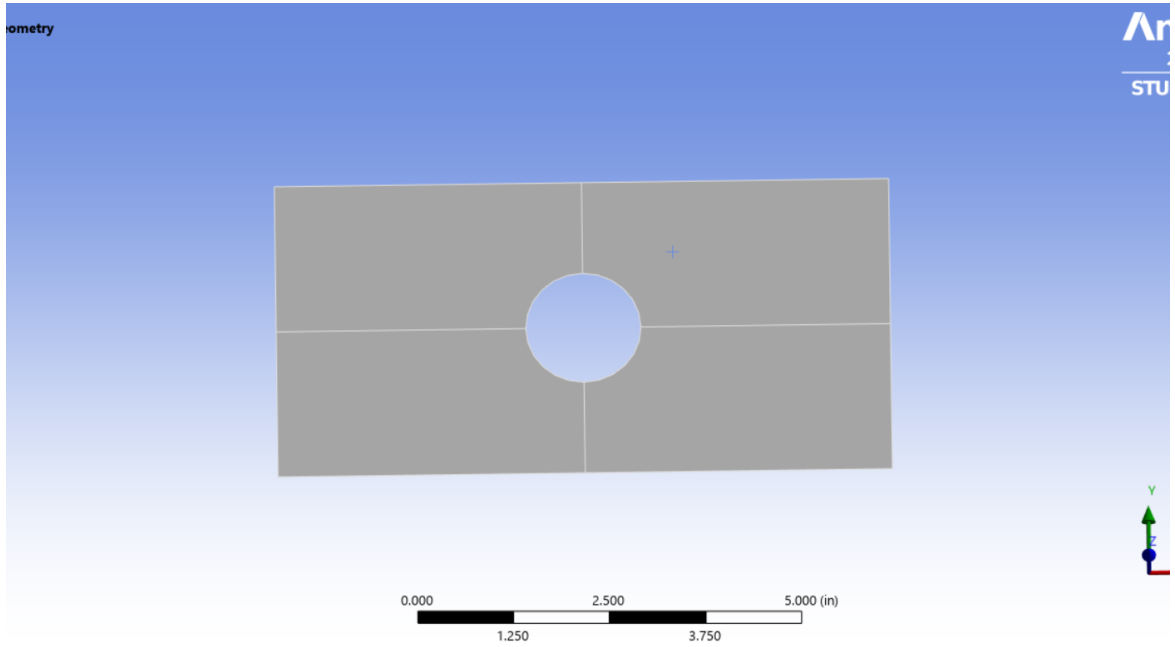
MECH-6605
FINITE ELEMENT METHODS IN ENGINEERING

FE Analysis – Mesh Evaluation and Correlation Assignment

Yashwanth Kanthala

00921675



objectives

1. Understand Mesh Sensitivity: Evaluate how mesh size and element type influence the accuracy of finite element results.
2. Verify Accuracy of FEM: Compare FEM results with theoretical stress values to validate model correctness.
3. Study Stress Concentrations: Focus on predicting stress magnification around the hole in a plate under load.
4. Learn Element Behavior: Analyze how linear (4-node) and quadratic (8-node) elements perform under various mesh densities.
5. Develop Best Practices: Establish guidelines for mesh quality and density for future engineering simulations.
6. Build Confidence in CAE Tools: Enhance skills in setting up, solving, and interpreting FEA simulations in Ansys Workbench.

Element types

	Element Type	Description
1	**4-Node Linear Quadrilateral**	First-order elements with nodes at the four corners. Suitable for basic meshing, faster computation, but lower accuracy near stress concentrations.
2	**8-Node Quadratic Isoparametric**	Second-order elements with mid-side nodes. Provide more accurate stress distribution, especially useful in curved or high-gradient regions.

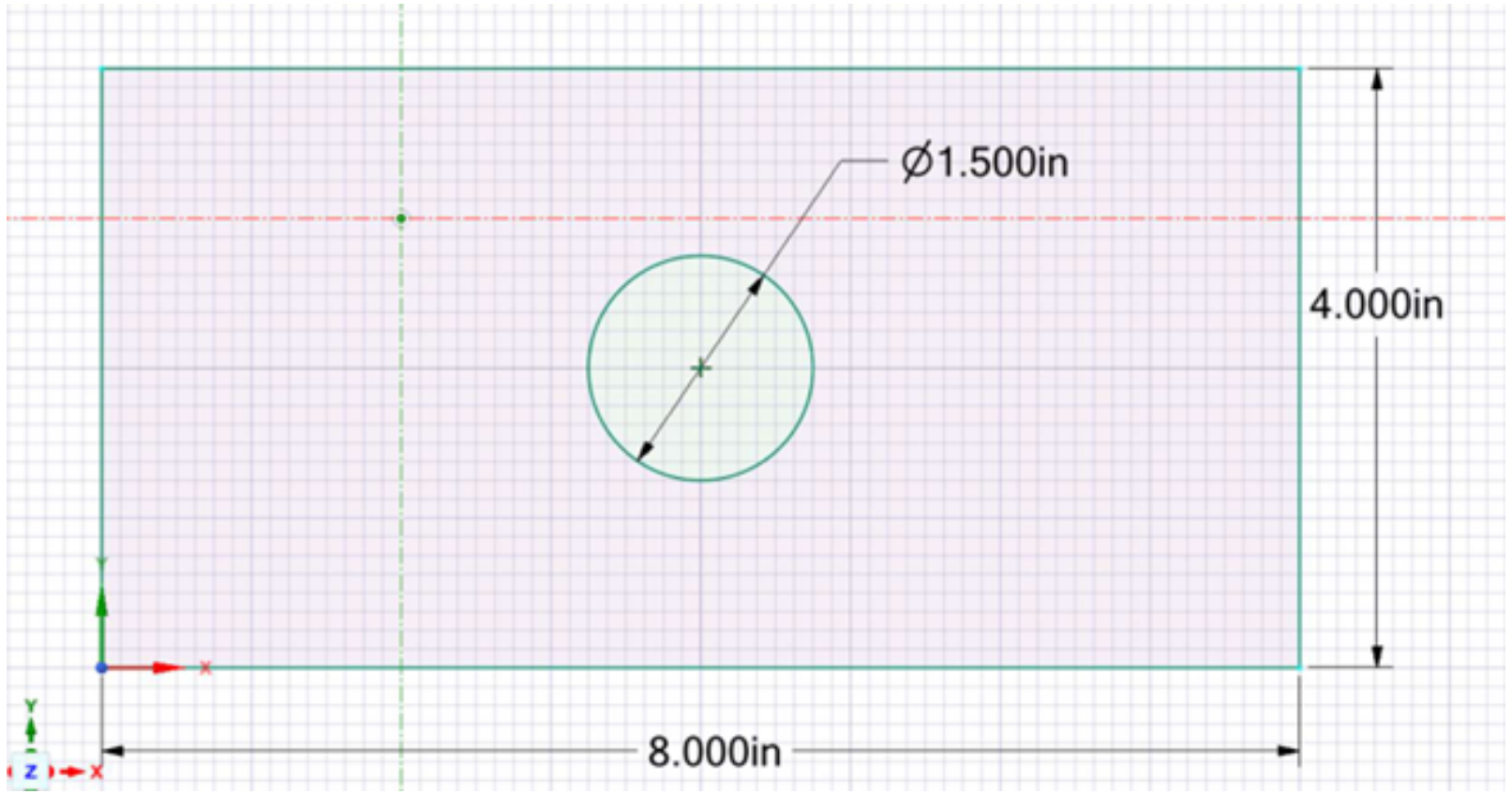
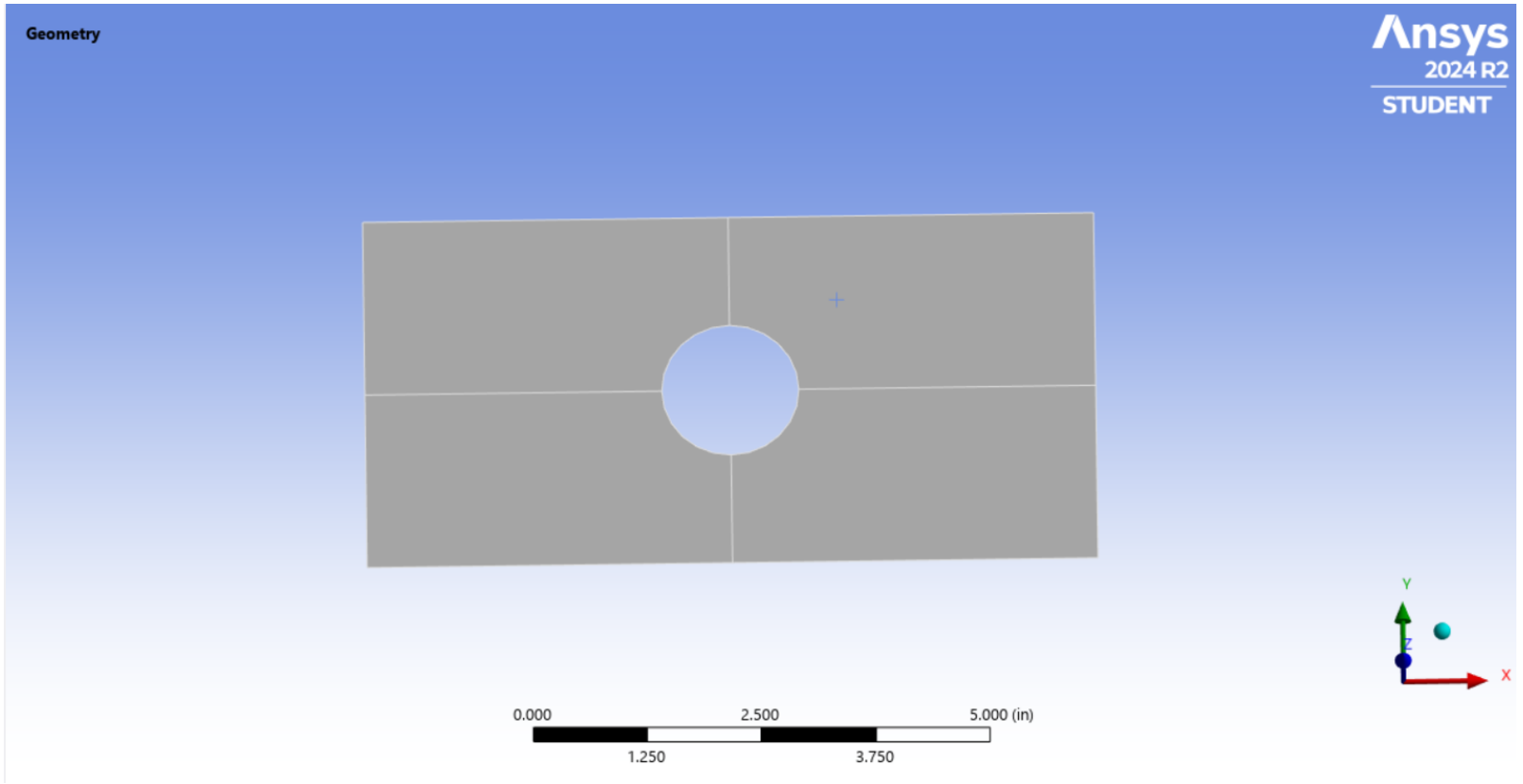


