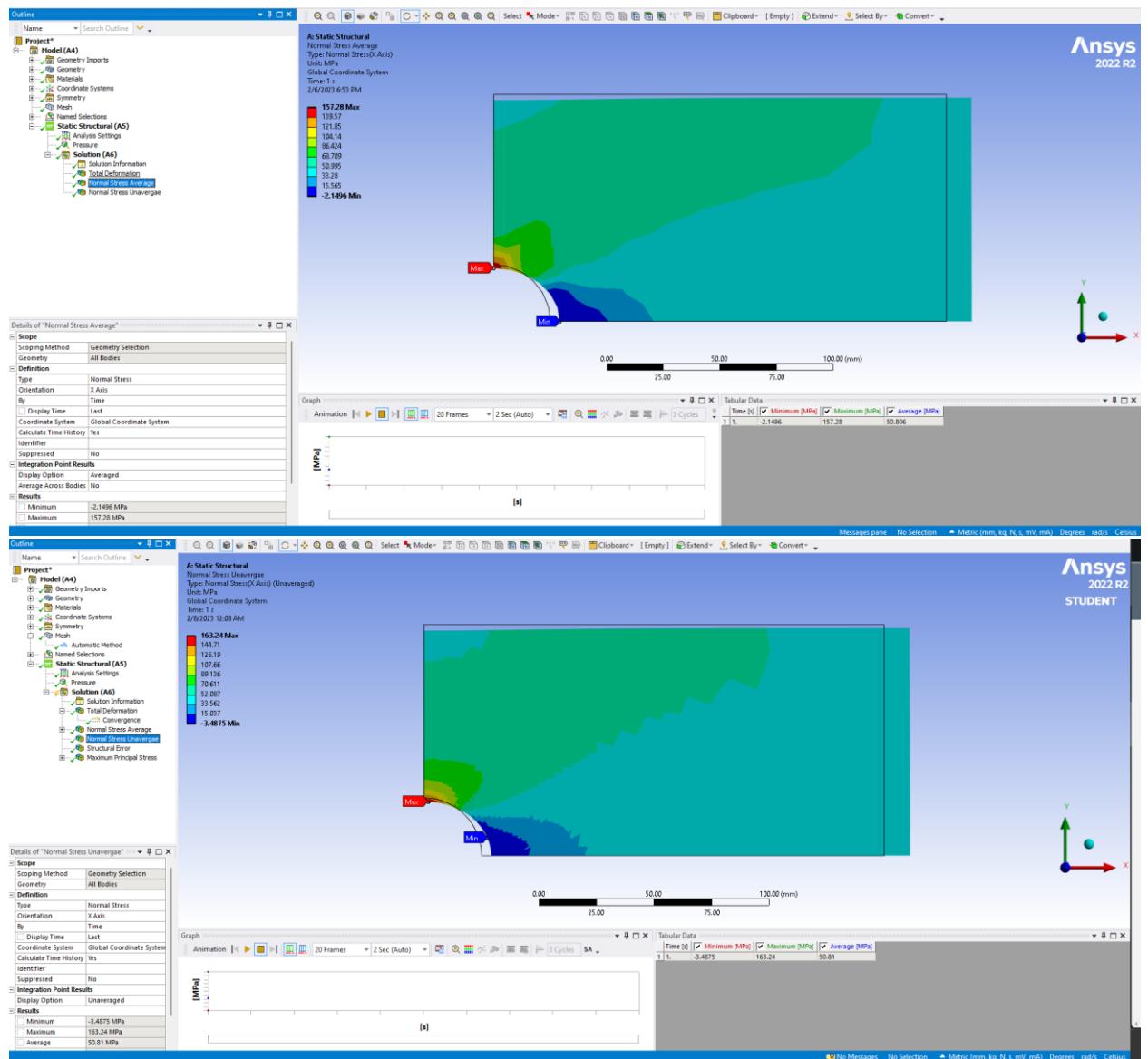
The meshing does not follow the “rules of thumb” because there are about 3 elements around the circular hole.



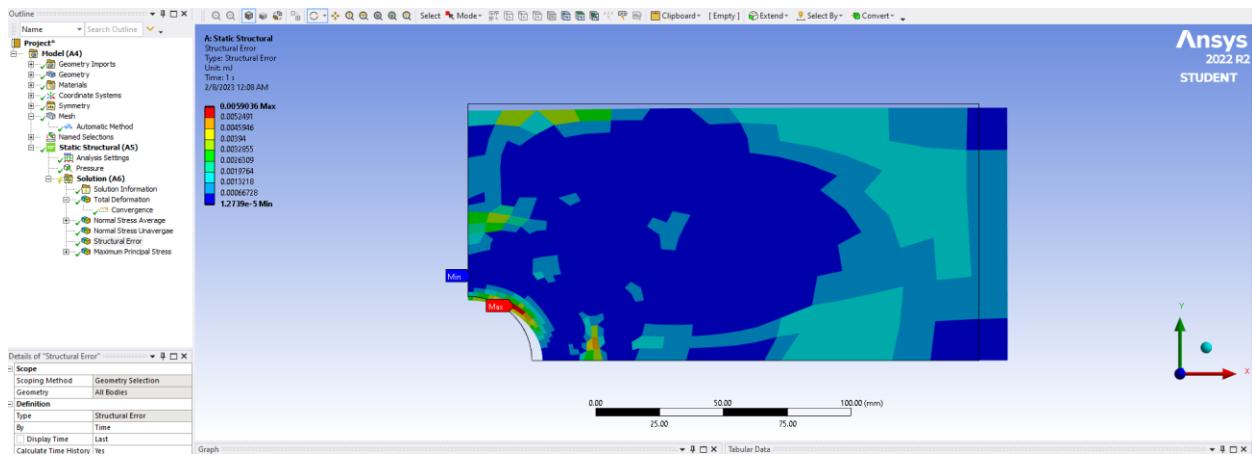
- (b) Show the figure of the deformed shape with undeformed wireframe. What happens to the hole when the plate deforms? **The hole is deformed to the right.**



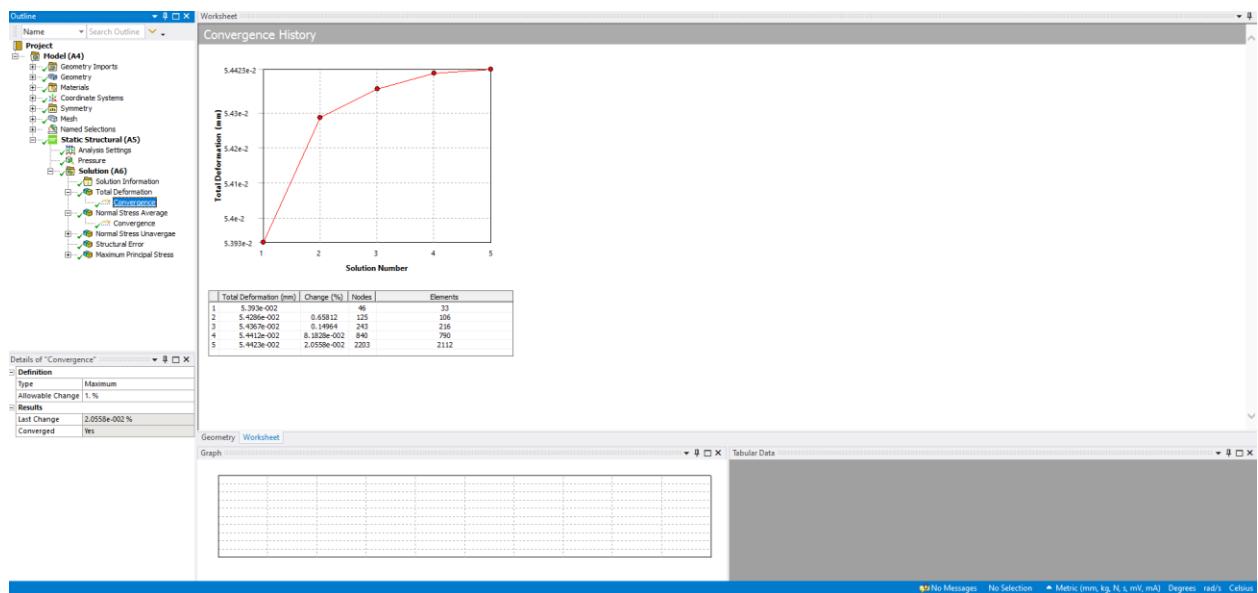
- (c) Show the figure of the normal stress in the direction of the loading. Show both the averaged and unaveraged stress contours and comment on what you see. **In the average stress the lines are more smooth since it gives a approximation. In the un average stress the lines are not at smooth because it is calculated more at that spot.**

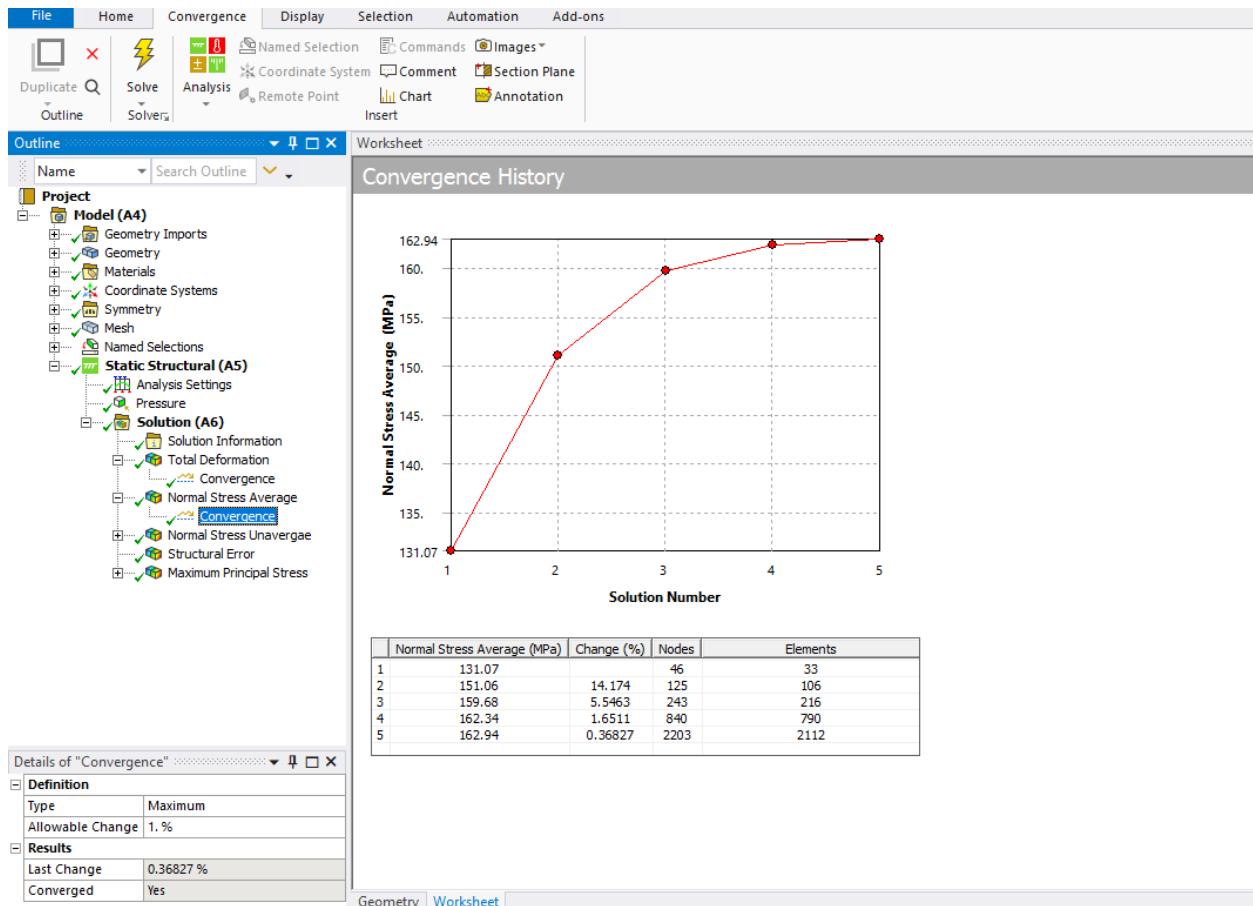


- (d) Show the figure of the structural error computed by ANSYS. What does this tell you? Where would the mesh have to be refined? Where could the mesh be less refined? Explain why in your own words. **The error is more towards the circular shape. This tells me there can be more error because the stress changes more due to stress concentration meaning it will need more meshes for it to be more accurate. This is where it need to be refined where away from the circle it can be less refined because there is no error.**



- (e) Carry out an automatic/adaptive convergence analysis (see section 5.3-6 on page 221 of the textbook for step-by-step instructions) on both the maximum displacement and the maximum averaged normal stress (you may want to do each separately). Use a convergence criterion of 1% for both. Comment on the difference that you see between displacement and stress convergence. Why do you see a difference? **There is a difference because displacement will not change as much which will cause it to converge more.**





- (f) Repeat the analysis of (e) but with quadrilateral elements with quadratic element order (higher order shape functions). Compare the total number of nodes and elements required for convergence when the element order used is linear vs. quadratic.

Deformation

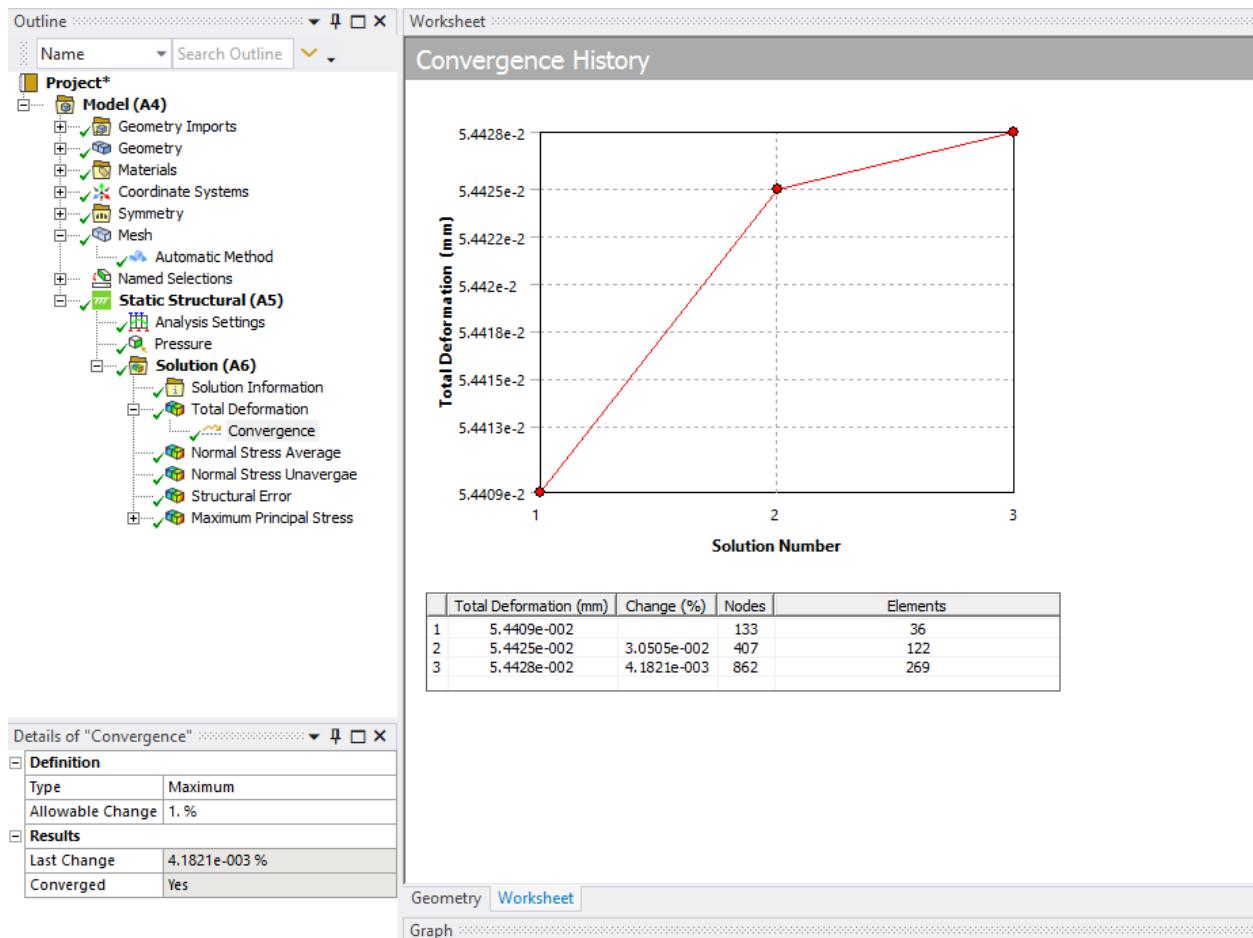
Linear Nodes: 125

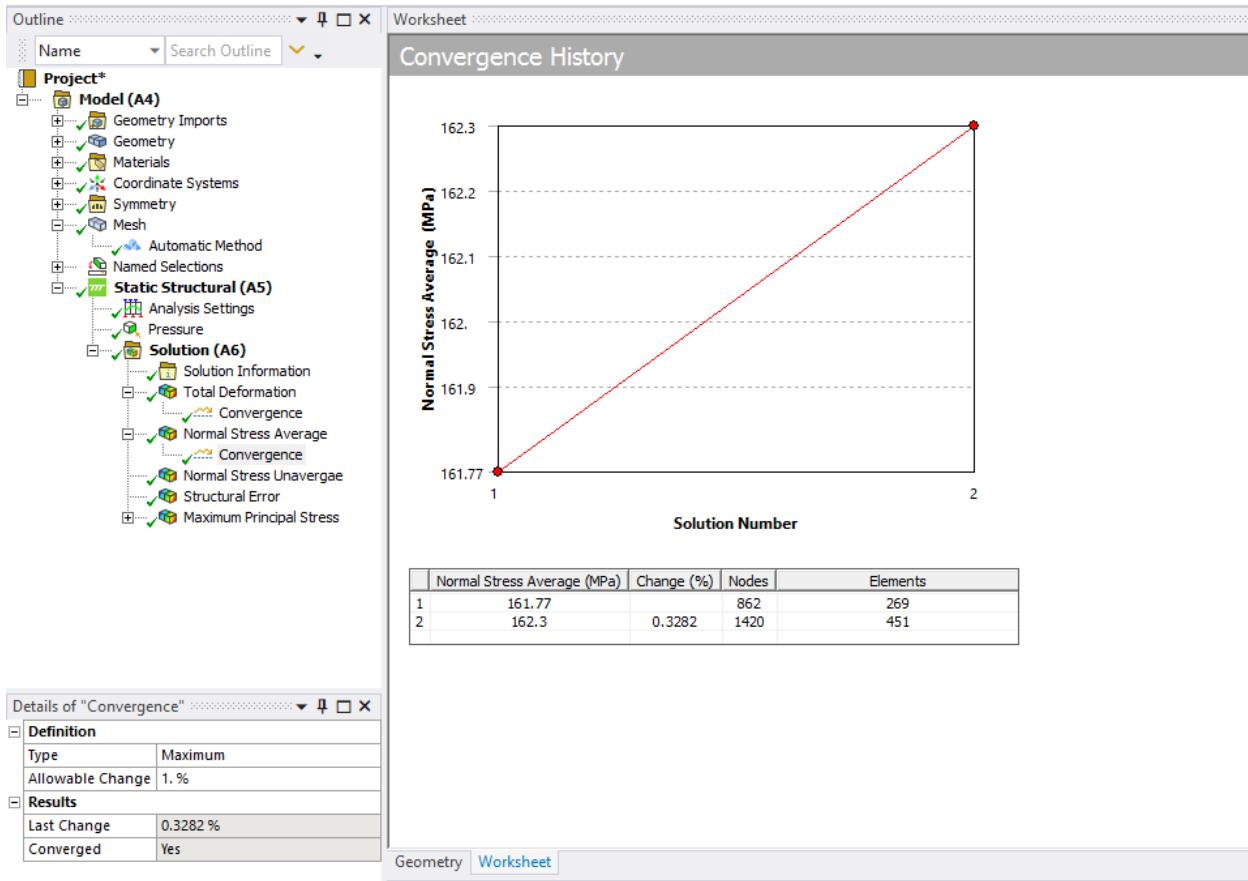
Quadratic Nodes: 3096

Normal Stress

Linear Nodes: 840

Quadratic Nodes: 3096





- (2) For the analysis 1(f) with quadrilateral elements with quadratic element order, show the contour plot of the maximum principal stress. What is the maximum value of the stress? Use the attached stress concentration factor table to calculate the theoretical value of maximum stress and validate your FE model results. **Maximum stress is 162 MPa.**

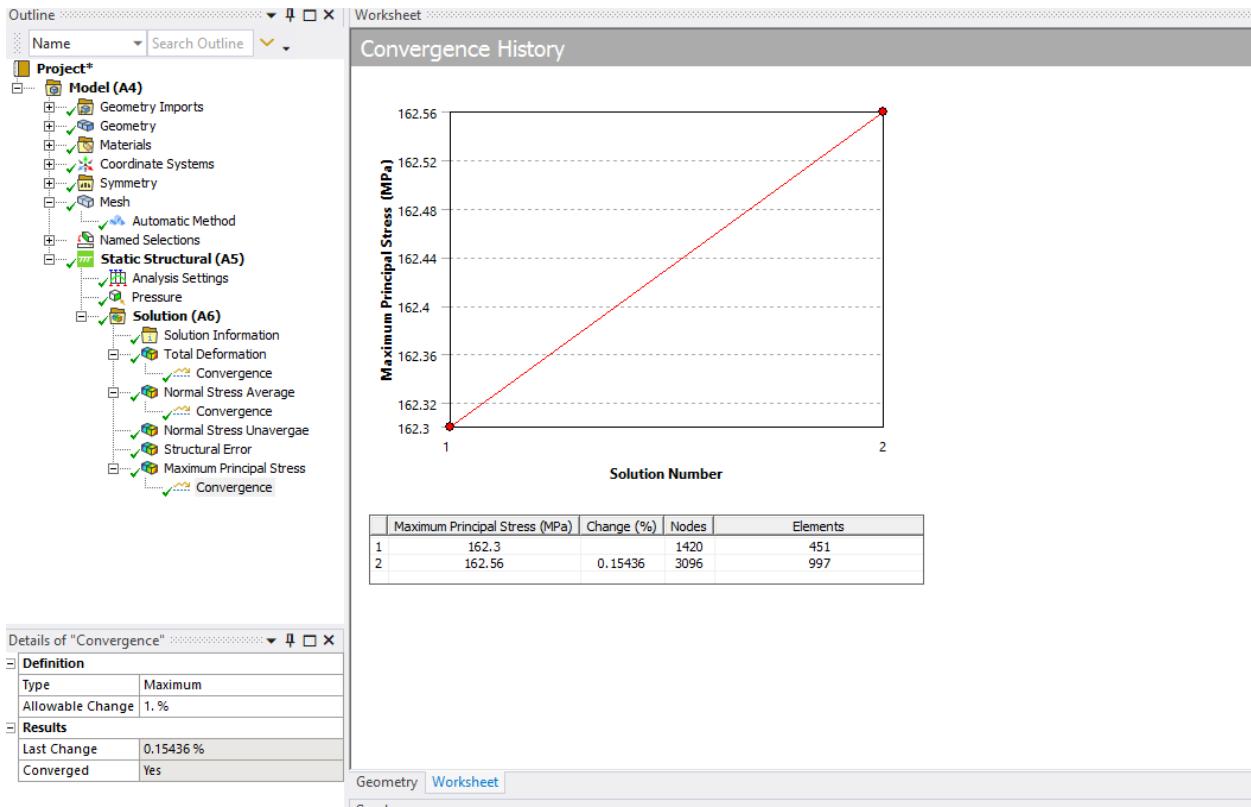
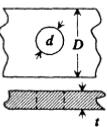
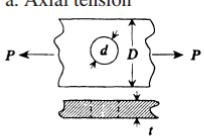
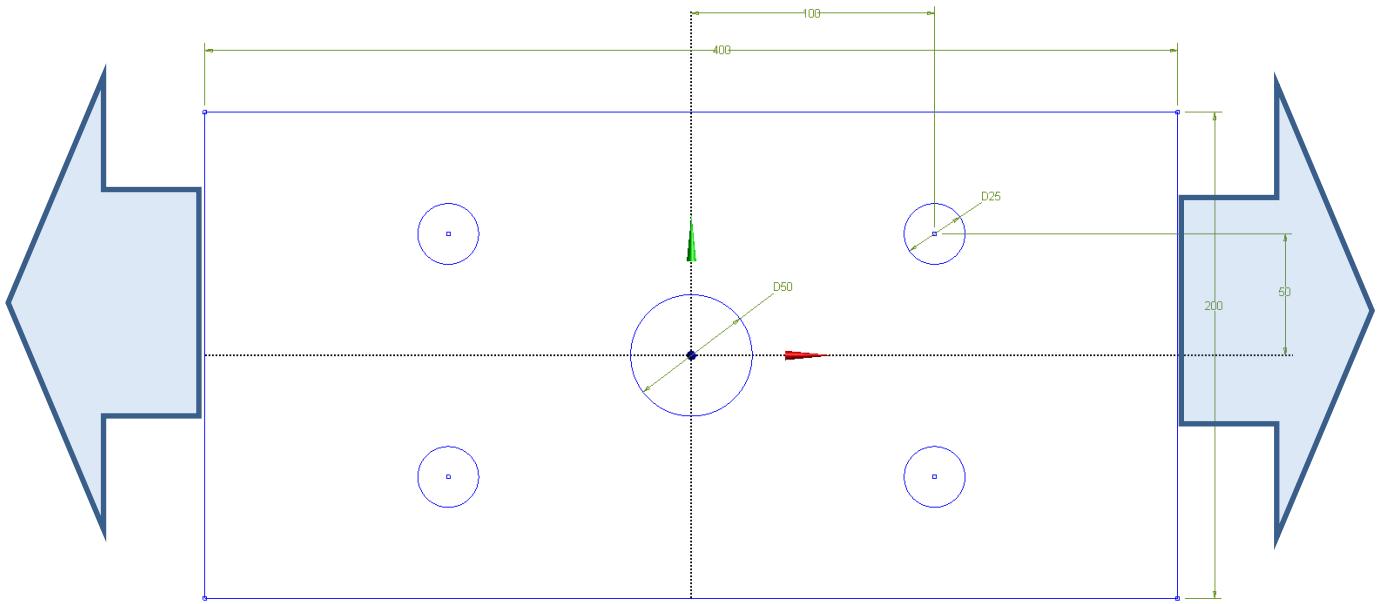
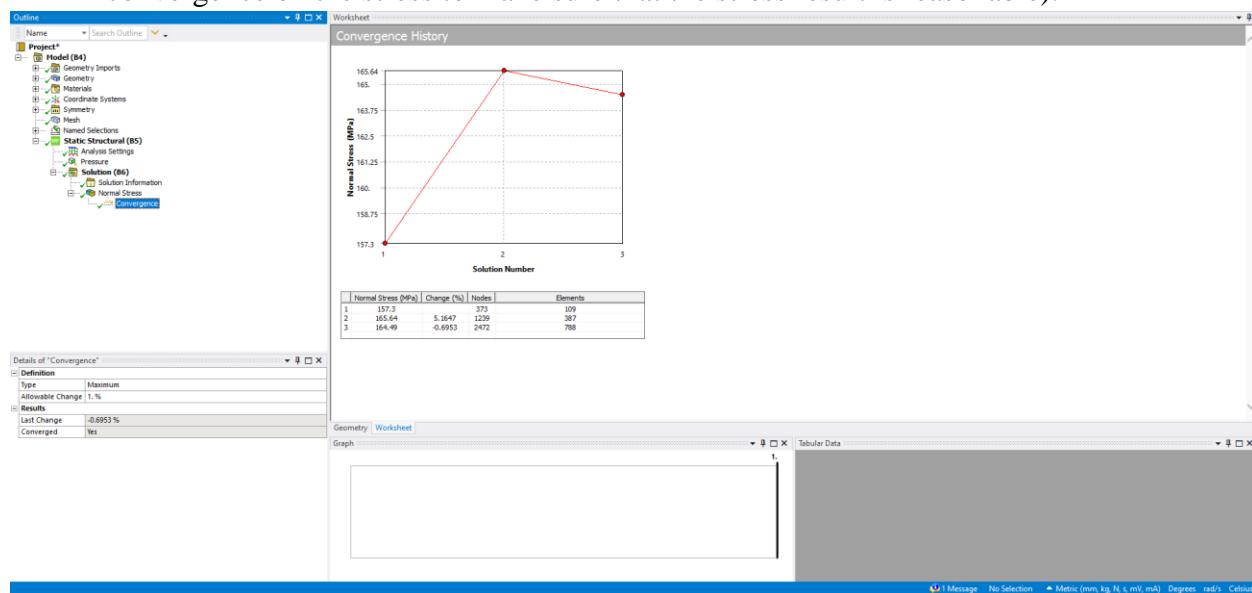


