MAE 503: Project 2

Hannah Seyler, Isabel Schinella, Steven Van Overmeiren, Sagardas Gireesh Pai

Task 1. Element Matrices and Global System Assembly

Kd = f

- K: Global Stiffness matrix (2n x 2n matrix (n ≣ # of nodes in mesh), symmetric, positive definite)
  - Assembled from all element stiffness matrices; relates displacements to internal forces.
  - Derived from terms like $\int_{\Omega_e} B^{e\,T}_i \, D \, B^e_j \, d\Omega$
    - B^e_i & B^e_j: derivatives of shape functions (depend on polynomial degree) and has size of 3 x 2n (n ≣ # of nodes in mesh) for an element with n nodes

$$\mathbf{B}^e = \begin{bmatrix} \frac{\partial N_1^e}{\partial x} & \frac{\partial N_2^e}{\partial x} & \frac{\partial N_3^e}{\partial x} & \frac{\partial N_4^e}{\partial x} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{\partial N_1^e}{\partial y} & \frac{\partial N_2^e}{\partial y} & \frac{\partial N_3^e}{\partial y} & \frac{\partial N_4^e}{\partial y} \\ \frac{\partial N_1^e}{\partial y} & \frac{\partial N_2^e}{\partial y} & \frac{\partial N_3^e}{\partial y} & \frac{\partial N_4^e}{\partial y} & \frac{\partial N_1^e}{\partial x} & \frac{\partial N_2^e}{\partial x} & \frac{\partial N_3^e}{\partial x} & \frac{\partial N_4^e}{\partial x} \end{bmatrix}$$

    - D: constant for isotropic linear materials (E: Young's modulus & ν: Poisson's ratio)

$$\mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

- d: Displacement Vector (2n x 1, n ≣ # of nodes in mesh)
  - Unknown nodal displacements (in x, y directions for 2D analysis).
  - Computed as d = K \ f in Matlab

$$\mathbf{d} = [u_1, v_1, u_2, v_2, \ldots, u_n, v_n]^T$$

    - u_i: x-displacement at node i
    - v_i: y-displacement at node i

- f: Force Vector (2n x 1 vector, n ≣ # of nodes in mesh)
  - External nodal forces (from loads, traction, boundary conditions).
  - This is computed as:

$$\mathbf{f}^e = \int_\Gamma \mathbf{N}^T \bar{\mathbf{t}} \, d\Gamma$$

- where t is the traction vector in y direction and N is the shape function matrix

We have used plane stress due to the following reasons:

- Plate is thin (6mm thick, compared to 150mm length and 50mm height)
- Loading and constraints are in-plane
- The analysis doesn't have any through-thickness loading

Task 2. Model Setup Using a Commercial FEM Package (2D)

For the 2D FEM package ANSYS structural was used to model the 2D geometry. For this analysis the step was to create the geometry in the design modeler. Once the geometry was made the analysis was changed to 2D for analysis type as shown below:

Properties of Schematic A3: Geometry

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Geometry
4	Directory Name	SYS
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ Geometry Source	
10	Geometry File Name	C:\Users\hanna\AppData\Local\Temp\WB_hanna_12292_2\wbnew_files\dp0\SYS\DM\SYS.agdb
11	⊟ Advanced Geometry Options	
12	Analysis Type	2D
13	Compare Parts On Update	No

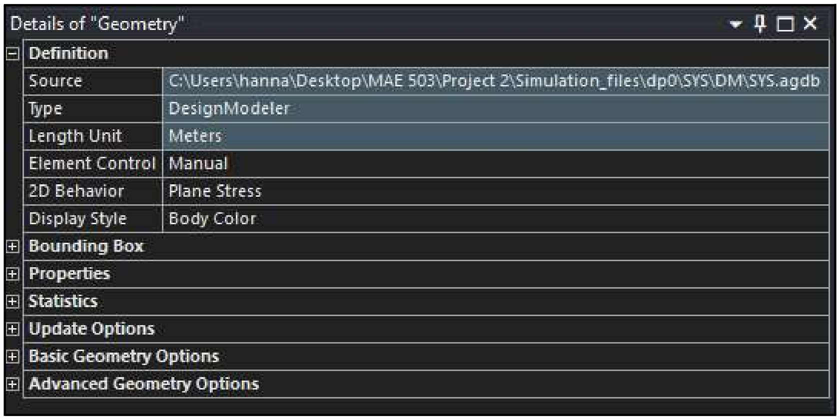
We also changed the material properties so that the Young's modulus was 105 GPa and the Poisson's ratio was 0.34. A screen shot is shown below of this new material named "titanium".

4	titanium				
*	Click here to add a new material				

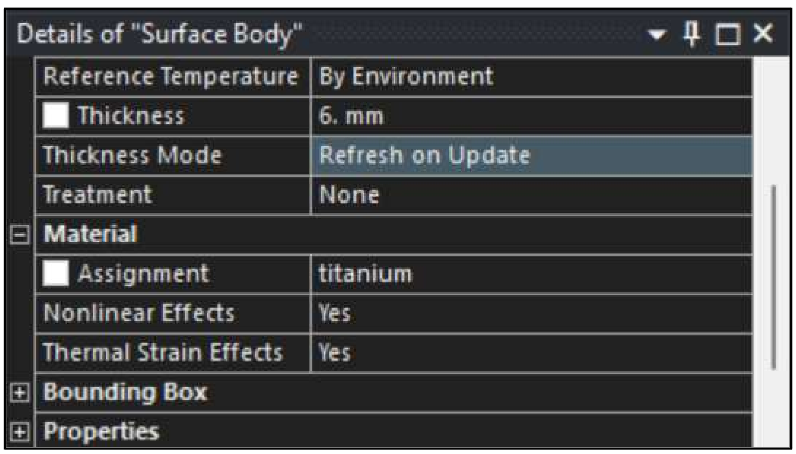
Properties of Outline Row 4: titanium

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	⊟ Isotropic Elasticity				
4	Derive from	Young'...			
5	Young's Modulus	105	GPa		
6	Poisson's Ratio	0.34			
7	Bulk Modulus	1.0938E+11	Pa		
8	Shear Modulus	3.9179E+10	Pa		

In the meshing step, we defined the 2D behavior, element type, element size, and material properties. We selected plane stress as the 2D behavior because the plate's thickness was small relative to its other dimensions. The plate had a thickness of 6 mm, length of 150 mm, and height of 50 mm. When the thickness is significantly smaller than the length and height, out-of-plane stresses become negligible, making plane stress an appropriate assumption. The screenshot below shows the selection of the plane stress option in the meshing setup.



The next screenshot is to show that the modified material "titanium" was selected and used for the 2D finite element analysis.



The last thing that was set for the mesh was the element order and element size. The element order chosen was linear with a quadrilateral dominant method. This will produce quadratic elements with 4-nodes. The element size for the analysis chosen was 0.5 mm. This size was used since there is a good rule of thumb to use an element size that is 10% of the longest length. The longest length for this simulation was 150 mm so 10% of that is 1.5 mm. Another rule that we thought was important to follow was to have at least 30 elements around the hole. With the element size of 0.5 mm this was met and ensured that the solution converged. The screenshot of the element order and element size are shown below:

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Quadratic
Element Size	0.5 mm
Sizing	
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	

Details of "Quadrilateral Dominant Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Quadrilateral Dominant
Element Order	Linear
Free Face Mesh Type	Quad/Tri

After creating the mesh Ansys said that the simulation uses PLANE 182 and SURF 153 for the element types as shown below.

Material IDs	Element Name IDs	Element Type IDs	Number of Elements
MAT_1	PLANE182	1	26442
MAT_2	SURF153	2	100

Element Name IDs	Material IDs	Element Type IDs	Number of Elements
SURF153	2	2	100
PLANE182	1	1	26442

Element Type IDs	Element Shape	Material IDs	Element Name IDs	Number of Elements
ETYPE_1	QUAD4	1	PLANE182	26442
ETYPE_2	LINE3	2	SURF153	100

The PLANE 182 element type is 4 nodes with two degrees of freedom and is used to model 2D structures. The SURF 153 is defined by two or three nodes and the material properties. This is used for load and surface effect applications. The information was obtained from mm.bme and shown below.

Figure 182.1: PLANE182 Geometry

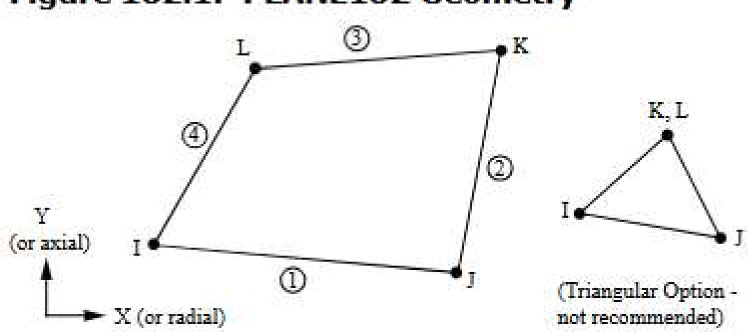
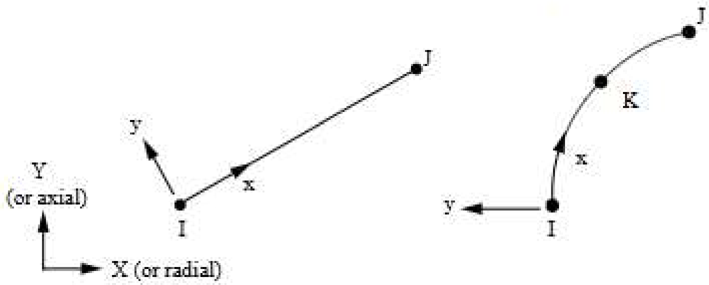
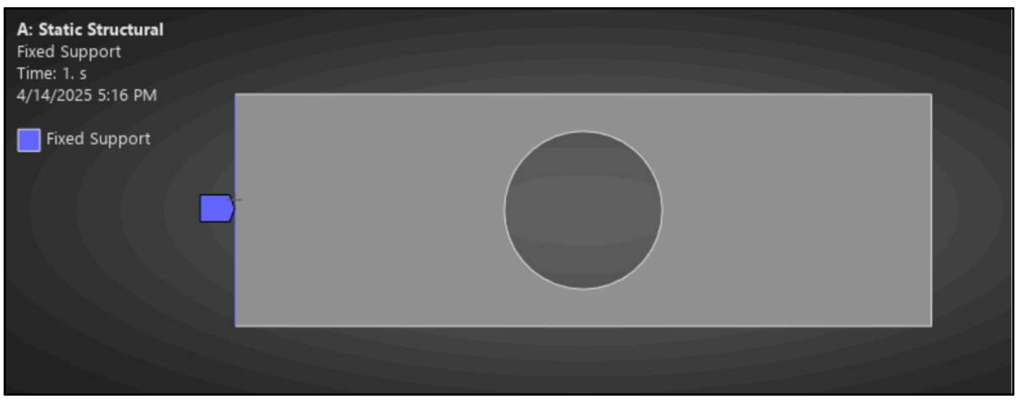


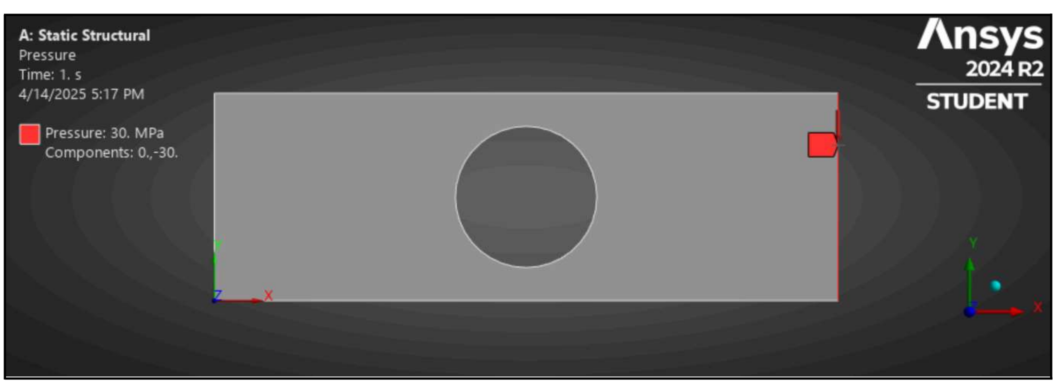
Figure 153.1: SURF153 Geometry



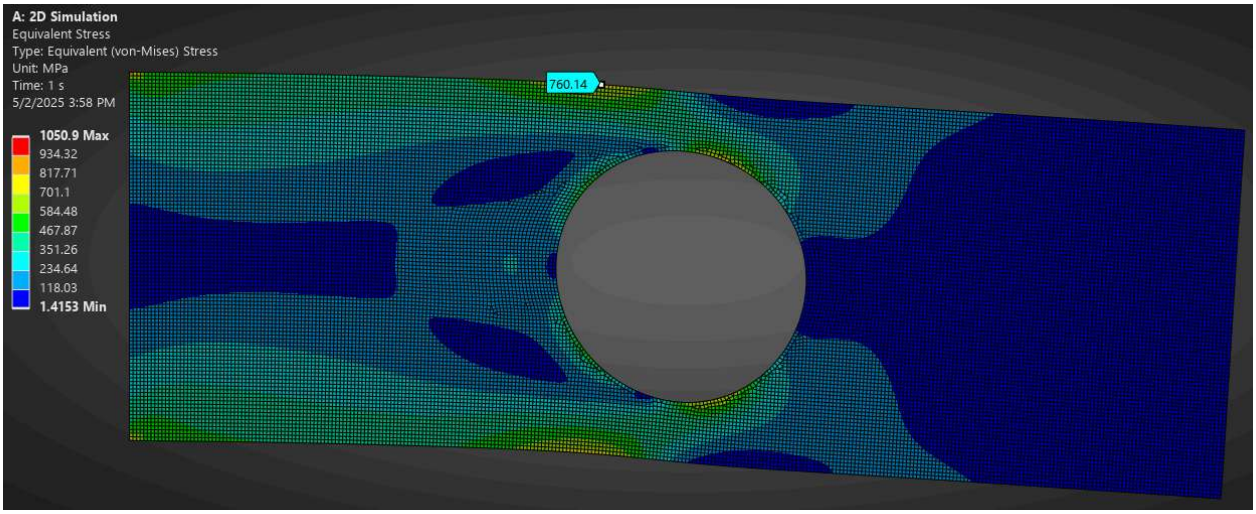
With the mesh done the next steps are to add the boundary conditions and applied loads of the system. For this simulation there is one boundary condition and one applied load. The boundary condition is a fixed support on the left side of the plate. This is shown in the screenshot below. The blue line on the geometry shows where the fixed support was added.