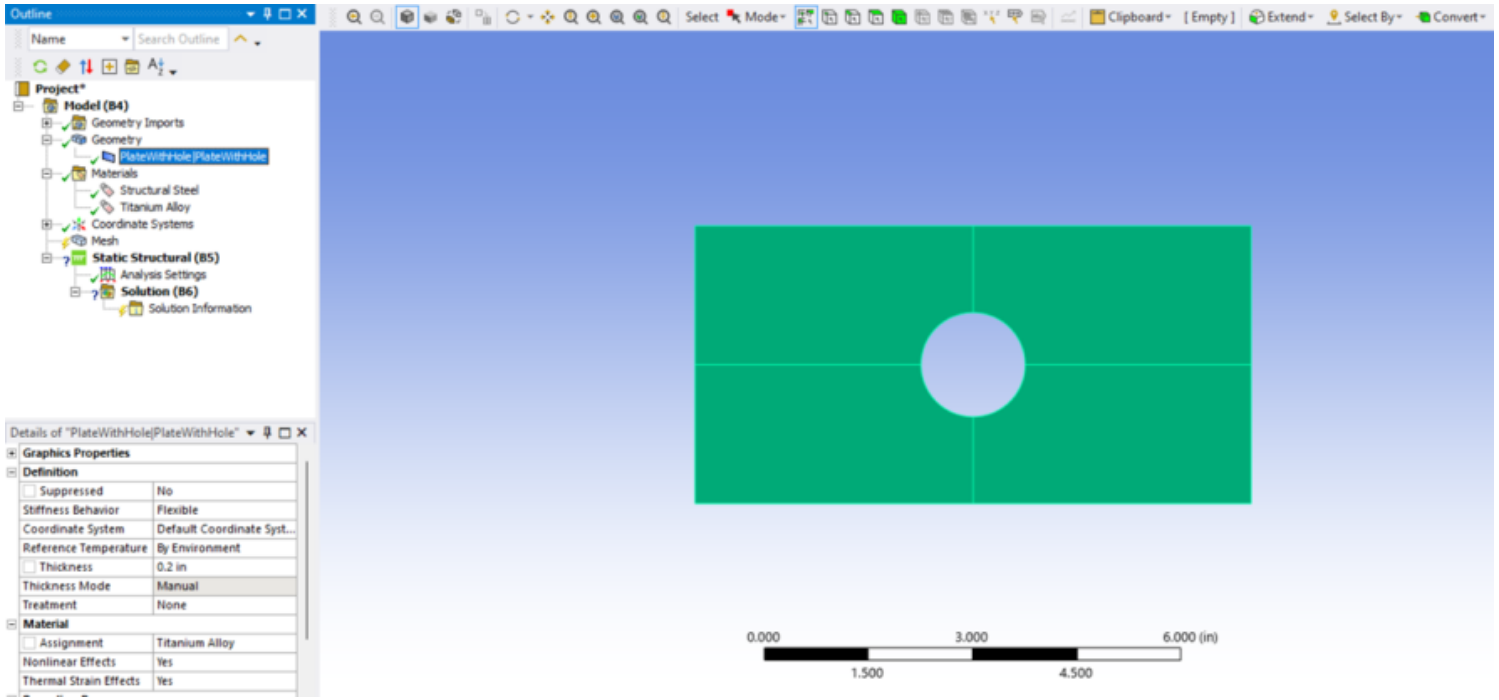
Plate dimensions


Geometry
Ansys
2024 R2
STUDENT
Y
Z
X
0.000
2.500
5.000 (in)
1.250
3.750

Geometry

Material assignment and thickness

In this I used the thickness of the plate as 0.2 inches and material as Titanium alloy



Materials properties

I used the the Titanium alloy as the material for the plate these are the properties of Titanium alloy

Titanium Alloy

Density	0.16691	lbm/in³

Structural

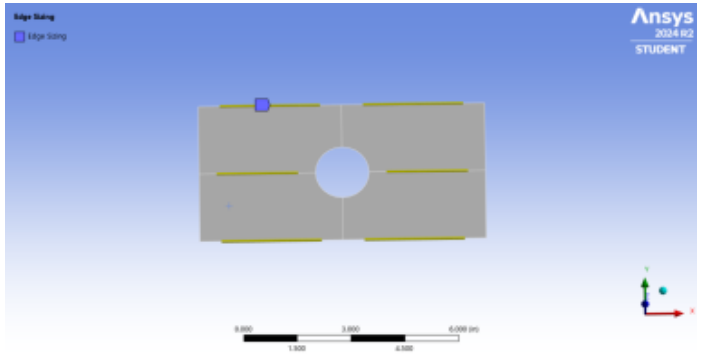
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	1.3924e+07	psi
Poisson's Ratio	0.36	
Bulk Modulus	1.6576e+07	psi
Shear Modulus	5.119e+06	psi
Isotropic Secant Coefficient of Thermal Expansion	5.2222e-06	1/°F
Compressive Ultimate Strength	0	psi
Compressive Yield Strength	1.3489e+05	psi
Tensile Ultimate Strength	1.5519e+05	psi
Tensile Yield Strength	1.3489e+05	psi

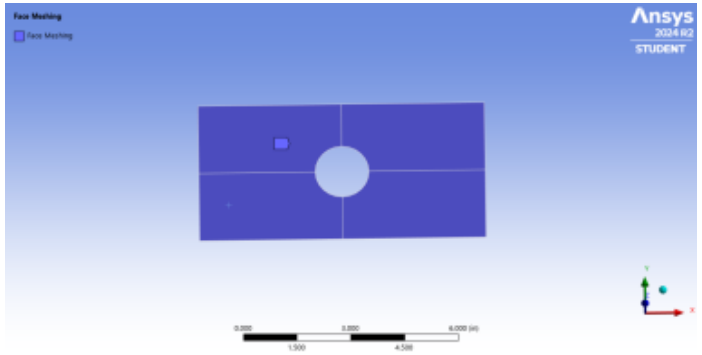
Thermal

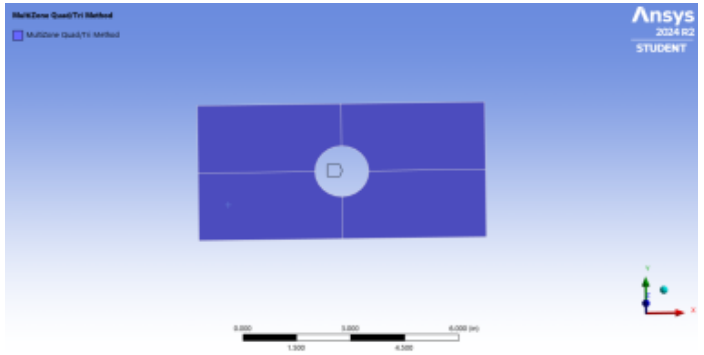
Isotropic Thermal Conductivity	0.00029291	BTU/s·in·°F
Specific Heat Constant Pressure	0.12468	BTU/lbm·°F

Electric

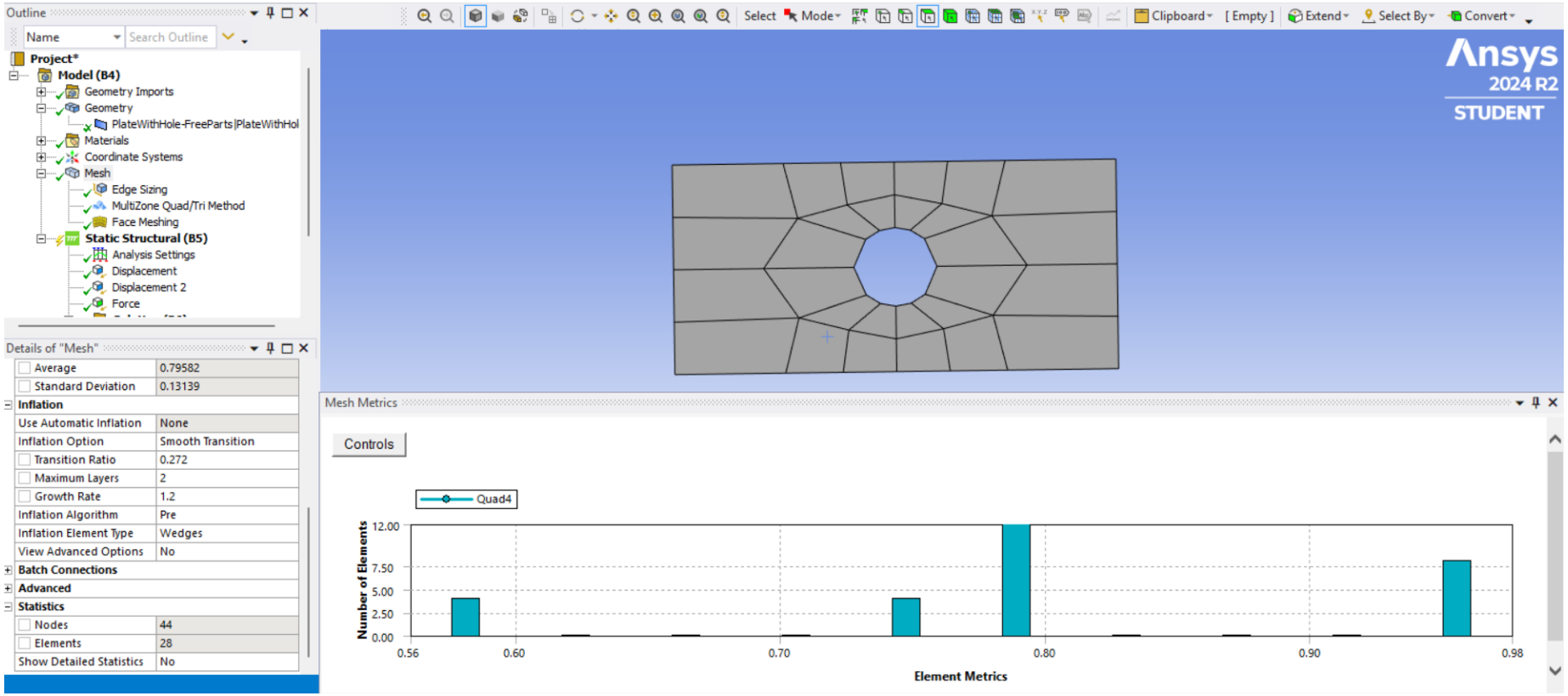
Isotropic Resistivity	85.235	ohm·cmil/in







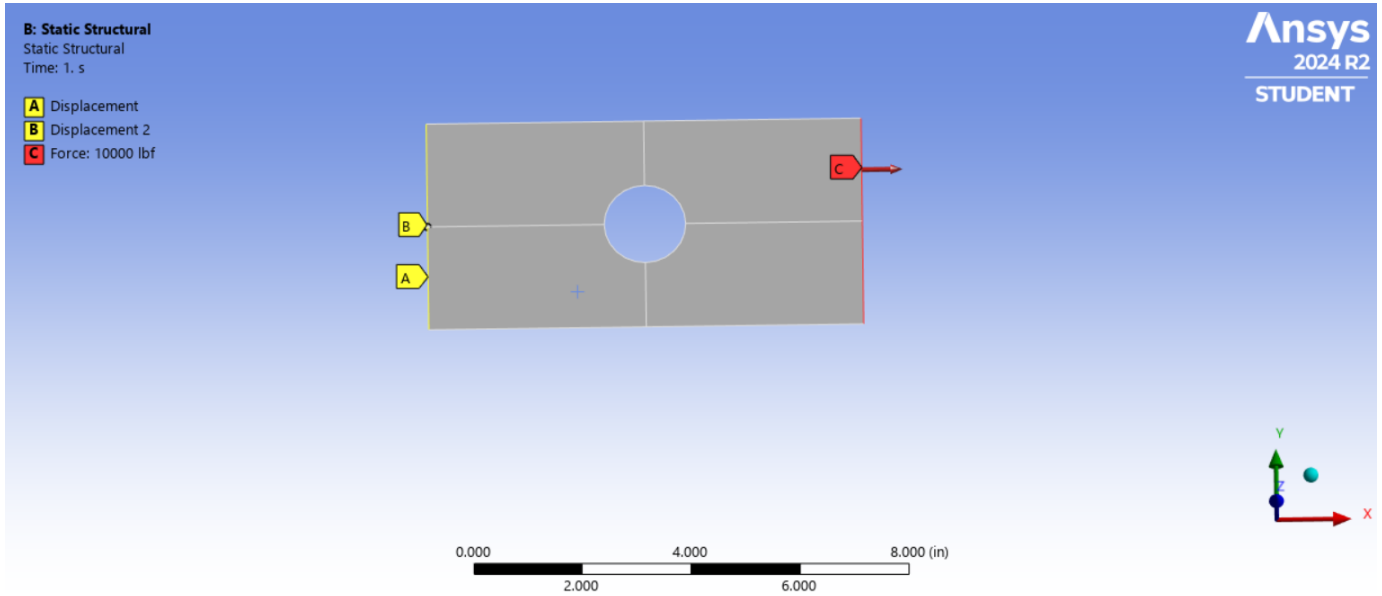
Different meshing tools used like face meshing ,edge size meshing and multi zone meshing

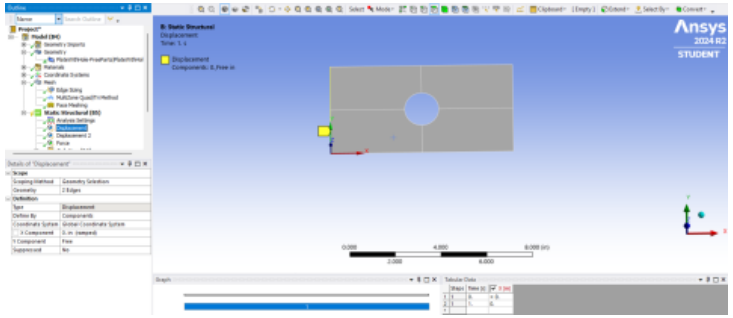


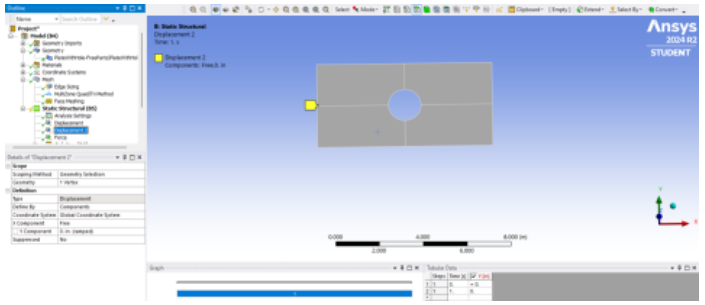
4 CORE Linear Quadrilateral (course)