TABLE 6-1 (continued) STRESS CONCENTRATION FACTORS: Holes		
2. Central single circular hole in finite-width plate 	a. Axial tension 	$\sigma_{\max} = \sigma_A = K_t \sigma_{\text{nom}}, \quad \sigma_{\text{nom}} = P/[t(D-d)]$ $K_t = 3.000 - 3.140(d/D) + 3.667(d/D)^2 - 1.527(d/D)^3$ $\text{for } 0 \leq d/D \leq 1$

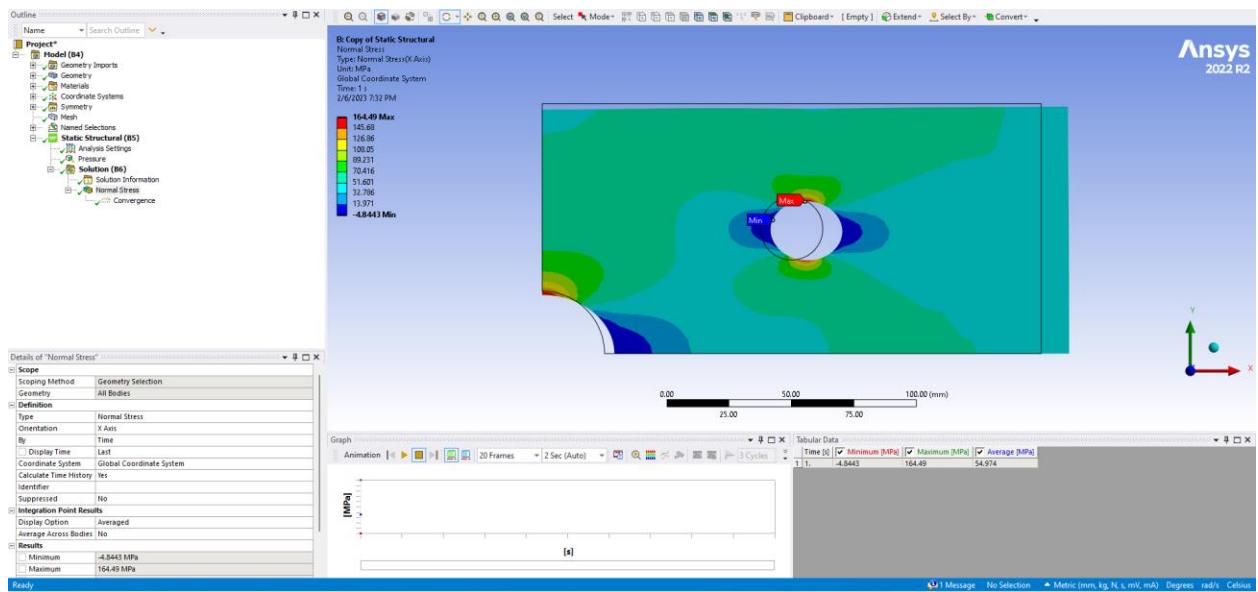
- (3) Place four 25mm diameter holes as shown below. Once again, reduce the model using two planes of symmetry and analyze one-quarter of the plate. Assume plane stress condition and model the plate as a 2-D surface model.



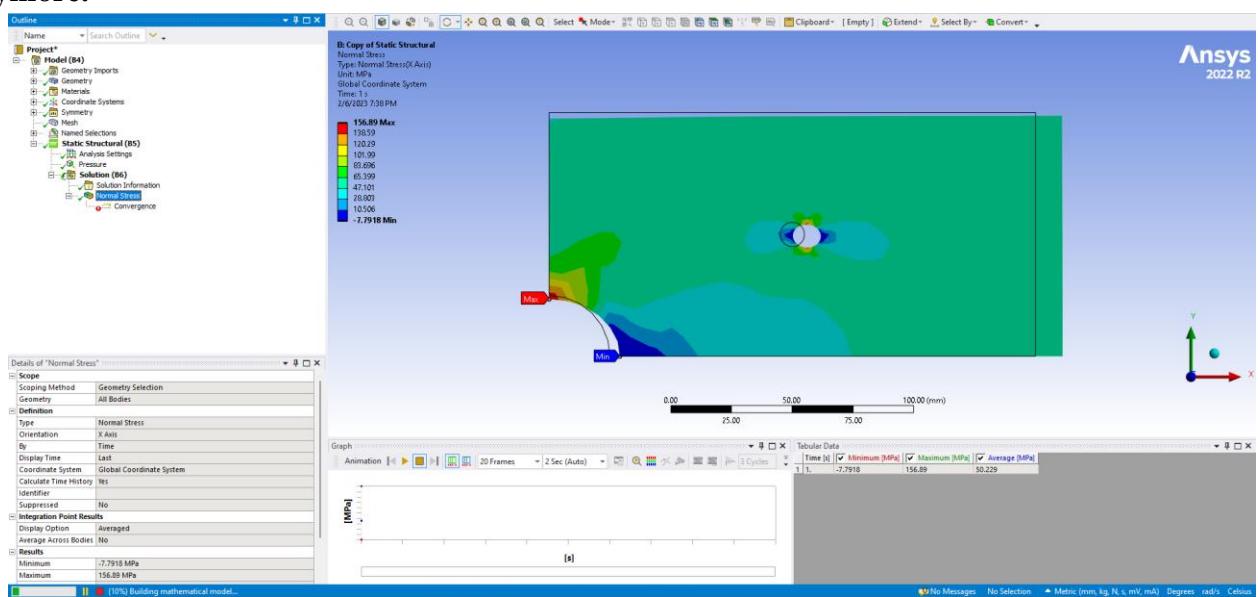
- (a) Find the magnitude and location of the maximum normal stress (run a quick automatic convergence on the stress to make sure that the stress result is reasonable).



- (b) Show the figure of the normal stress in the direction of the loading.



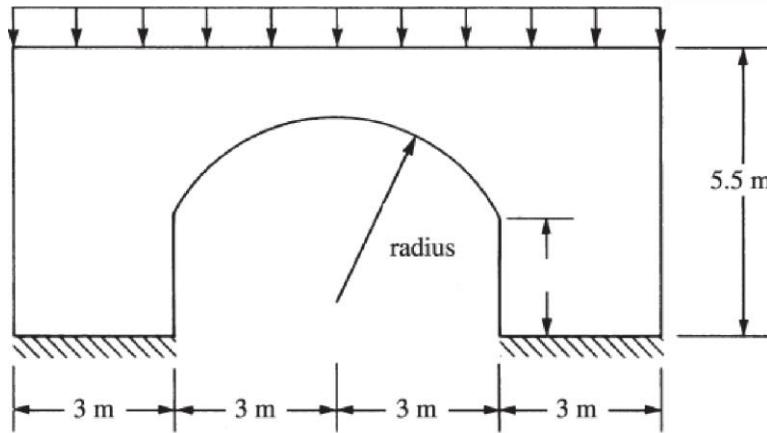
- (c) Compare this result to the maximum normal stress obtained in part (1) and explain the differences. Why does the smaller hole show a larger value of stress compared to the larger hole in the plate? How would you validate this result if you had to? **The smaller hole has a larger value because the change in geometry is higher.**
- (d) Reduce the diameter of the smaller hole to 10mm. Repeat the analysis. What do you see now? Where is the maximum stress in the plate and why? **The maximum stress is now in the larger whole because the small hole is smaller now so stress isn't as concentrated anymore.**



Question 2:

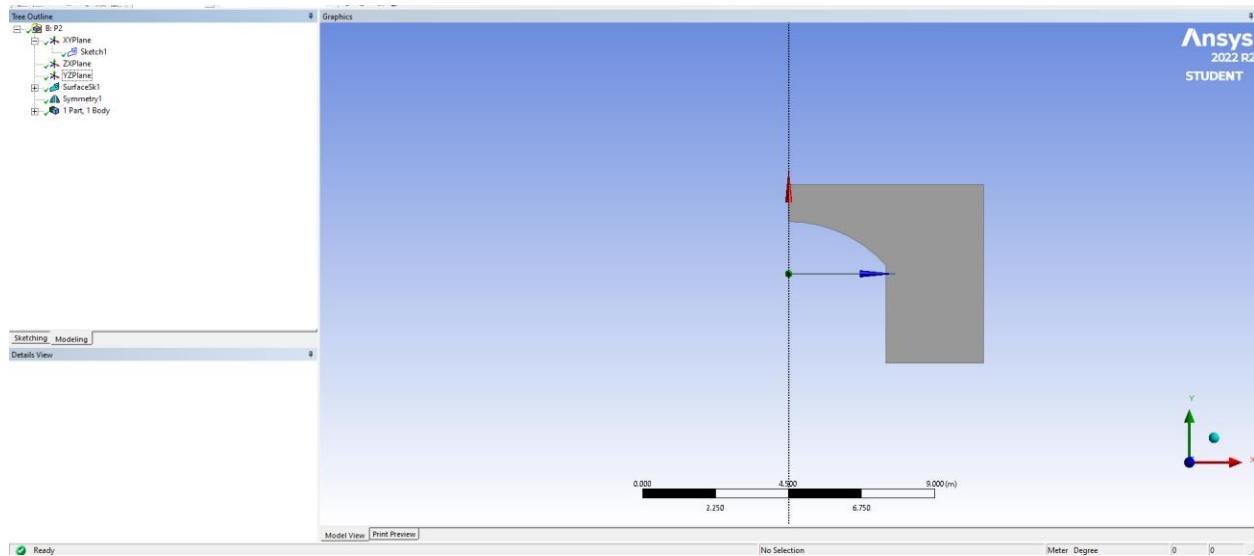
Consider the **concrete** overpass structure with dimensions shown in the figure below.

See the last page below on how to change materials used in the simulation.

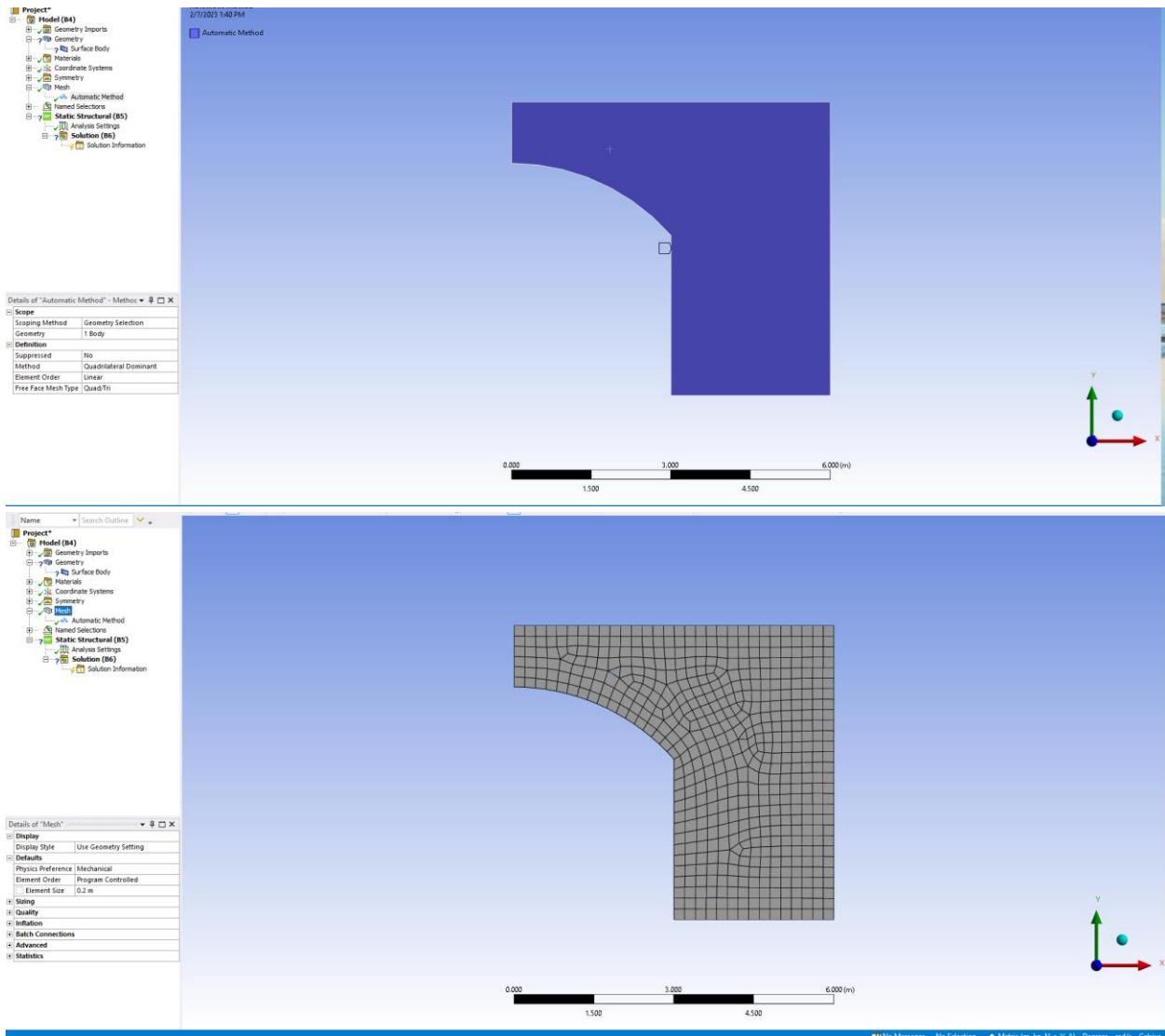


Analyze the structure in 2-D assuming plane strain conditions. There is a reflective plane of symmetry in the structure – use it appropriately and model just half of the structure. Make sure to account for the effect of using symmetry on the loading of the structure.

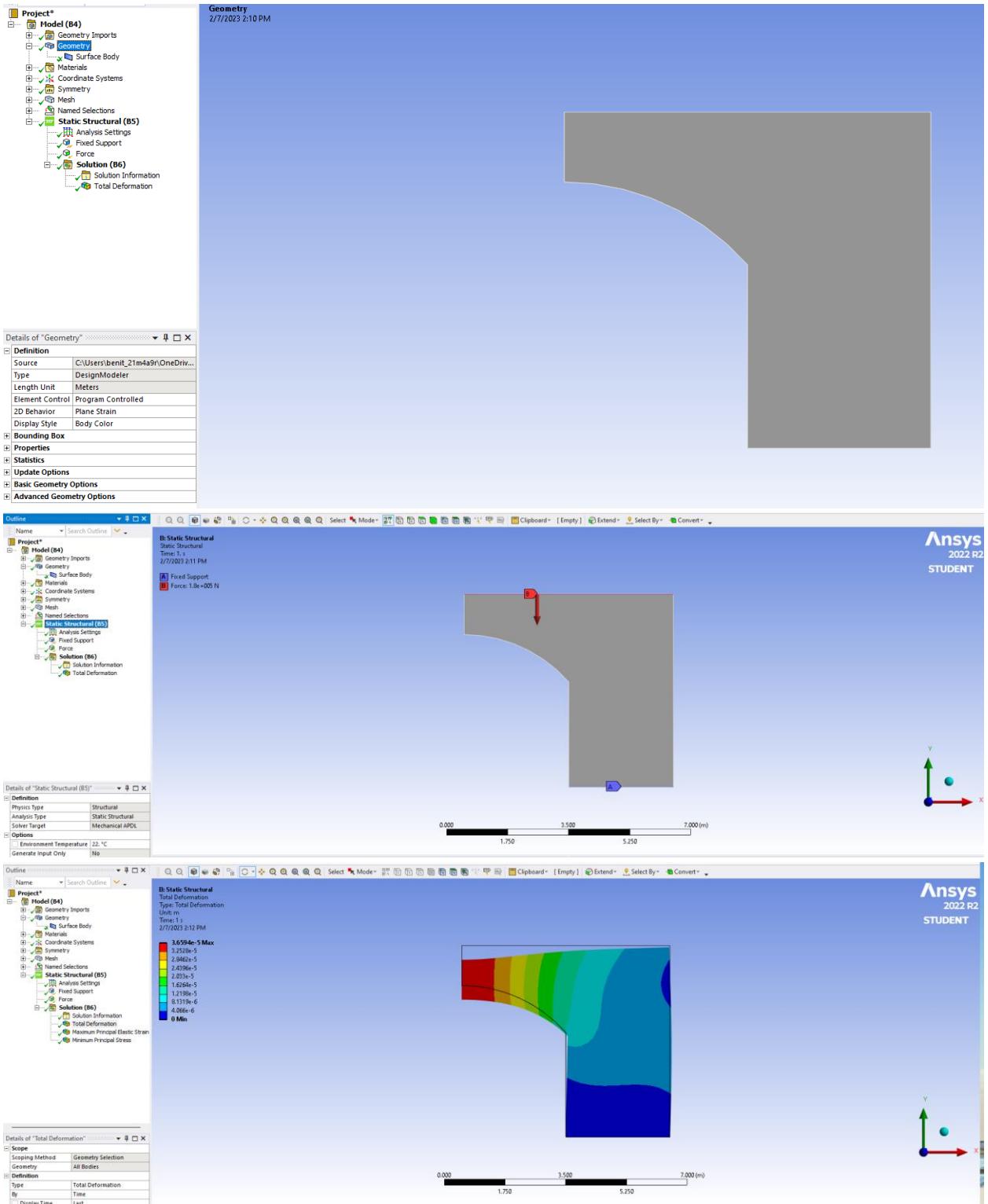
- (a) Create a 2-D surface body by creating a sketch on the XY plane. Show a figure of the surface body after having used symmetry.

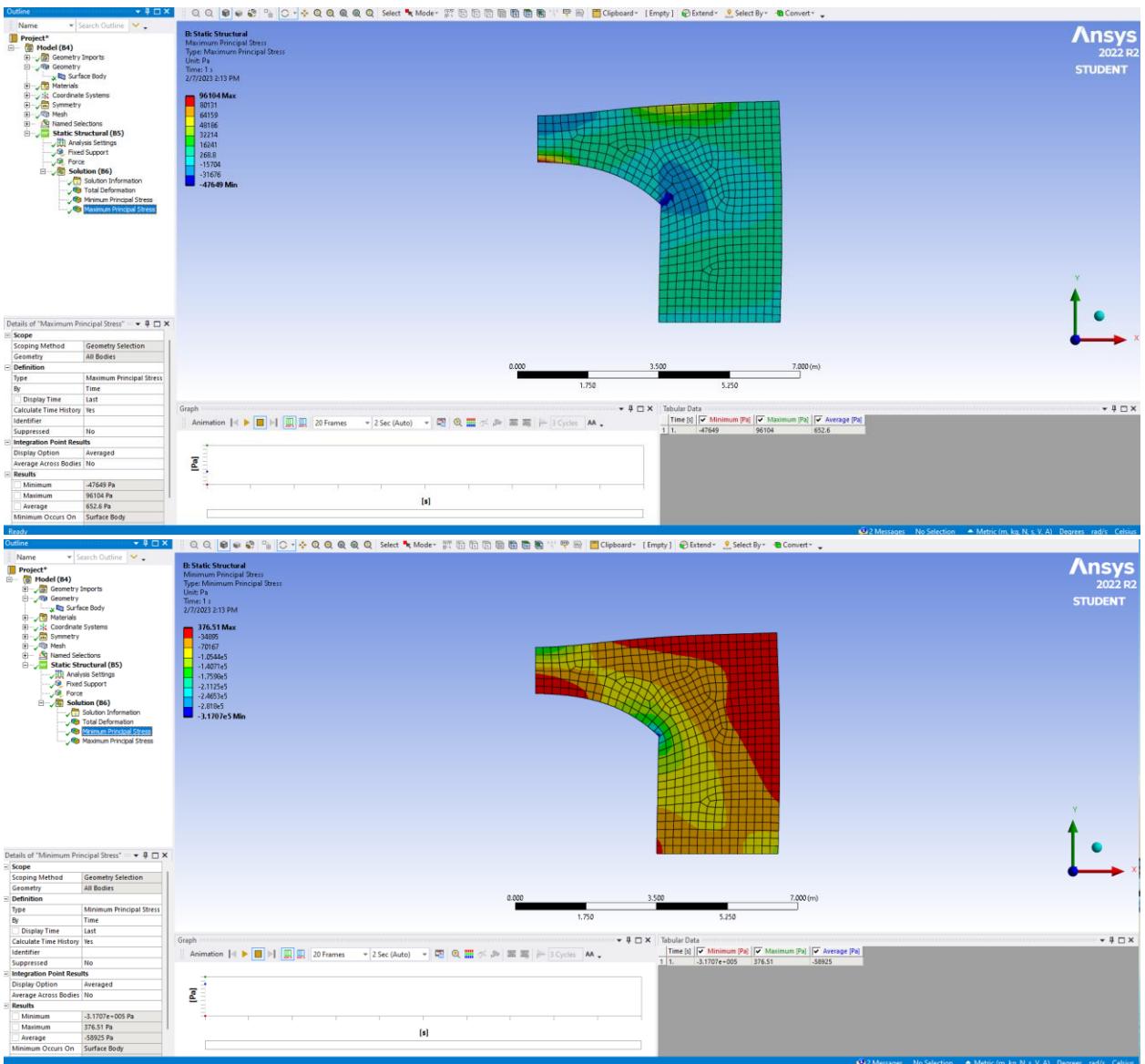


- (b) Mesh the model with a mesh size of 0.2 m using quadrilateral elements with linear element order. Show a screenshot of the mesh method control with quadrilateral elements and linear element order. Also show a figure of the mesh.

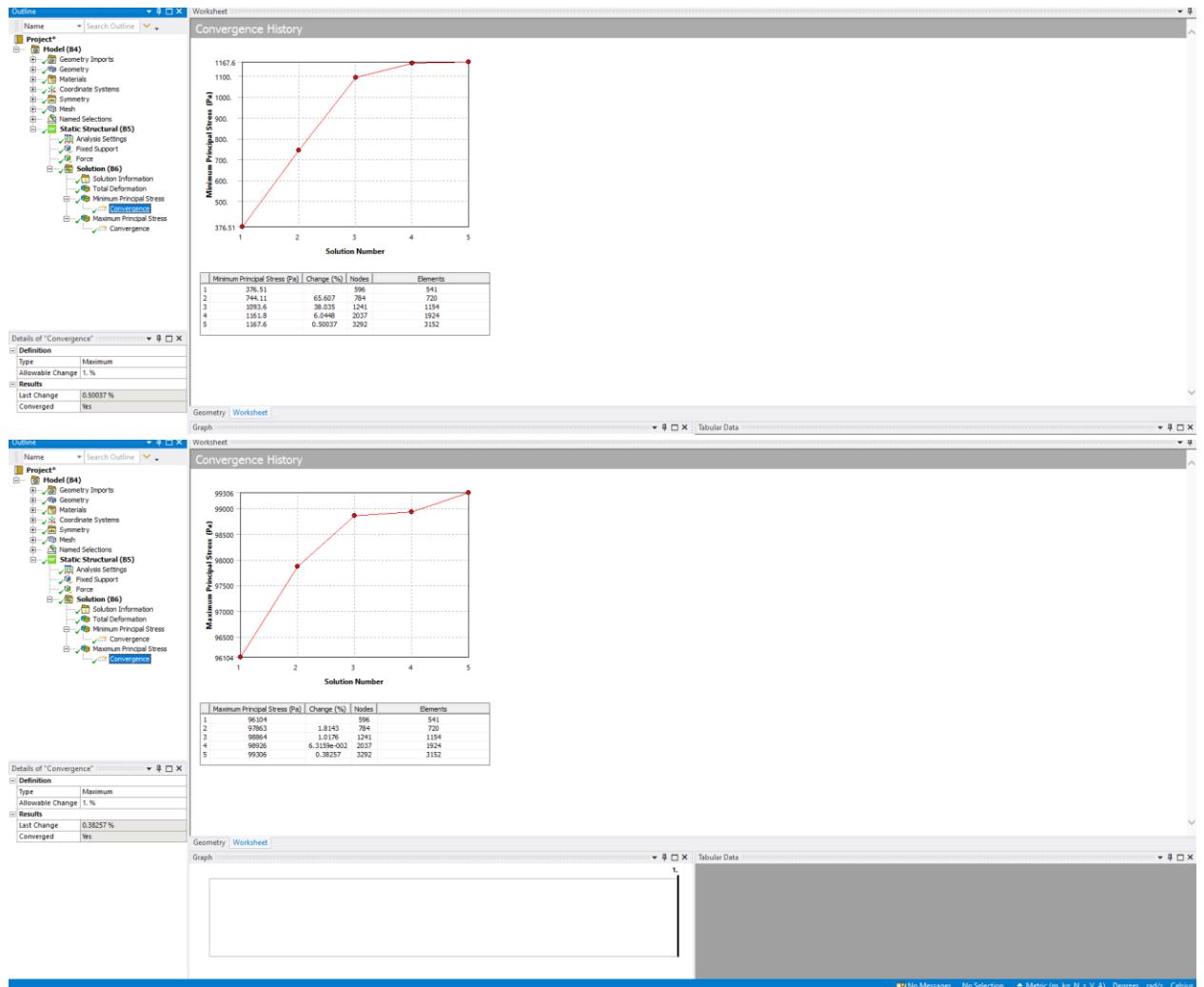


- (c) Apply the boundary conditions and loads and determine the maximum total deformation, maximum principal stress, and minimum principal stress. Show a screenshot of the geometry details showing that you are indeed running a plane strain analysis. Show figures for the applied boundary conditions & loads as well as each of the required results.