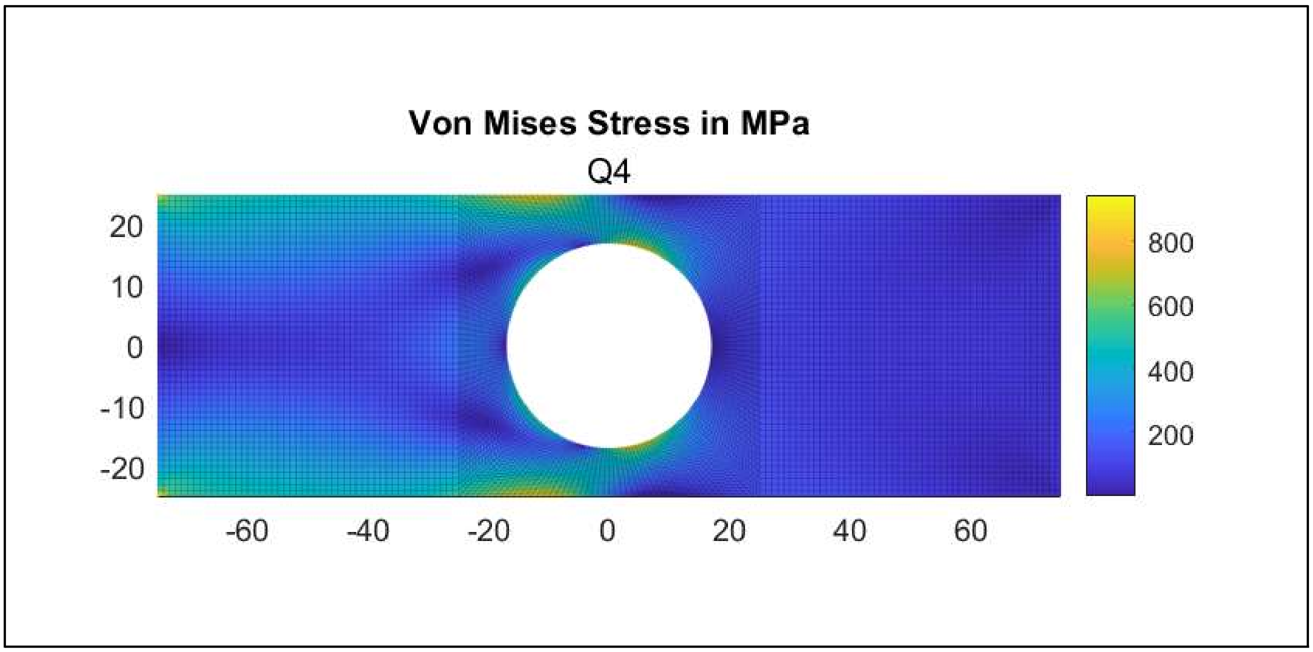
The applied load was from the applied moment exerted via a distributed traction. This is applied in the negative y direction on the right end of the plate. To do this a pressure force was applied to the right side as highlighted on the geometry shown below. To ensure the 30 N mm^{-2} was applied properly a pressure was used to represent the distributed traction. For pressure 30 MPa is equal to 30 N mm^{-2}. We also made sure to split the force into components so that we could specify the force acting in the negative y direction and have no force in the x direction.



A solution from the simulation is shown below. This is the von Mises stress.



The fine mesh from our code was used to plot the von Mises of a Q4 plot as seen below:



When looking at the contours of both plots they are the same. The scales are slightly different between the plots so the contour might not follow the exact same shape but the overall contour is the same and the values are the same when looking at specific areas. An example is on the top of the plate near the hole. The Ansys solution shows a value of around 700 MPa when a probe is placed. The matlab solution has a dark yellow spot there which should also represent around 700 MPa. Another spot along the top of the beam is the lighter blue or teal looking color. The probe in Ansys says this is about 430 MPa. When looking at the matlab plot the light blue/teal falls around 400 MPa or slightly above as it turns to teal. Another way to show that they are the same is through the mesh convergence done later. In this convergence the max von mises was found to be near 760 MPa and in the Ansys simulation the max von-mises was 760.14 MPa. The maximum value of 1050.9MPa found on the ansys simulation is at the points of singularity (top left & bottom left corners) where it goes to infinity theoretically, and thus value is avoided.

Task 3. Numerical Quadrature Justification

The weak form integrals for stiffness matrix computation are of the type:

$$Ke_{ij} = \int_{\Omega_e} B_i^T \, D \, B_j \, d\Omega$$

- B_i & B_j: derivatives of shape functions (depend on polynomial degree)
- D: constant for isotropic linear materials

So, the integrand is a polynomial involving shape function derivatives. Quadrature must exactly integrate a polynomial of degree = 2 * (degree of shape function -1)

- TRI3 (Linear Triangle): 1 point quadrature
    - Shape function degree: 1
    - Derivatives are constant and this means integrand is constant
    - Need to integrate 0^{th} degree polynomial (max integrand degree 0)
    - 1 point quadrature is used for linear triangles

- TRI6 (Quadratic Triangle): 3 point quadrature
    - Shape function degree: 2 (derivatives are linear)
    - B^TDB involves products of linear terms, therefore the maximum degree is 2
    - Need to integrate 2^{nd} degree polynomial (max integrand degree 2)
    - Because of this a 3 point quadrature integrates 2 degree triangles

Table 7.7 Gauss quadrature weights and points for triangular domains.

Integration order	Degree of precision	ξ_1	ξ_2	Weights
Three-point	2	0.1 666 666 666	0.1 666 666 666	0.1 666 666 666
		0.6 666 666 666	0.1 666 666 666	0.1 666 666 666
		0.1 666 666 666	0.6 666 666 666	0.1 666 666 666
		0.1 012 865 073	0.1 012 865 073	0.0 629 695 903
		0.7 974 269 853	0.1 012 865 073	0.0 629 695 903
		0.1 012 865 073	0.7 974 269 853	0.0 629 695 903
Seven-point	5	0.4 701 420 641	0.0 597 158 717	0.0 661 970 764
		0.4 701 420 641	0.4 701 420 641	0.0 661 970 764
		0.0 597 158 717	0.4 701 420 641	0.0 661 970 764
		0.3 333 333 333	0.3 333 333 333	0.1125

- QUAD4 (Bilinear Quadrilateral): 2x2 Gauss quadrature
    - Shape function degree: 1 in each direction (bilinear)
    - Derivatives are constants or linear in ζ, η (product terms are max of 2^{nd} degree)
    - Max Integrand degree is 2
    - 2x2 Gauss quadrature integrates degree-3 polynomials in 2D

- QUAD8 (Serendipity): 3x3 Gauss quadrature

- Shape function degree: 2 (derivatives are linear)
- B^TDB involves products of quadratic terms, therefore the maximum degree is 4
- For 2D quad, 3x3 Gauss quadrature integrates up to degree-5 polynomials

The weak form integrals for nodal force arising from the prescribed traction is as follows:

$$\mathbf{f}^e = \int_\Gamma \mathbf{N}^T \bar{\mathbf{t}}\, d\Gamma$$

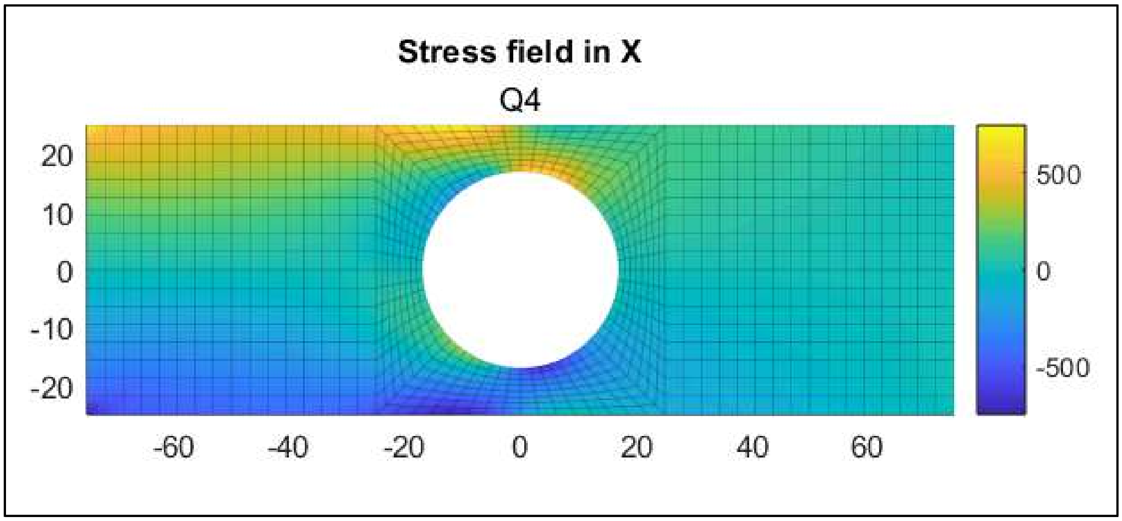
- where t is the traction vector in y direction and N is the shape function matrix

Here a 2-point gauss quadrature is used to perform the integration. The 2-point Gauss quadrature is used here because it's the exact integration rule for polynomials of degree up to 3 over a 1D interval. N will be linear or quadratic depending on the element and t is constant so at most a degree of 2 will be used. Therefore, the degree of 2 is covered by the 2-point quadrature that integrates up to 3.

Task 4. Least-Squares Stress Projection

The least squares projection of the finite element stress fields was implemented into our Matlab code. A normal stress plot in the x direction for a Q4 element type is shown below.

Quadrilateral Element (Q4)

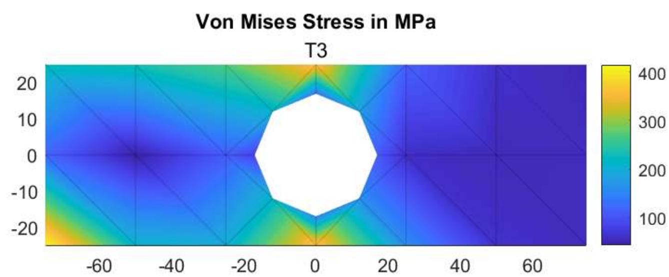
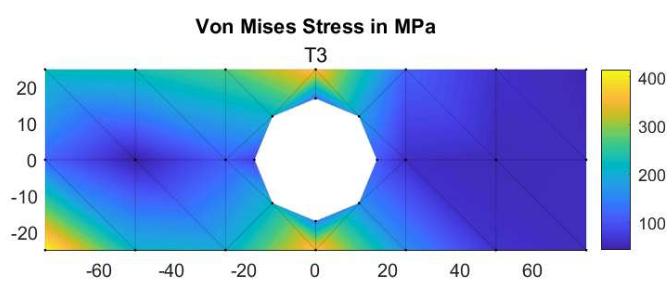


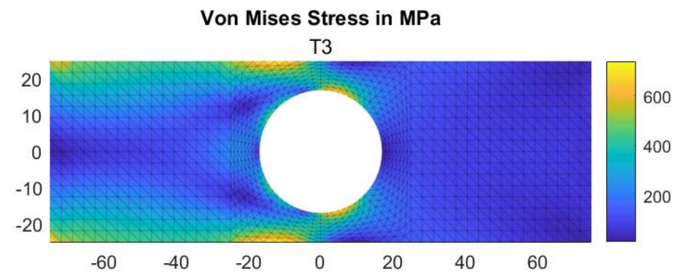
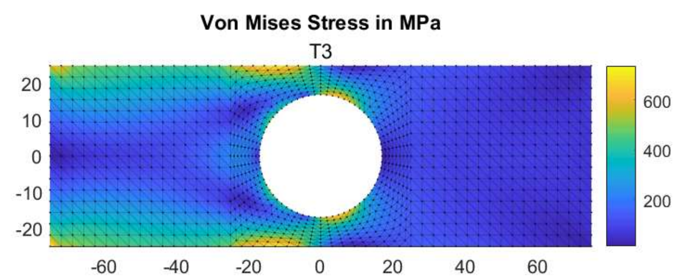
Task 5. Comparison of Element Types

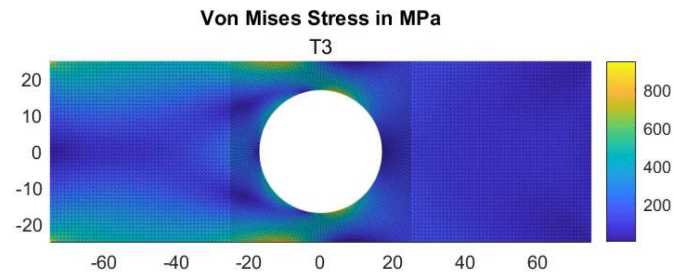
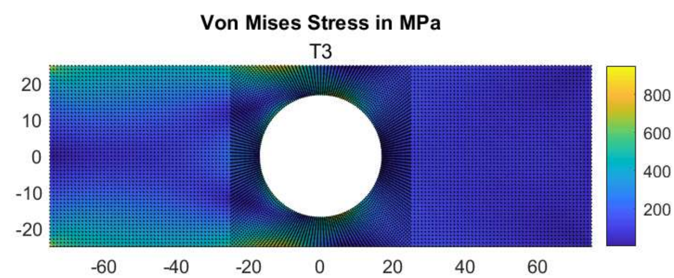
(there are two contour plots which show the von Mises stress for each type of element, the second one has the nodes highlighted)

3-node triangle

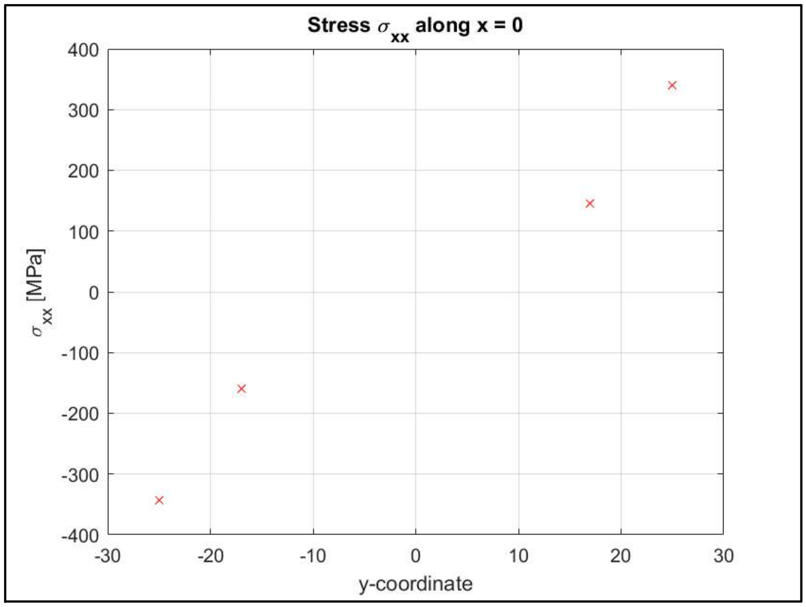
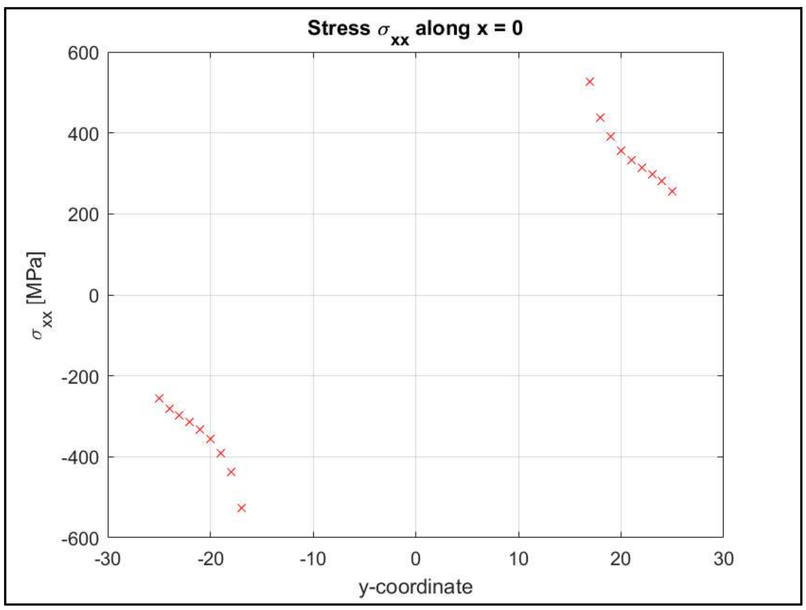
a) contour plot of the von Mises (equivalent) stress