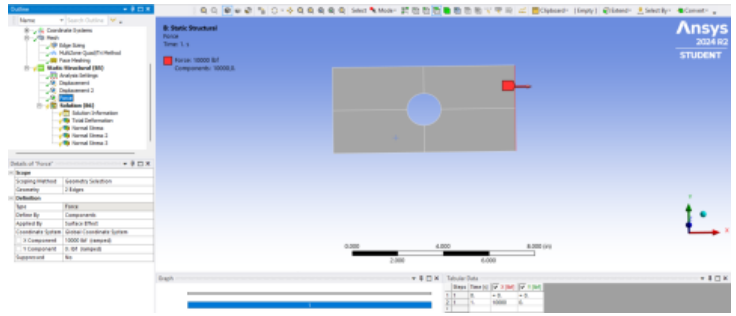
- In this I used different mesh controls and also the linear method in the multizone mesh and element as 28 and nodes as 44

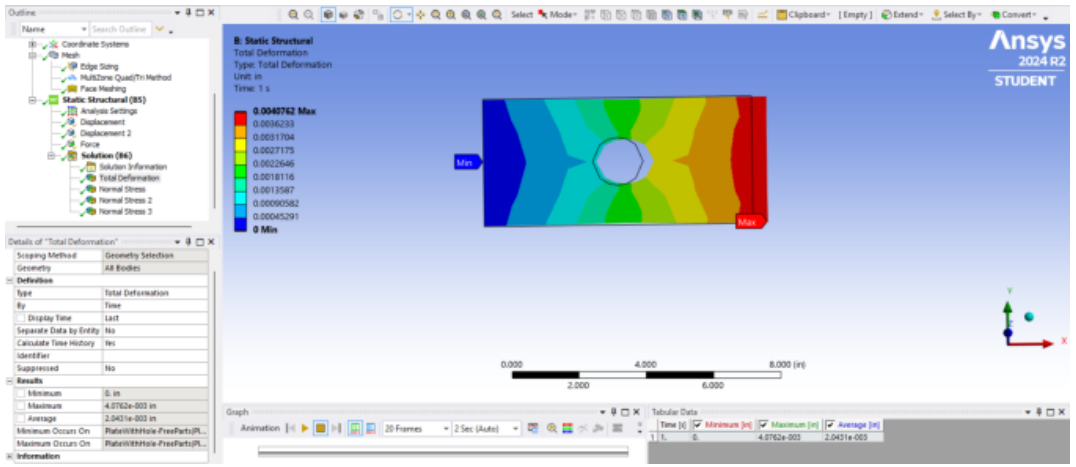
These are the boundary conditions
Applied the displacement boundary condition on left edge to constrain the plate in x direction
Applied the displacement boundary condition y constraint only at the left middle vertex
Applied force of 10000lbs on right edges

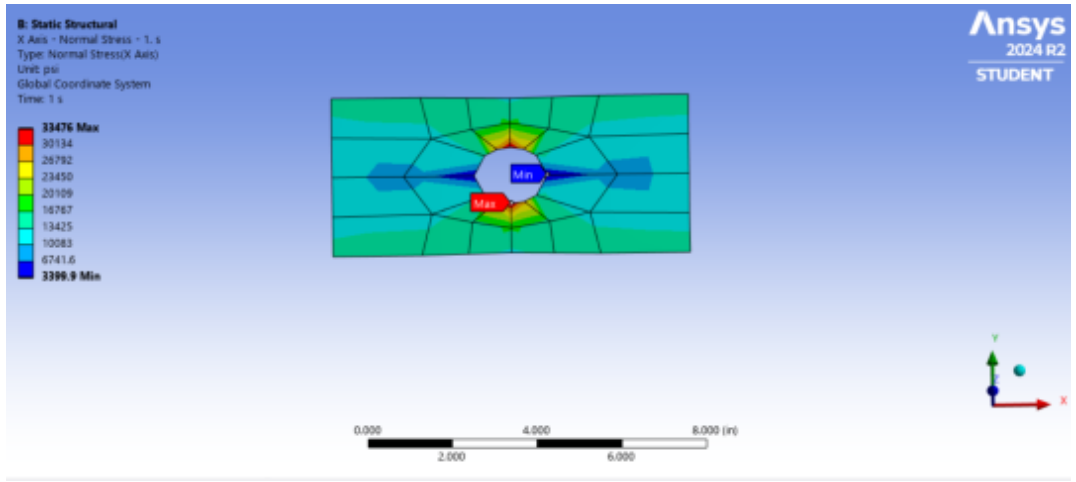








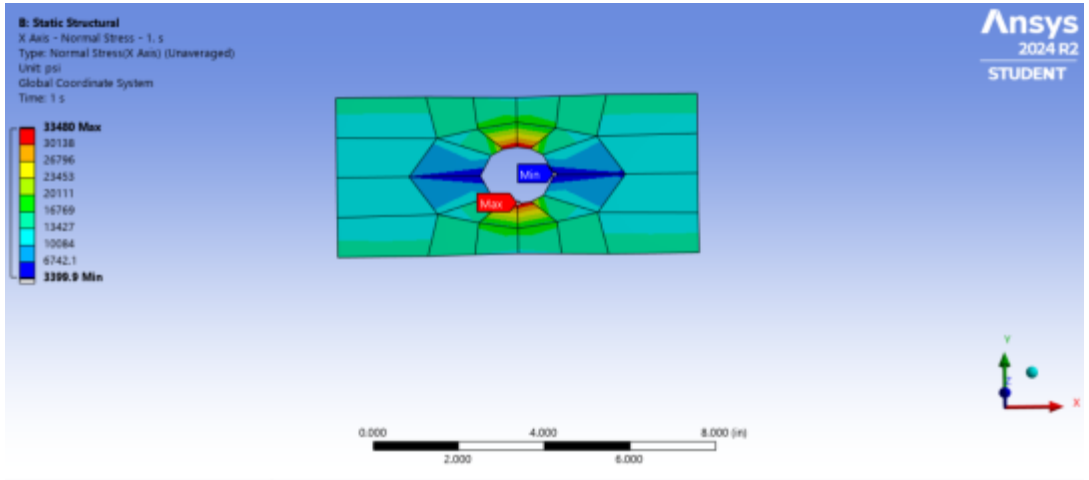




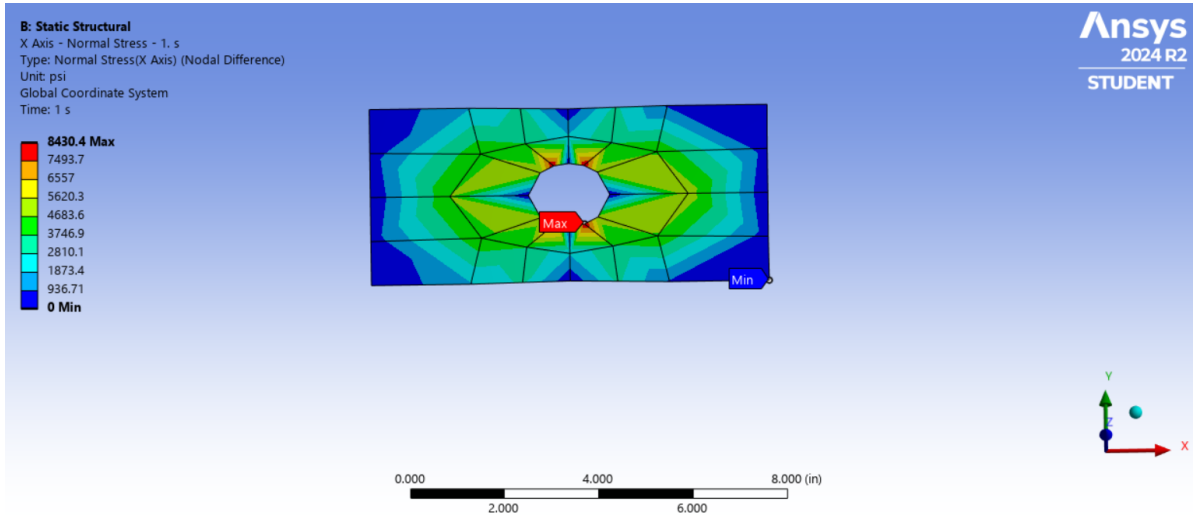
These are total deformation and normal stress (averaged)

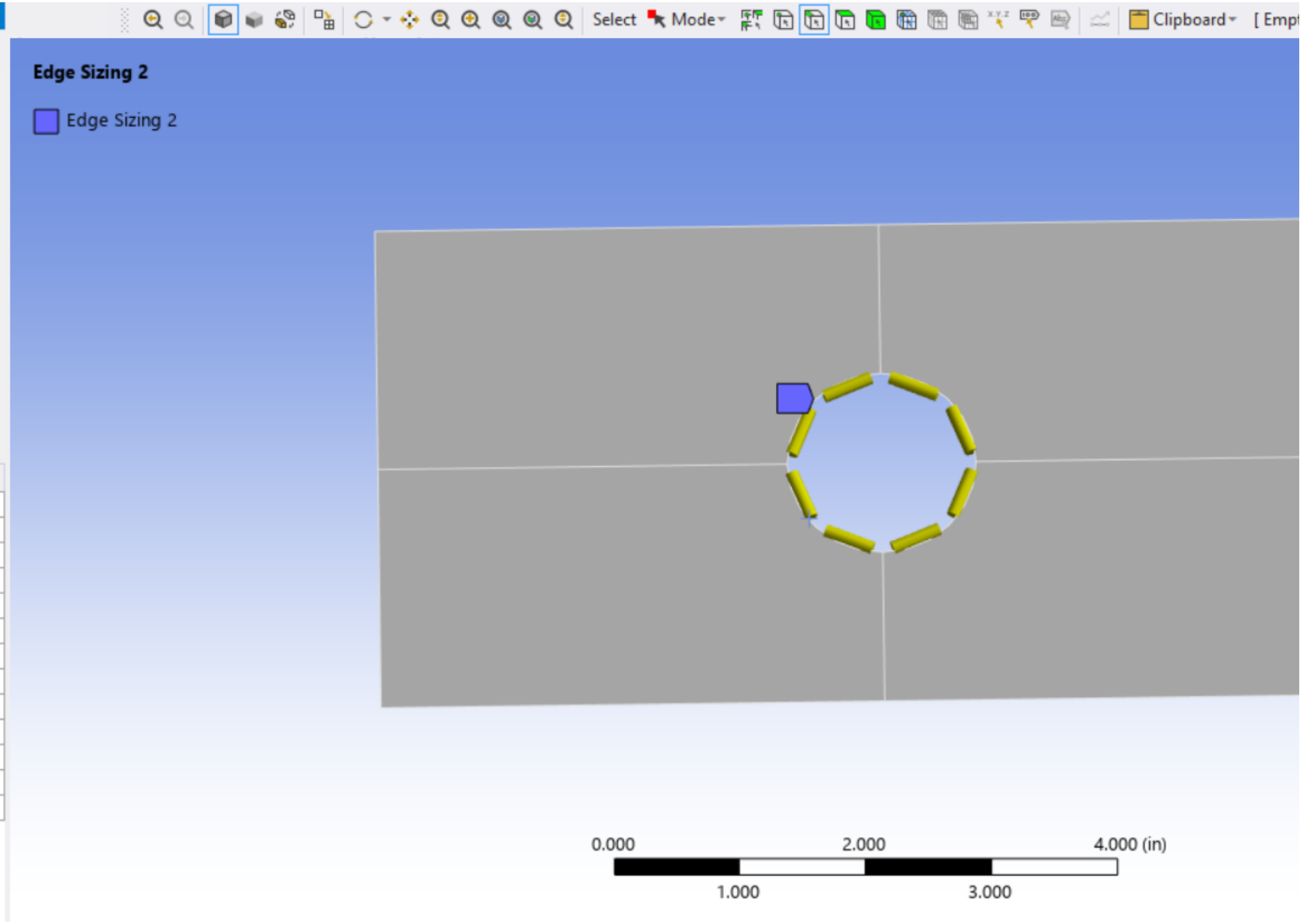
Normal stress

(unaveraged)



(nodal difference)