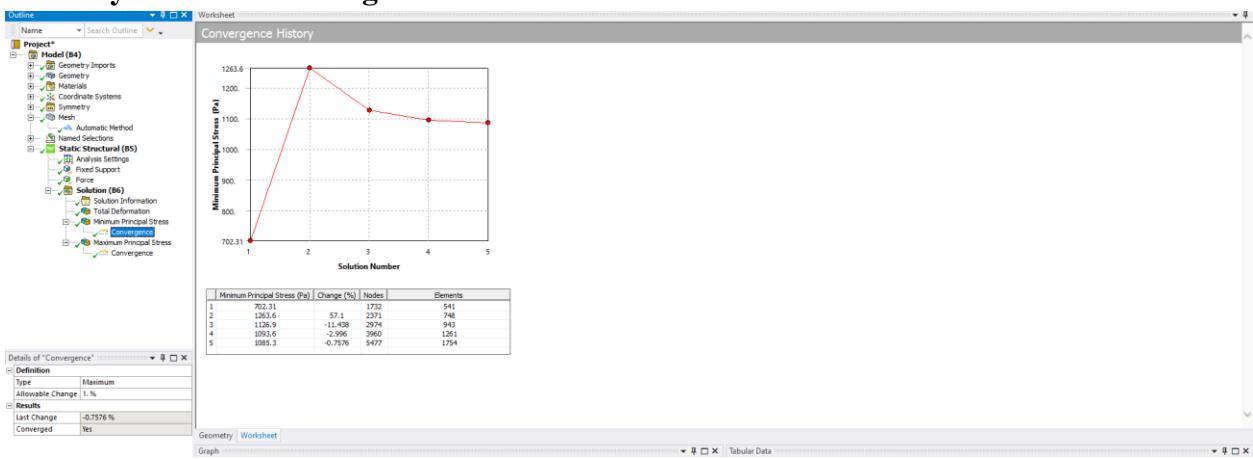


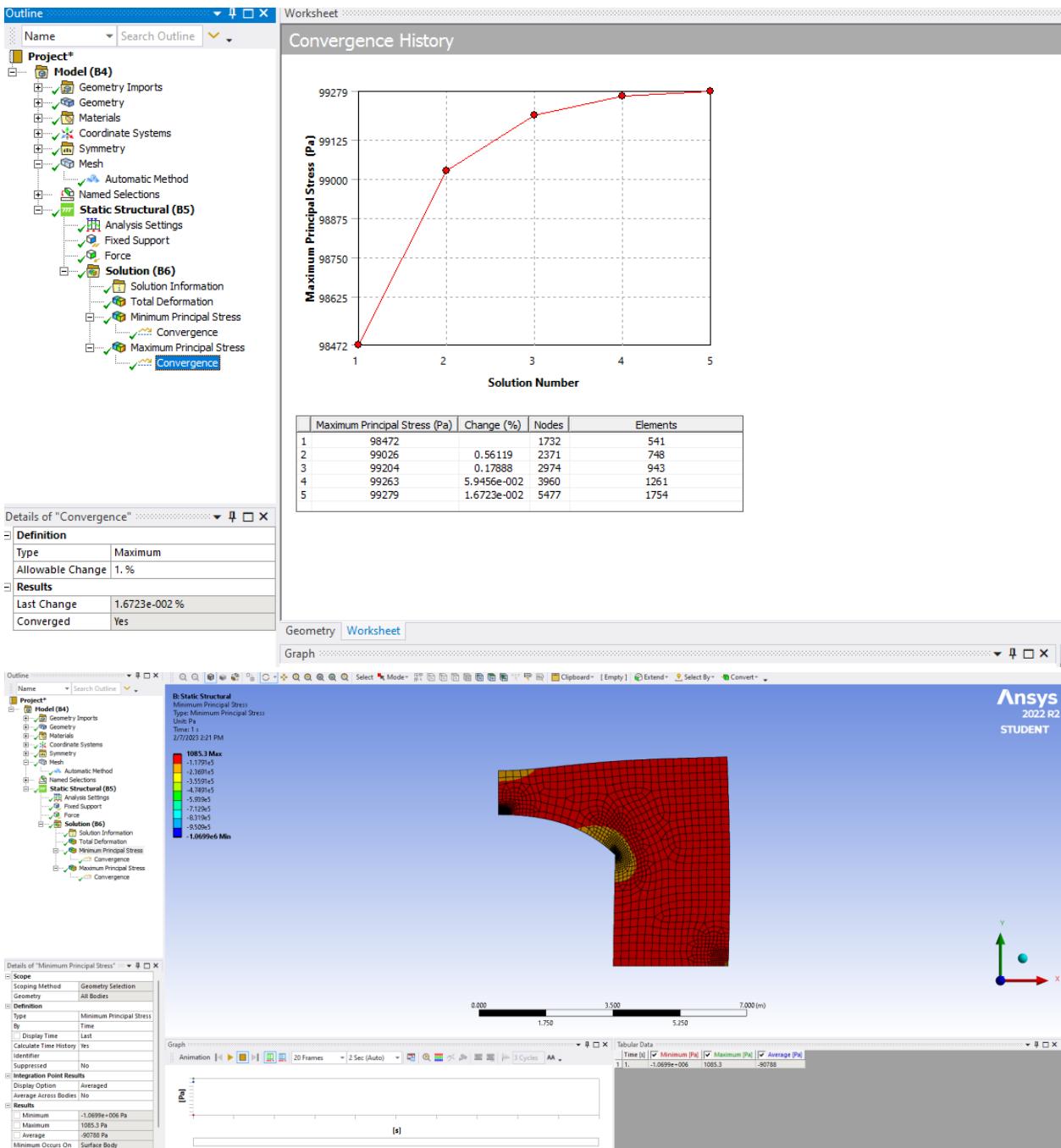


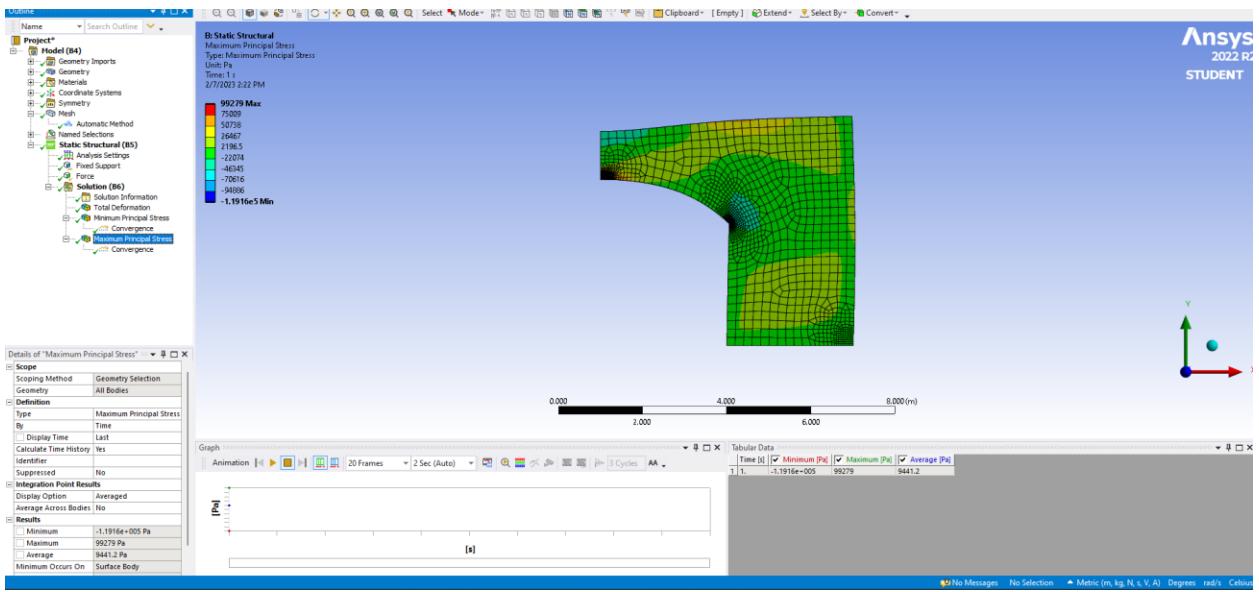
- (d) Carry out a convergence analysis on (i) the maximum value of the maximum principal stress and (ii) the minimum value of the minimum principal stress. You may carry out the convergence study for both simultaneously. Choose an allowable change in the stress value of 1% with the maximum number of refinement loops set to 5. Show convergence plots for both (i) and (ii) and explain the results. Why does one value converge while the other one diverges? How can you prevent the divergence (in words only)? **The maximum stress diverges because the stress will keep going into infinity due to stress concentration.**



- (e) Repeat the analysis of (d) but mesh the model with a mesh size of 0.2 m using quadrilateral elements with quadratic element order. Comment on the differences seen. **The difference is that they now both converge.**

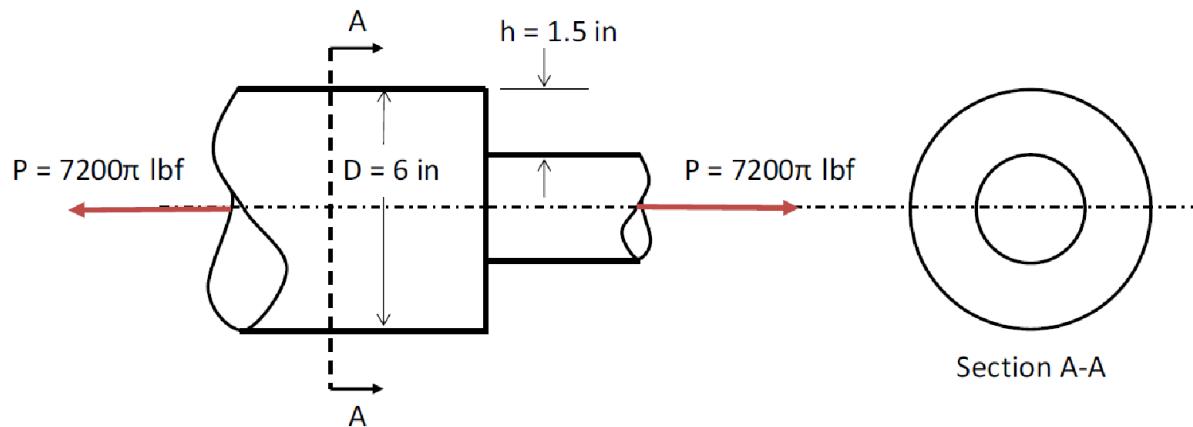






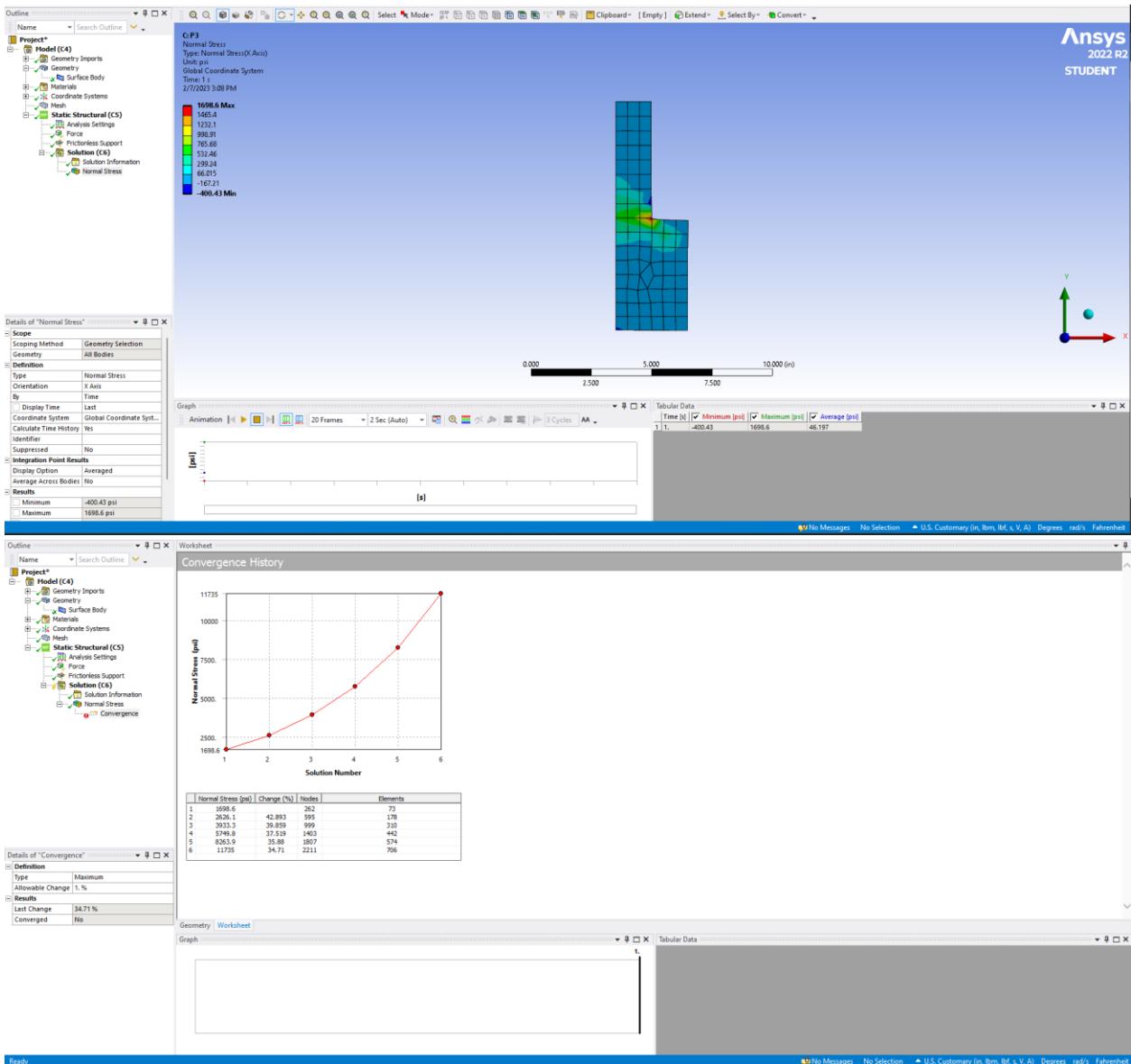
Question 3:

Consider the structural steel stepped shaft with the dimensions shown below. The shaft is subject to a uniform axial tensile load of $P = 7200\pi \text{ lbf}$.



Simplify the model as a 2D axisymmetric problem. Carry out the following analyses. Please make any required reasonable assumptions (for e.g., on the geometry, analysis type, mesh parameters, convergence criteria, etc.) with justifications (*Hint: take lengths of the two sides of the shaft to be about 1.5 times the larger diameter. Explain why this is a good assumption*).

- (a) The stepped shaft designed as shown above will show a stress singularity. Where will this be seen and why? Carry out a finite element analysis with a convergence study and show that indeed this is the case. Show a figure of the convergence behavior (including number of nodes and elements) and explain what you see. Set the number of refinement loops in the convergence study judiciously. **The stress concentration is been seen in the middle because of the geometry change. I see that the values diverge because it will keep increasing due to stress singularity.**



(b) To remove the stress singularity above, add a fillet of radius 0.75 in at the appropriate location (duplicate the Static Structural analysis block and make the modification in the duplicate block). Repeat the analysis ensuring that the results have converged. Show results for and explain what you see for:

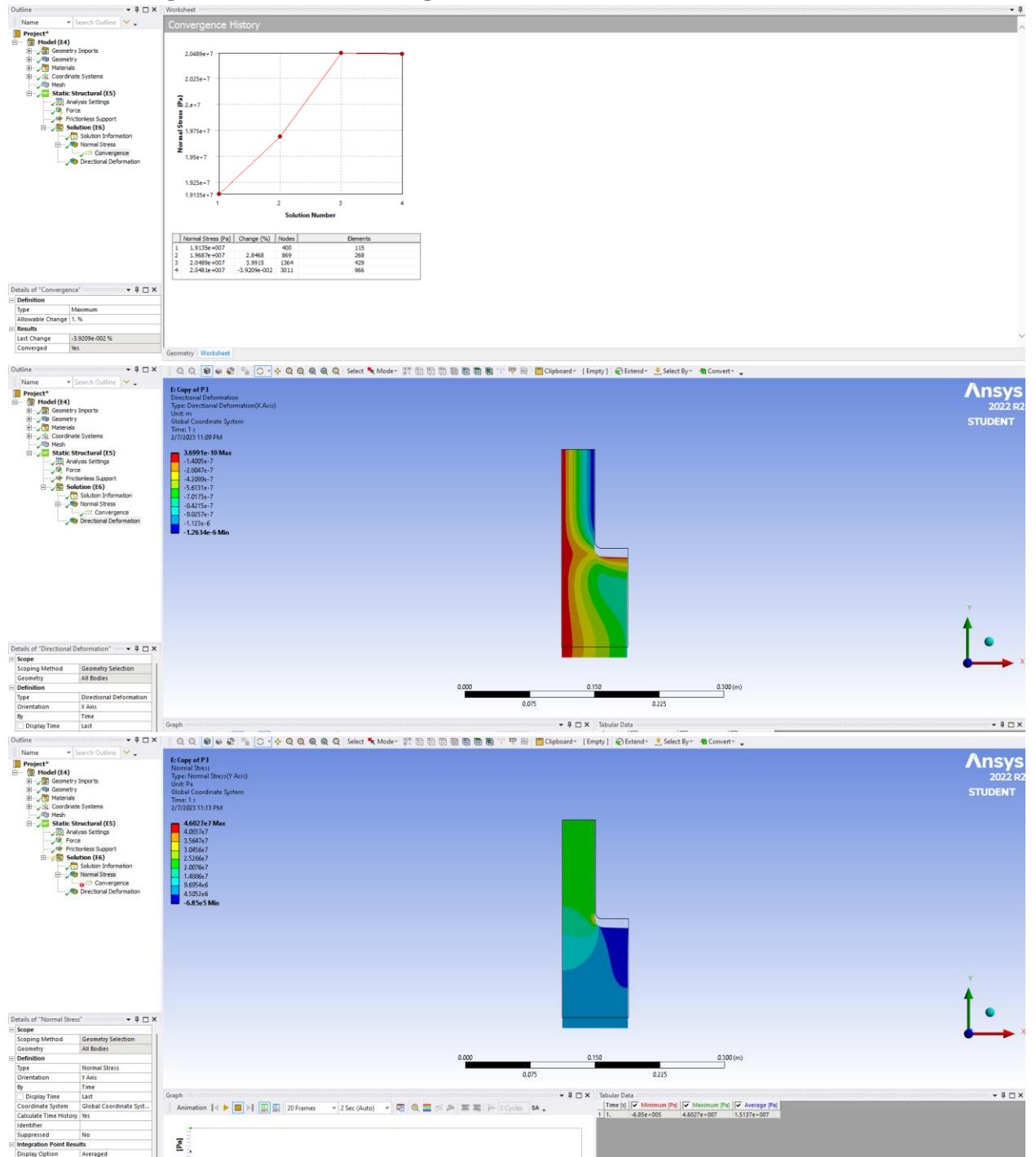
- (i) Directional deformation in the direction of loading.
The deformation goes upwards.
- (ii) Normal stress in the direction of loading.
The max stress reduced around the fillet.
- (iii) Structural error with an explanation of what it tells you.
The structural error is not as much around the fillet.

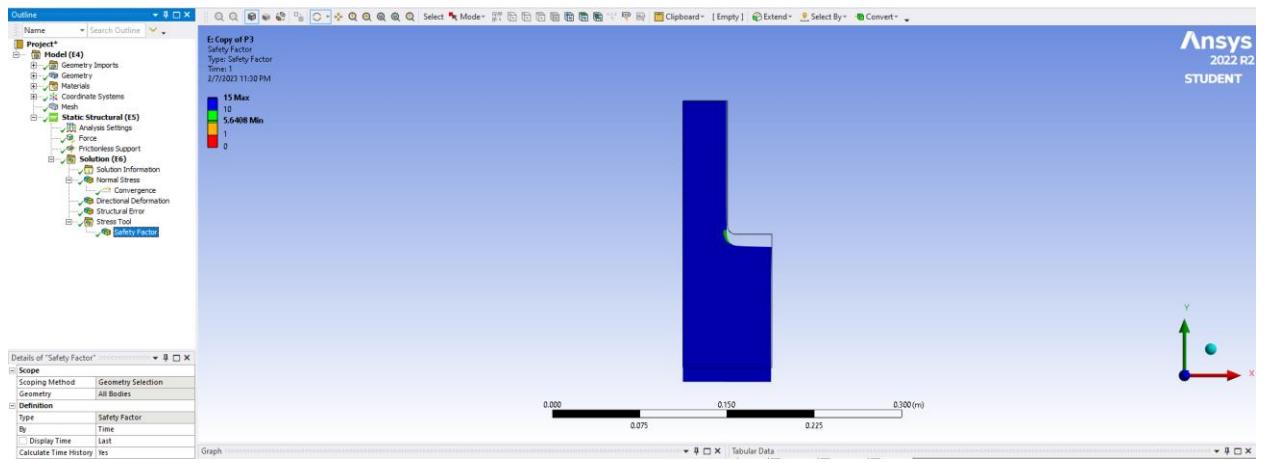
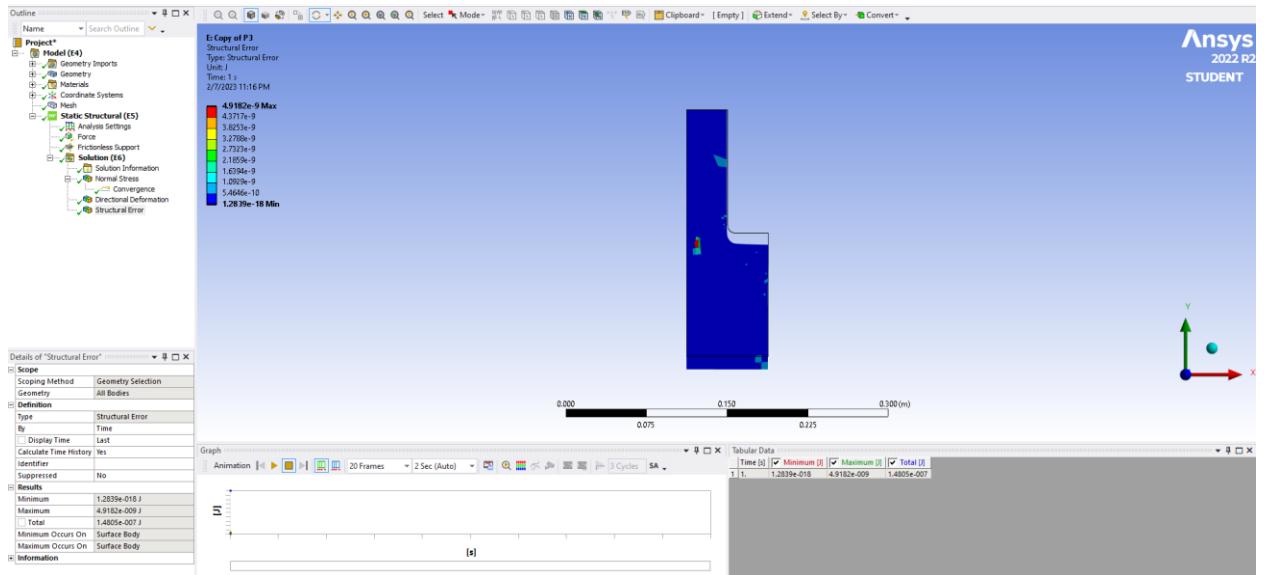
- (iv) Safety factor (under “Stress Tool”). What is the minimum safety factor for the geometry and loading given?

5.6 is the minimum safety factor.

- (v) Convergence plots with number of nodes & elements.

Values converge now instead of diverge.





- (c) Verify the results of your analysis using the stress concentration formulae given below. The given formula is accurate to within 5% - check the difference between what the formula predicts and your converged finite element result and confirm this.

where	$\sigma_{\max} = K_t \frac{4P}{\pi(D-2h)^2} \text{ where } K_t = C_1 + C_2 \left(\frac{2h}{D} \right) + C_3 \left(\frac{2h}{D} \right)^2$	
	$0.25 \leq h/r \leq 2.0$	$2.0 \leq h/r \leq 20.0$
C_1	$0.927 + 1.149\sqrt{h/r} - 0.086h/r$	$1.225 + 0.831\sqrt{h/r} - 0.010h/r$
C_2	$0.011 - 3.029\sqrt{h/r} + 0.948h/r$	$-1.831 - 0.318\sqrt{h/r} - 0.049h/r$
C_3	$-0.304 + 3.979\sqrt{h/r} - 1.737h/r$	$2.236 - 0.522\sqrt{h/r} + 0.176h/r$
C_4	$0.366 - 2.098\sqrt{h/r} + 0.875h/r$	$-0.630 + 0.009\sqrt{h/r} - 0.117h/r$

$$C_2 = 0.011 - 3.029\sqrt{2} - 0.948(2) \rightarrow C_2 = -6.17$$

$$C_3 = -3.04 + 3.979\sqrt{2} - 1.737(2) \rightarrow \underline{\underline{C_3 = 1.85}}$$

$$C_4 = 0.366 + 2.098\sqrt{2} + .875(2) \rightarrow \underline{\underline{C_4 = 5.083}}$$

$$K_6 = 2.34 - 6.17 \left(\frac{2(1.5)}{6} \right) + 1.85 \left(\frac{2(1.5)}{6} \right)^2 - 5.08 \left(\frac{2(1.5)}{6} \right)^3$$

$$\underline{K_6 = -0.92}$$

$$\sigma_{max} = 0.12 \left(\frac{4(7200\pi)}{\pi(6in - 2(1.5in))^2} \right) \quad \sigma_{max} = 2944$$

$$100 \times \frac{2944 - 2970.1}{2944} = \boxed{0.8\%} \quad \checkmark$$

MEE 323 – Computer Aided Engineering II
Homework Assignment #6 – Shell Models and Assemblies

Instructions:

- Use this Word file as a template for your homework report. Add screenshots of your modeling/analysis or any required explanations below the appropriate question. Turn in the homework report (converted to PDF) on the course Gradescope before the deadline.
- Upload a copy of your ANSYS files to the appropriate assignment on the course Canvas. The uploaded ANSYS files may be used to check your work and/or ensure academic integrity.