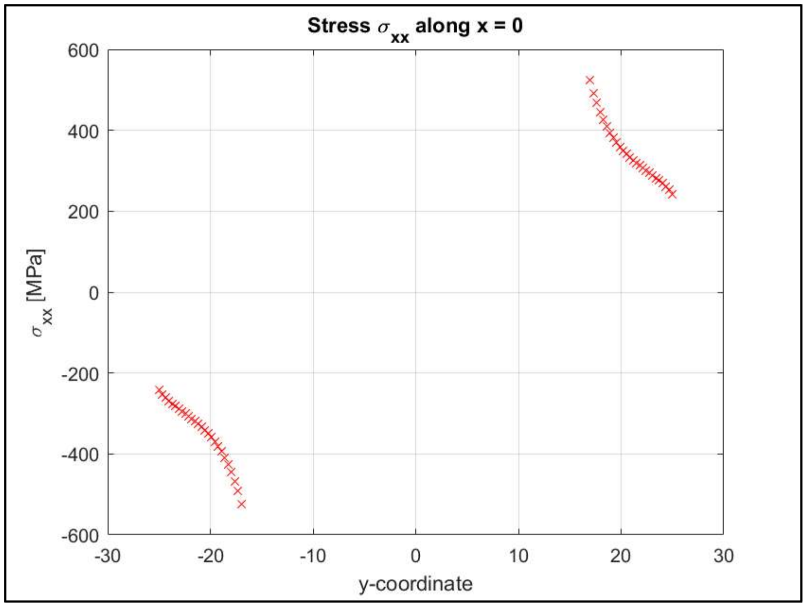
Number of elements	Contour Plots
Coarse (ey = 2)	 

Medium (ey = 16)	 Von Mises Stress in MPa T3 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 600 400 200  Von Mises Stress in MPa T3 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 600 400 200

Fine (ey = 50)	Von Mises Stress in MPa T3 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 800 600 400 200 Von Mises Stress in MPa T3 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 800 600 400 200

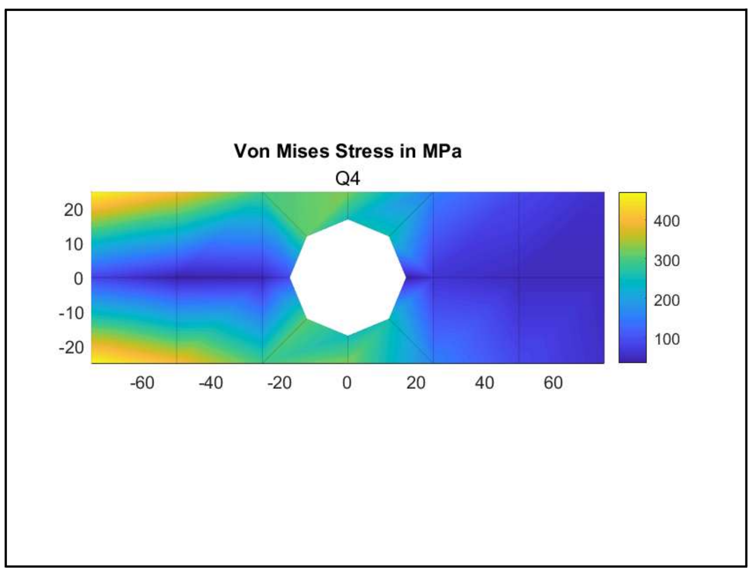
b) a line plot of the normal stress along cross-section AA

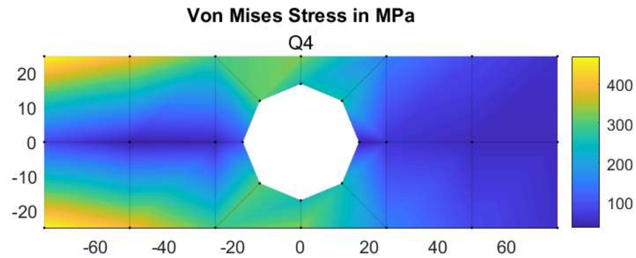
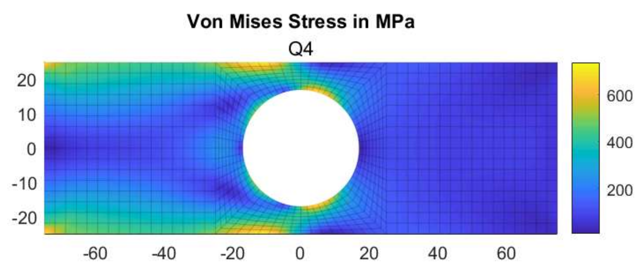
Number of elements	Line Plots
Coarse (ey = 2)	
Medium (ey = 16)	

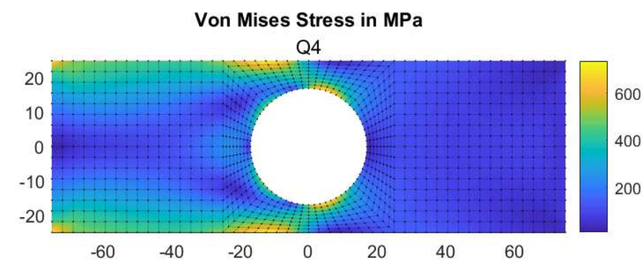
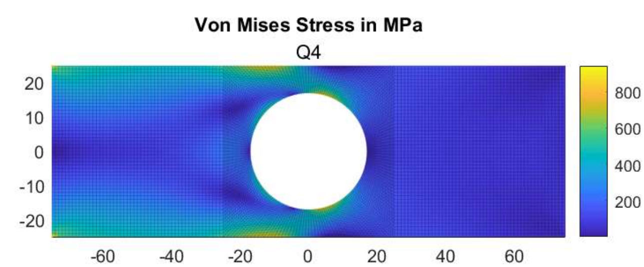
Fine (ey = 50)	

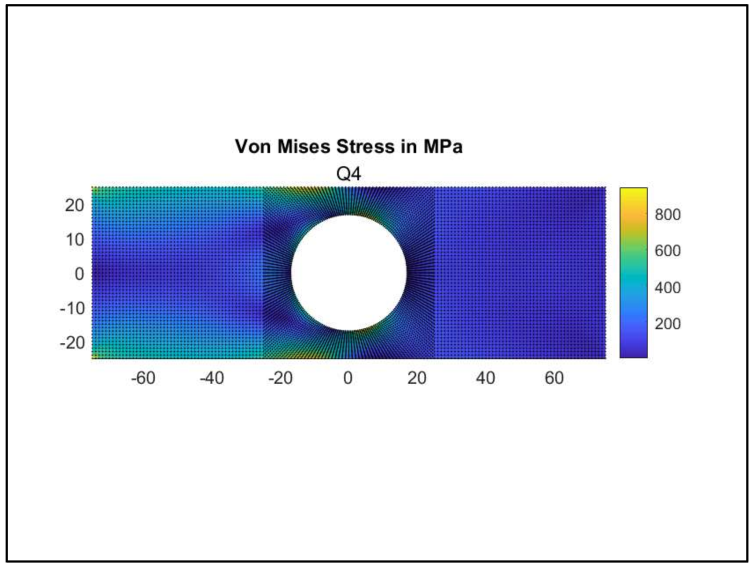
4-node quadrilateral

a) contour plot of the von Mises (equivalent) stress

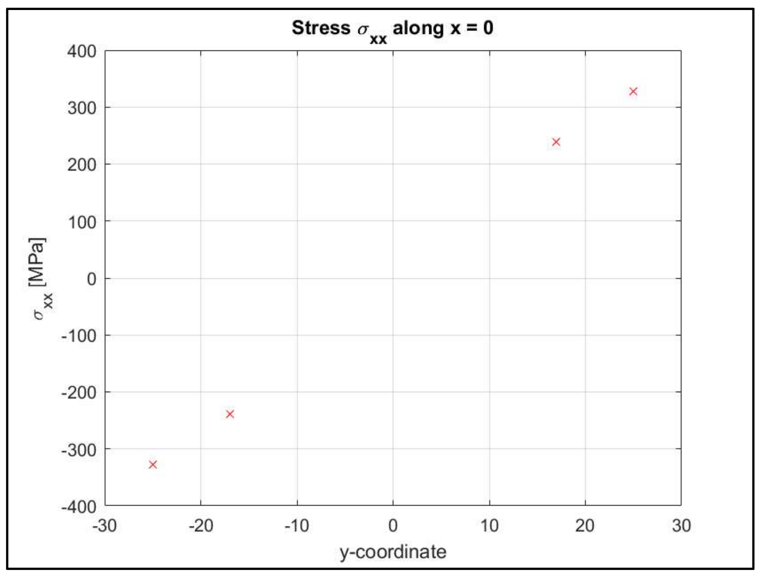
Number of Elements	Contour Plots
Coarse (ey = 2)	

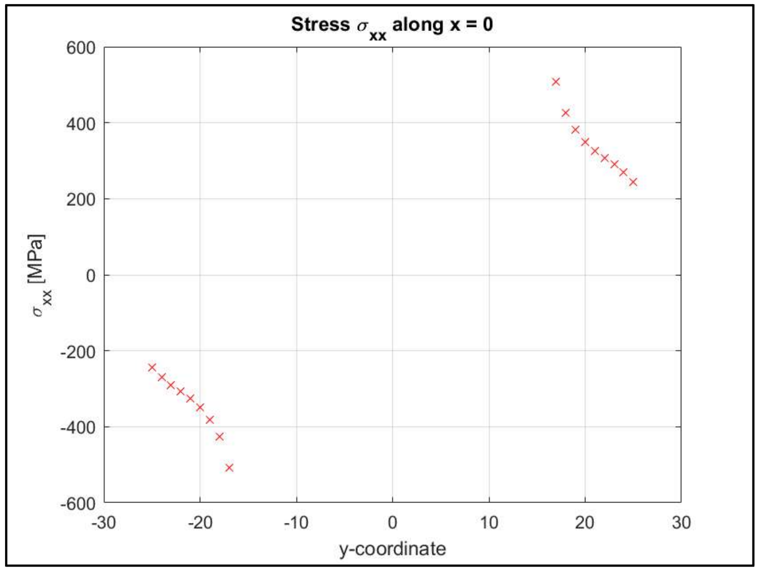
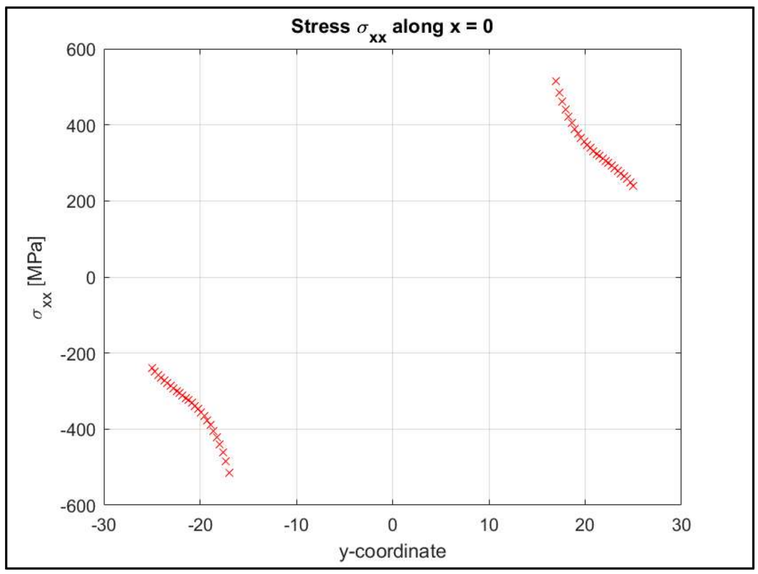
<table>
<tr>
<td></td>
<td>
</td>
</tr>
<tr>
<td>Medium (ey = 16)</td>
<td>
</td>
</tr>
</table>

	Von Mises Stress in MPa Q4 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 600 400 200
Fine (ey = 50)	Von Mises Stress in MPa Q4 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 800 600 400 200

<table>
<tr><td></td><td>
</td></tr>
</table>

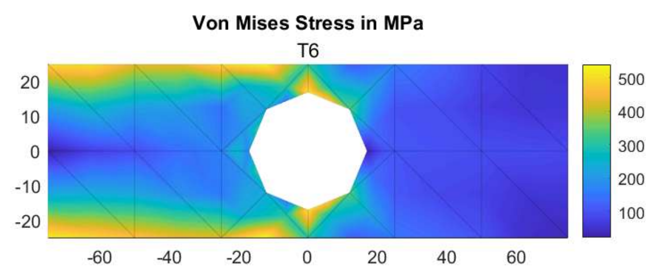
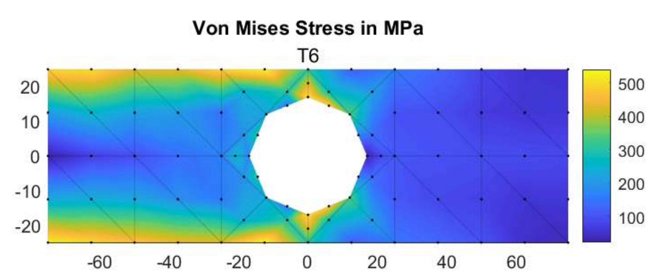
b) a line plot of the normal stress along cross-section AA

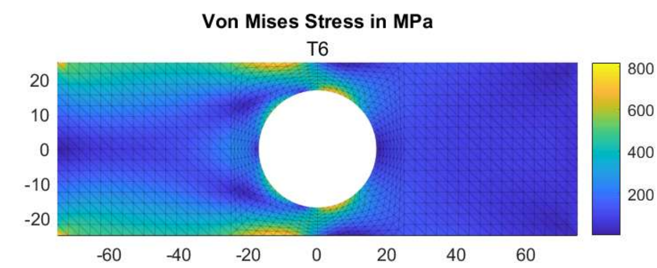
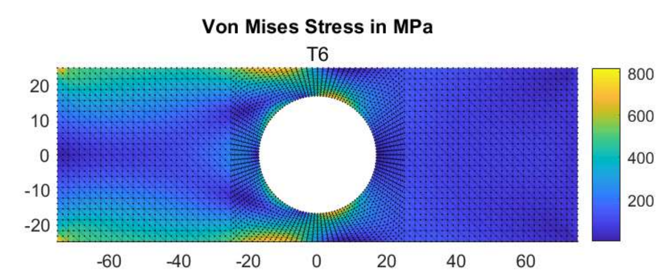
<table>
<tr><td>Number of Elements</td><td>Line Plots</td></tr>
<tr><td>Coarse (ey = 2)</td><td>
</td></tr>
</table>

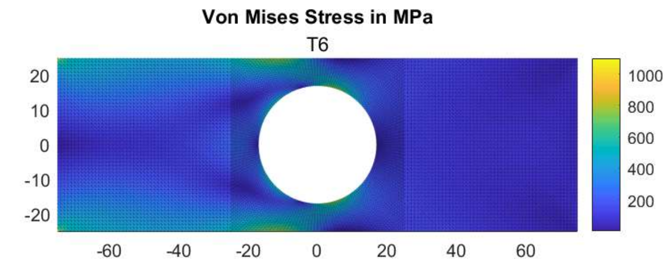
Medium (ey = 16)	
Fine (ey = 50)	

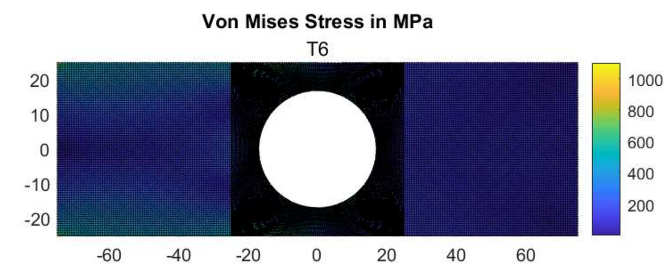
6-node triangle

a) contour plot of the von Mises (equivalent) stress

Number of Elements	Contour Plots
Coarse (ey = 2)	 