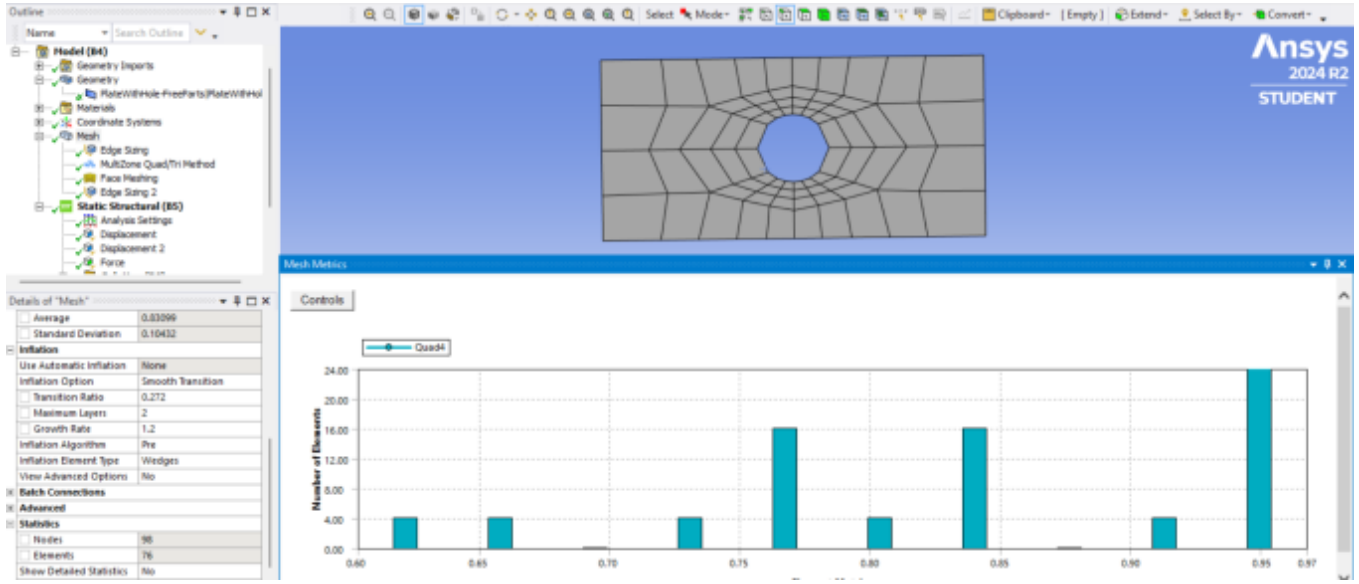


4 CORE Linear Quadrilateral (medium)

- In this another mesh sizing to the hole of the plate and used number of divisions as 2

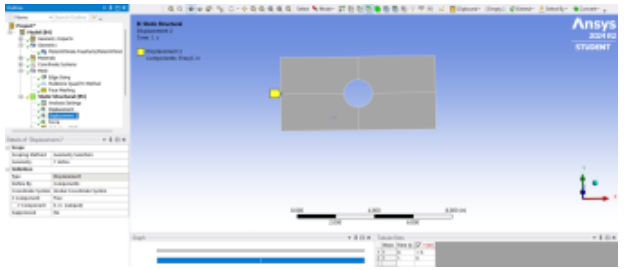
- 4 CORE Linear Quadrilateral (MEDIUM)

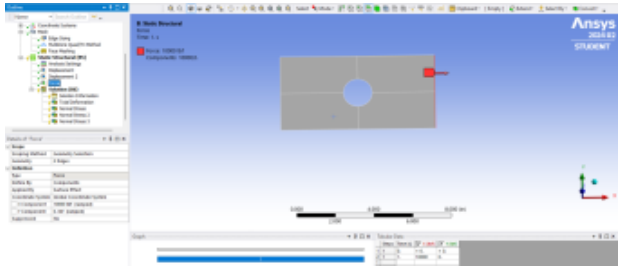
Medium mesh having the elements as 76 and nodes as 98 and got all the quad4 as the mesh element quality

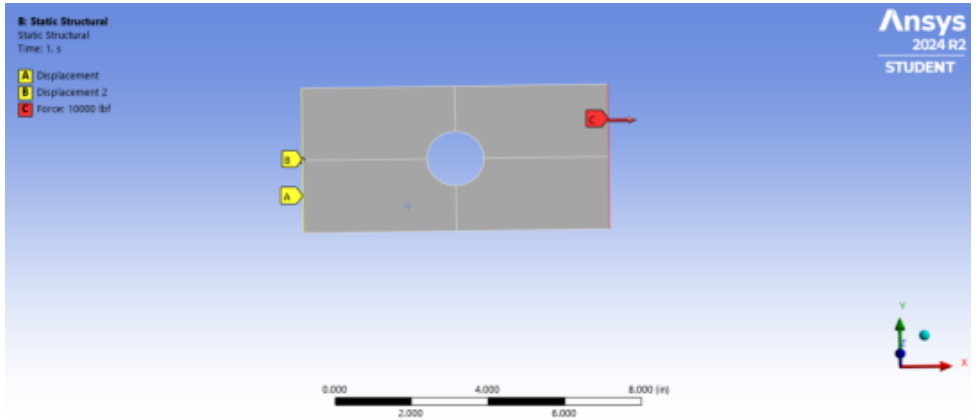


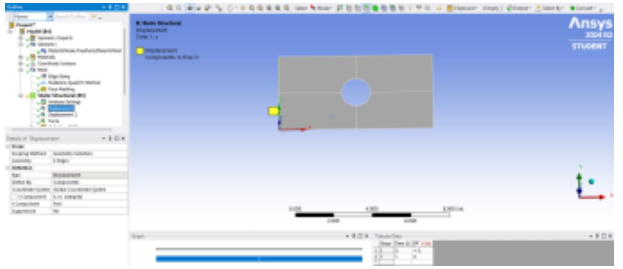
BOUNDARY CONDITIONS

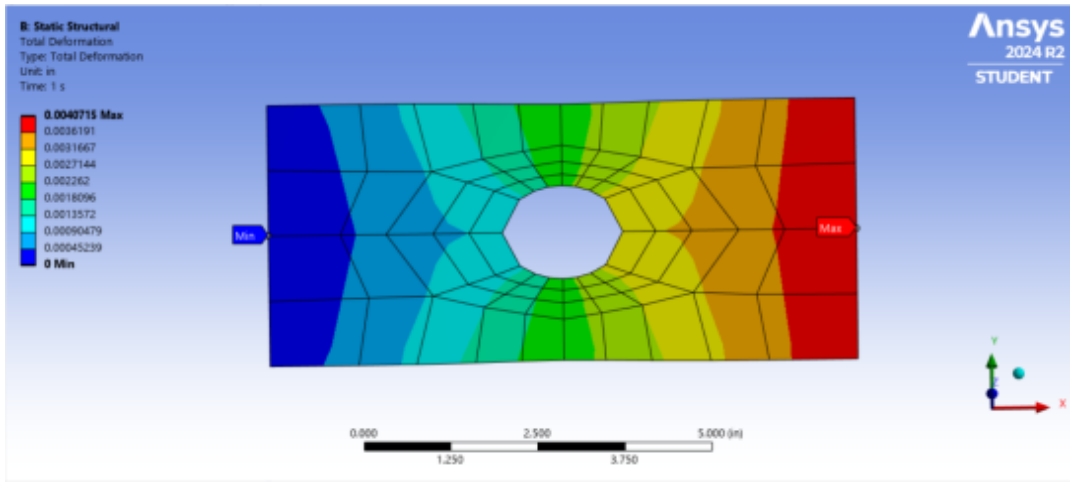
These are the boundary conditions
Applied the displacement boundary condition on left edge to constrain the plate in x direction
Applied the displacement boundary condition y constraint only at the middle left vertex
Applied fore of 10000lbs on right edge