Homework Objectives:

Learn about 2D shell models and assemblies simulations in ANSYS Workbench.

Reading Assignment:

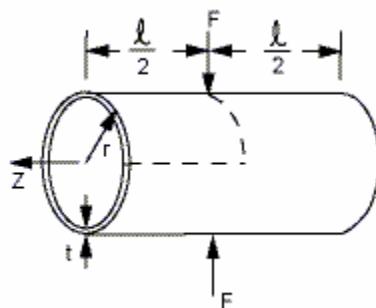
Look through **Chapter 9** in the textbook for meshing techniques and tips.

Read **Chapter 6** in the textbook for examples of surface models and simulation techniques.

ANSYS Exercises:

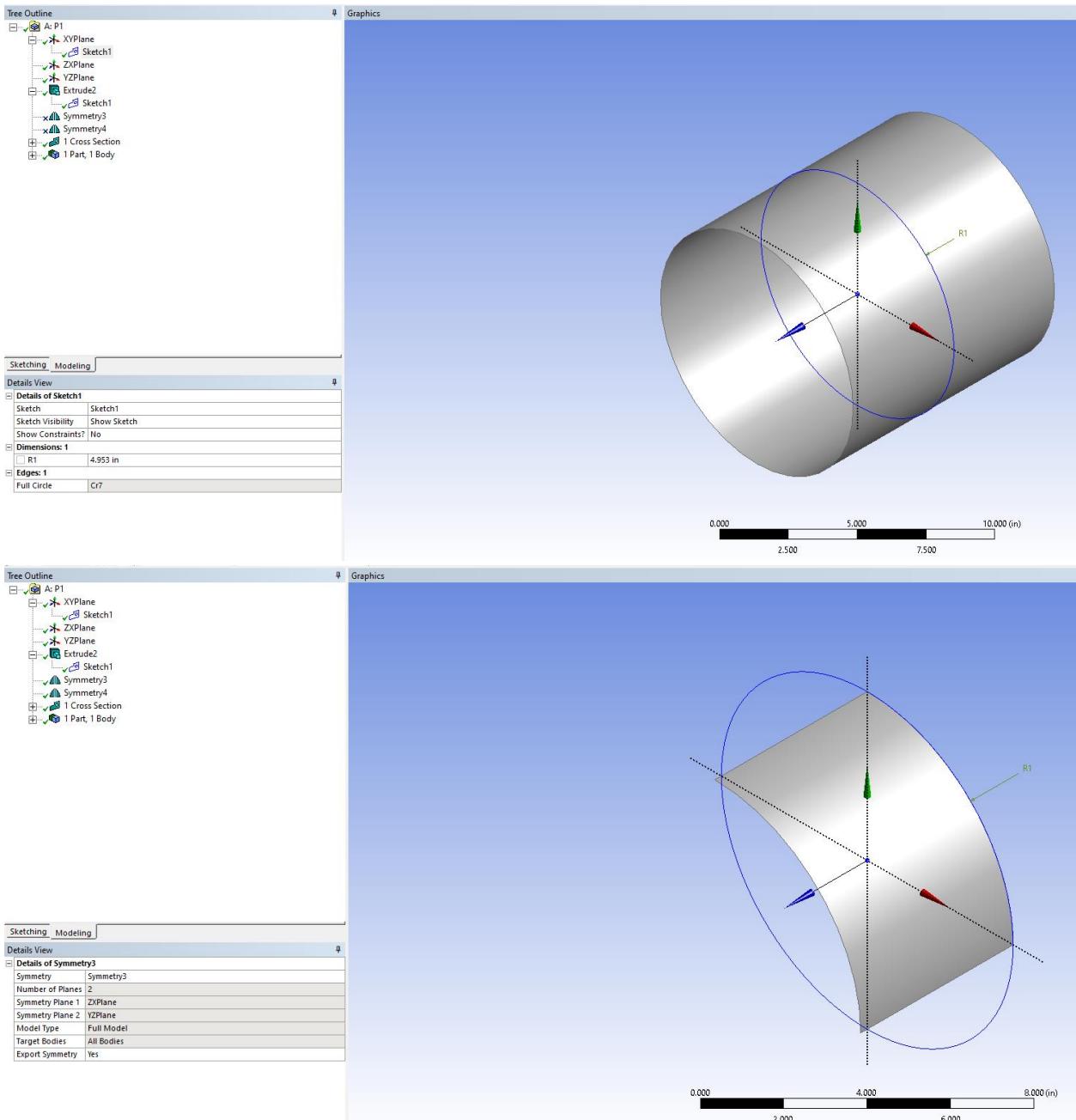
Question 1:

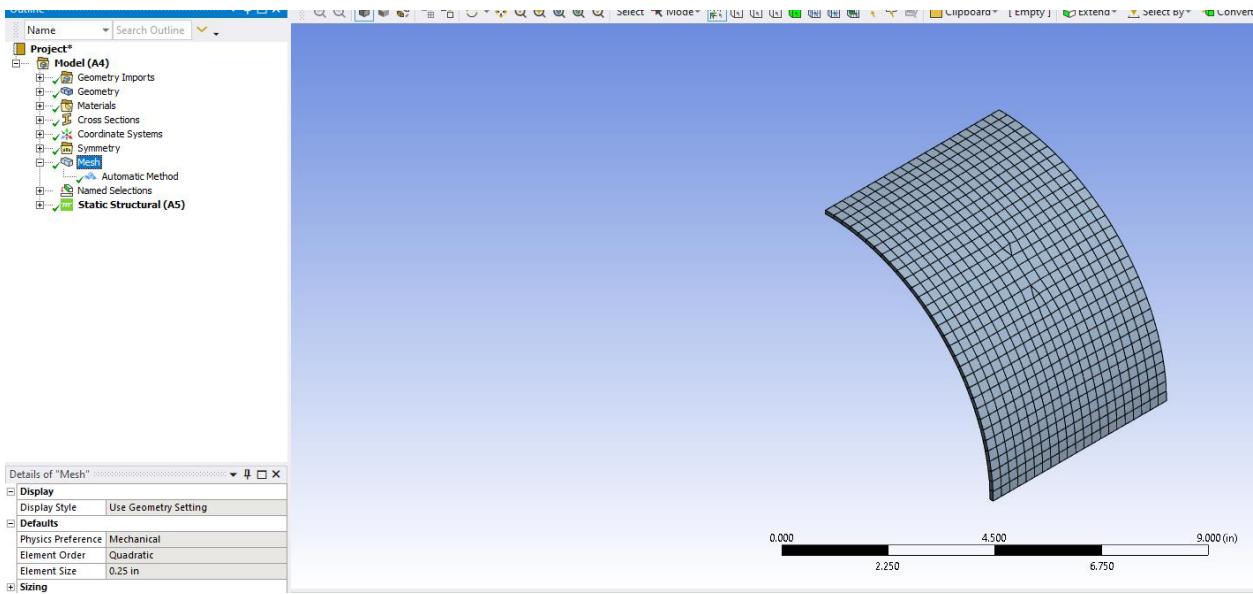
A thin-walled cylinder is pinched by a force F at the middle of the cylinder length as shown below. The cylinder dimensions and material properties are given in the table below.



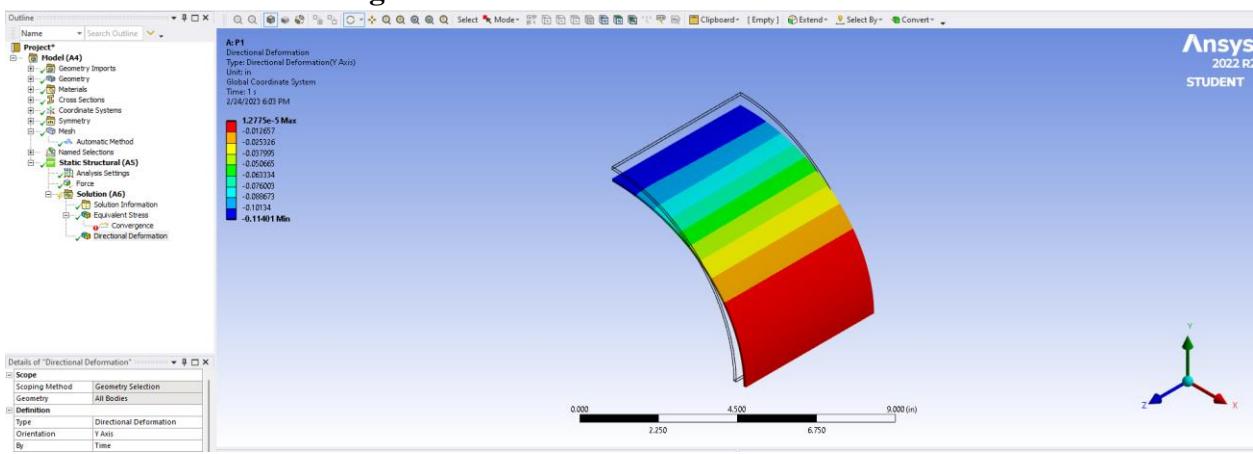
Material Properties	Geometric Properties	Loading
$E_s = 10.5e6 \text{ psi}$ $\nu_s = 0.3125$	$l = 10.35 \text{ in}$ $r = 4.953 \text{ in}$ $t = 0.094 \text{ in}$	$F = 100 \text{ lbf}$

- (1) Model the thin-walled cylinder as a shell model (2D model / 3D simulation). Use the three planes of symmetry in the model to cut the model down to a one-eighth symmetry model. Make sure to scale the load appropriately to one-fourth of the original value. Show a figure of the final geometry and the symmetry conditions. Mesh the model using appropriate mesh controls to get a neat quadratic mesh on the surface body and show a figure of the mesh.

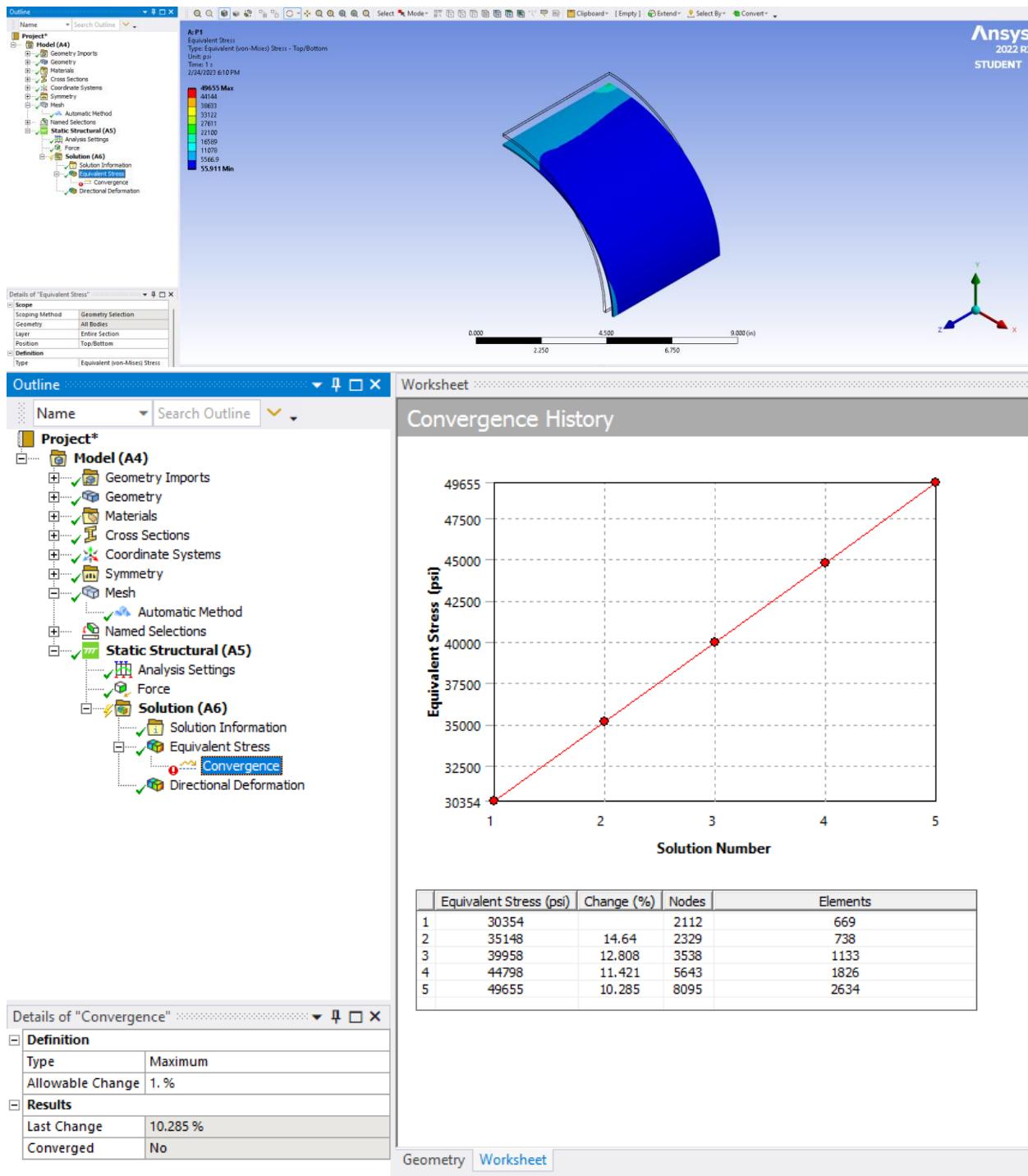




- (2) Determine the radial displacement δ at the point where the load is applied. Show a figure of the total displacement. Check the solution (calculate a percentage difference) versus the theoretical value of displacement of -0.1139 in . You may use the beta options and graphical expansion to visualize the full cylinder (as we did in class). If you do, make sure you present both the one-eighth symmetry result as well as the graphically expanded result. **Displacement at load: -0.11401. Percentage difference: 0.096 %**

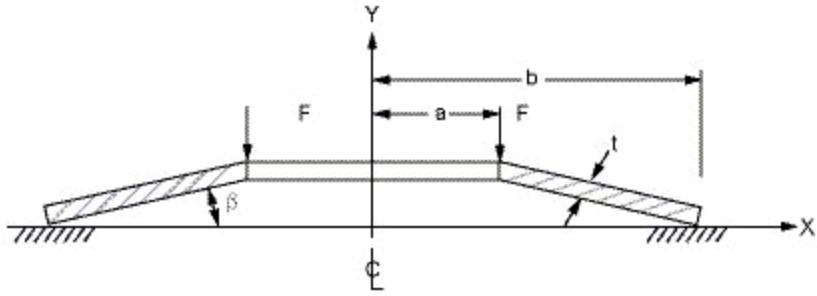


- (3) Also show a figure of the von Mises stress in the cylinder. Carry out a convergence analysis and check if the solution converges. If it does not, what could be the issue? **It does not converge because the point at the load is causing the stress to increase.**



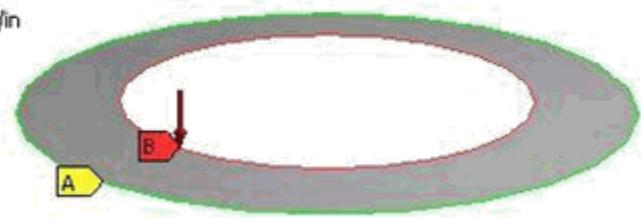
Question 2:

The conical ring shown below represents an element of a Bellville spring. The spring dimensions and material properties are given in the table below.



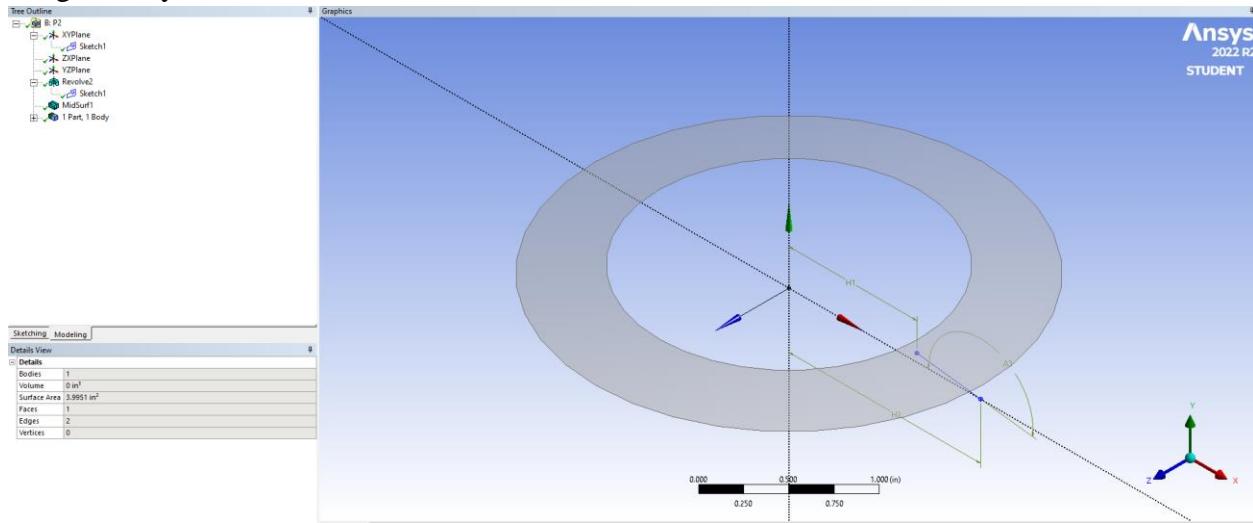
A Displacement Components: Free, 0., Free in

B Line Pressure: 100. lbf/in

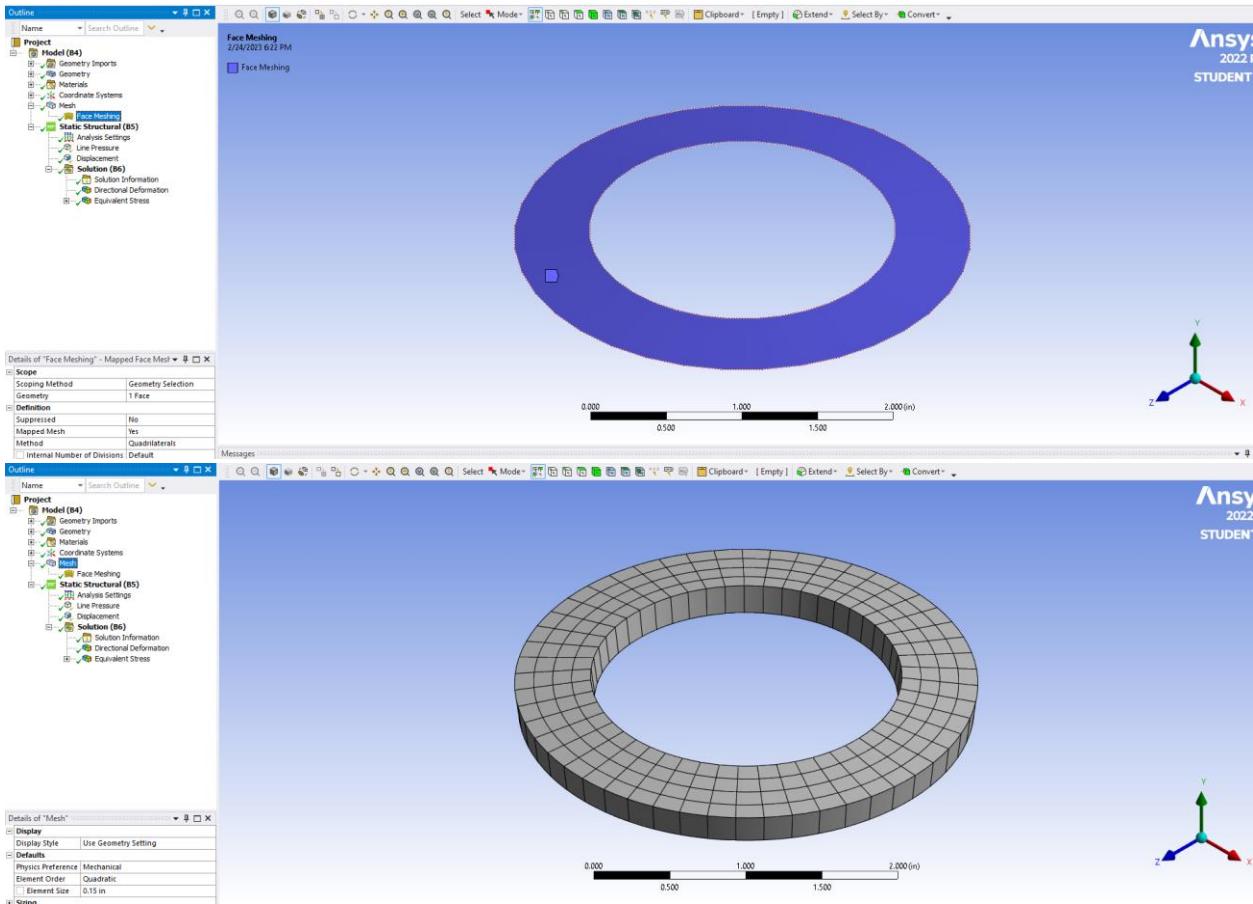


Material Properties	Geometric Properties	Loading
$E = 30e6 \text{ psi}$ $\nu = 0.0$	$a = 1 \text{ in}$ $b = 1.5 \text{ in}$ $t = 0.2 \text{ in}$ $\beta = 7^\circ = 0.12217 \text{ rad}$	Line pressure $= -100 \text{ lb/in}$

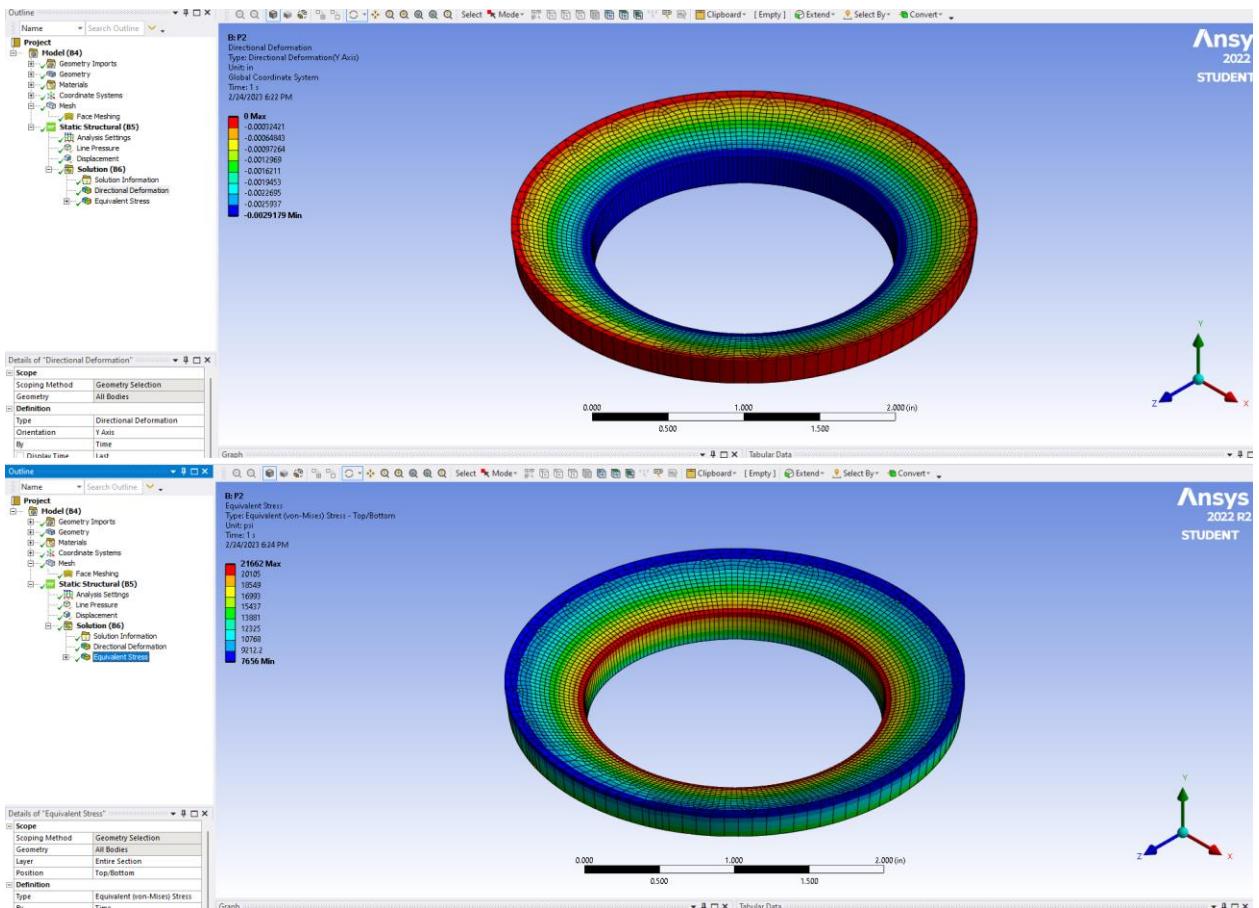
- (1) Model the Bellville spring as a shell model (2D model / 3D simulation) by revolving a line of appropriate dimensions. Do not use any symmetry conditions. Show a figure of the final geometry.