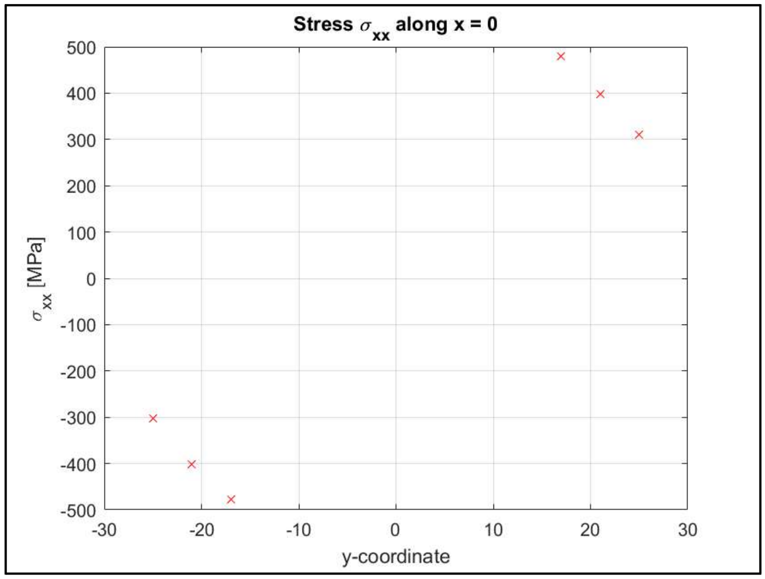
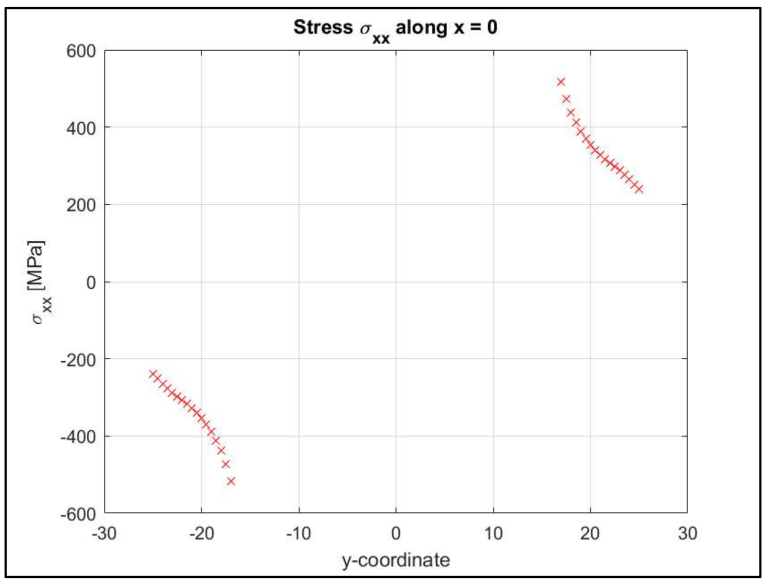
Medium (ey = 16)	 

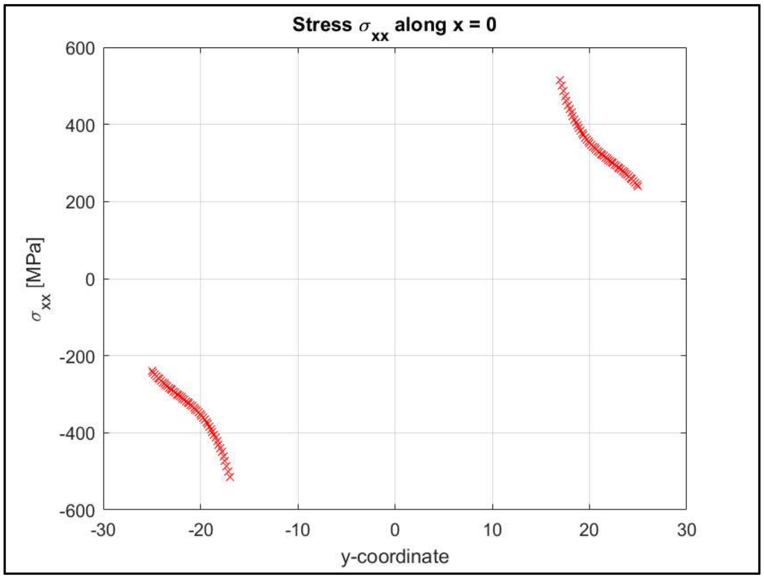
<table>
<tr>
<td>Fine (ey = 50)</td>
<td>




</td>
</tr>
</table>

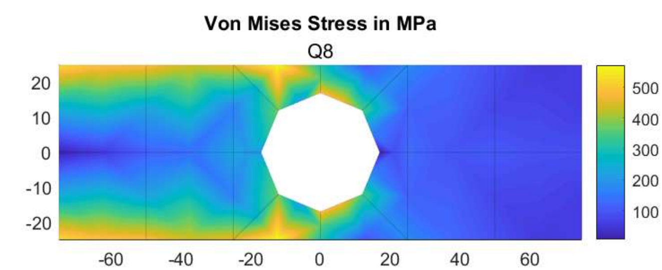
b) a line plot of the normal stress along cross-section AA

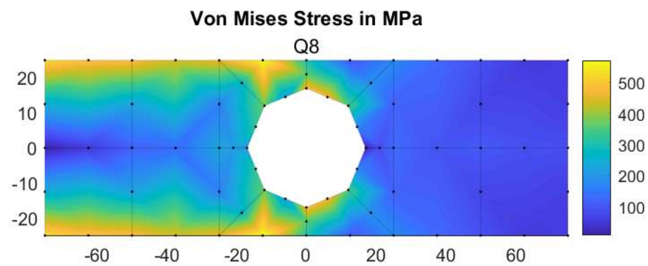
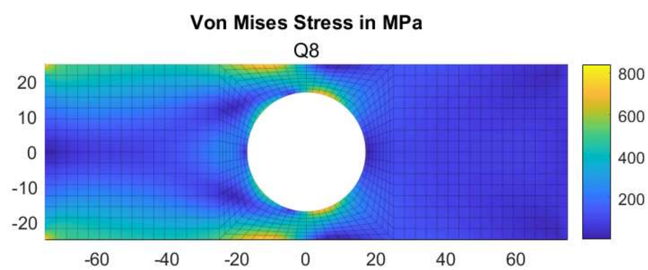
Number of Elements	Line Plots
Coarse (ey = 2)	
Medium (ey = 16)	

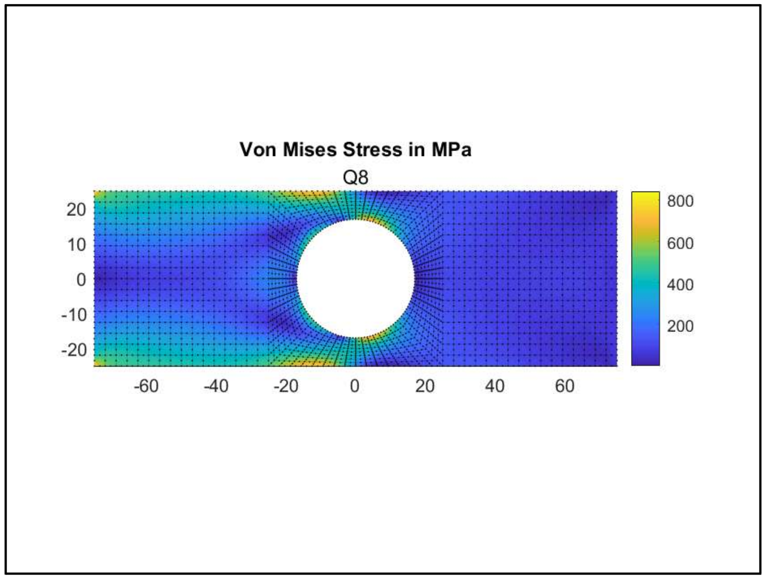
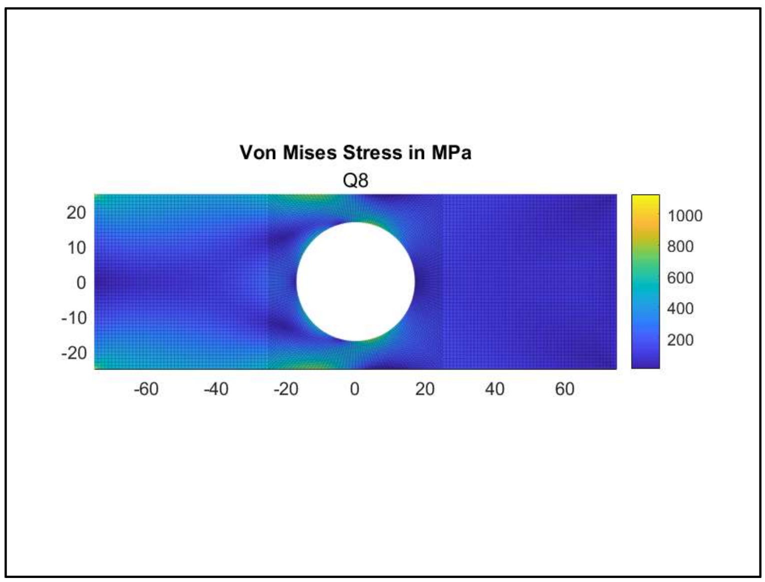
Fine (ey = 50)	

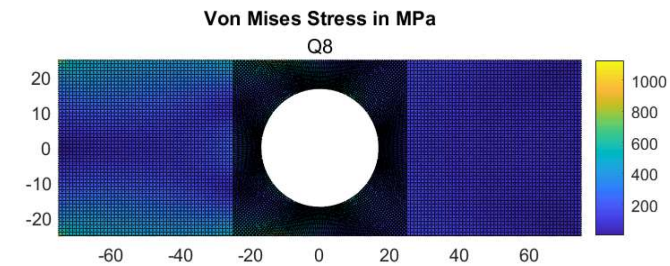
8-node quadrilateral

a) contour plot of the von Mises (equivalent) stress

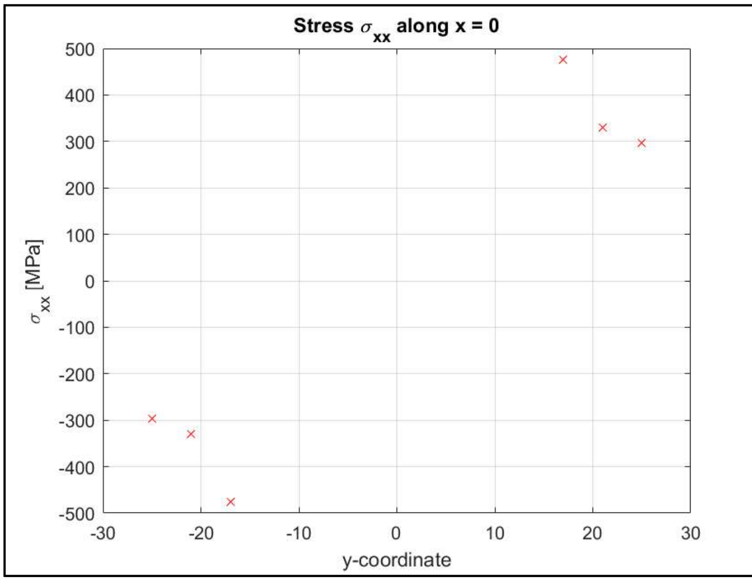
Number of Elements	Contour Plots
Coarse (ey = 2)	

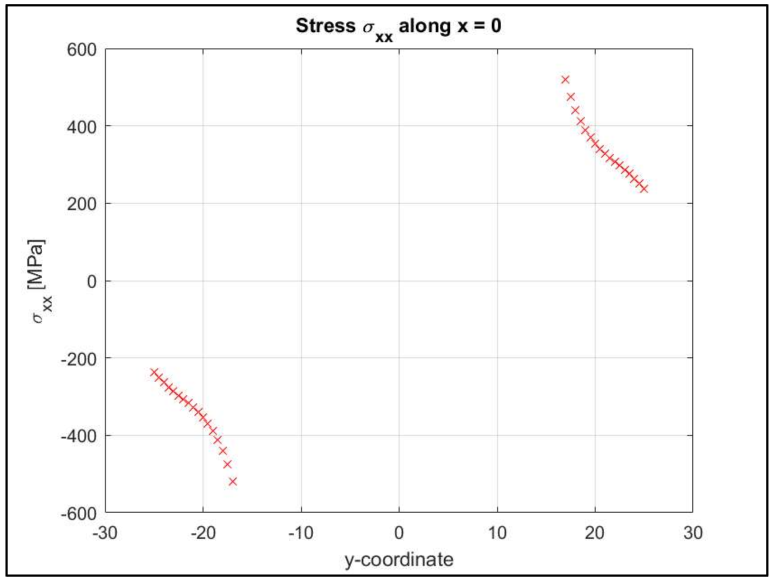
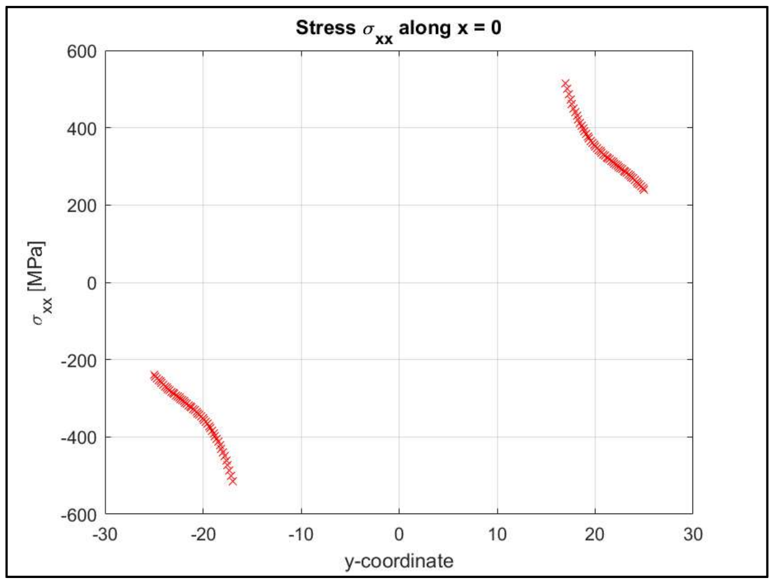
<table>
<tr><td></td><td>
</td></tr>
<tr><td>Medium (ey = 16)</td><td>
</td></tr>
</table>

	Von Mises Stress in MPa Q8 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 800 600 400 200
Fine (ey = 50)	Von Mises Stress in MPa Q8 20 10 0 -10 -20 -60 -40 -20 0 20 40 60 1000 800 600 400 200

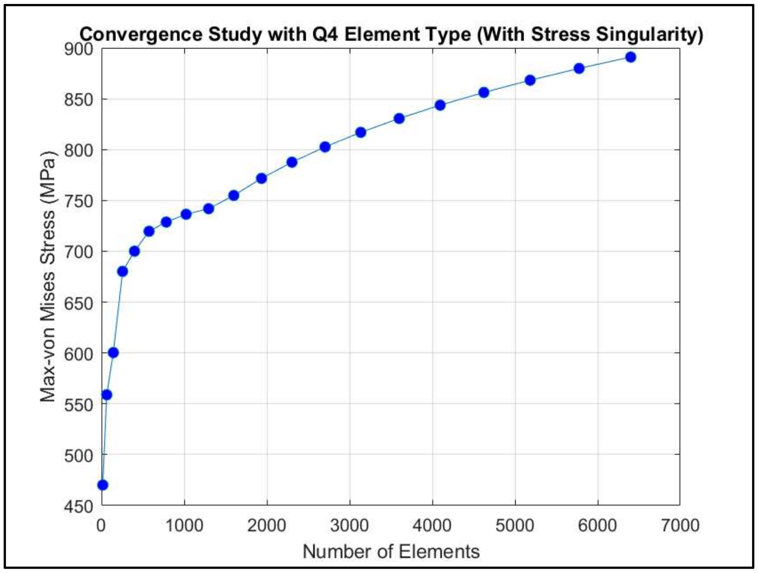
b) a line plot of the normal stress along cross-section AA