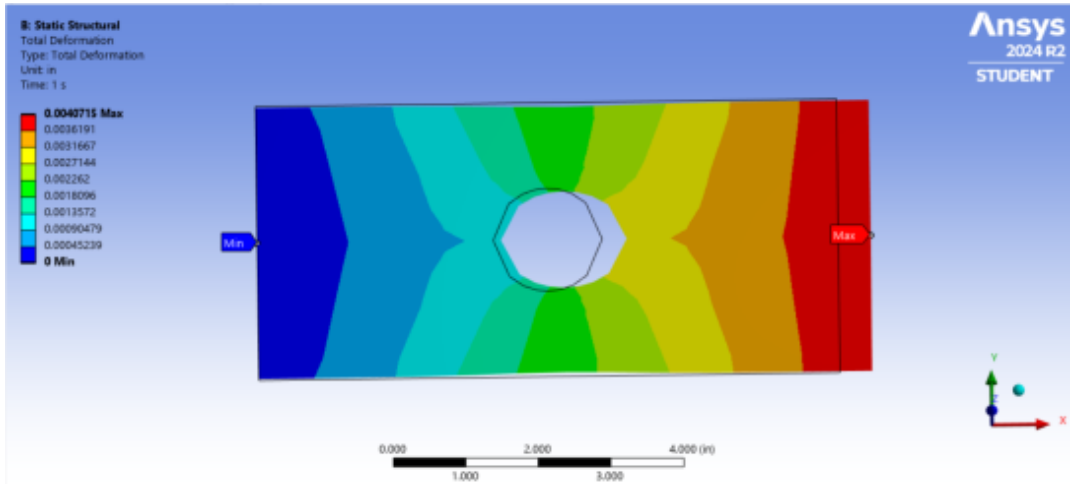




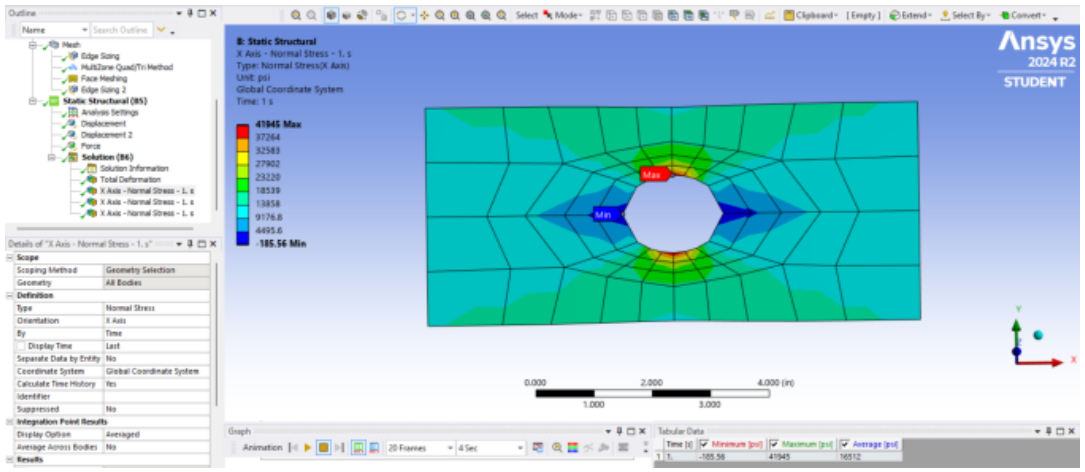


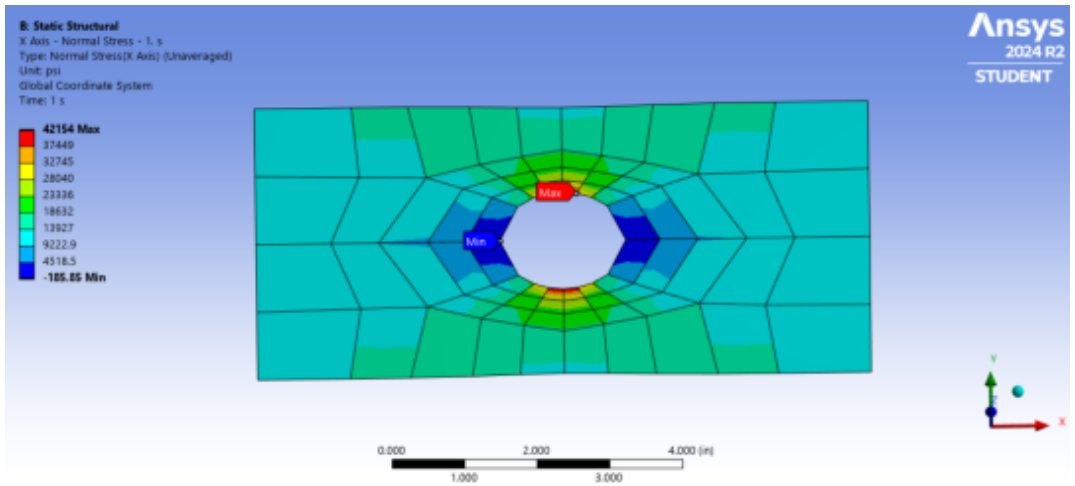




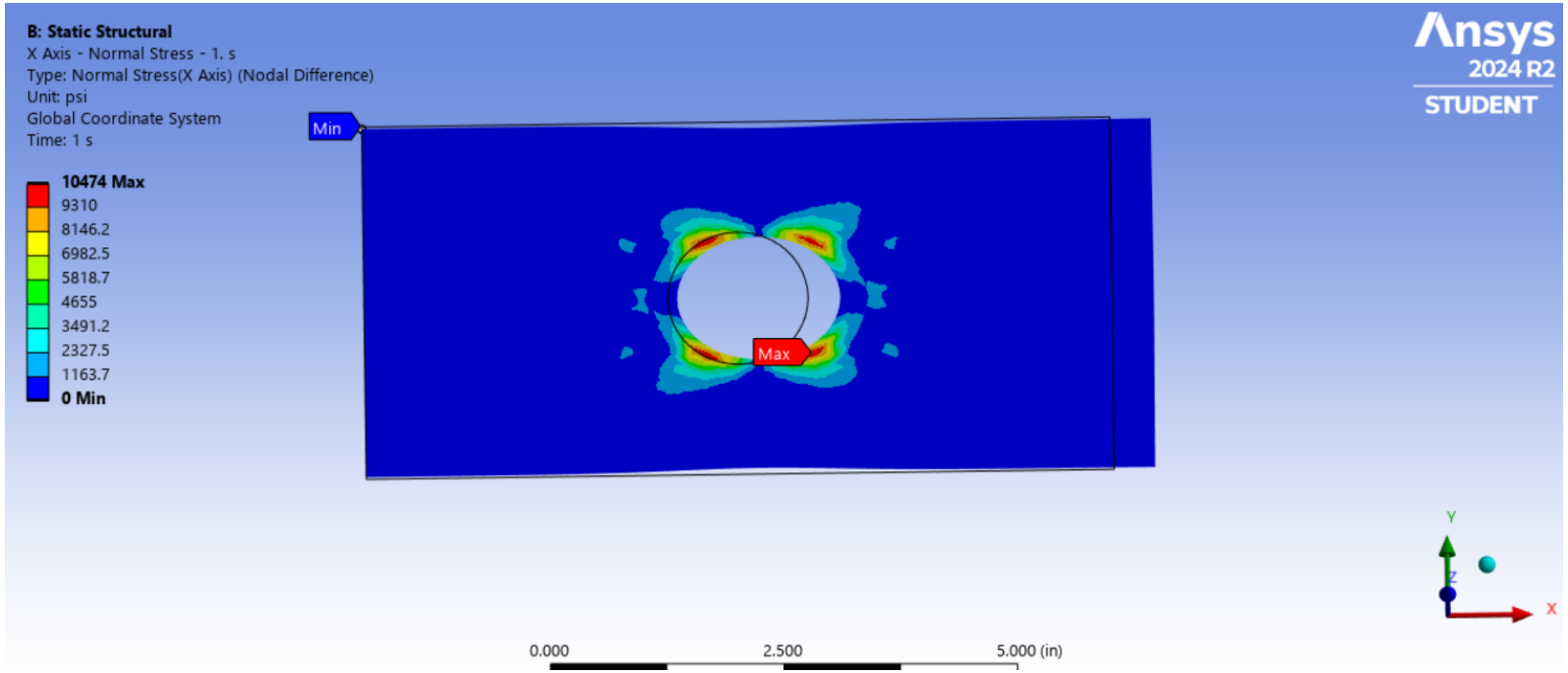


Total deformation

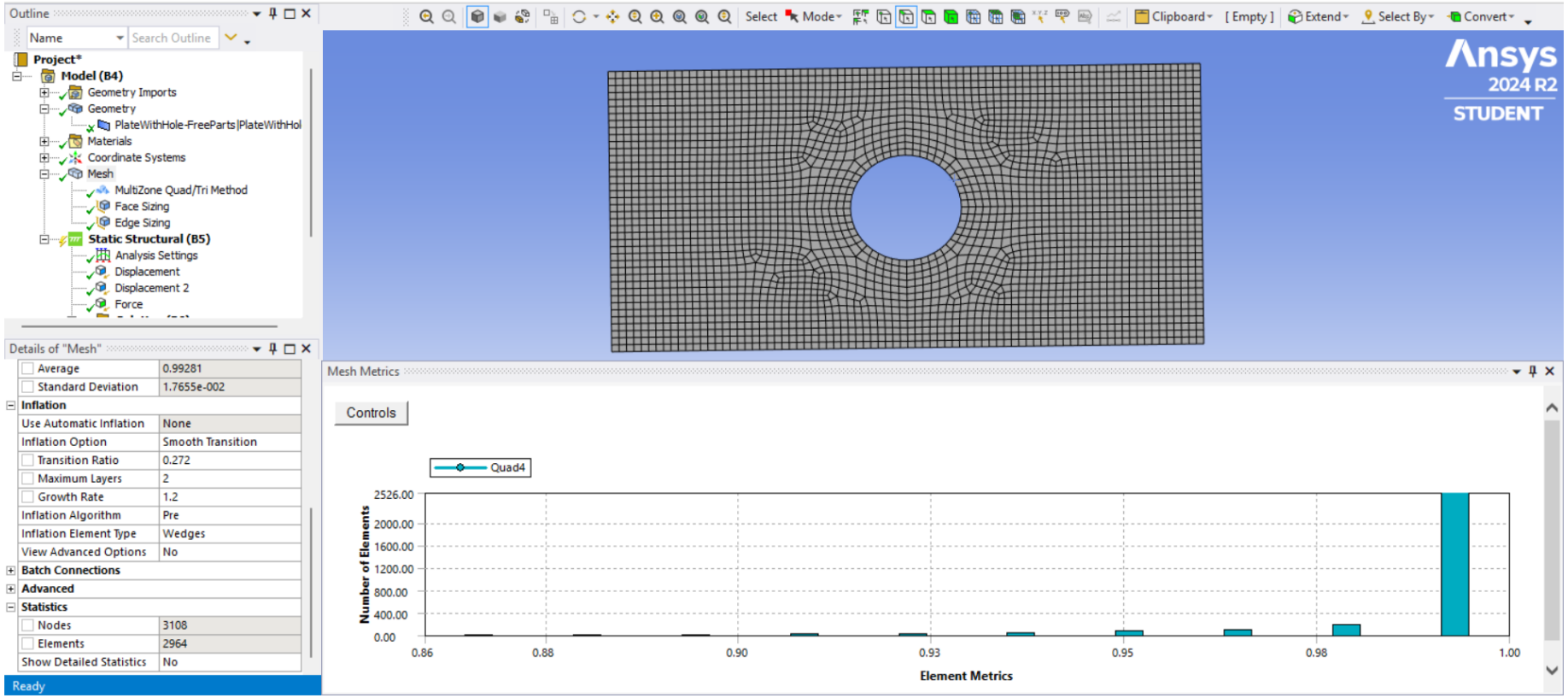




Normal stress
(averaged) & (unaveraged)

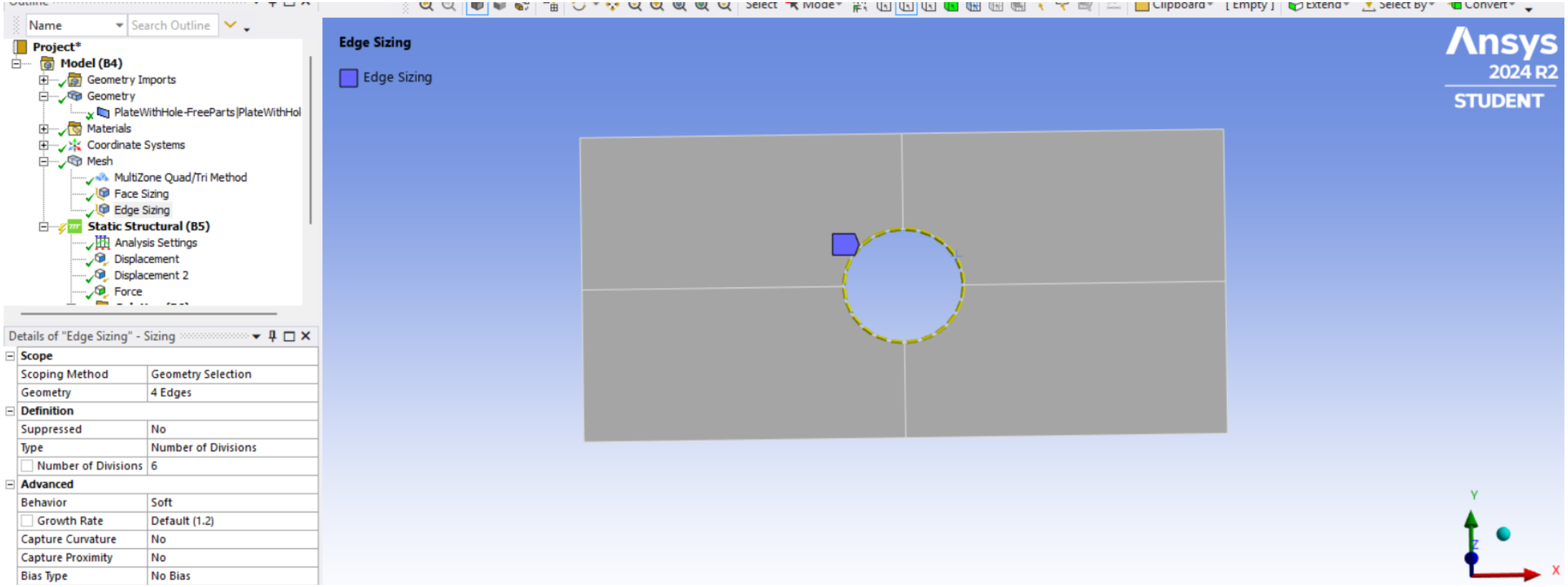


Normal stress(nodal difference)

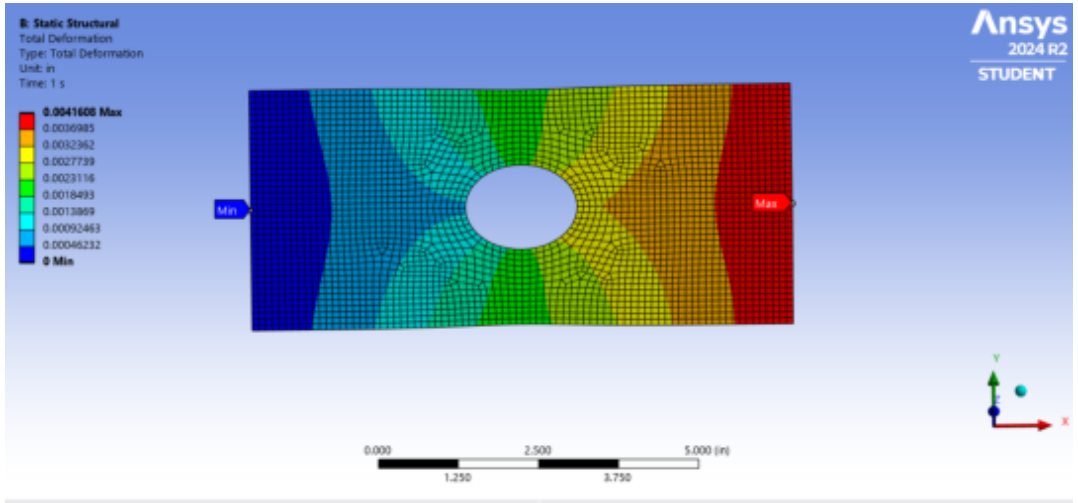


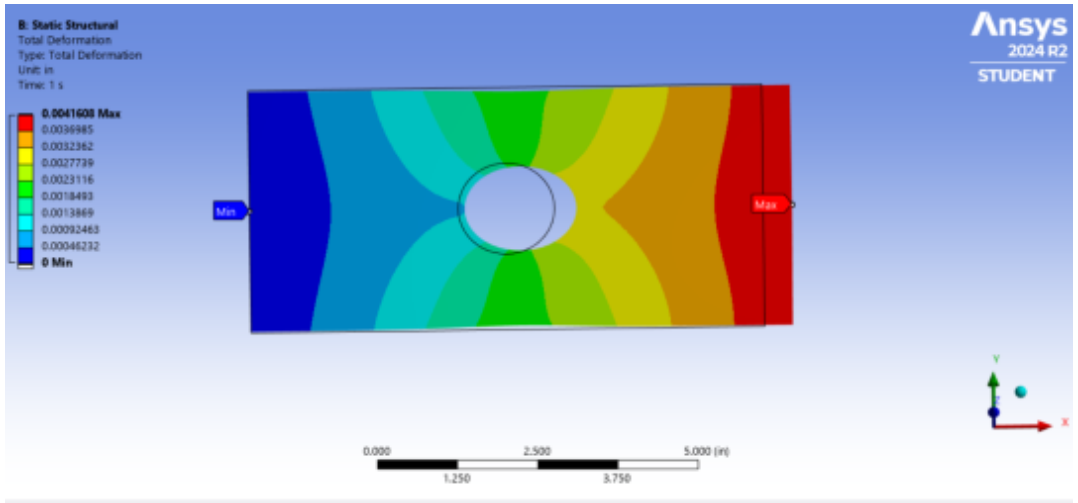
4 CORE Linear Quadrilateral (Fine)

- In this I used different mesh controls and also the linear method in the multizone mesh and element as 2964 and nodes as 3108