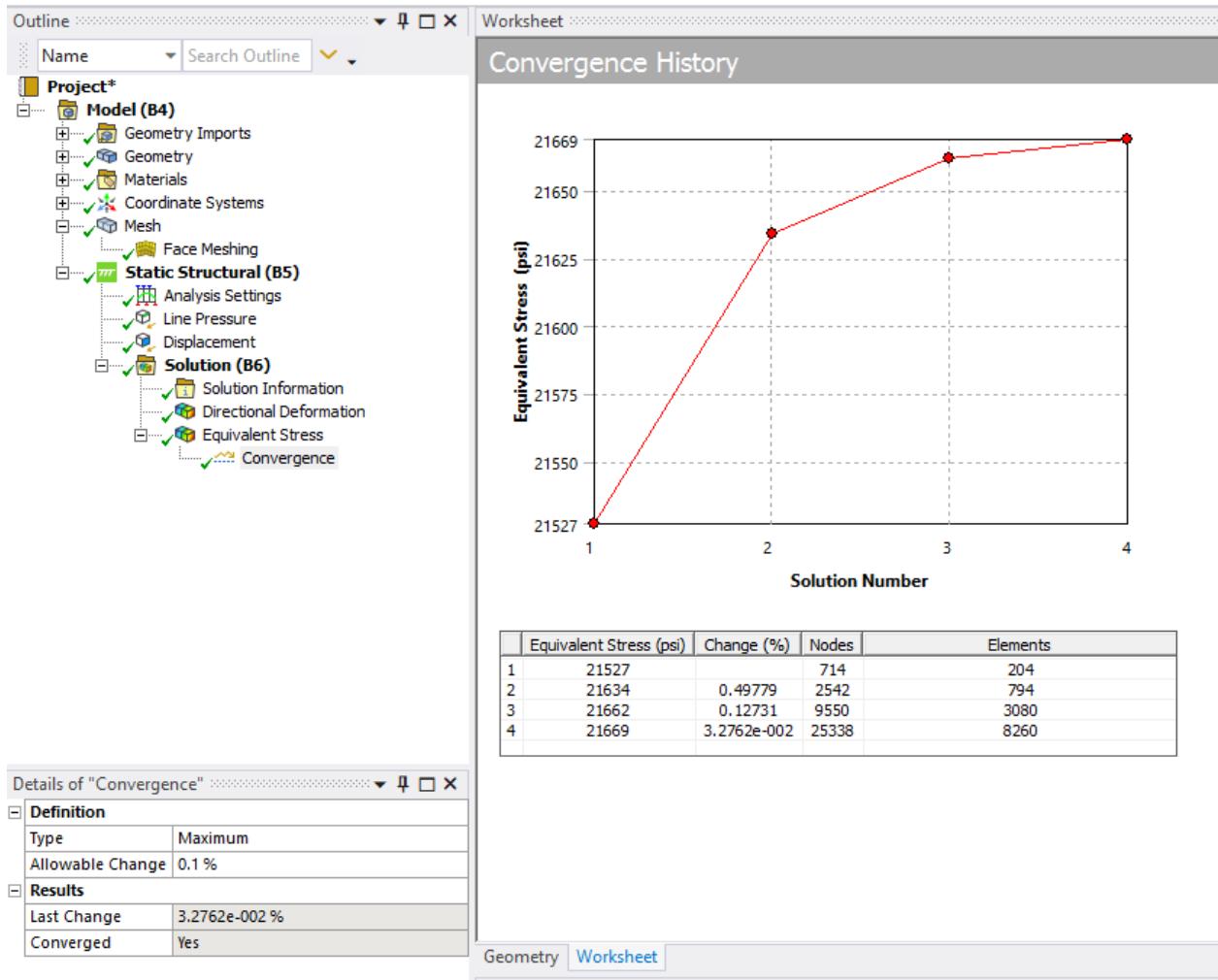


- (2) Mesh the shell body using appropriate mesh controls to get a neat quadratic mesh on the surface. Show a figure of the mesh and details of the mesh controls used to get this mesh.

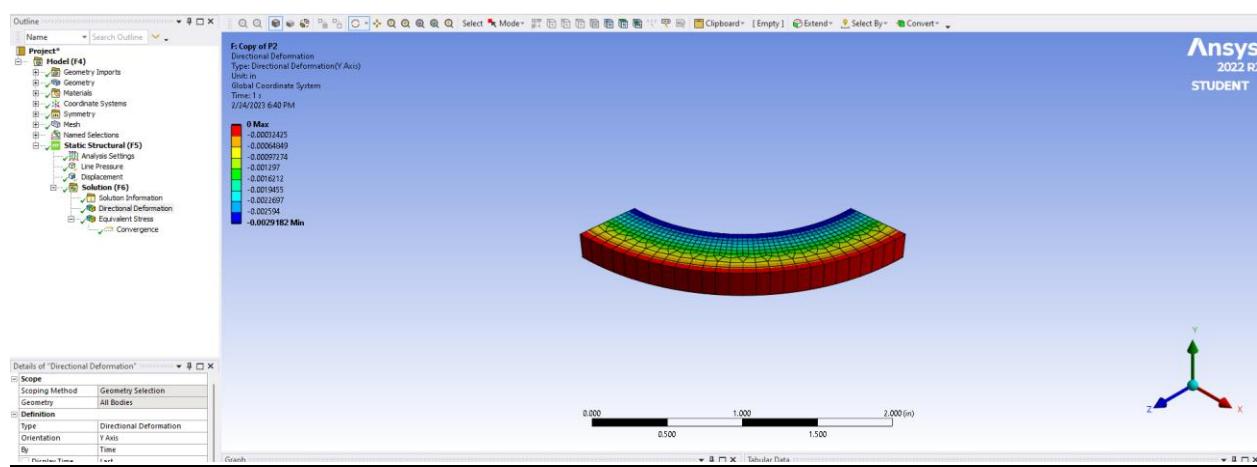
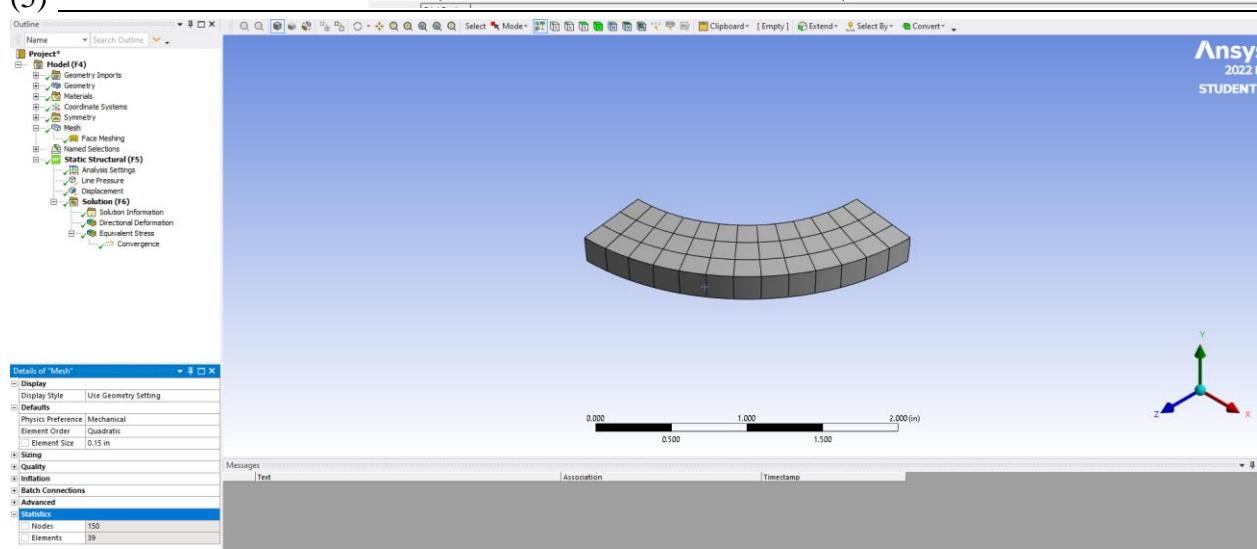
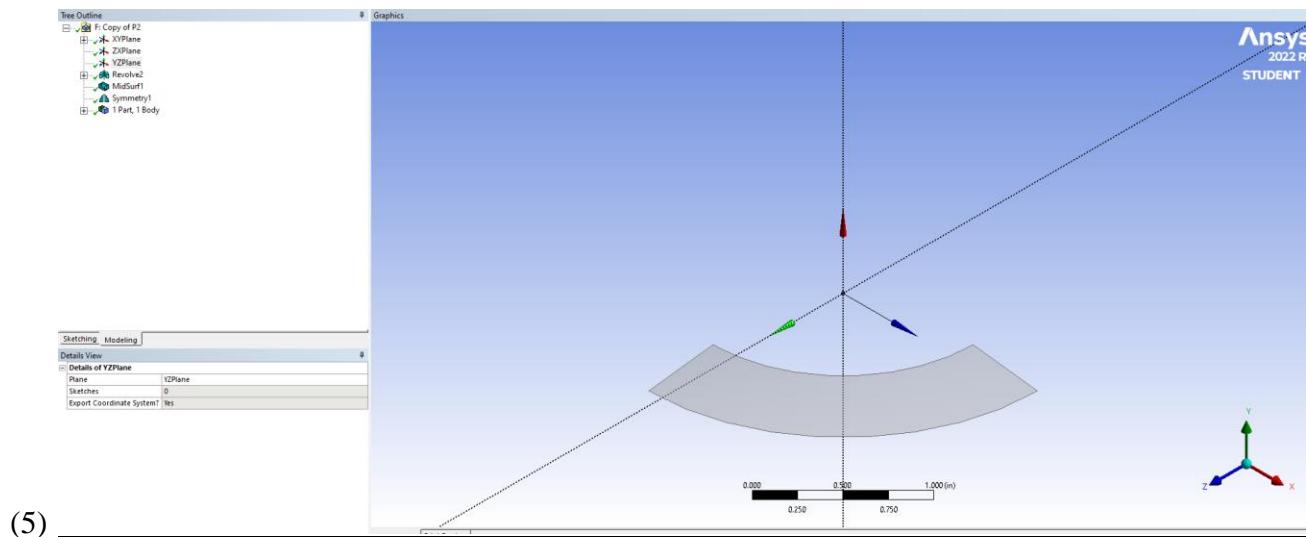


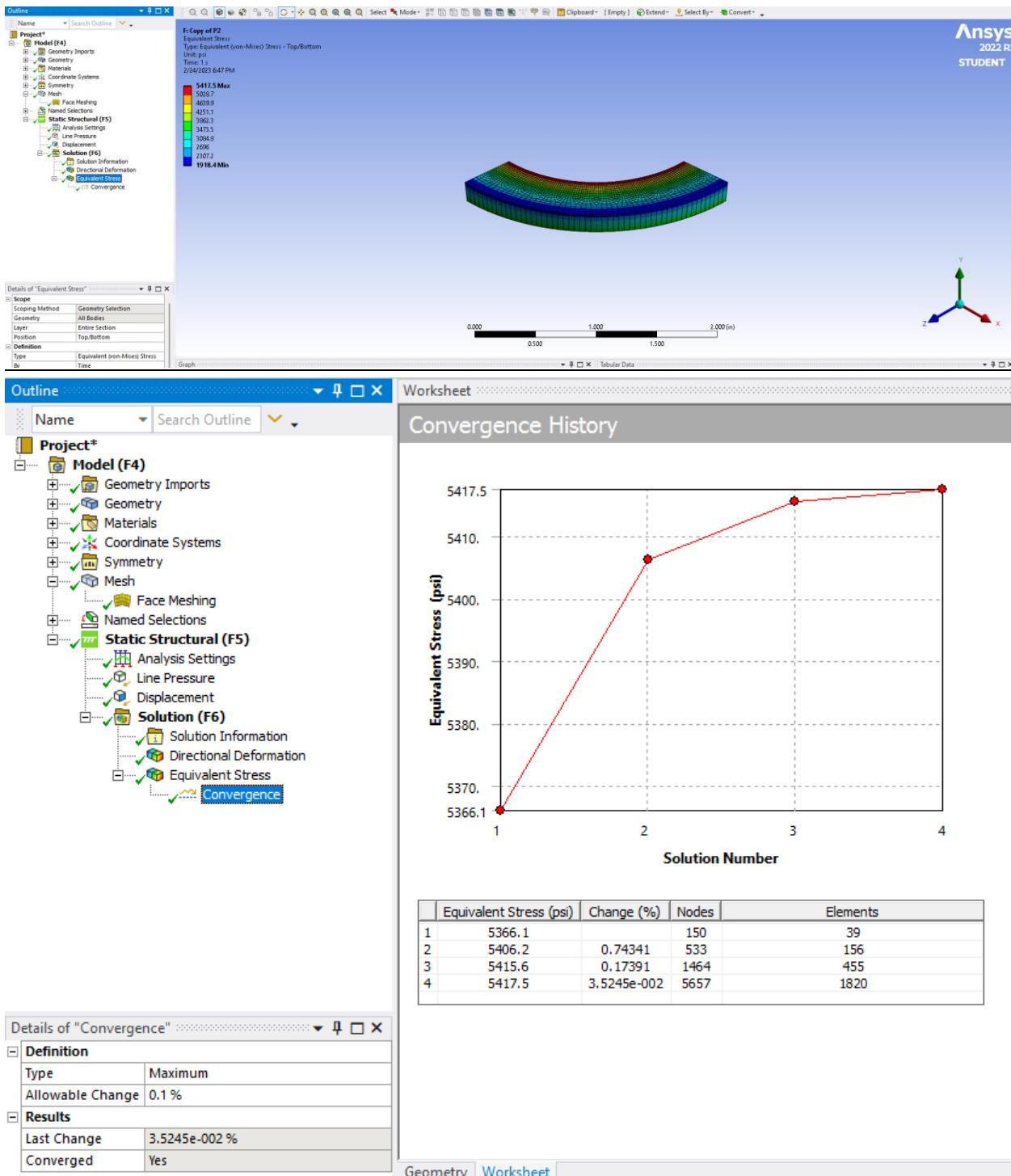
- (3) Determine the vertical displacement y of the spring when loaded with a downward ($-y$ direction) line pressure of 100 lbf/in. Show a figure of the directional displacement – is the displacement of the spring actually as large as the figure shows it to be? Check the solution (calculate a percentage difference) versus the theoretical value of displacement of -0.00282 in . Also show a figure of the von Mises stress and carry out convergence to check if the solution converges. **Displacement: -0.0029179. Percentage difference: 3.47%**
The displacement of the spring is not as large as the figure shows.





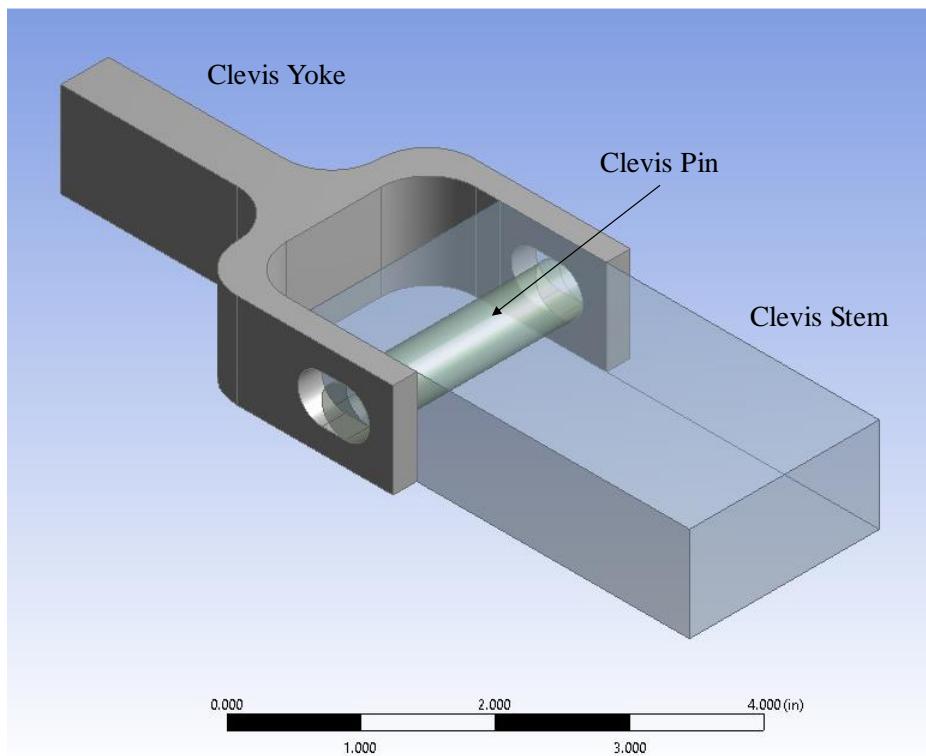
- (4) Use symmetry to reduce the model to a one-fourth symmetry model and repeat the analysis. Compute the reduction in the number of nodes and elements and calculate a percentage difference in the final deformation solution. **Nodes: 150. Number of Nodes Reduction: $714 - 150 = 564$ Nodes Reduced.**
Final Deformation: -.0029182 Percentage difference: 3.5%



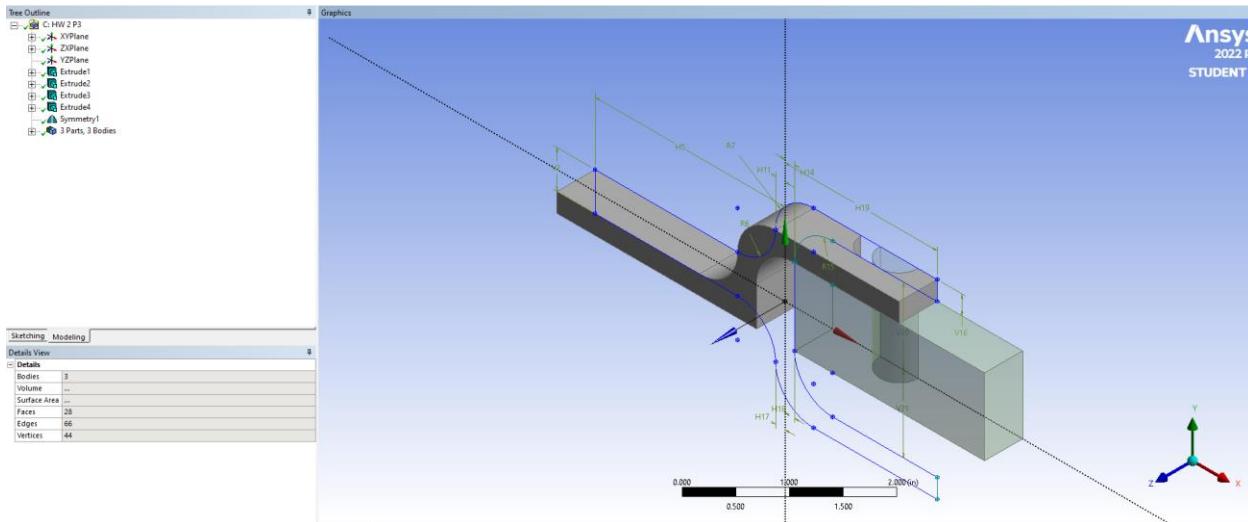


Question 3:

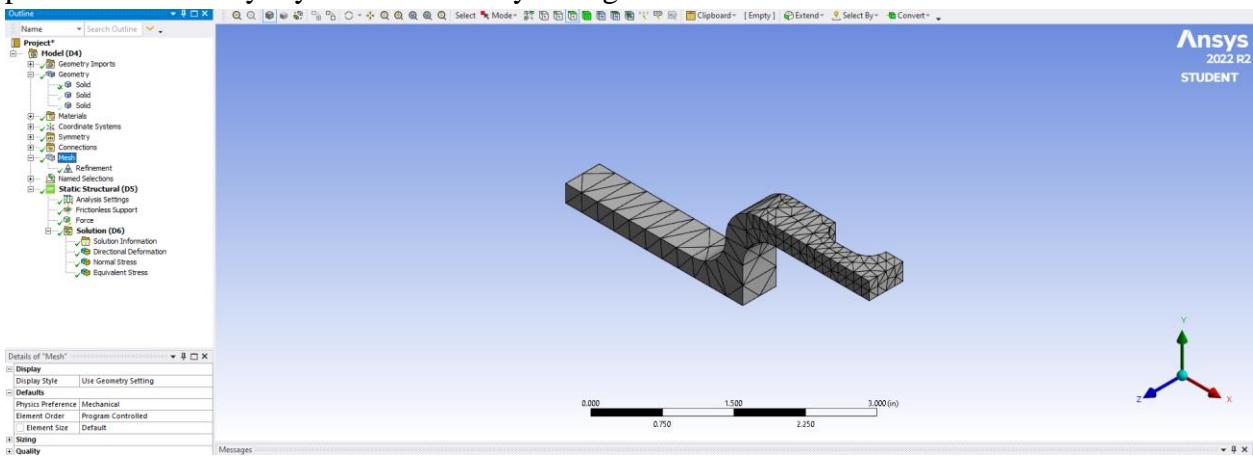
Consider the clevis assembly shown below. You may use the clevis yoke model you created in homework #2. The rectangular stem is 4 inches long, 2 inches wide, and 1 inch thick and contains a circular through hole of diameter 0.5 inch. The 0.5 inch diameter pin is 2.5 inches long and connects the clevis and stem. All components are made of structural steel and all contacts are frictional with a coefficient of friction of 0.2.

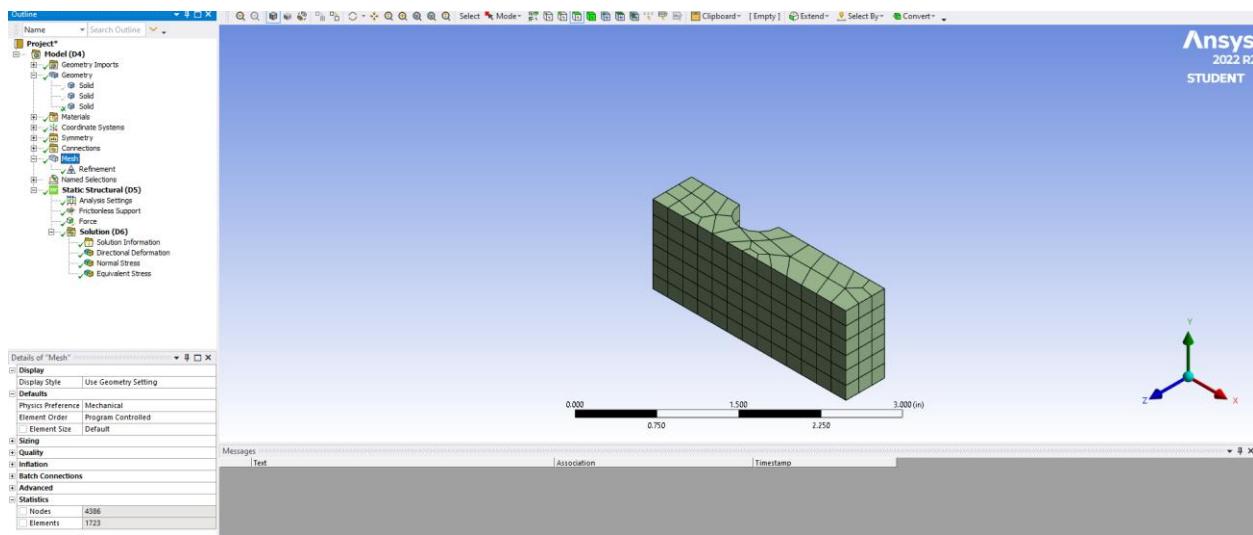
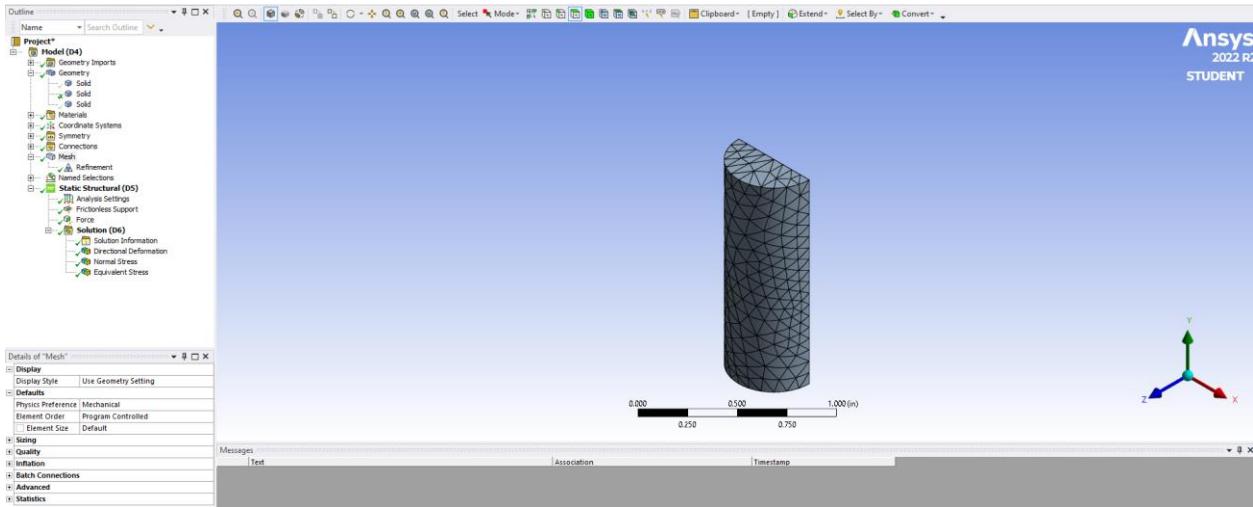


- (1) Create the clevis pin and stem as frozen bodies. Use the 2 planes of symmetry in the model to create a one-fourth model of the assembly. Show a figure of the final symmetric assembly geometry.