Number of Elements	Line Plots
Coarse (ey = 2)	

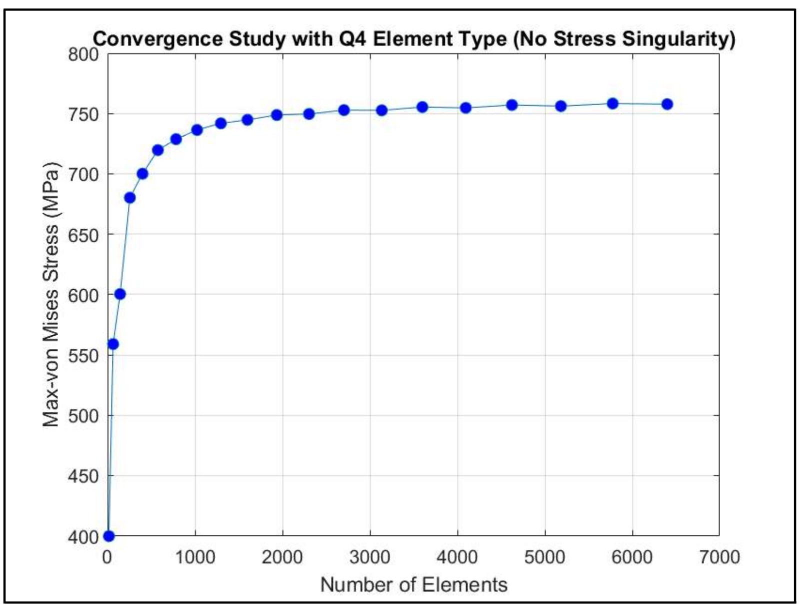
Medium (ey = 16)	
Fine (ey = 50)	

Task 6. Convergence Analysis

A mesh convergence study was performed for the Q4 element type. The max von Mises stress was calculated with a different number of elements and graphed below:



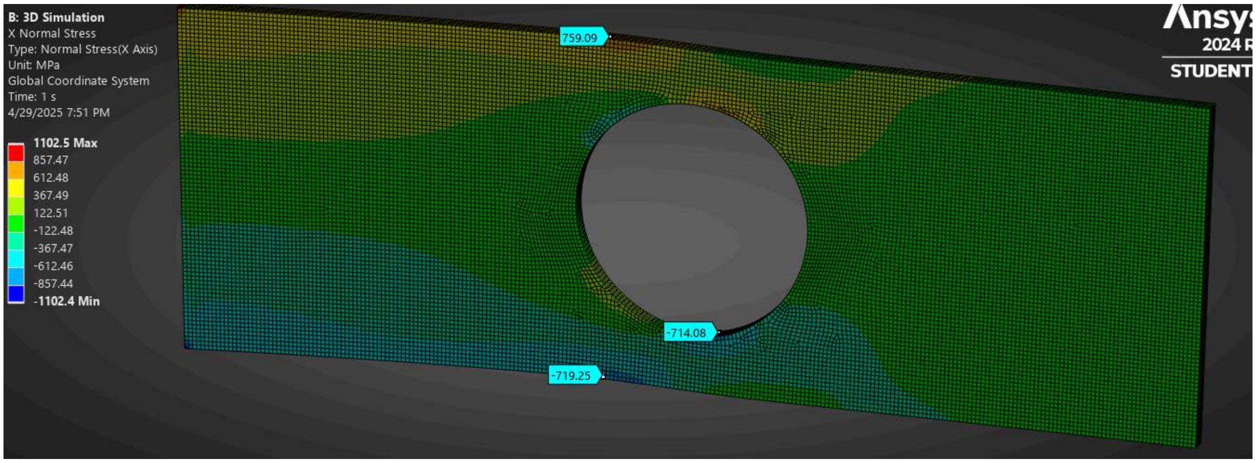
This graph did not converge since there was a stress singularity on the left side of the plate where there was a sharp corner and a fixed end. The convergence study was then done excluding elements 5 mm from the left edge. This one did converge as shown below:



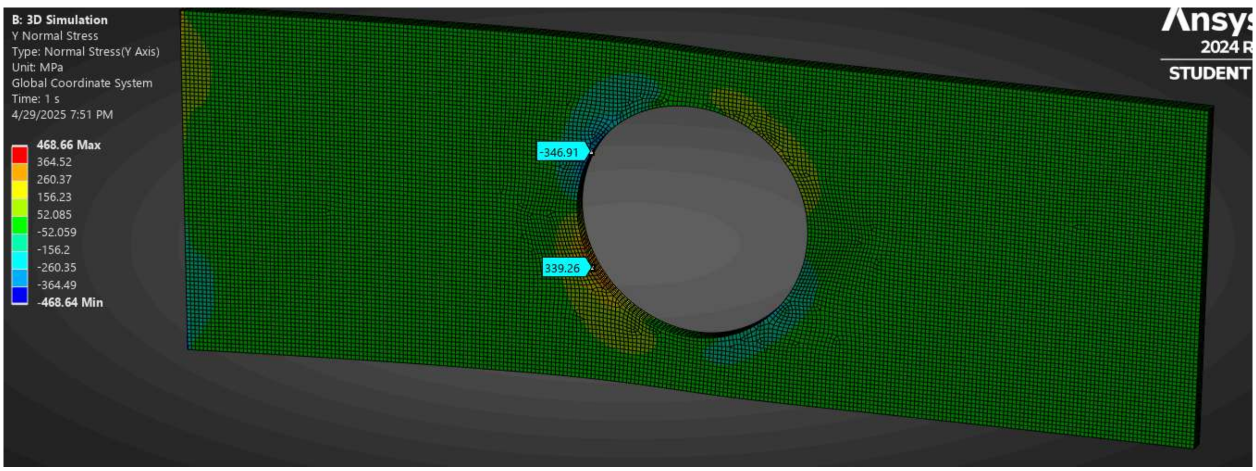
Task 7. 3D Finite Element Analysis

To determine whether a plane stress or plane strain assumption is more appropriate the in-plane and out-of-plane stress and strain components were found and plotted below.