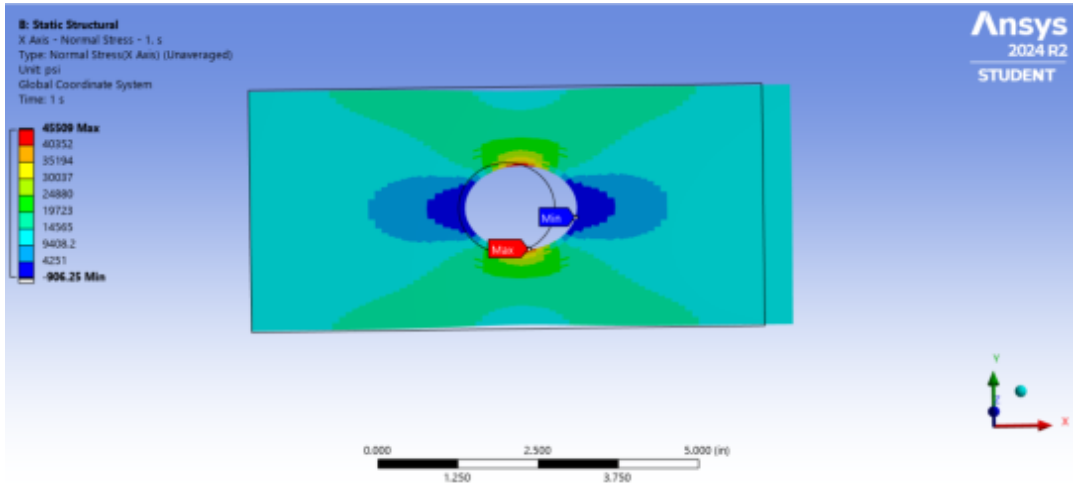


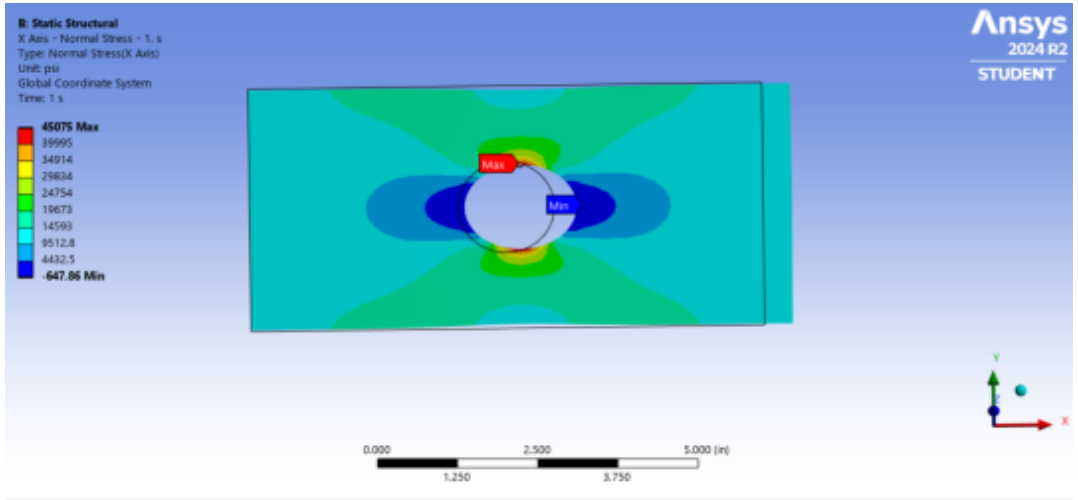
- In this another mesh sizing to the hole of the plate and used number of divisions as 6





Total deformation



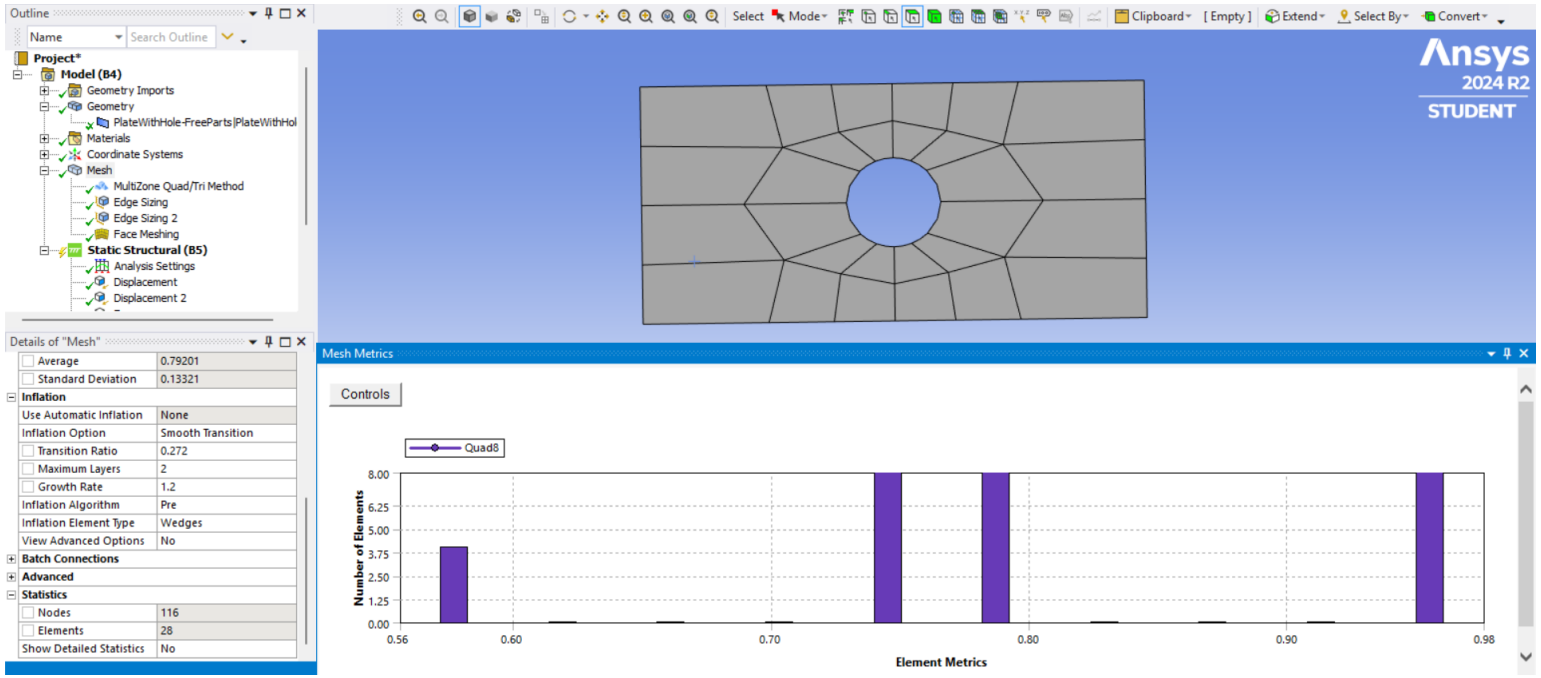


Normal stress averaged and unaveraged

8 node plate model

8 Node Quadratic Isoparametric (coarse)

It is a coarse model here I got the no of element as 28 and nodes as 116 element mesh quality

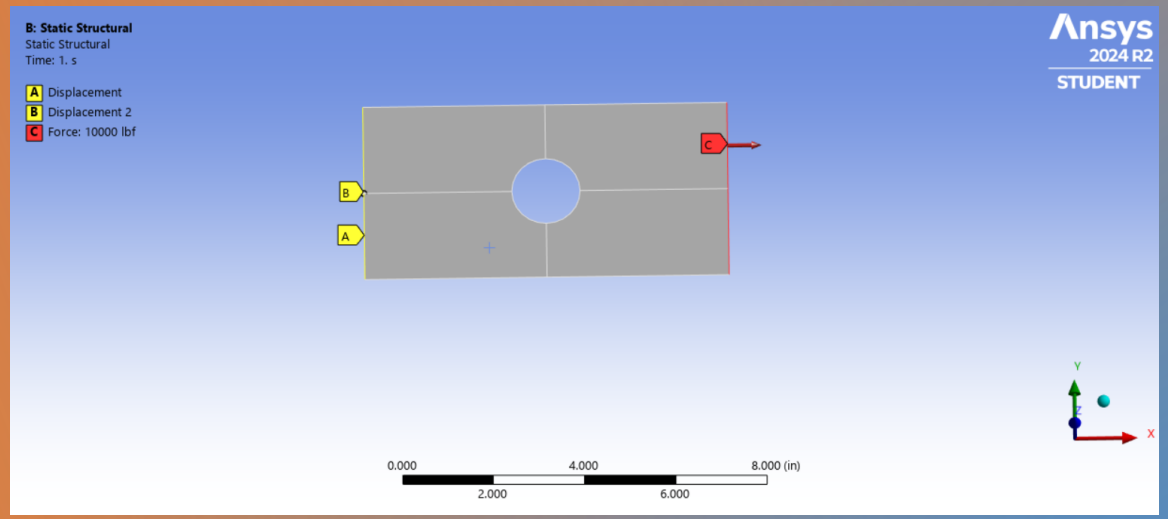


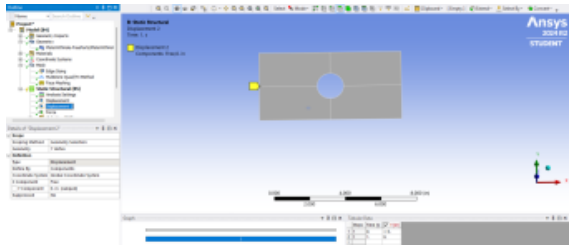
- These are the boundary conditions.

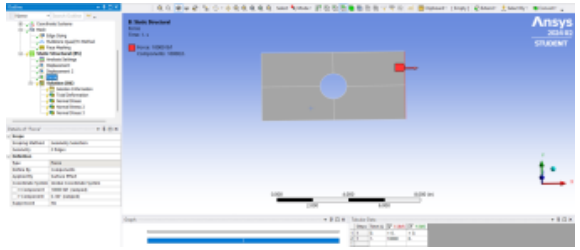
  Applied the displacement boundary condition on left edge to constrain the plate in x direction.

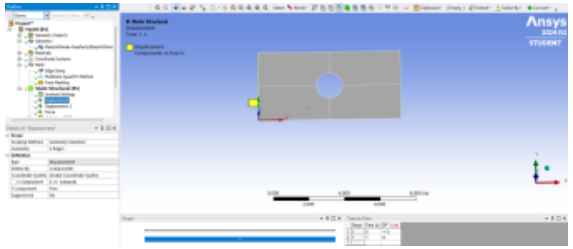
  Applied the displacement boundary condition y constraint only at the left middle vertex

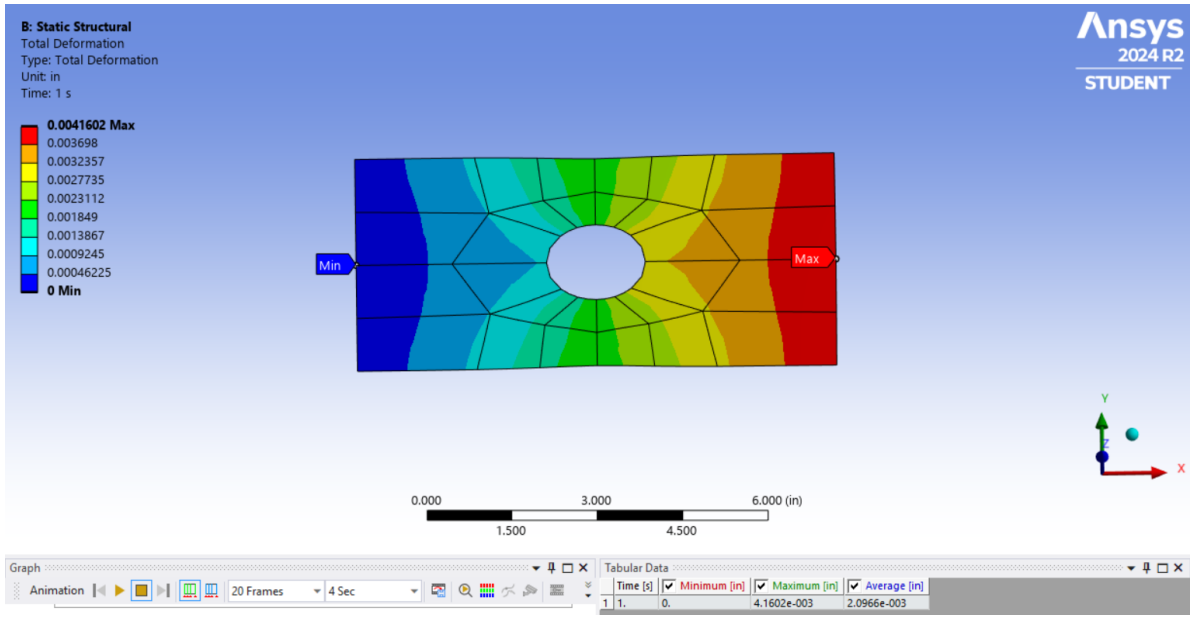
  Applied a force of 10000lbs on right edge