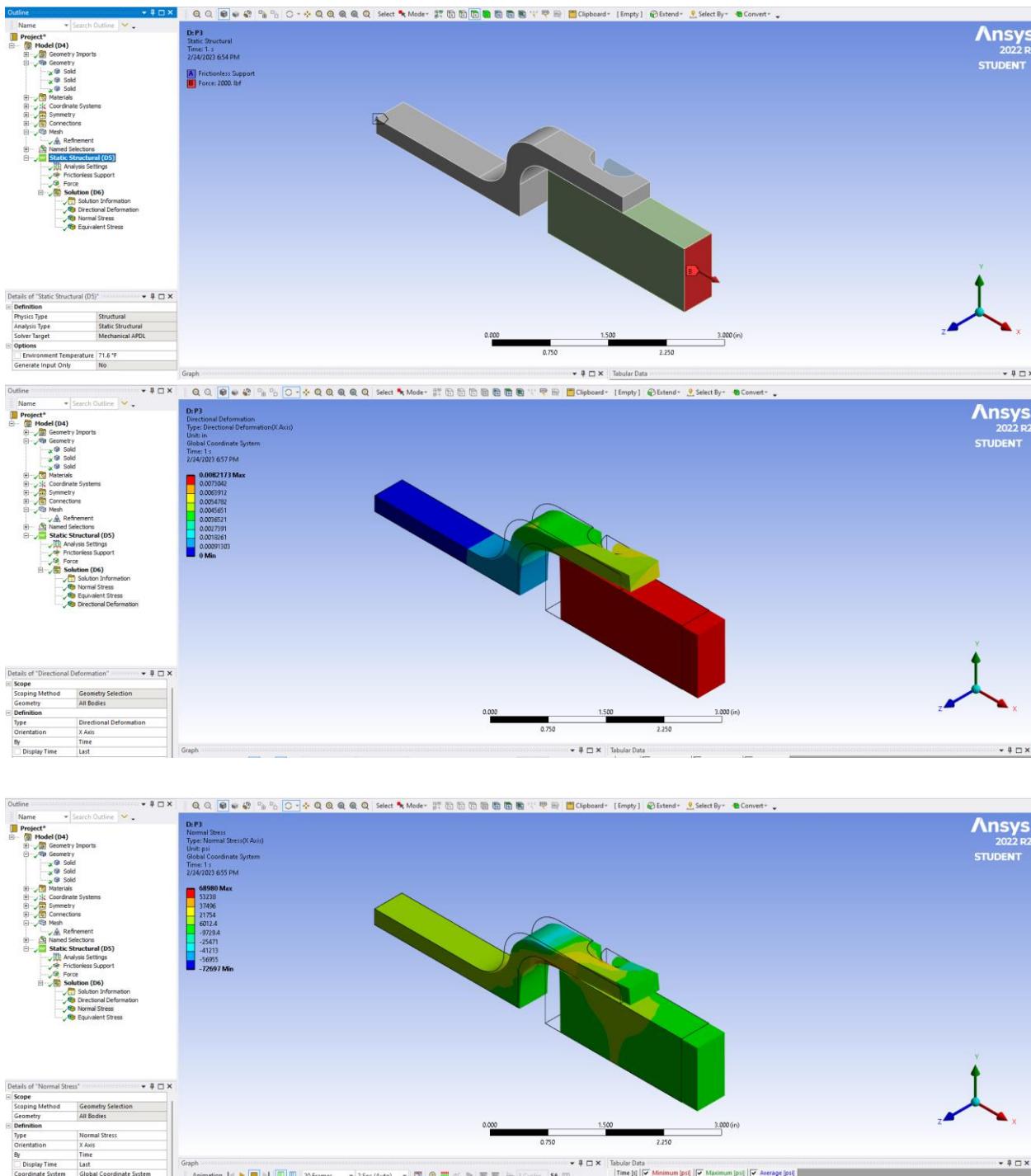


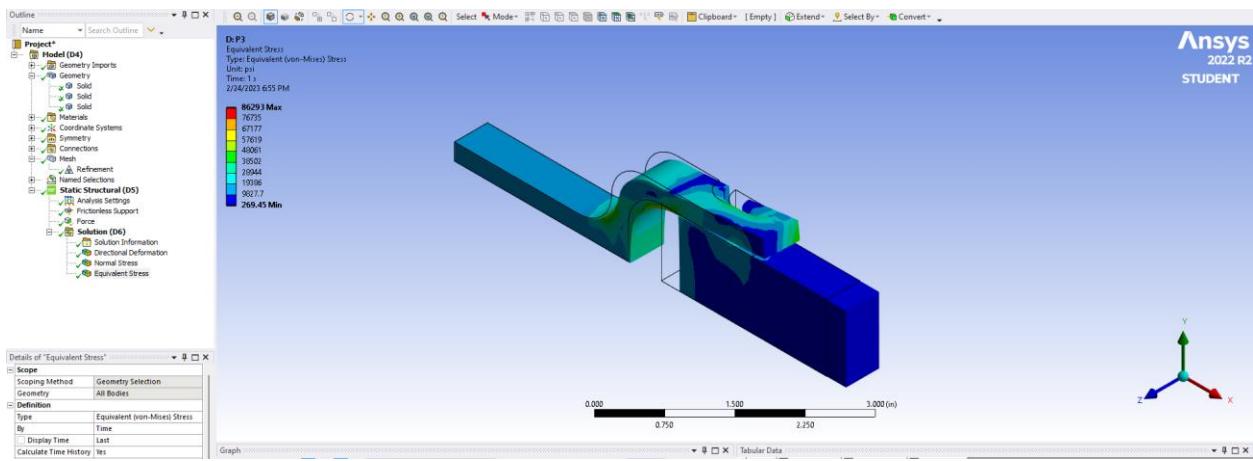
- (2) Set all contact conditions to frictional contacts with a coefficient of friction of 0.2. Mesh the model including “Refinement” conditions for each contacting surface (you will have 3 contacting surfaces – one on each body of the assembly). Show figures of the mesh of each part in the assembly – you can do this by hiding individual bodies.



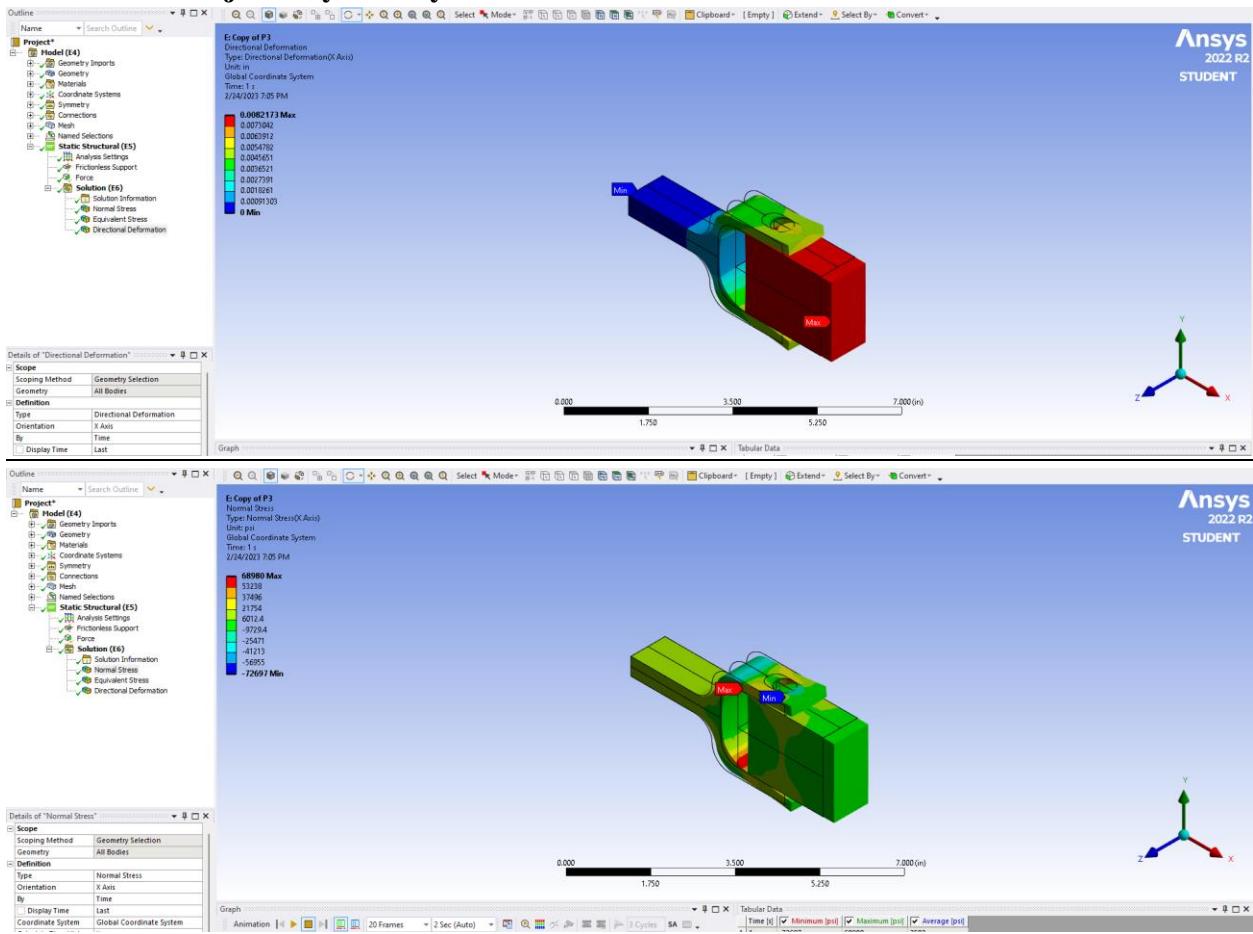


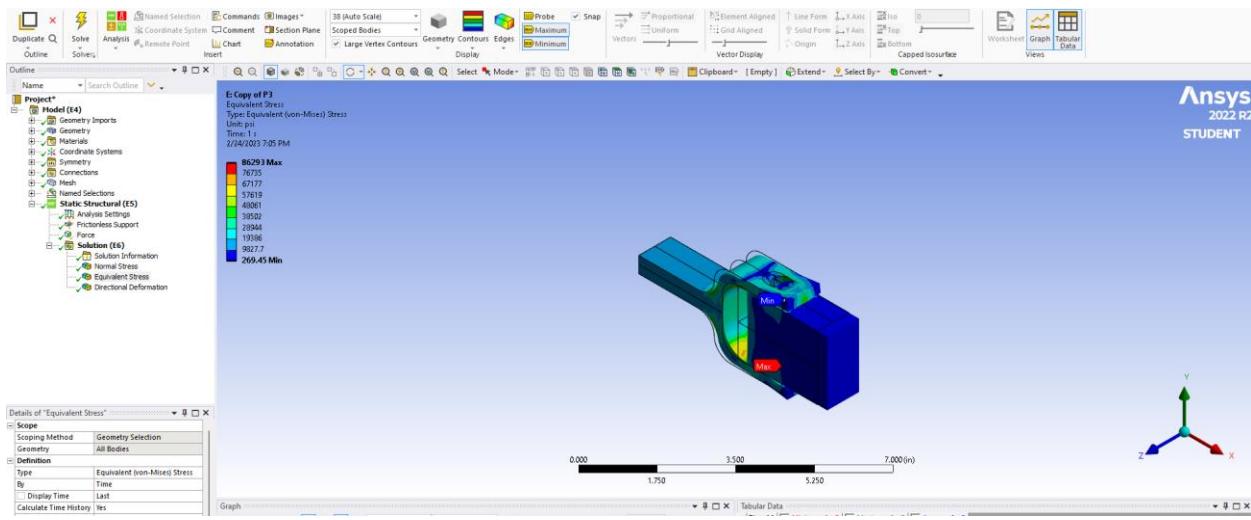
(3) The back end of the clevis yoke is supported by a frictionless support. The opposite end of the stem is loaded with a tensile pressure of 1000 psi (which corresponds to a tensile load of 2000 lbf). Apply the load and boundary condition and extract the directional deformation in the direction of the load, the normal stress in the direction of the load, and the von Mises stress. Show figures of all results and comment. **Directional deformation is the highest at the stem. The normal stress is highest at the inside curve of the Yoke. The equivalent stress is highest at the pin.**





(4) Carry out the same simulation with the full model instead of the symmetric model. Make sure to turn on the weak springs in “Analysis Settings” to make sure ANSYS takes care of the rigid body motions. Compare the results of the full analysis with the symmetric model – what differences do you see and does the full model accurately predict the deformations and stresses? **There is no difference in the models, only that now there are the 3 other parts that are being mirrored. This model can accurately predict the deformation and stresses because it is just a symmetry and it should have the same values.**





MEE 323 – Computer Aided Engineering II
Homework Assignment #7 – Line Models and Simulations

Instructions:

- Use this Word file as a template for your homework report. Add screenshots of your modeling/analysis or any required explanations below the appropriate question. Turn in the homework report (converted to PDF) on the course Gradescope before the deadline.
- Upload a copy of your ANSYS files to the appropriate assignment on the course Canvas. The uploaded ANSYS files may be used to check your work and/or ensure academic integrity.

Homework Objectives:

Learn to create and validate line models (beams and trusses).

Reading Assignment:

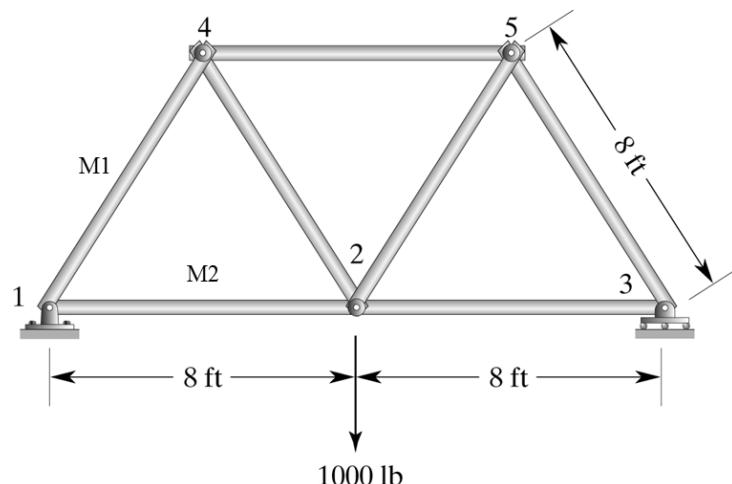
Look through **Chapter 7** in the textbook for examples of line models and simulation techniques.

ANSYS Exercises:

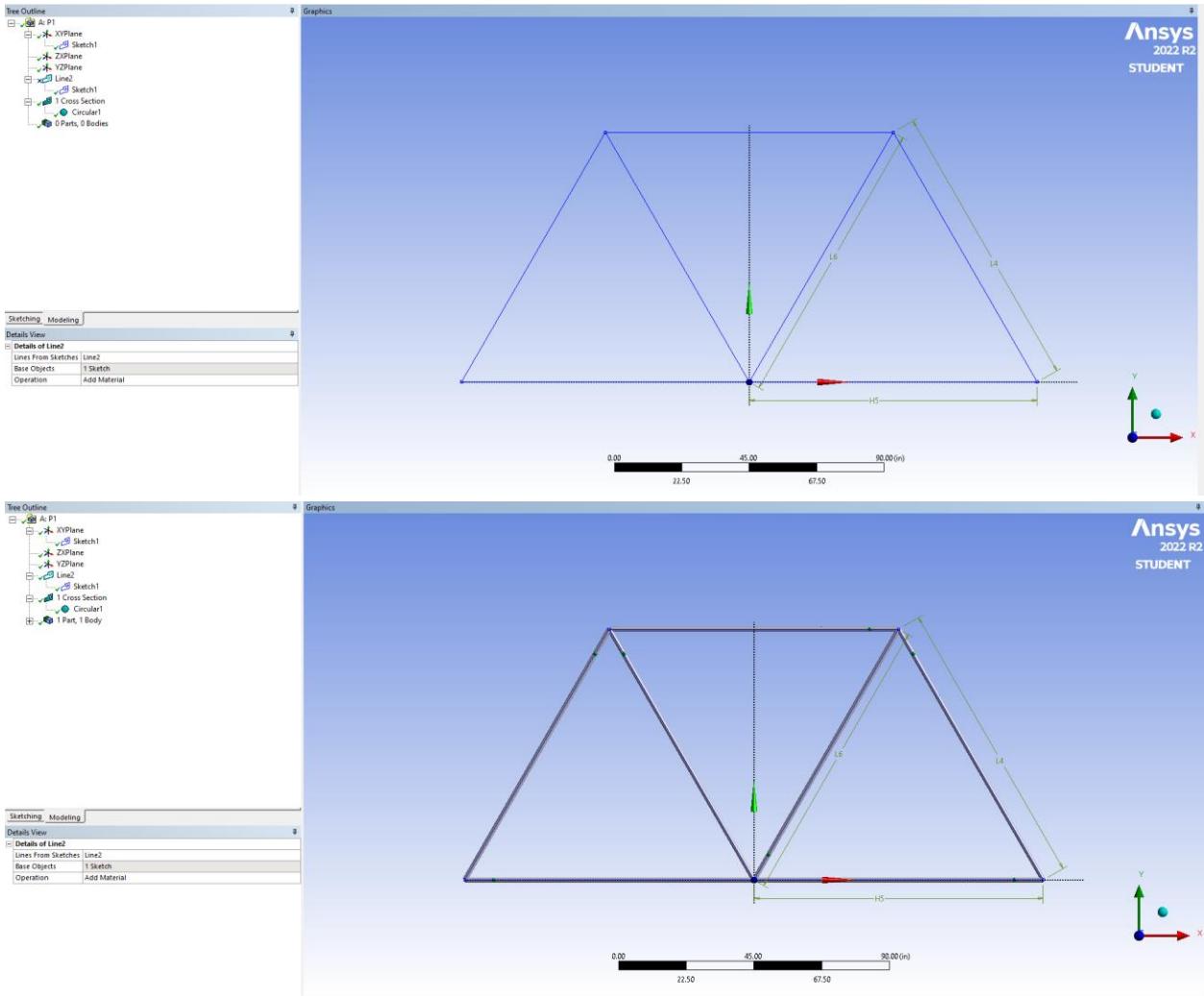
Question 1:

The members of the structural steel truss shown in the figure below have a cross sectional area of 1.96 in^2 and are made of structural steel. The left end is fixed in translation but allowed to pivot, and the right end is allowed to freely move horizontally as well as pivot (no vertical motion is allowed); use appropriate supports to model the two boundary conditions.

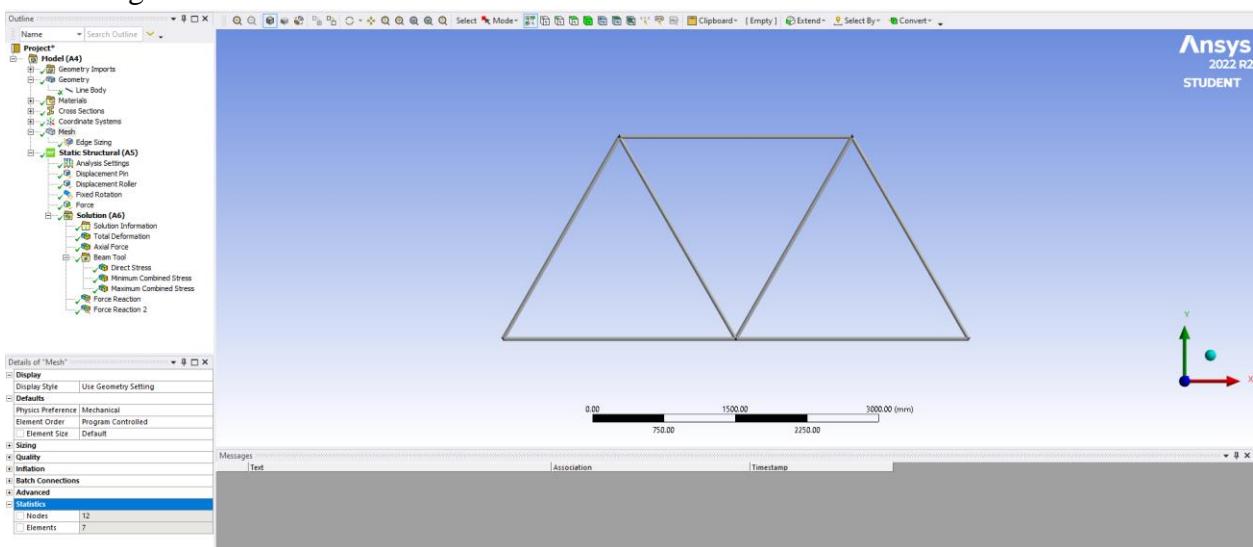
Model the truss structure in ANSYS by creating lines on a sketch and then using the "Lines from Sketches" tool to generate the line model. Use a mesh "Edge Sizing" control to ensure that each member of the truss consists of only one element (as was done in class).



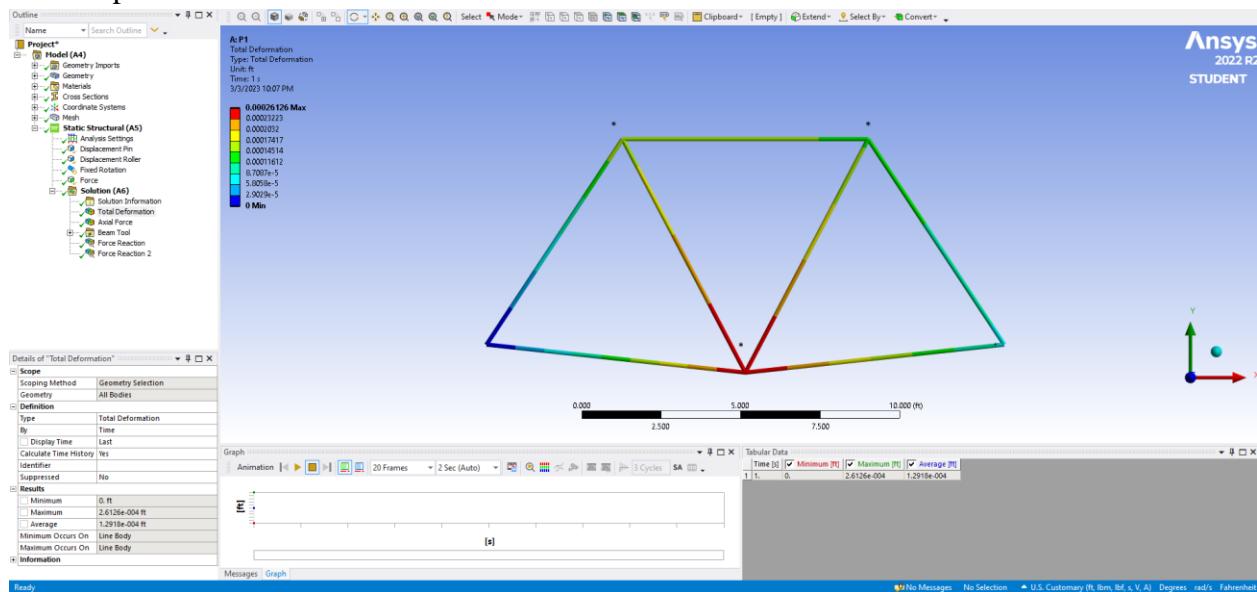
- (a) Show a figure of the sketch appropriately dimensioned and fully constrained.



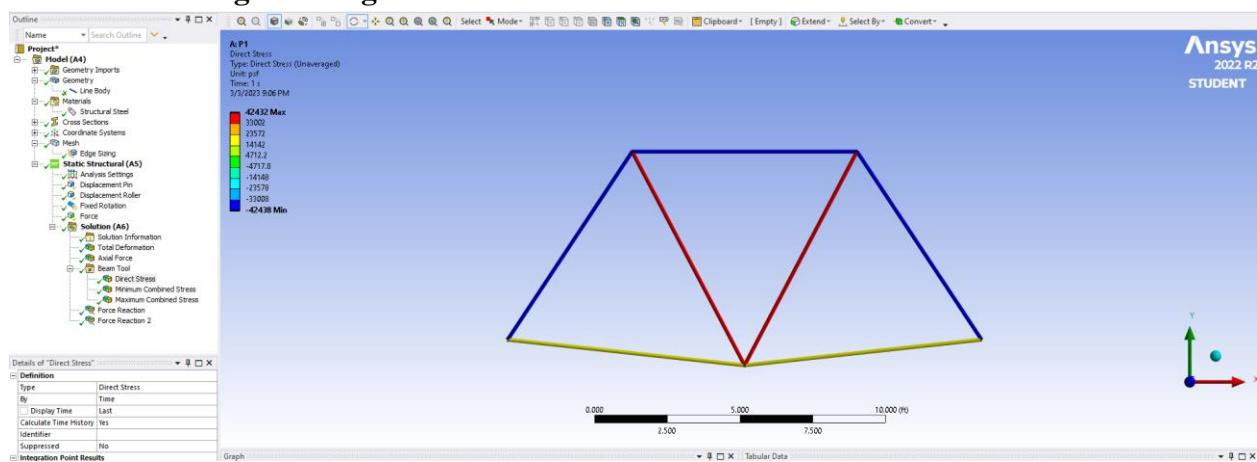
(b) Show a figure of the mesh with one element for each member of the truss.



(c) Show a plot of the total deformation with the undeformed wireframe.



(d) Insert a beam tool and show the figure of the direct stress (unaveraged) in the members of the truss. Right-click on the direct stress and choose "Export" to save the stress data in a file. Show the stress data in a table below. Will the truss support the given load without failing? What is the safety factor? Why is the safety factor so large? What other form of failure will this structure experience that justifies this high value of safety factor? **The truss support will not fail with the given load. The safety factor will be 122.9. It is so high because in this case it is not considering buckling.**



Node Number	Element Number	Direct Stress (psf)
1	1	-42438
2	1	-42438
3	2	-42434
4	3	-42438
5	4	21223
1	2	-42434
1	7	42432
2	6	21223
3	3	-42438
3	5	42432
4	4	21223
5	5	42432
5	6	21223
5	7	42432

- (e) Insert a probe and extract the reaction forces at the supports and tabulate them below. Verify these values by carrying out a global verification, i.e., sum of forces in the two directions must be zero, and the sum of the moments taken about any point on the structure must also be zero.

$$\sum F_y = R_1 + R_2 - 1000lb = 0 \Rightarrow R_1 = \underline{500 \text{ lb}}$$

$$\sum M_A = -(8ft)(1000lb) + R_2(16ft) = 0 \Rightarrow R_2 = \underline{500lb}$$

	x	y (lb)	z
Force Reaction 1	0	500	0
Force Reaction 2	0	500	0

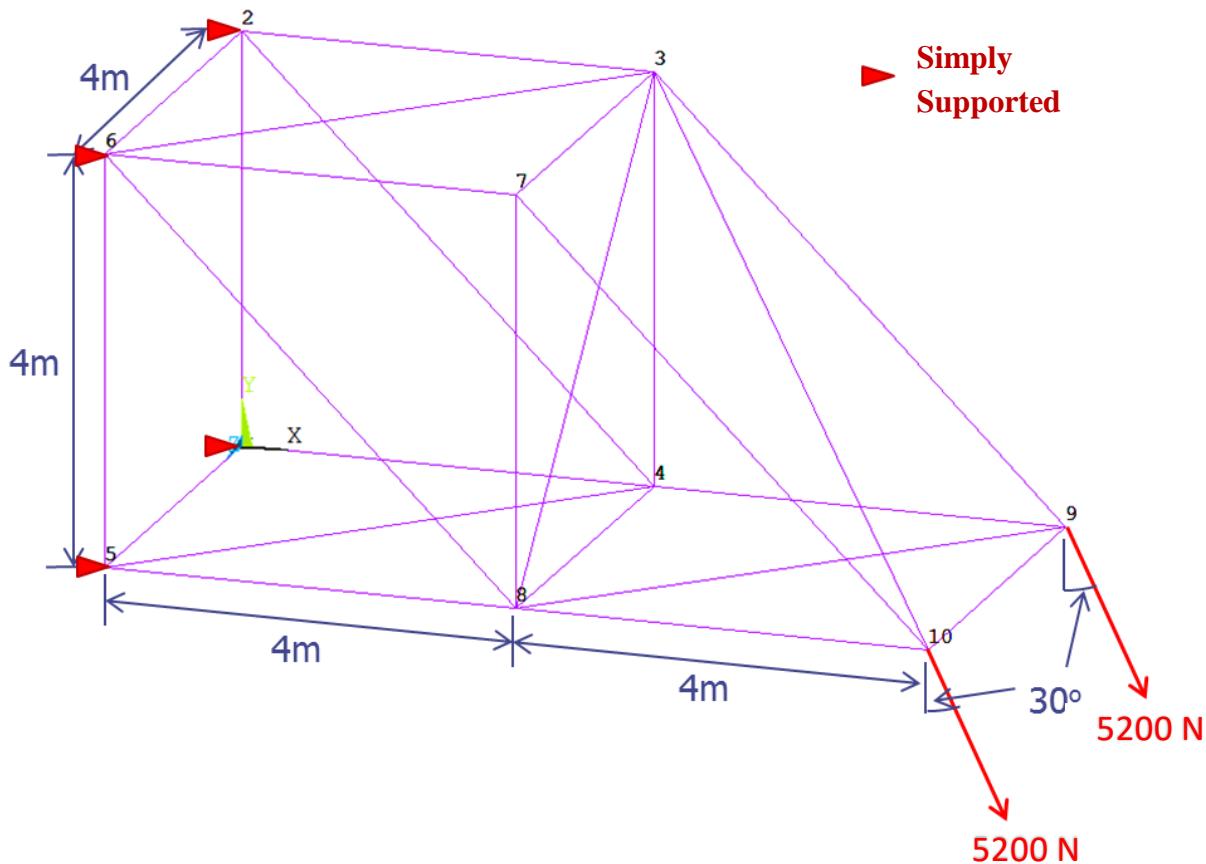
- (f) Insert a beam result to extract the axial forces in each member of the truss and tabulate them below. Use static equilibrium hand calculation (at node 1) to verify that this is indeed correct for the members M1 and M2 as labeled on the figure (remember to consider the reaction at the node as well).