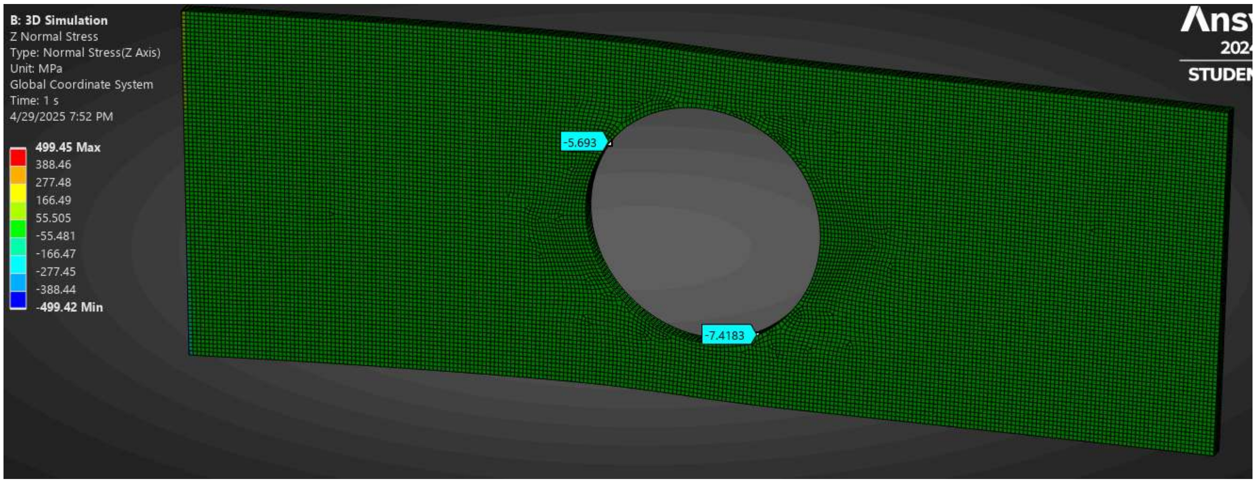
X Normal Stress



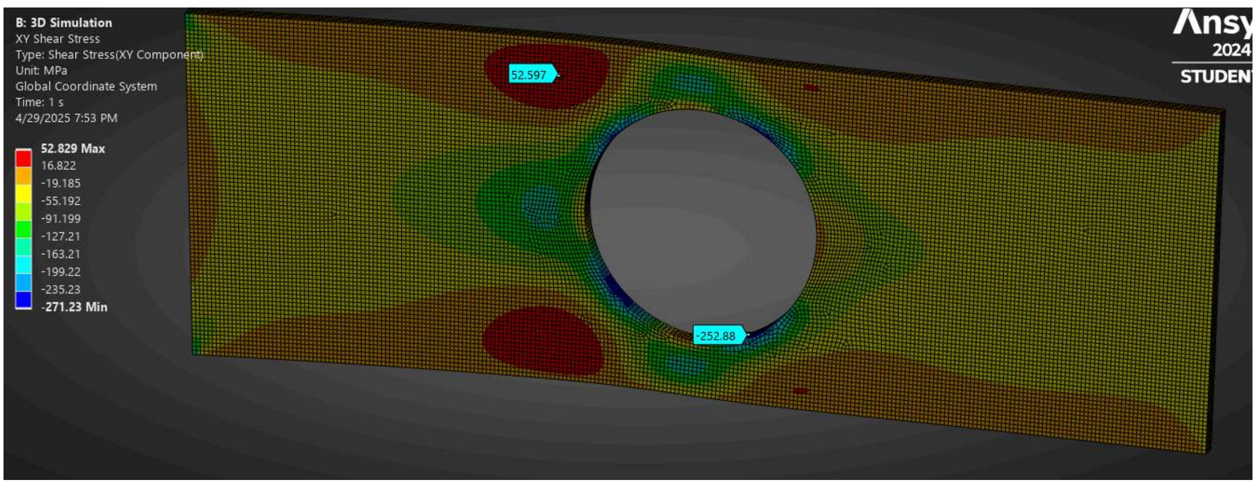
Y Normal Stress

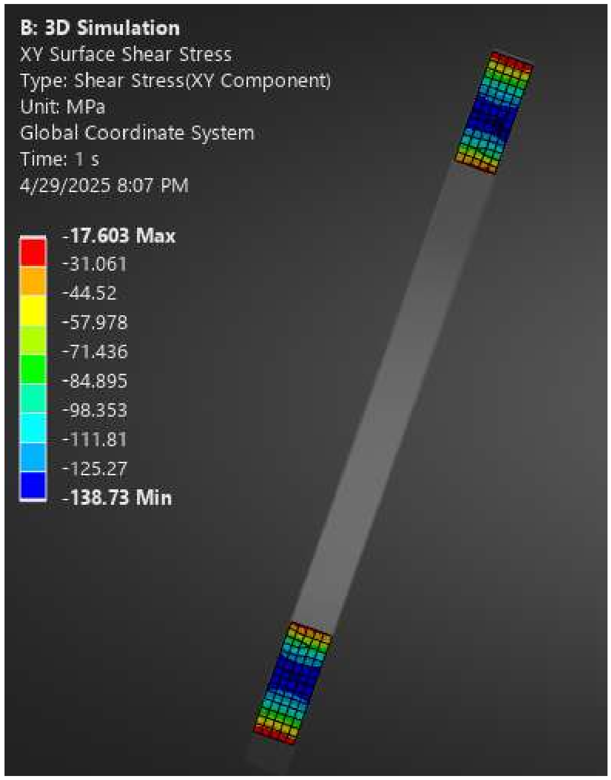


Z Normal Stress

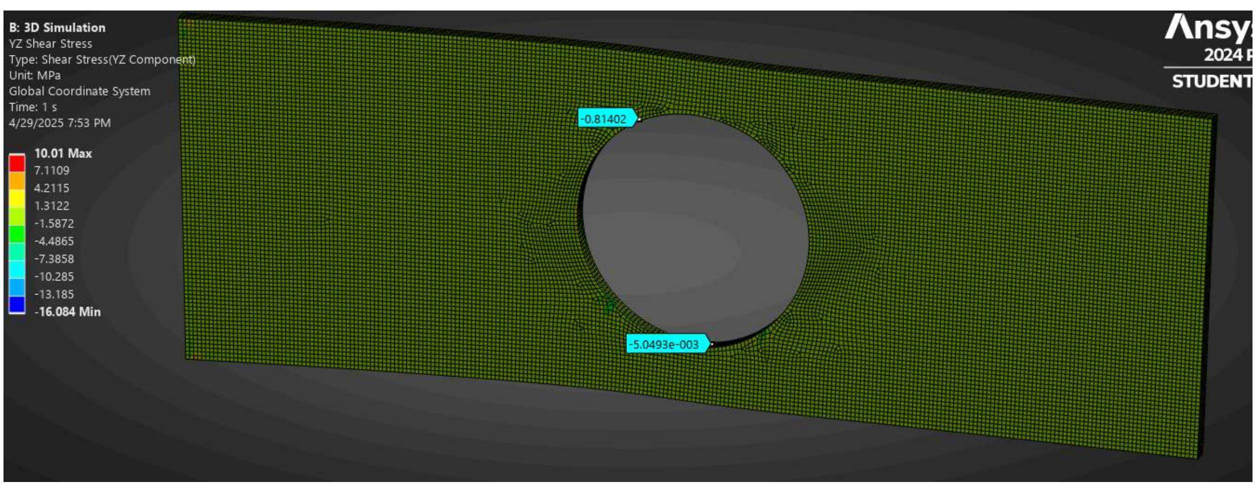


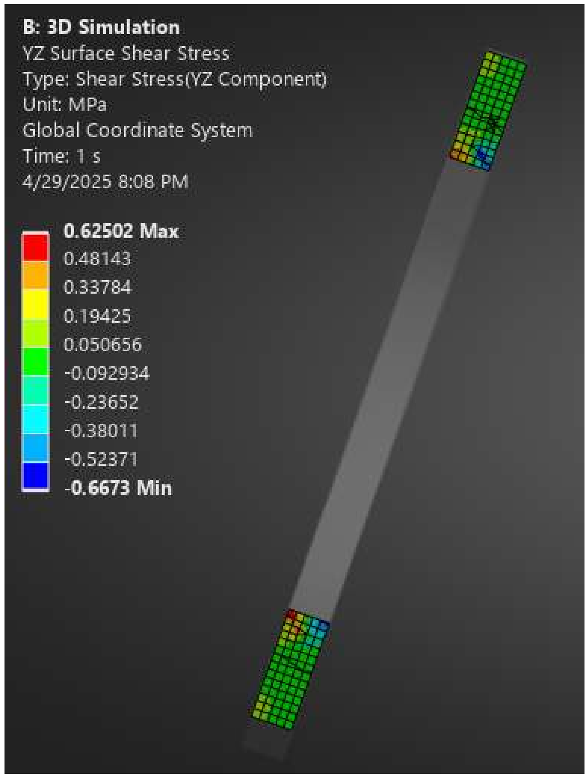
XY Shear Stress



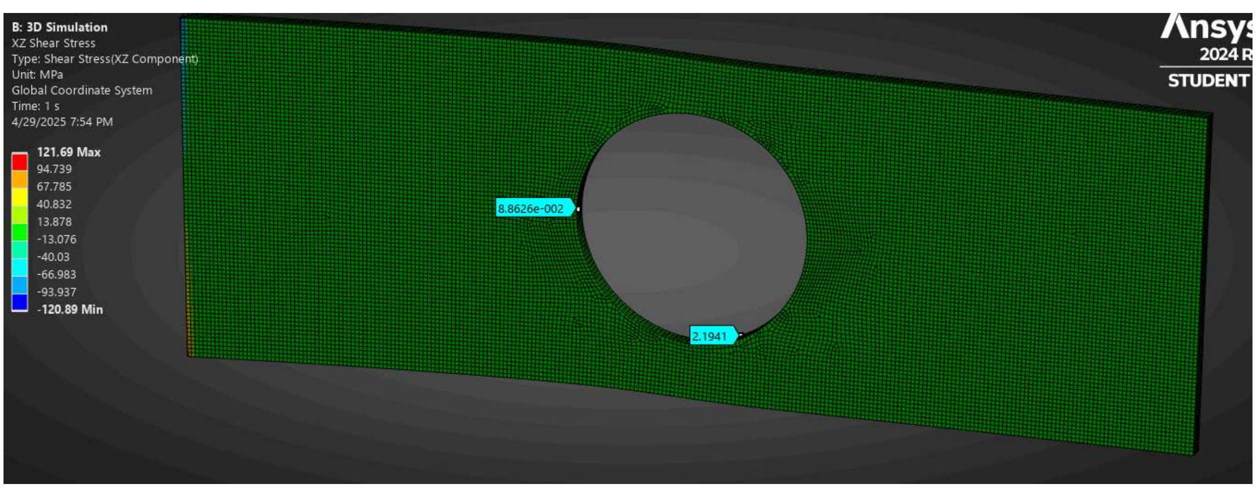


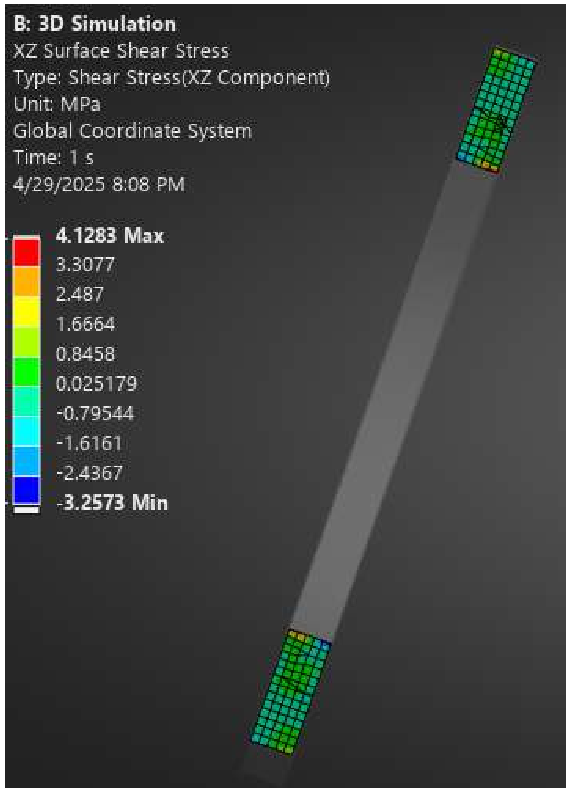
YZ Shear Stress



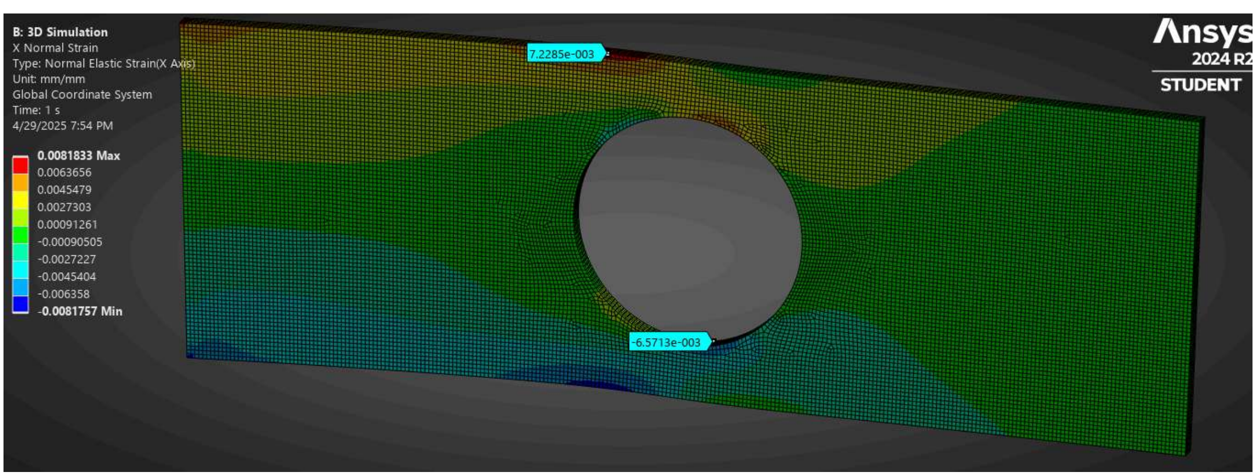


XZ Shear Stress

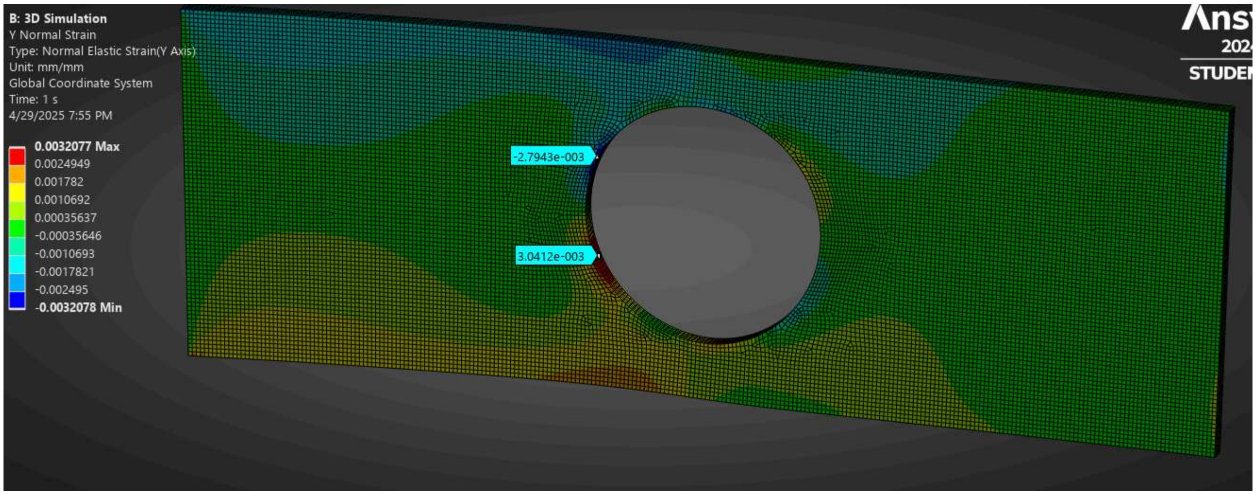




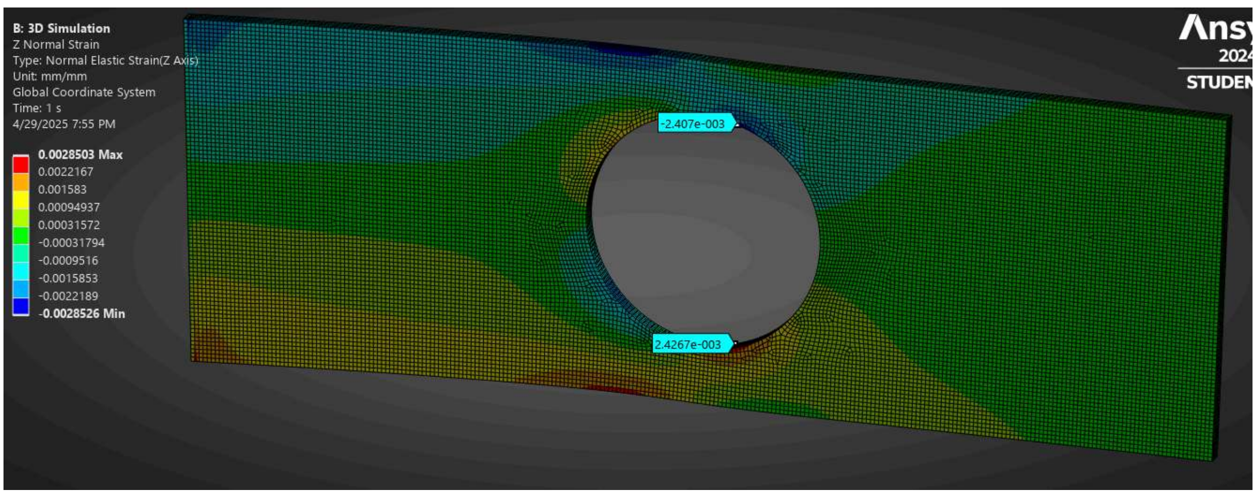
X Normal Strain



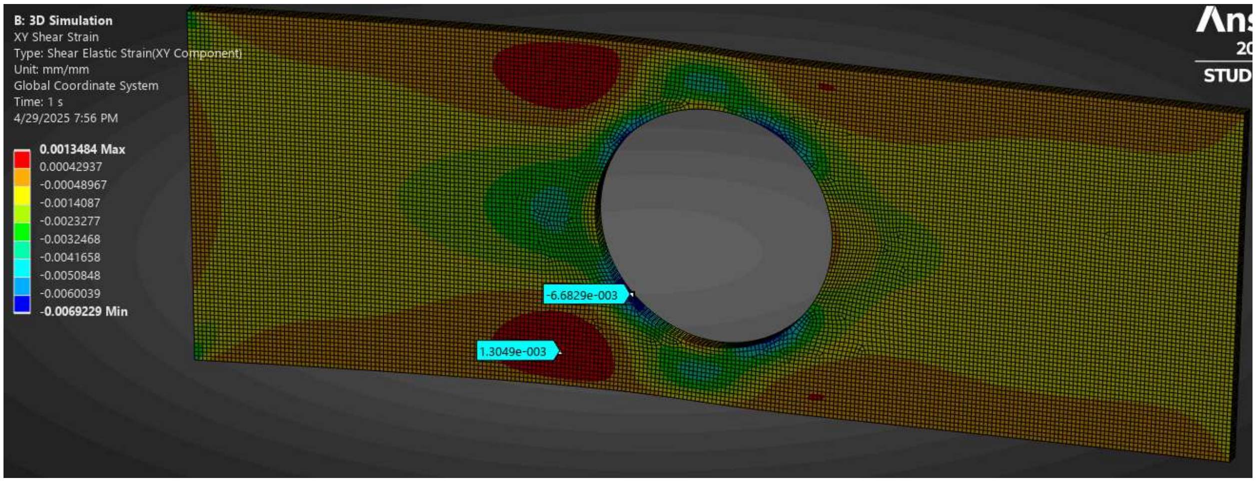
Y Normal Strain

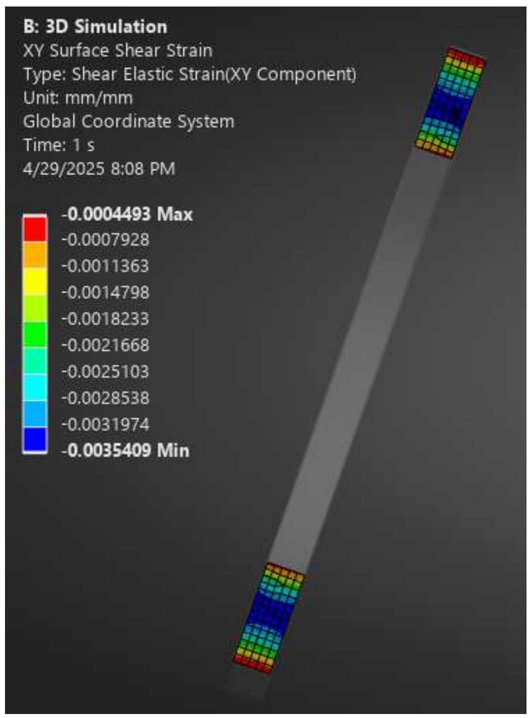


Z Normal Strain

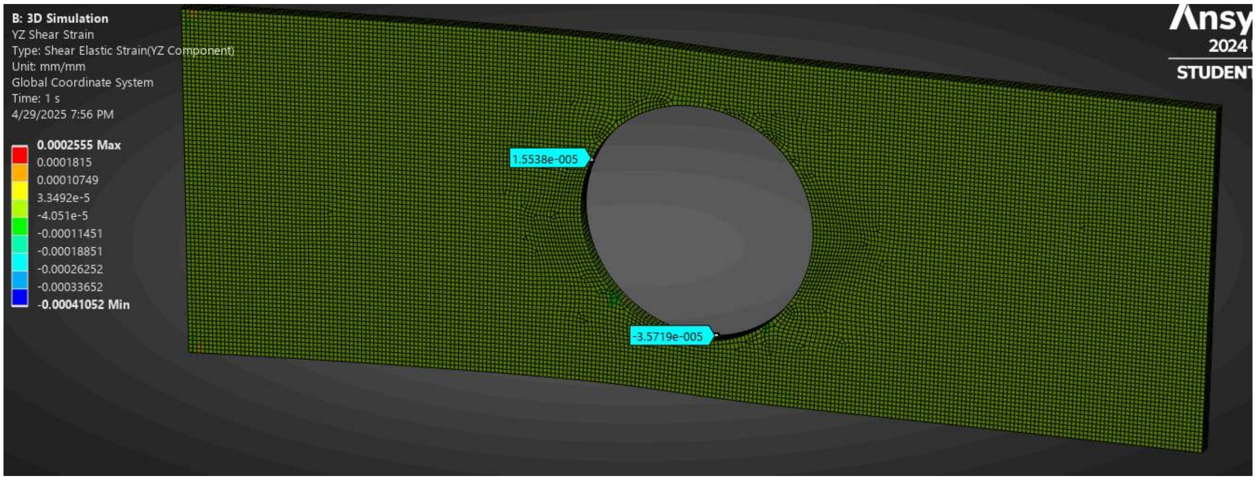


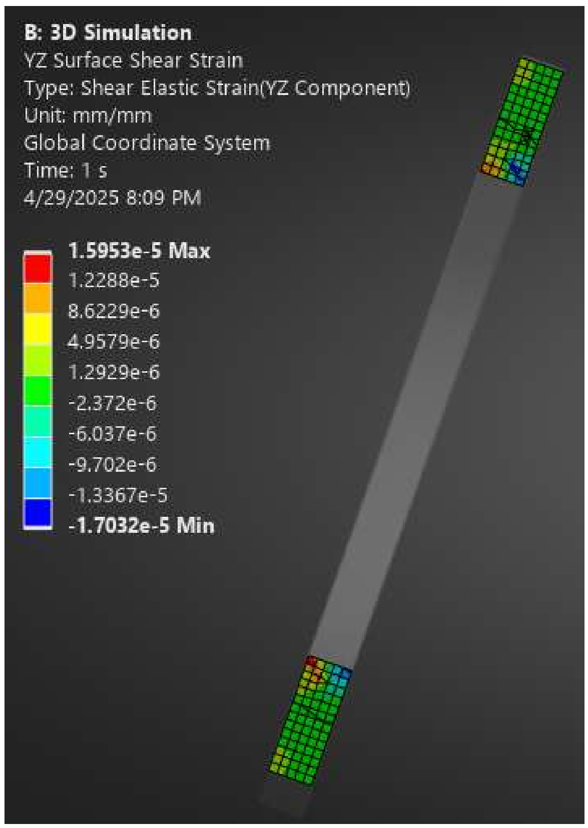
XY Shear Strain



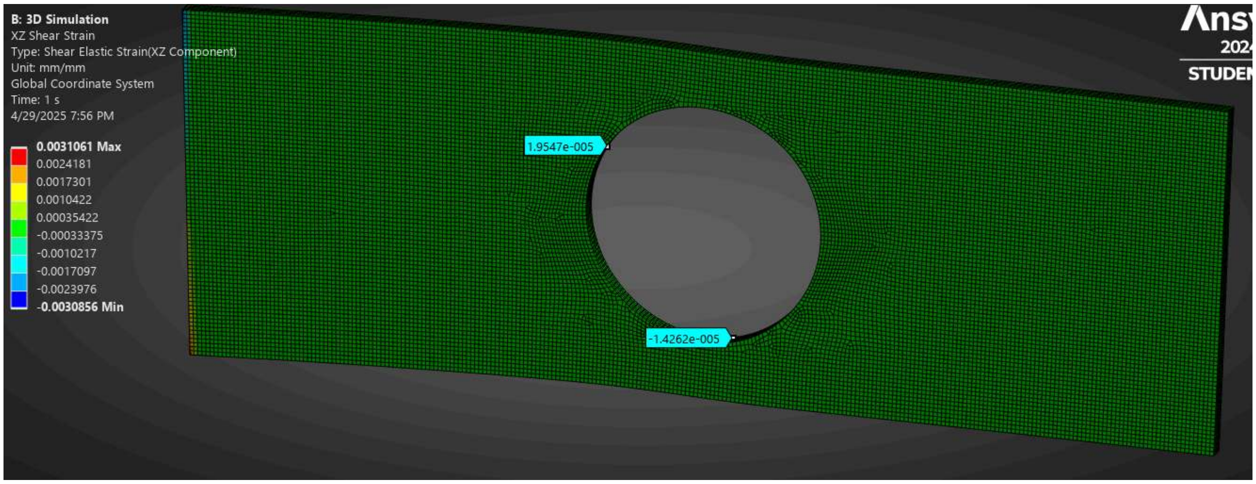


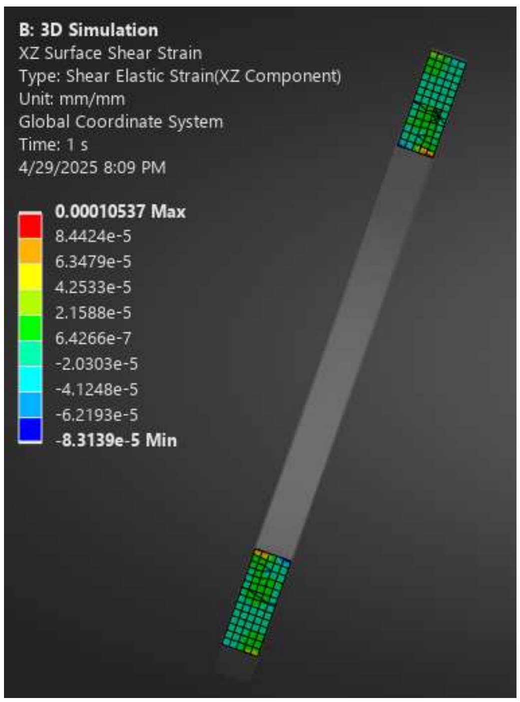
YZ Shear Strain





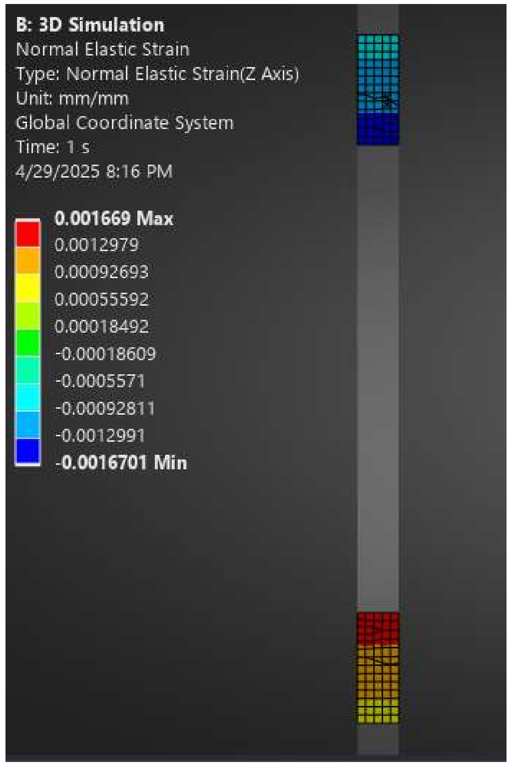
XZ Shear Strain



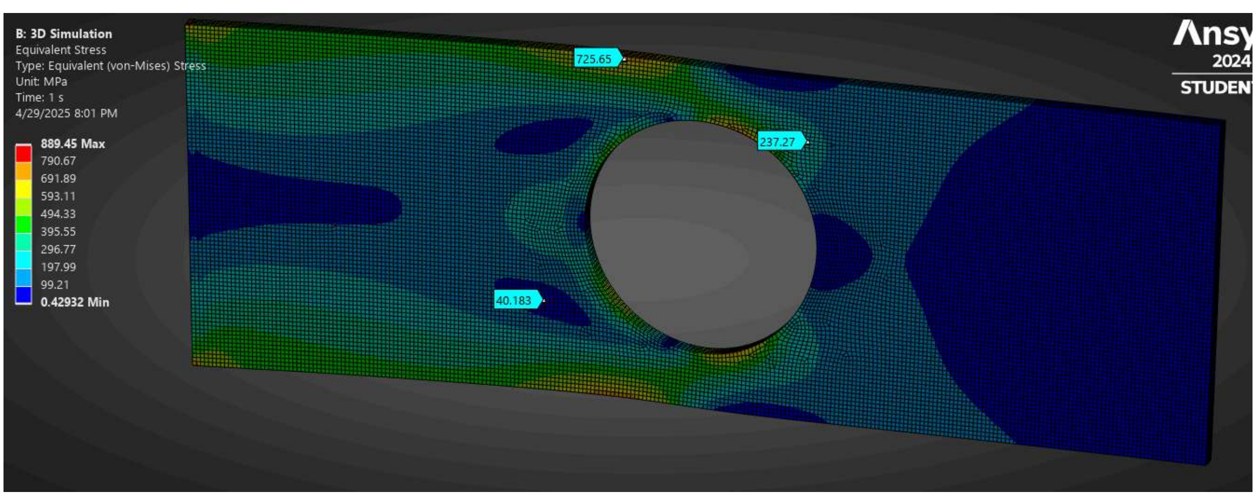


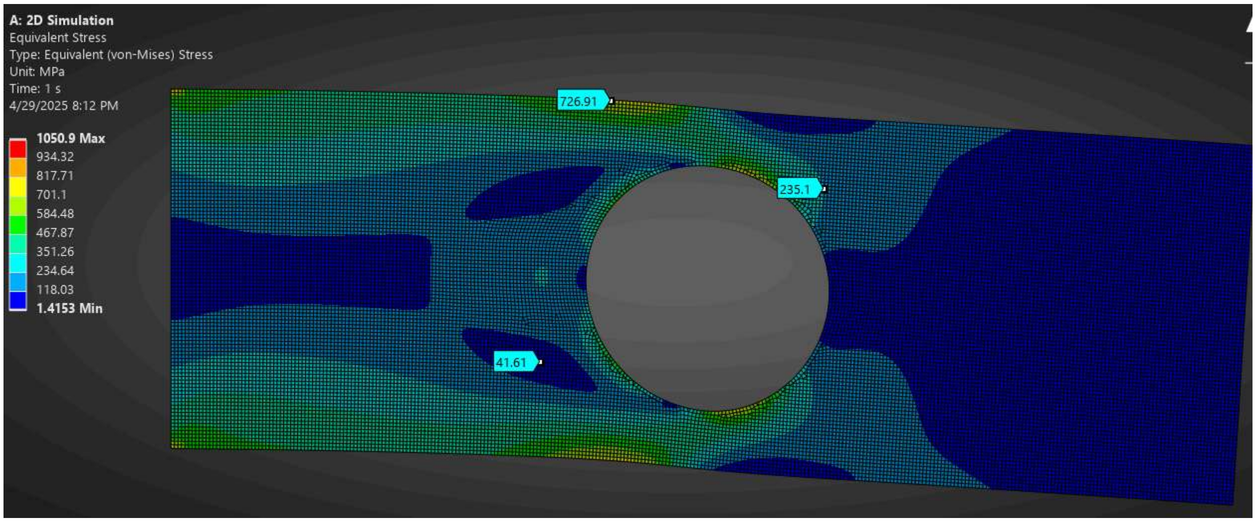
When looking at all the stress and strain contour plots the out-of-plane results were mostly negligible compared to in-plane. For some of the out-of-plane contour plots there looked to be significant values but those occurred at a stress singularity. The only out-of-plane result that gave significant values was the Z normal strain. This shows that it is a plane stress problem. Plane stress is used when the thickness is small and the material is free to expand or contract in the Z direction, meaning the out-of-plane strain is significant and the stress in that direction is negligible.