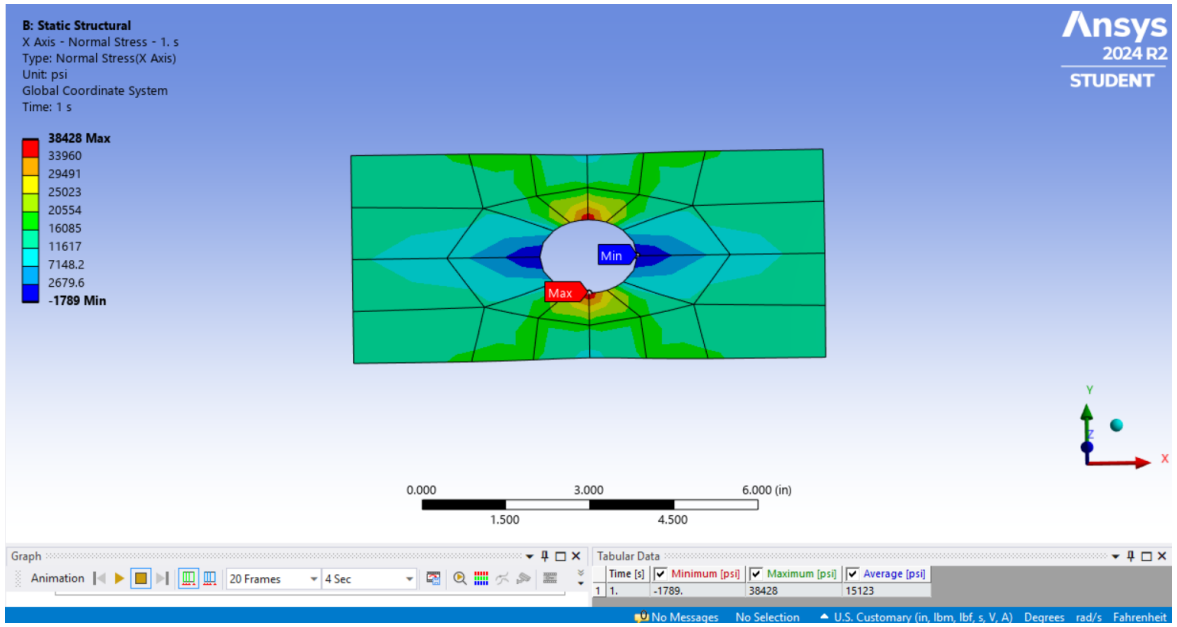




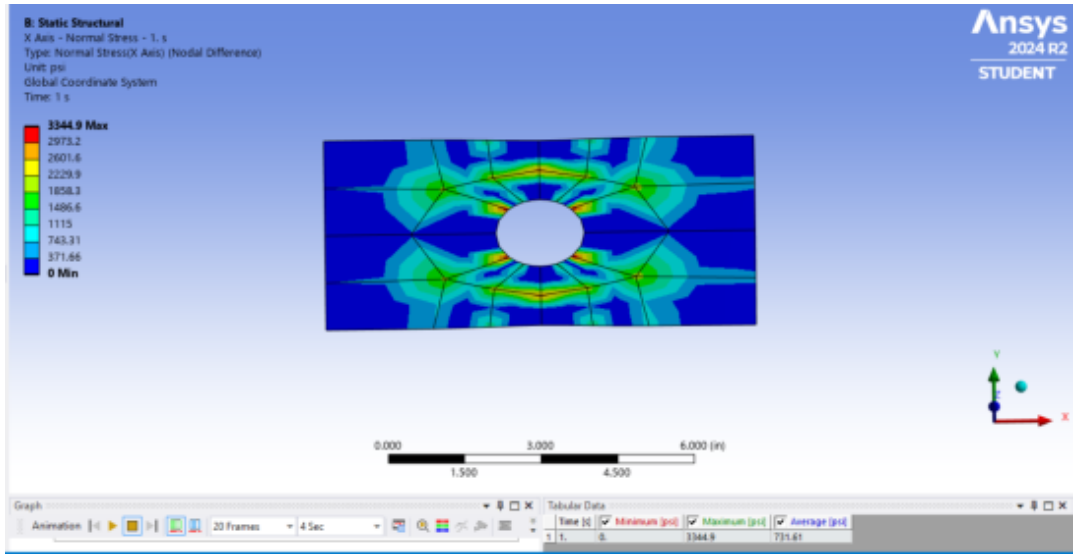


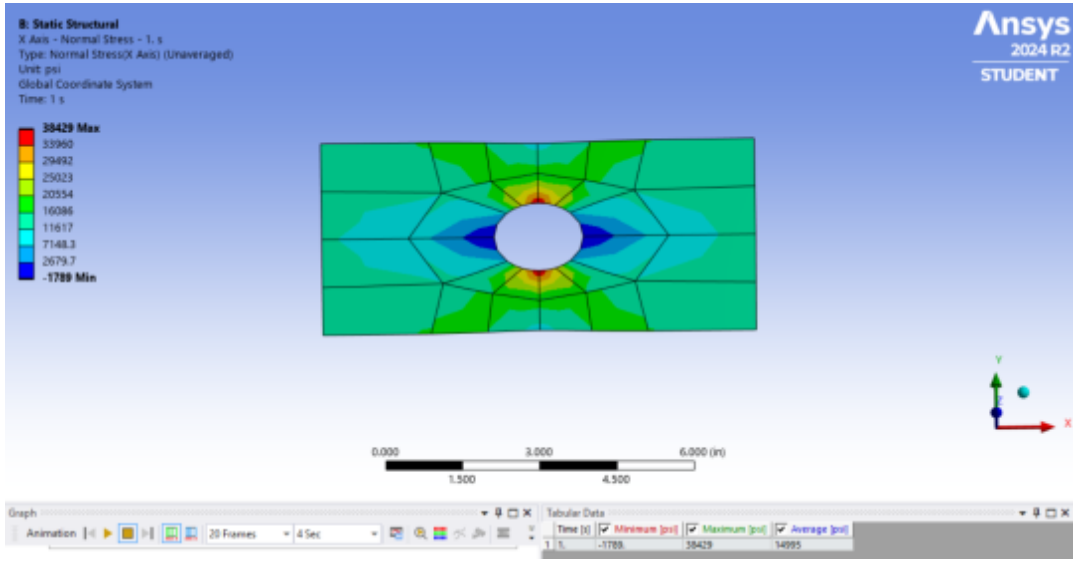




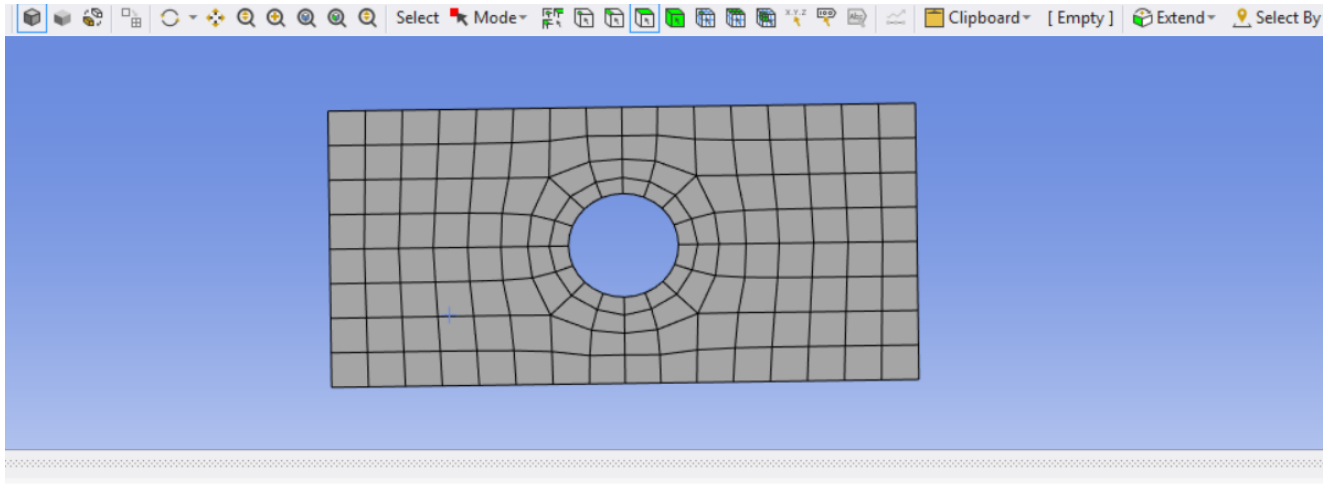


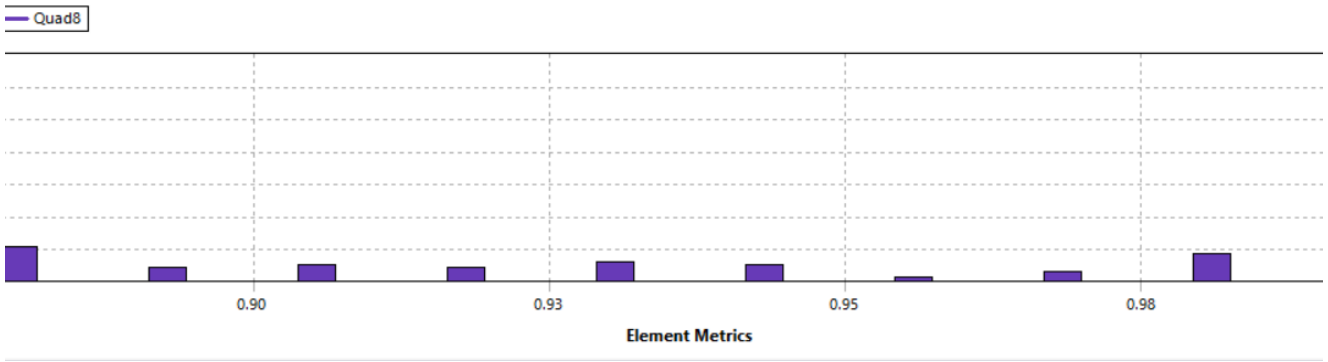
Normal stress (averaged)
Total deformation





Normal stress
Unaveraged and Nodal difference

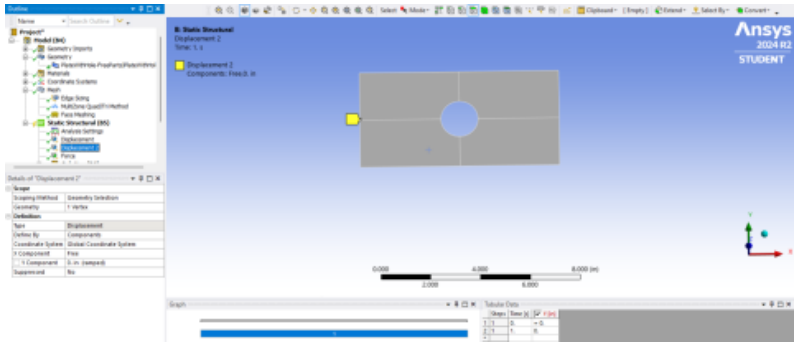


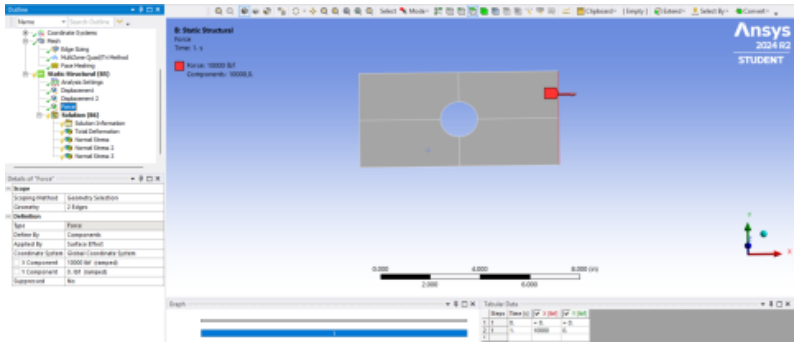


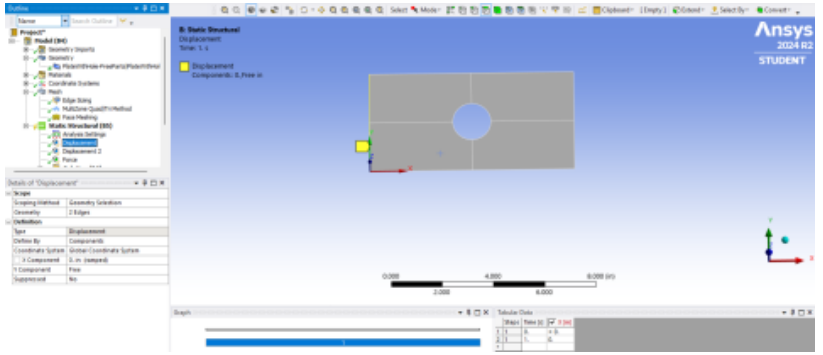
8 Node Quadratic Isoparametric (MEDIUM)