Node Number	Element Number	Directional Axial Force (lbf)
1	1	-577.19
2	1	-577.19
3	2	-577.13
4	3	-577.19
5	4	288.64
1	2	-577.13
1	7	577.11
2	6	288.64
3	3	-577.19
3	5	577.11
4	4	288.64
5	5	577.11
5	6	288.64
5	7	577.11

Node 1: $-577.19 \text{ lbf} + 577.1 \text{ lbf} \approx 0 \text{ lbf}$

Question 2:

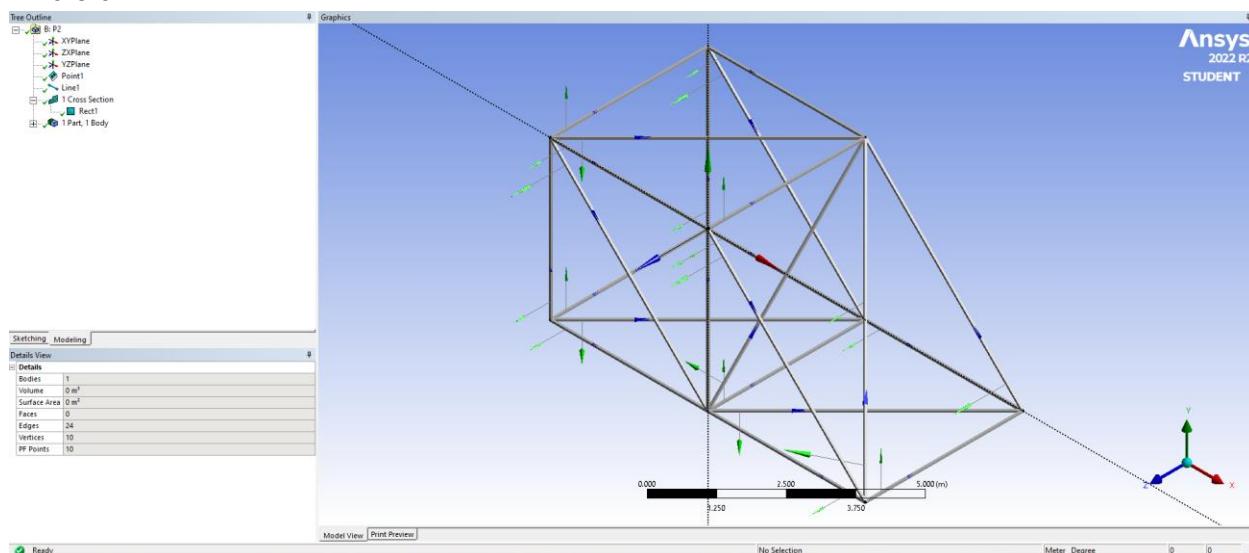
The members of the 3-D space truss shown in the figure below are made of aluminum alloy and have a square cross-sectional area of $2.50 \times 10^{-3} m^2$. The 4 left endpoints are fixed but allowed to pivot (use appropriate supports), and the two endpoints on the right end are subjected to a load of 5200 N oriented at 30° from the vertical as shown in the figure. The node numbers and coordinates are given in the following table.



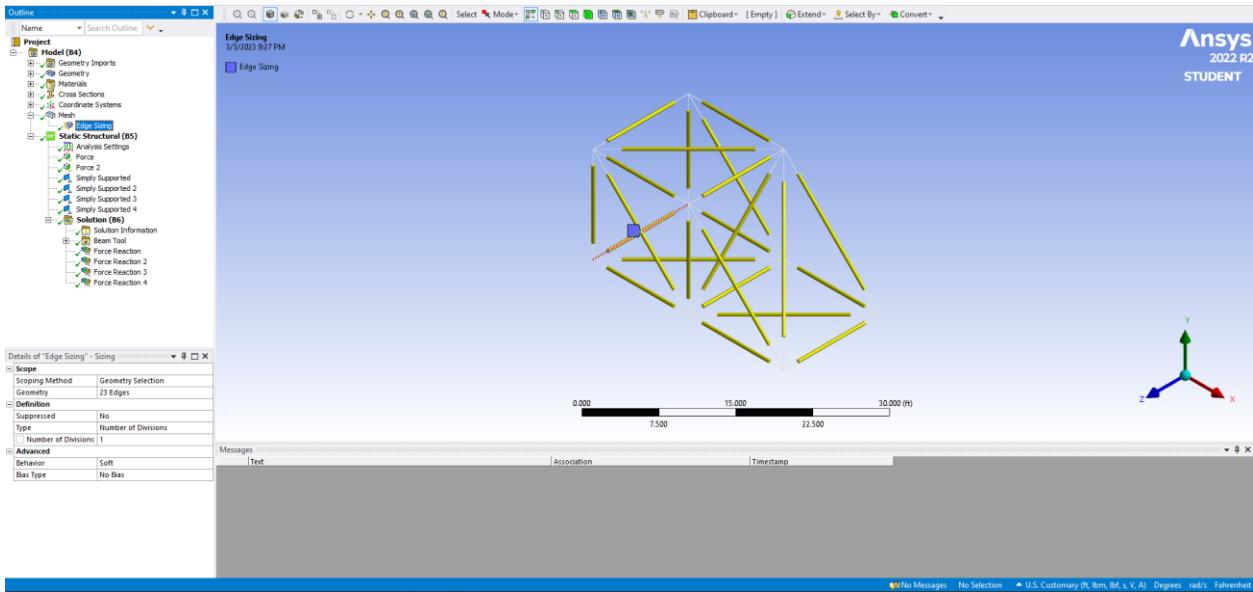
Node Number	X Coord. (m)	Y Coord. (m)	Z Coord. (m)
1	0	0	0
2	0	4	0
3	4	4	0
4	4	0	0
5	0	0	4
6	0	4	4
7	4	4	4
8	4	0	4
9	8	0	0
10	8	0	4

- (a) Create the geometry of the truss by creating a "Point" feature in Design Modeler and using the "Lines From Points" to create the truss elements. Specify a rectangular cross section for the truss members. Create a coordinates text file and read that in when you create the "Point" feature. Show your coordinates text file below (copy and paste what you have in your coordinates text file below). Also show a figure of the final geometry.

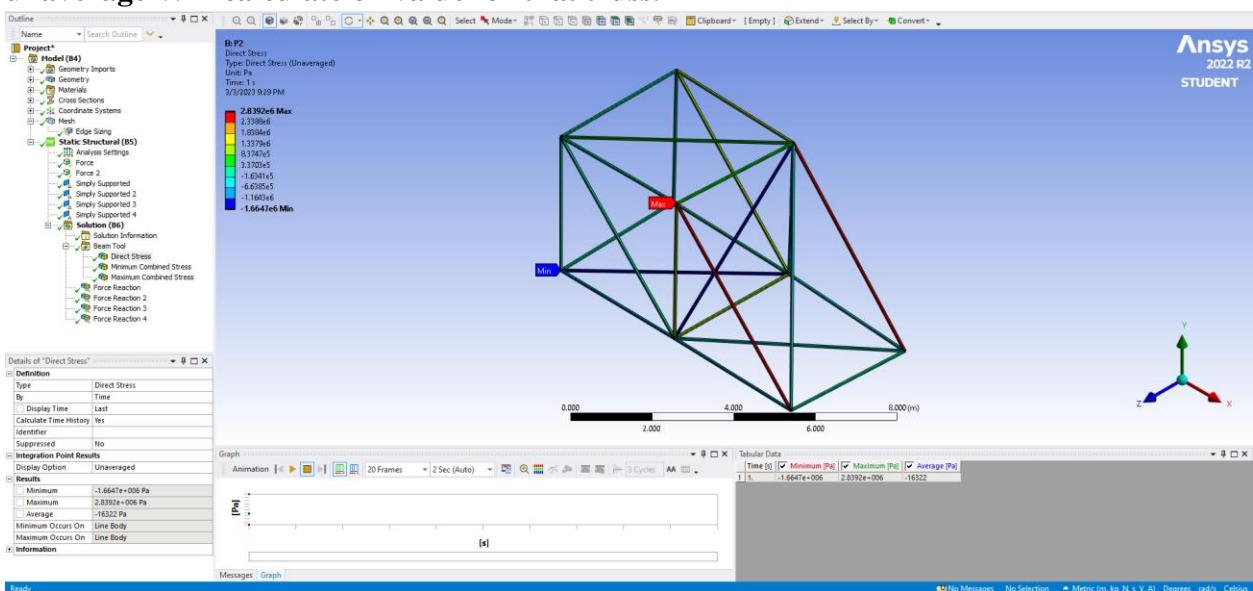
```
1 1 0 0 0
1 2 0 4 0
1 3 4 4 0
1 4 4 0 0
1 5 0 0 4
1 6 0 4 4
1 7 4 4 4
1 8 4 0 4
1 9 8 0 0
1 10 8 0 4
```



- (b) Use a mesh "Edge Sizing" control to ensure that each member of the truss consists of only one element. Show a figure of the mesh.



- (c) Insert a beam tool and show the figure of the direct stress (unaveraged) in the members of the truss. Use the "Max" and "Min" probes to show the location of the maximum and minimum stresses. Explain why we use the unaveraged stresses and not the averaged stresses. **We used unaveraged stresses because it is just on element of mesh size on each truss therefore unaverage Will calculate on value for that truss.**



- (d) Insert a probe and extract the reaction forces at the supports and tabulate them below. Carry out a global verification and make sure that the sum of forces in all three directions comes to zero. Also insert a probe and extract the reaction moments at the supports and show that they sum to zero as well (note where the reaction moment is output and make sure to use vector analysis for the moment balance equation since the moment will have components in all 3 directions).

Force(N)	x	y	z
R1	-1437	1.1197	1435.2
R2	5366.3	-0.36937	-2944.5
R3	-5020.1	7447.1	1509.2
R4	-4109.2	1558.8	0

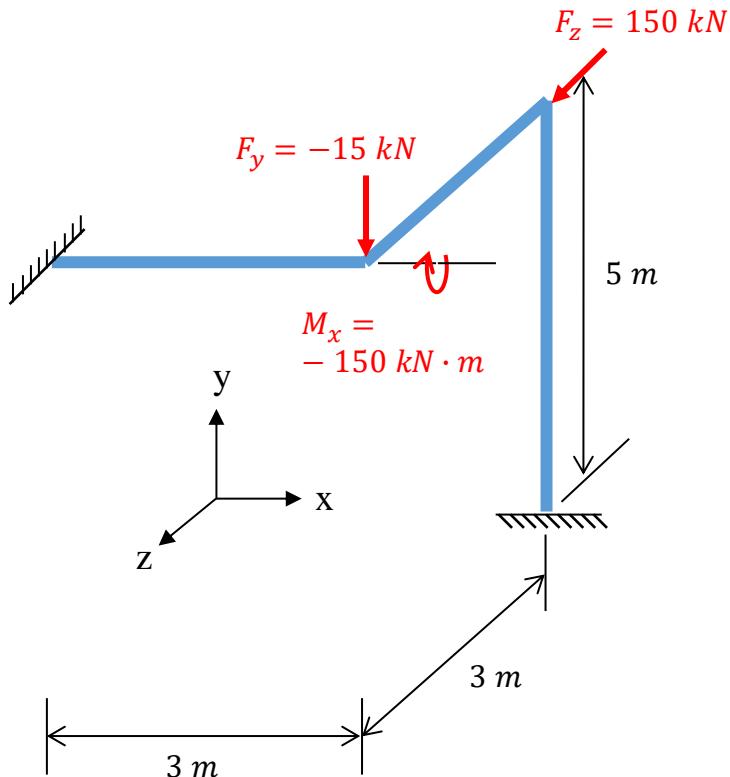
Moment (N.m)	x	y	z
R1	0	0	0
R2	0	0	0
R3	0	0	0
R4	0	0	0
Total	0	0	0

<u>x</u>	<u>y</u>	<u>z</u>
-1437	1.1191	14355.2
+5366.3	-0.3693	-2944.5
-5020.1	+7447.1	+1509.2
-4109.2	+1558.8	+0
<u>-2(5200 \sin 30)</u>	<u>-2(5200 \cos 30)</u>	<u>$\approx 0 N$</u>
<u>$\approx 0 N$</u>	<u>$\approx 0 N$</u>	<u>$\approx 0 N$</u>

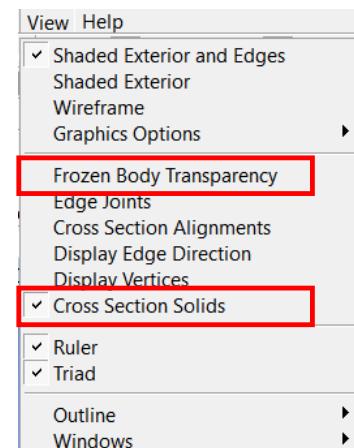
Hint: You will have to consider both reaction forces and moments to check moment equilibrium. Carefully choosing the point about which to calculate moments will make the computation simpler/faster.

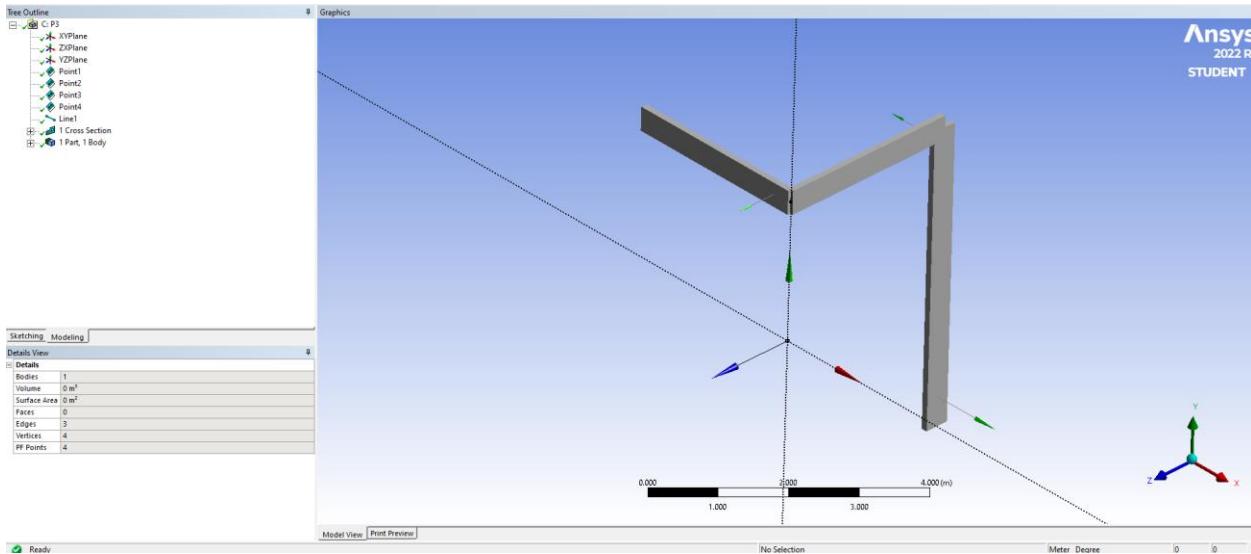
Question 3:

Consider the structural steel space load frame shown in the figure below. A frame is different from a truss in that the members are modeled as beams, and the mesh needs to have multiple elements per member.



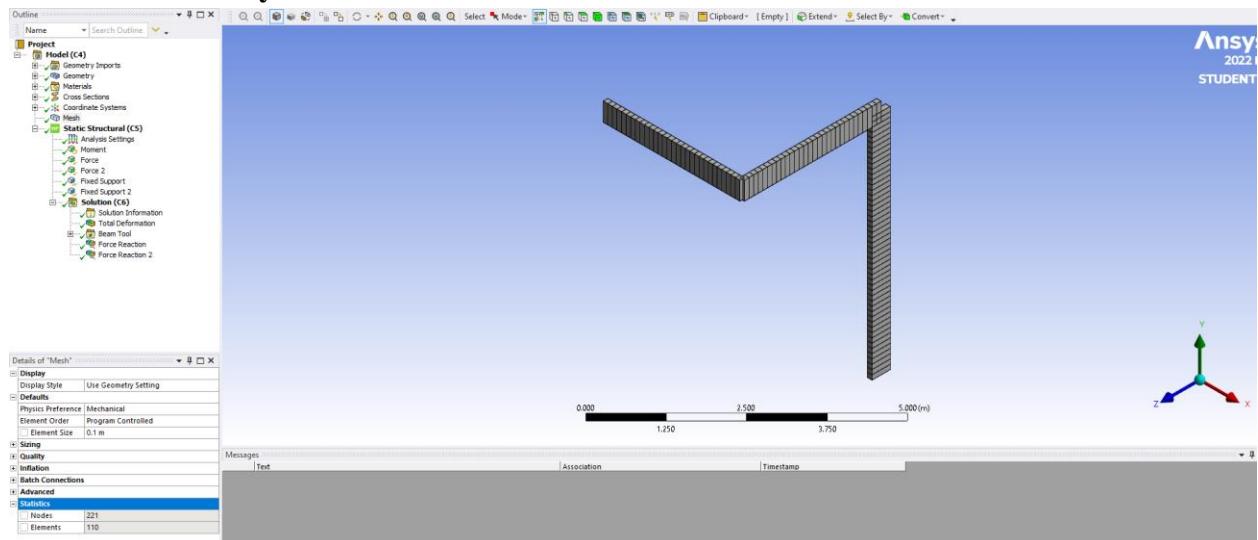
- (a) Create a set of points for the frame in Design Modeler (either from a coordinates file or one by one). Use the Lines from Points tool to create a line body of the geometry shown. Use a rectangular cross section of $0.4 \times 0.1 \text{ m}^2$ for all the beams. Show a figure of the final line body **with the cross section assigned such that the bending stiffness is maximized in the appropriate direction** (i.e., in the X-Y plane for the horizontal beam, and in the Y-Z plane for the vertical beam as well as the angled middle beam). Set the View options in the menu bar as shown on the right.

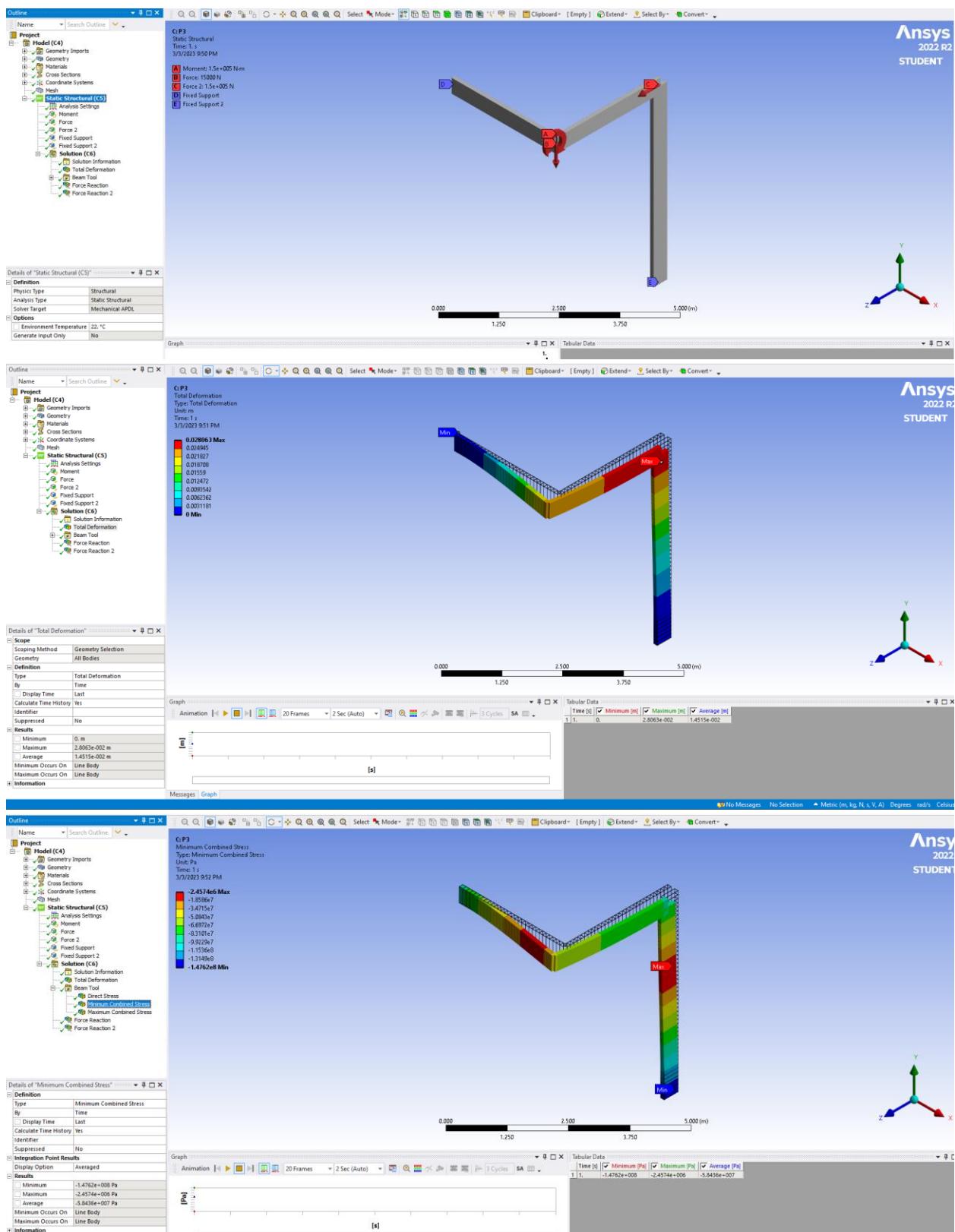


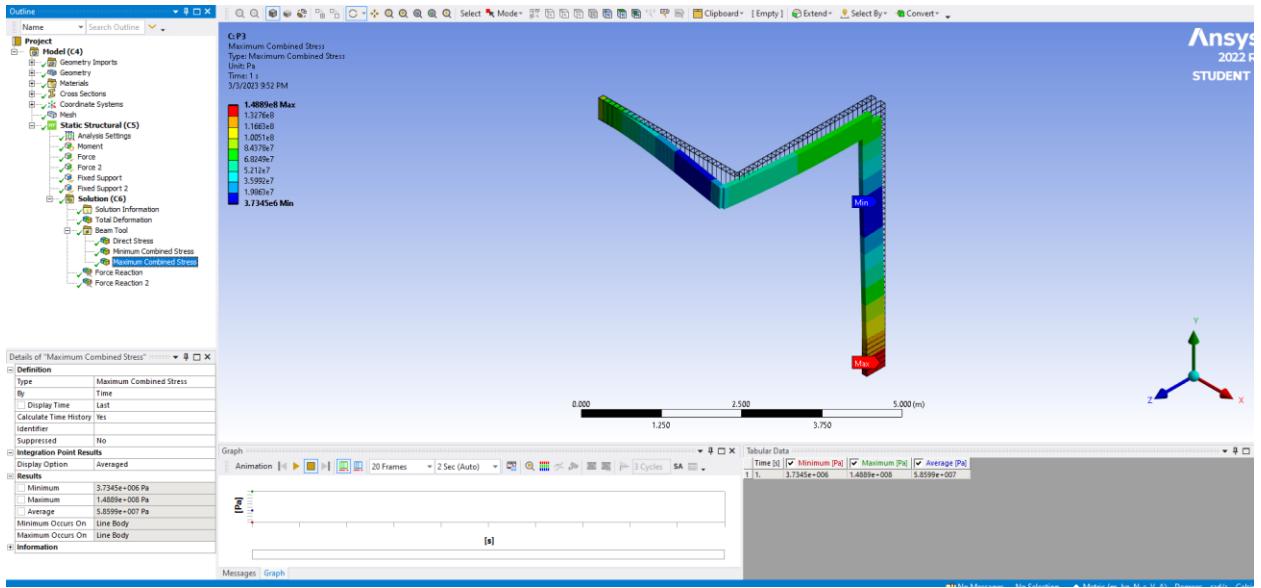


- (b) Mesh the model with a reasonable (medium-sized) mesh. Apply the loads and boundary conditions shown in the figure. Show plots of the mesh, total deformation, minimum combined stress, and maximum combined stress. Set the display options to also show the undeformed model. What is the factor of safety for this design? You can calculate this by dividing the yield strength of structural steel by the maximum absolute value of the stress results.