To determine whether the simulation can be done in 2D or if it needs to be done in 3D the strain in the Z direction was looked at. Since the other out-of-plane stresses and strains are negligible then they would not matter in a 2D simulation. The normal strain in the Z direction did matter so it is important to see if it changes along the thickness. Below is a screenshot of the normal strain in the Z direction.



This shows that the strain does not change with the thickness and therefore a 2D simulation can be done and a full 3D analysis does not need to be done. To further show this, the 2D & 3D von Mises stress are shown below with a probe at relatively the same spot. In both of these contour plots the values are similar (the contour plots range is different between the two and the colors will not be the exact same for both).

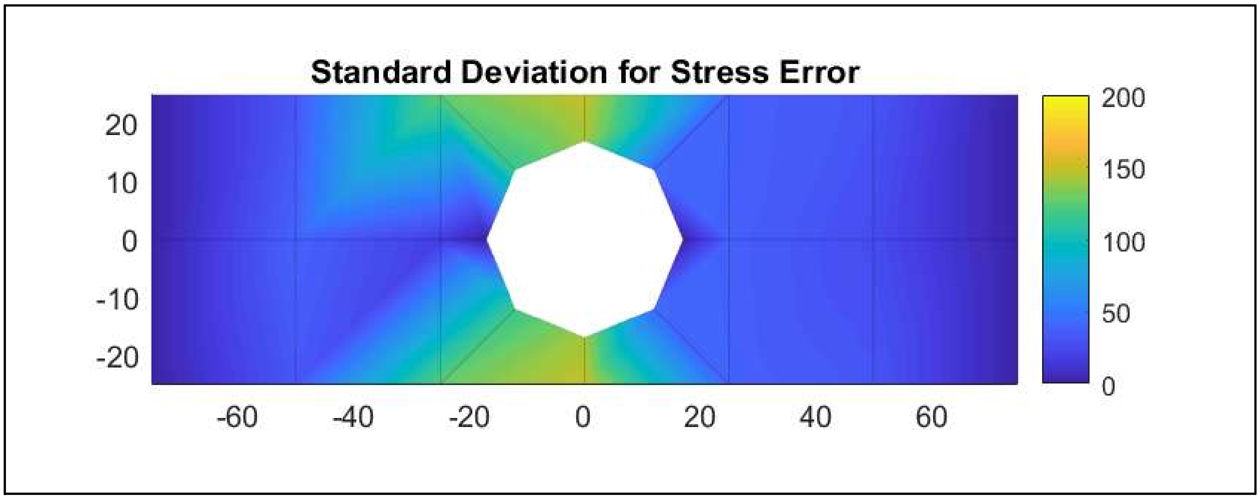



A: 2D Simulation
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1 s
4/29/2025 8:12 PM
1050.9 Max
934.32
817.71
701.1
584.48
467.87
351.26
234.64
118.03
1.4153 Min
726.91
235.1
41.61

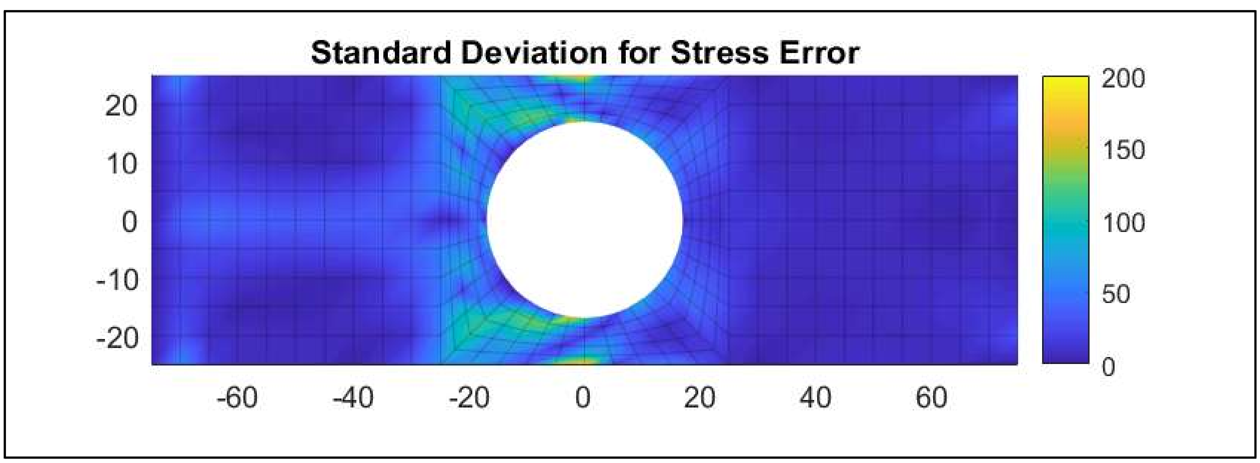
Task 8. Stress Error Estimation Function

In order to calculate the stress error at each node, the standard deviation of the unaveraged von Mises stress was calculated and plotted across the plate. The below plot shows the stress error for the Q4 element type and will decrease in element size. The first plot will be when ey is equal to 2 elements and go up to ey equaling 50 elements. As the number of elements increases the standard deviation will go to zero. You can see that originally the areas with a higher stress error lie towards the center of the plate around the hole and to further reduce this error, a finer mesh can be applied around the hole to minimize the standard deviation between nodes. To see this clearly the same scale was applied for the contour range and as the mesh becomes more defined the contour becomes darker showing a smaller standard deviation and lower error.

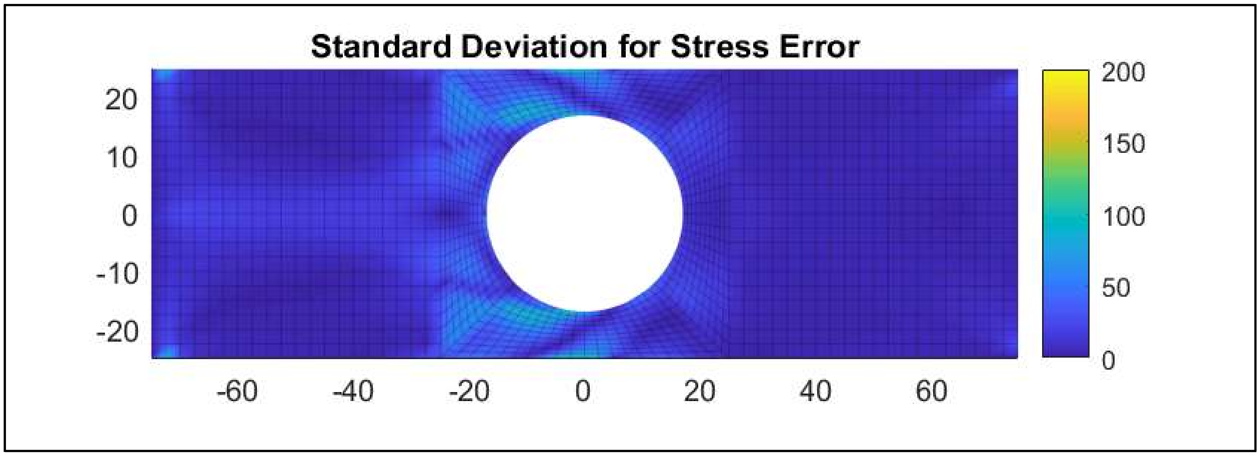
ey = 2



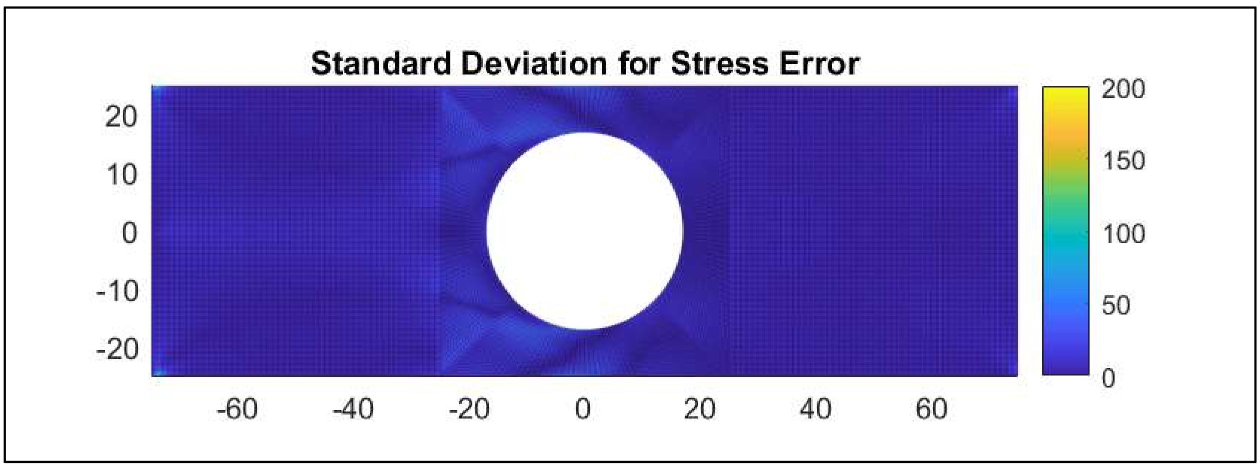
ey = 10



ey = 20

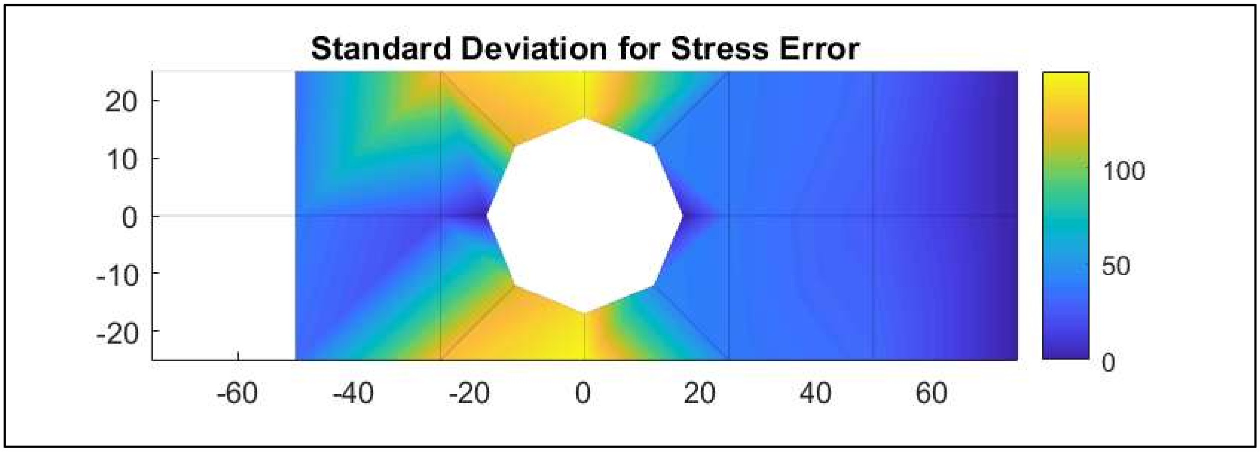


ey = 50

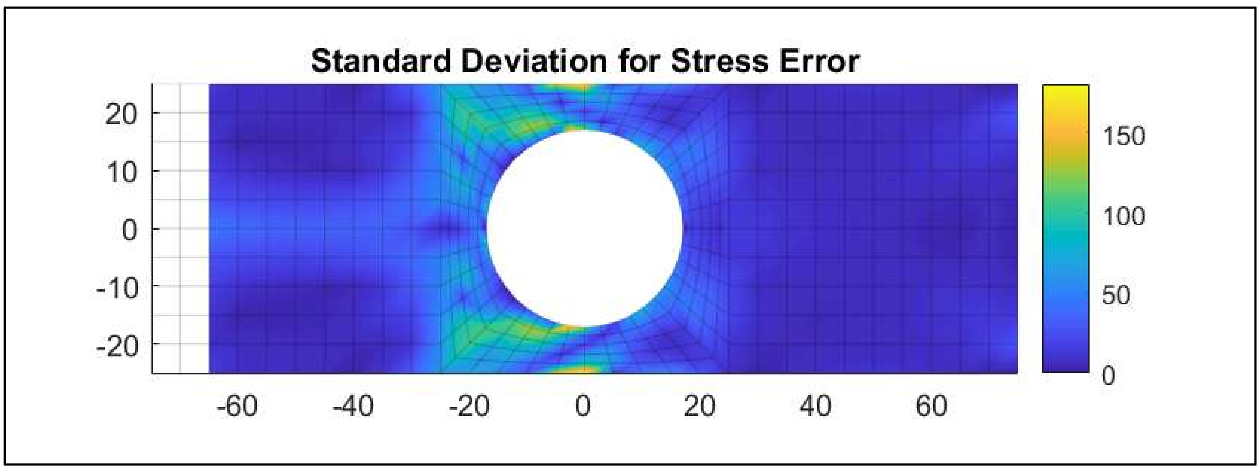


One important note is that near the stress concentration there is a concentration there as well which is why there is a bright spot on the contour. Near the top and bottom of the plate on the left side no matter how small the elements get it will not decrease the error. The stress error analysis was done again without the inclusion of elements within 5 mm of the left side to show the decrease in standard deviations with smaller element size.

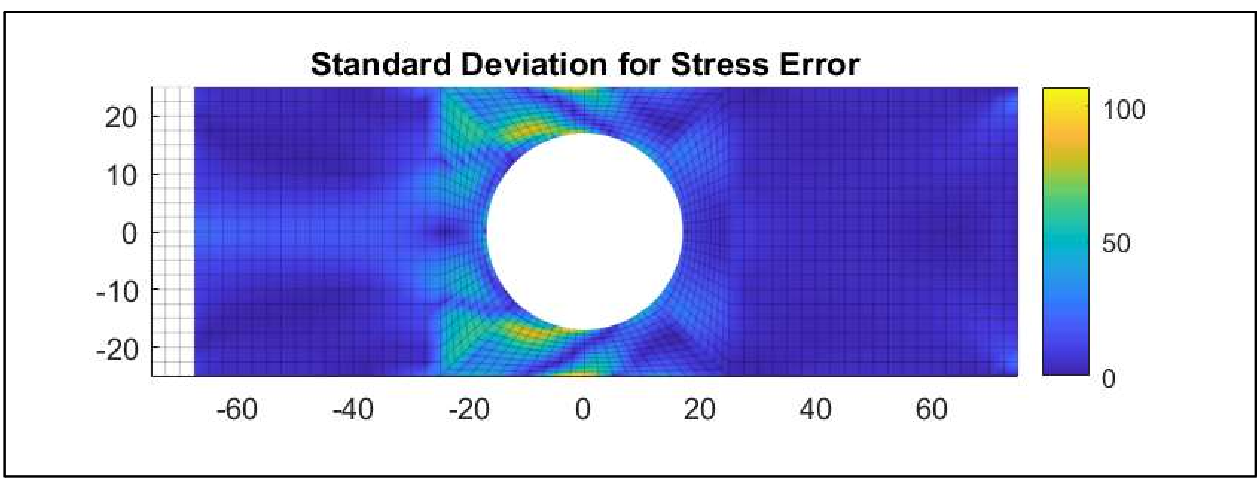
ey = 2



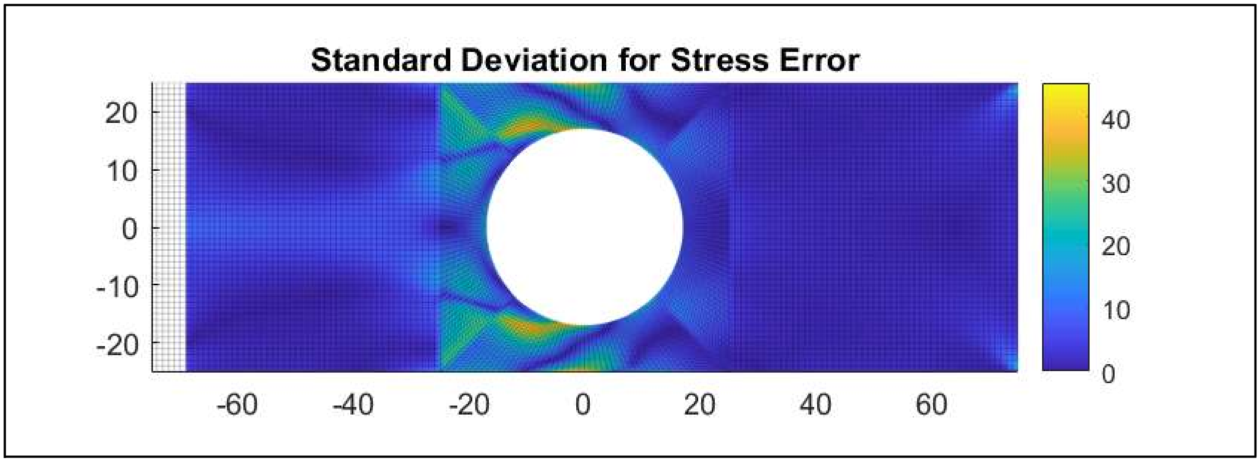
ey = 10



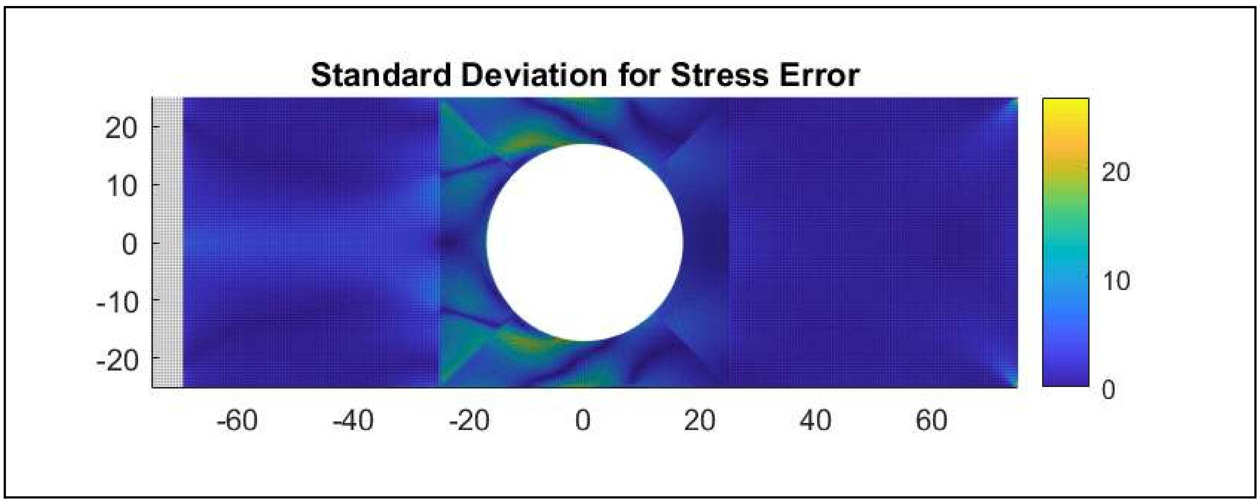
ey = 20



ey = 50



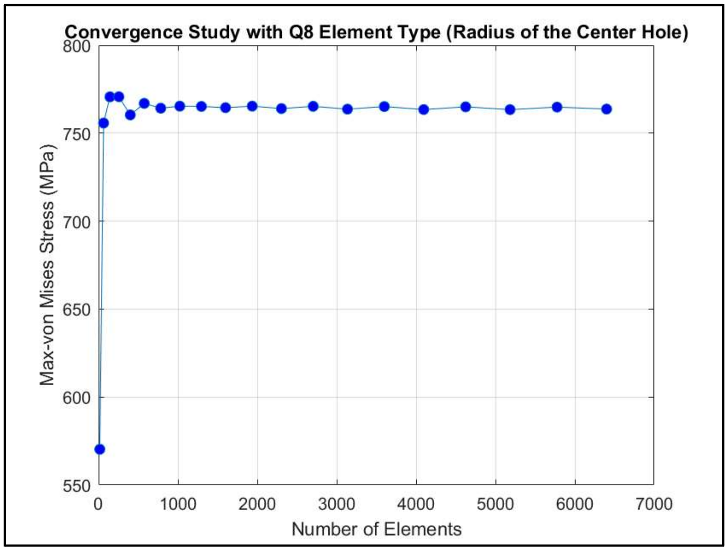
ey = 100



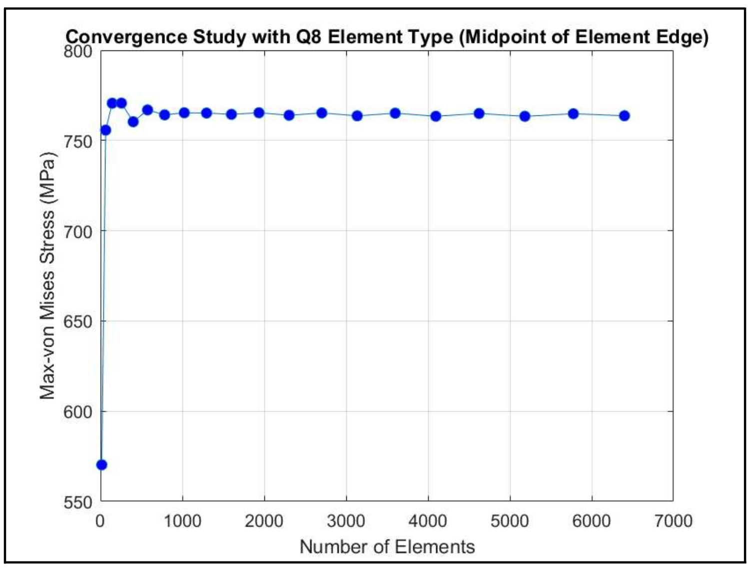
For these contour plots the range of the contour was not restricted to show that the critical region stayed the same no matter how small the elements got but the standard deviation did decrease. Originally with ey=2 the contour range was 0-150. When ey=100 the contour range was much smaller and was 0-25. This shows that if the mesh was to continue getting finer then eventually the standard deviation would approach 0.

Task 9. Accuracy of quadratic edges

The convergence study was done again with the Q8 element type. The graph below shows what would happen if the midside node was placed in the radius of the center hole.



The next plot shows the convergence study if the midside nodes are placed at the midpoint of the element edge.



Both of these results are the same and did not affect the convergence graph. We believe this is due to the max von Mises not occurring at the hole with the type of problem we have. Instead the max von Mises is likely to occur at the edge of the plate as shown earlier so this change would not affect the results of the convergence graph. If the problem was set up so that the max von Mises would occur at the hole then what is likely to happen is the situation where the midside nodes placed on the radius of the hole would converge quicker then if the midside nodes are at the midpoint of the element edge.