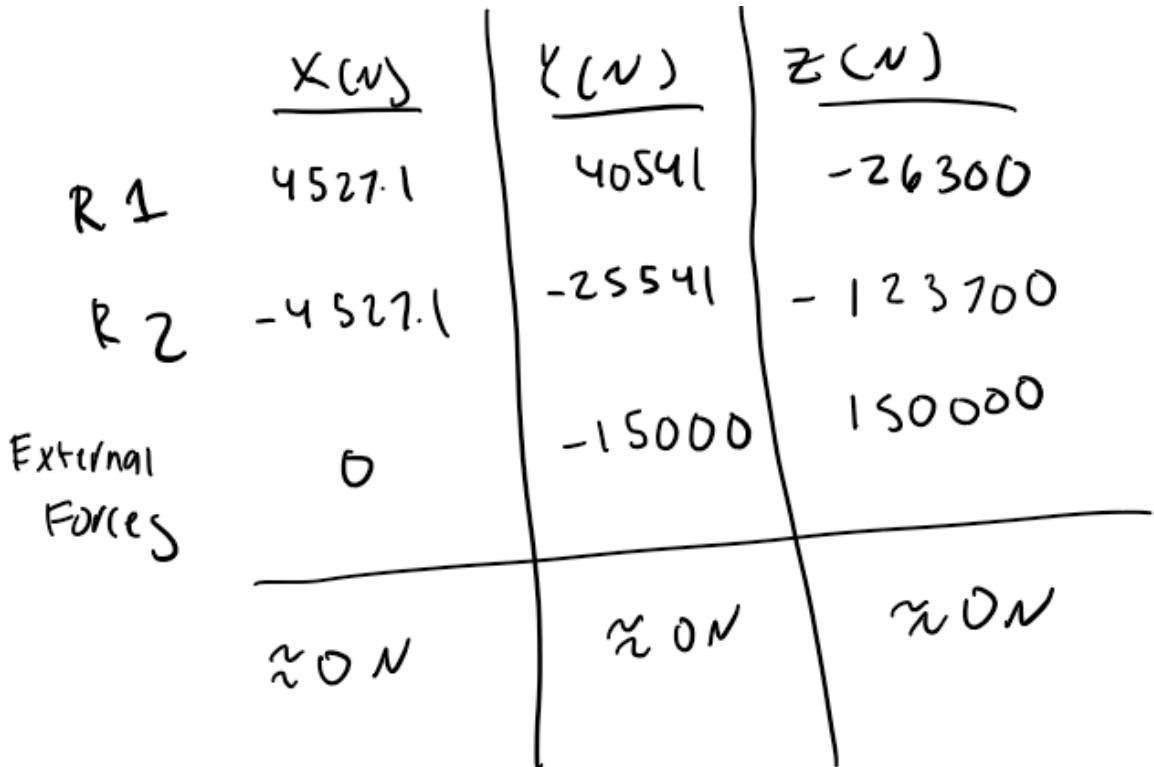
The factor of safety is: $2.5/1.4889 = 1.67$







- (c) Extract the force reactions at the supports and verify that global equilibrium of forces is satisfied.



MEE 323 – Computer Aided Engineering II
Homework Assignment #8 – Stress Stiffening and Buckling

Instructions:

- Use this Word file as a template for your homework report. Add screenshots of your modeling/analysis or any required explanations below the appropriate question. Turn in the homework report (converted to PDF) on the course Gradescope before the deadline.
- Upload a copy of your ANSYS files to the appropriate assignment on the course Canvas. The uploaded ANSYS files may be used to check your work and/or ensure academic integrity.

Homework Objectives:

Learn to create and validate line models (beams and trusses). Learn to carry out stress stiffening and buckling analyses in ANSYS Workbench.

Reading Assignment:

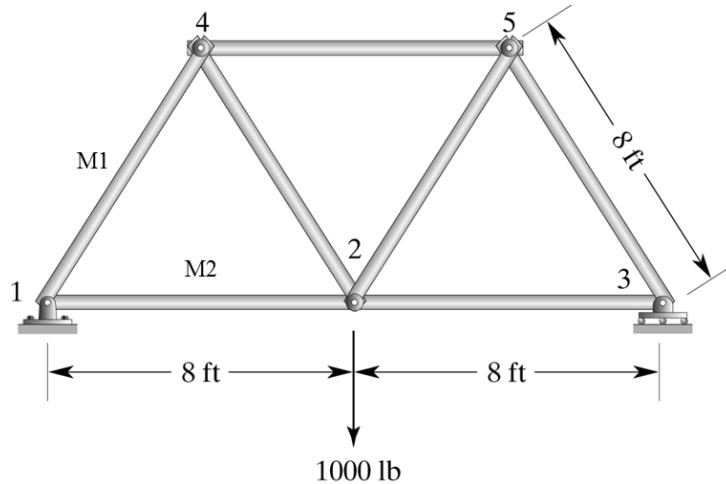
Look through **Chapter 10** in the textbook for examples of buckling and stress stiffening simulation techniques.

Question 1:

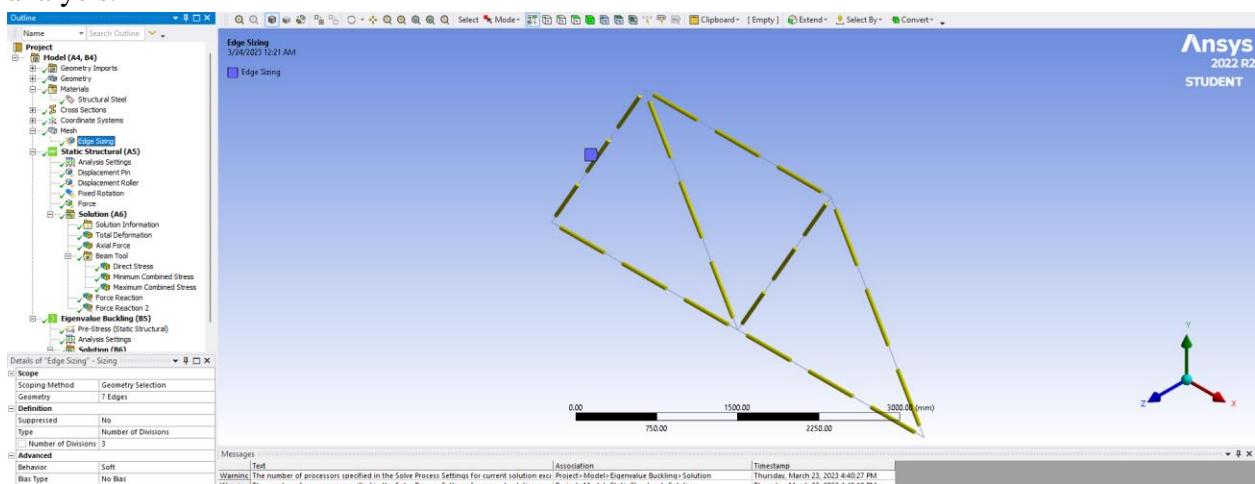
The members of the truss shown in the figure below have a **square** cross-sectional area of 1.96 in^2 and are made of structural steel. The left end is fixed but allowed to pivot, and the right end is allowed to freely move horizontally as well as pivot (no vertical motion is allowed); use appropriate supports to model the two boundary conditions.

Model the truss structure in ANSYS by creating lines on a sketch and then using the "Lines from Sketches" tool to generate the line model. Use a mesh "Edge Sizing" control to ensure that each member of the truss consists of only one element (as was done in class).

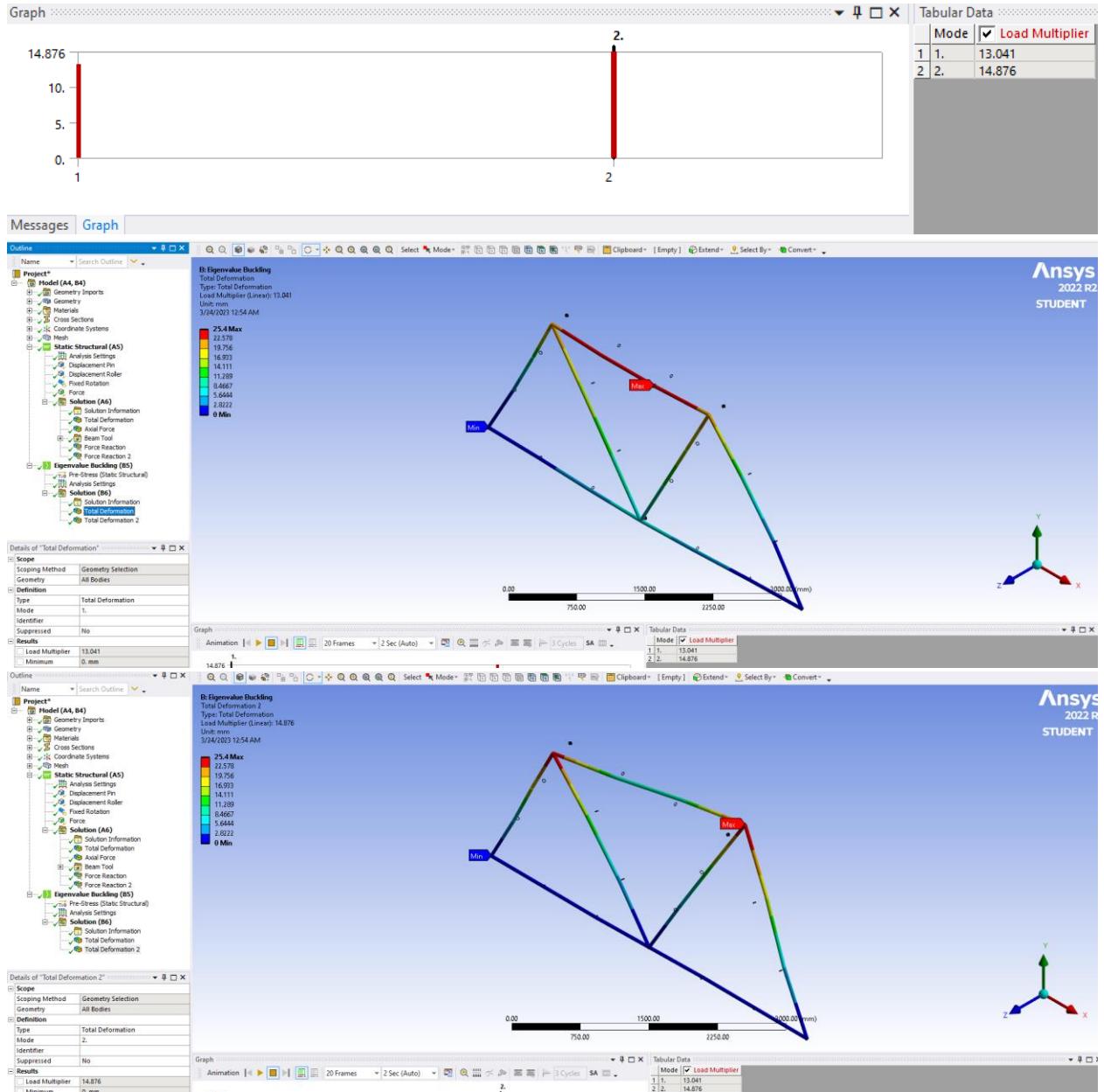
You may use the model you made for the previous homework. If you used a different cross section for the previous model, make sure to go back and change it to a square cross section.



- (a) Mesh the truss with multiple elements for each member and carry out an Eigenvalue Buckling analysis.



- (b) Find the first two **non-zero** load multipliers and show figures of the corresponding buckling mode shapes. Note that the buckling modes may be out of plane of the truss.



(c) Is the truss structure safe from a buckling standpoint (what is the factor of safety in buckling)?

How does this compare to the factor of safety from the standpoint of yielding?

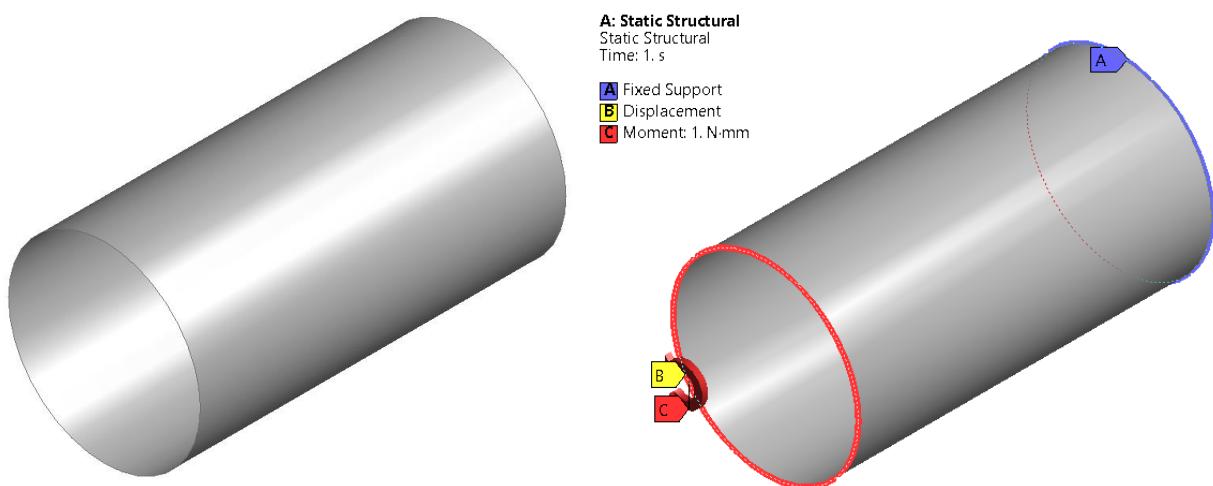
Buckling Factor of safety: 13.04

Standpoint of yielding Factor of safety: $250\text{Mpa}/2.0317\text{Mpa} = 123.05$

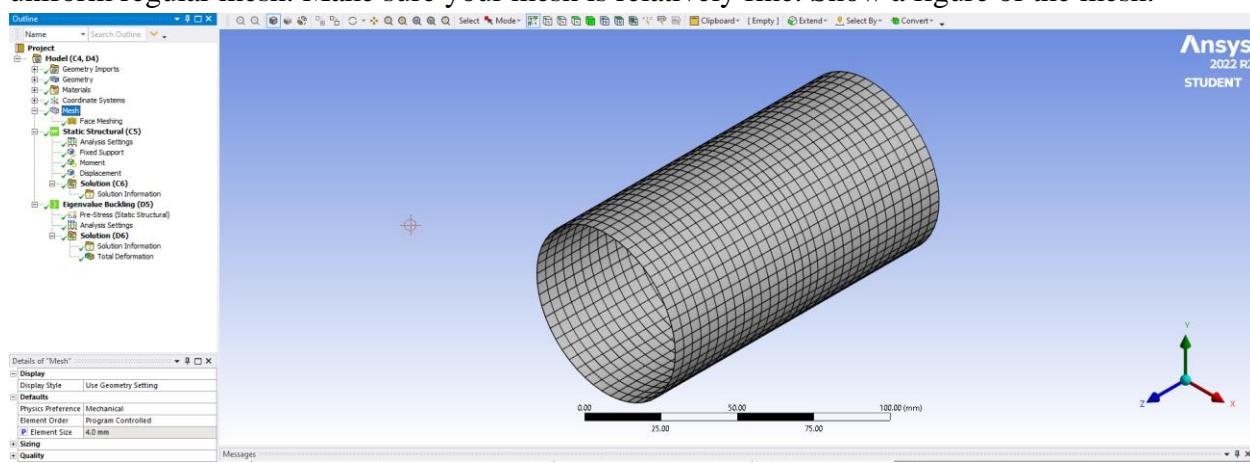
The factor of safety on the standpoint of yielding is far greater than the buckling factor of safety, therefore the truss structure is safe from buckling standpoint.

Question 2:

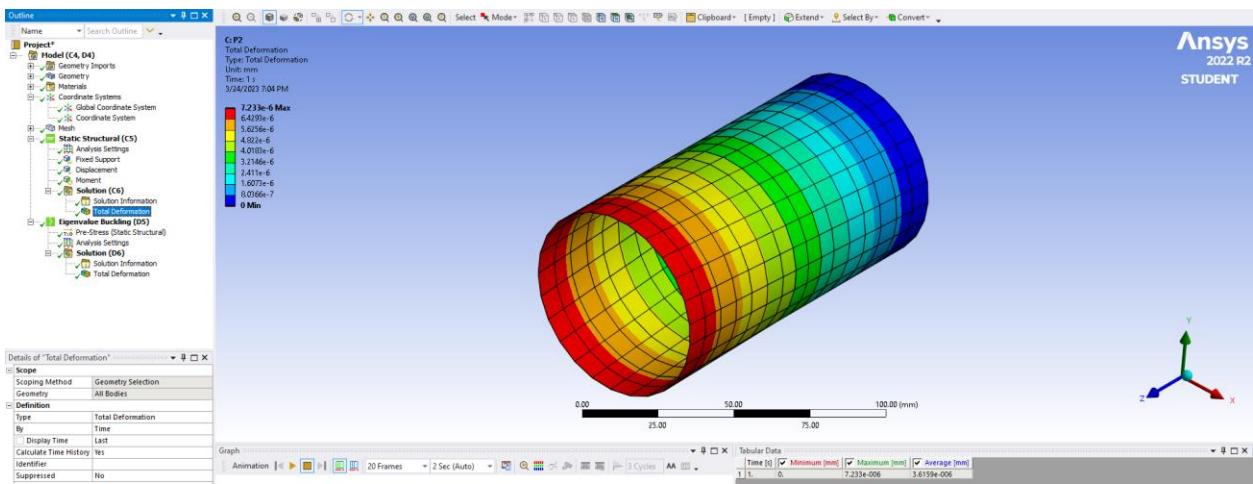
Apply a moment on the thin **aluminum alloy** cylinder shown below (Young's modulus of 71 GPa and a Poisson's ratio of 0.33). Model the can as a **surface body** with the appropriate thickness. The cylinder is of length 122 mm with a mean diameter of 64 mm and a wall thickness of 0.1 mm. The moment will cause a tensile stress on a principal direction in the can and a compressive stress in another principal direction causing the skin to buckle. Predict the moment that will cause the skin to buckle.



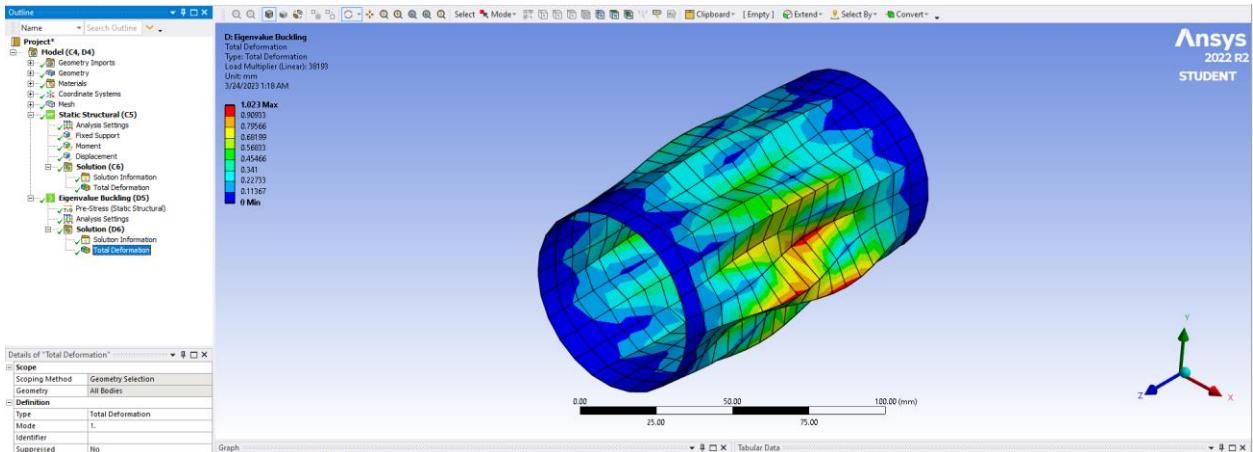
- (a) Use the model geometry provided. Mesh the model using a **face meshing** control to get a uniform regular mesh. Make sure your mesh is relatively fine. Show a figure of the mesh.



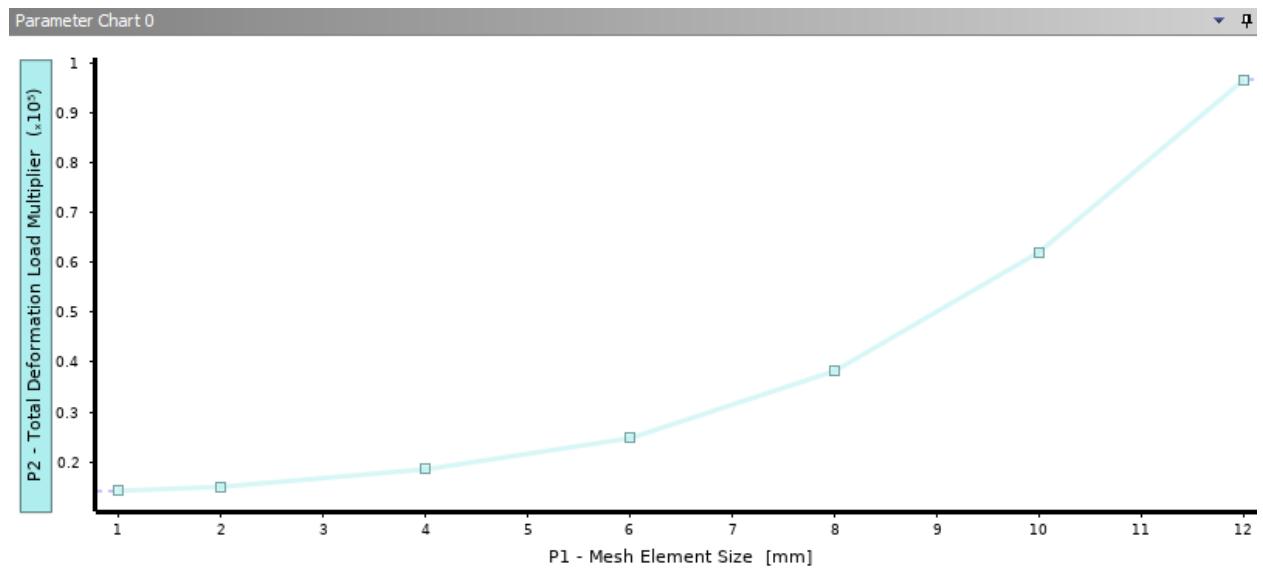
- (b) Apply a fixed support on one end. On the other end, create a **cylindrical coordinate system**. Then apply a displacement support with just the tangential (theta) direction free. Finally apply a moment of 1N-mm to this end. Show a figure of the total deformation.



- (c) Pass the solution of (b) to an eigenvalue buckling block and predict the moment for the first buckling mode. What is the value of the moment required for this buckling to happen? Also show the first buckling mode shape. **The value will be 38193 N*mm.**

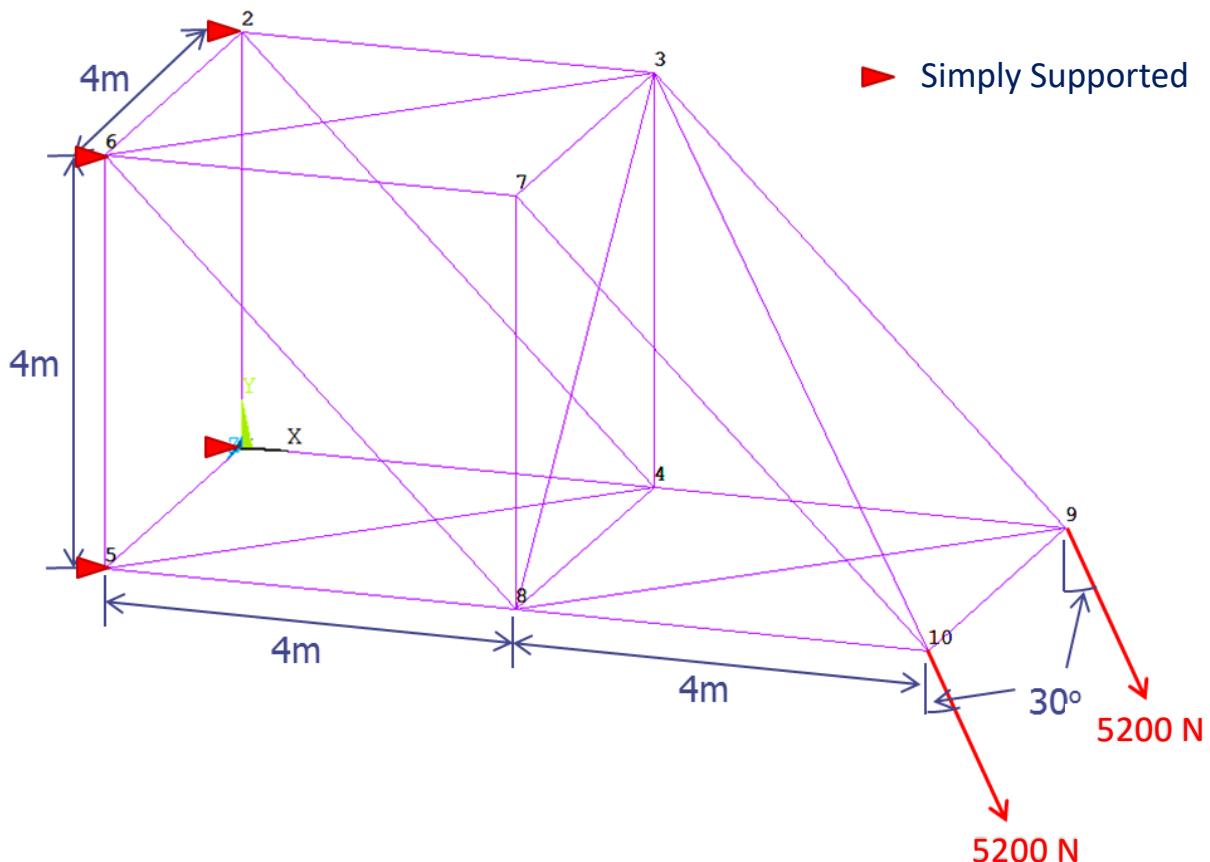


- (d) The buckling load prediction also shows convergence based on the size of the mesh. Run a convergence analysis by **manually** setting the mesh size to 12mm, 10mm, 8mm, 6mm, 4mm, 2mm, and 1mm (you can use parameters to have ANSYS simulate these all at once). Plot the buckling load multiplier vs. mesh size (or number of elements). What do you observe? **I observe as we increase the number of element size the load multiplier will converge.**



Question 3:

Consider the **aluminum alloy** space truss from the previous homework (shown below). It is desired to check the design from the standpoint of both yield and buckling failure. To this end, carry out the following analyses. Use the geometry you created for the previous homework and modify appropriately.



- (a) Determine the factor of safety considering yielding failure. Comment on the magnitude of the factor of safety – i.e., why do you think this structure warrants such a high factor of safety? Which truss member is the critical one from the standpoint of yielding failure?

Yielding Factor of safety: $280\text{Mpa} / 2.8392\text{Mpa} = 98.6$

The factor of safety is high because it is not considering buckling.

The critical truss is the one with the maximum stress which is at the point 7 within that truss.

- (b) Duplicate the analysis of (a) above and change the mesh so that you now have 25 elements per truss member. Repeat the analysis and check the safety factor of the design. Comment on any differences seen (or not seen).

Yielding Factor of safety: $280\text{Mpa} / 2.8392\text{Mpa} = 98.6$

I do not see a difference since the maximum stress is the same.

(c) Pass the solution of (b) to an eigenvalue buckling block and extract the first 4 **non-zero** buckling modes and load multipliers. Explain why the first 2 load multipliers are very close to each other. Considering the first buckling mode, what is the factor of safety of the design from the standpoint of buckling and which is the critical member in the system? **The first two multiplier will be the same because the critical truss will just bend the other way in the second run. The critical member is the bottom truss inside the cube from number 4 to 5.**