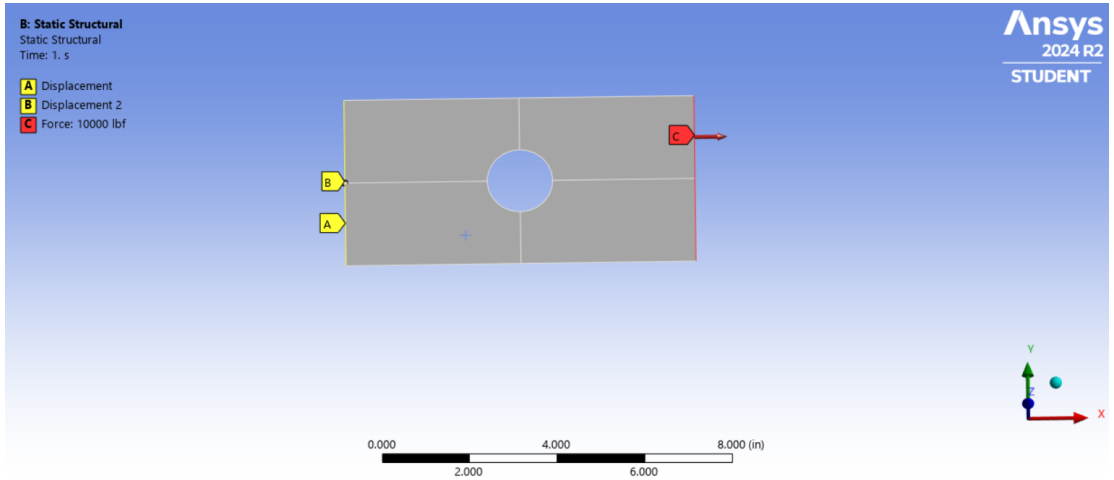
it has 76 elements

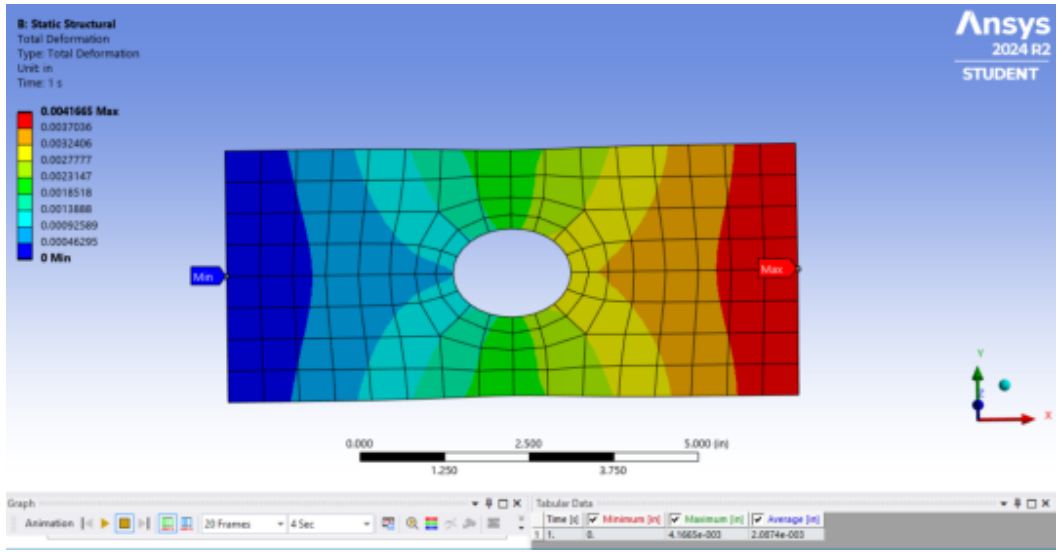
These are the boundary conditions
Applied the displacement boundary condition on left edge to constrain the plate in x direction
Applied the displacement boundary condition y constraint only at the middle left vertex

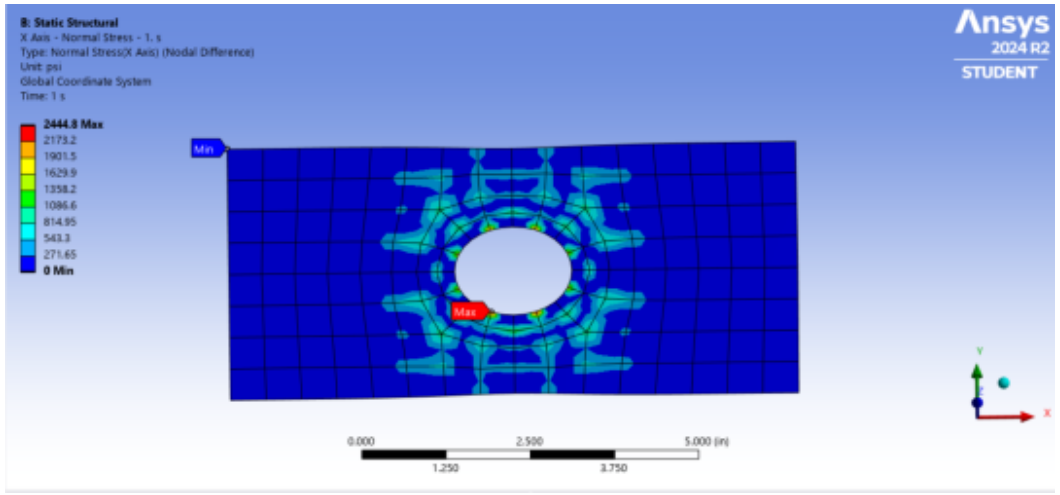




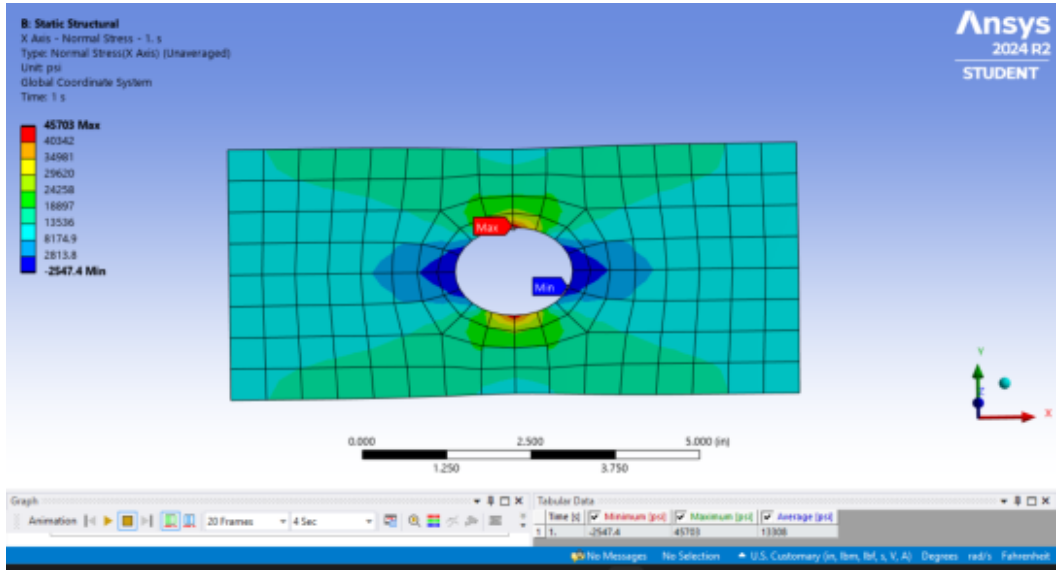


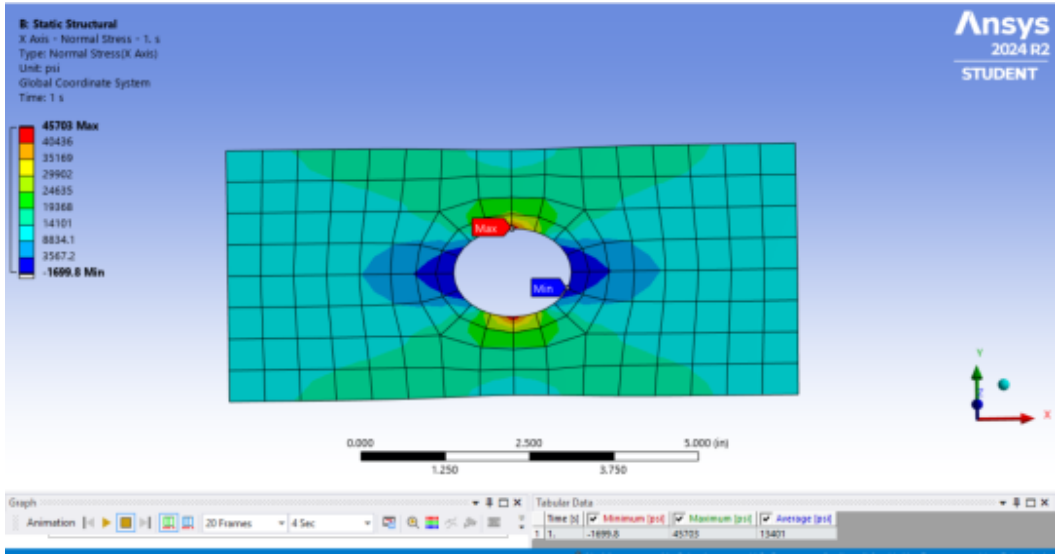




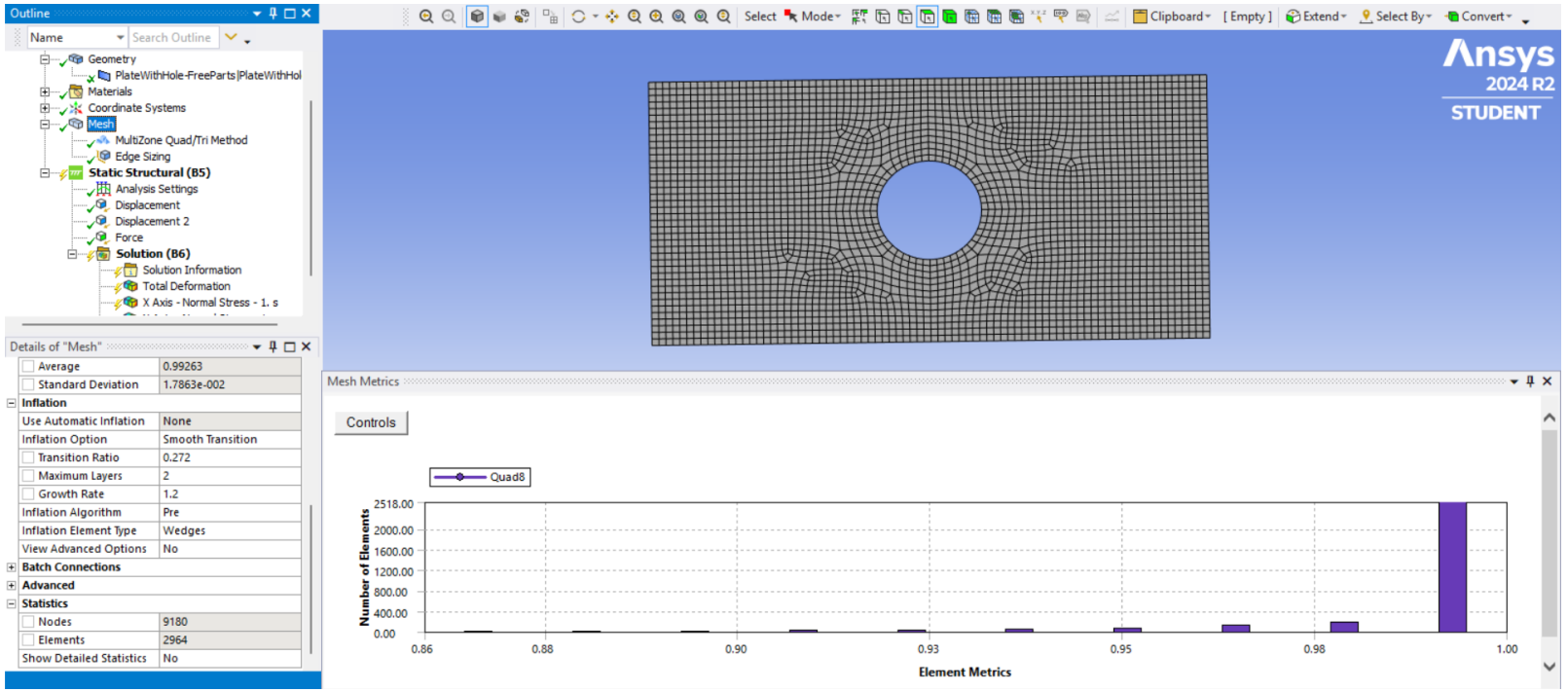


Total deformation & Normal stress(nodal differenece)

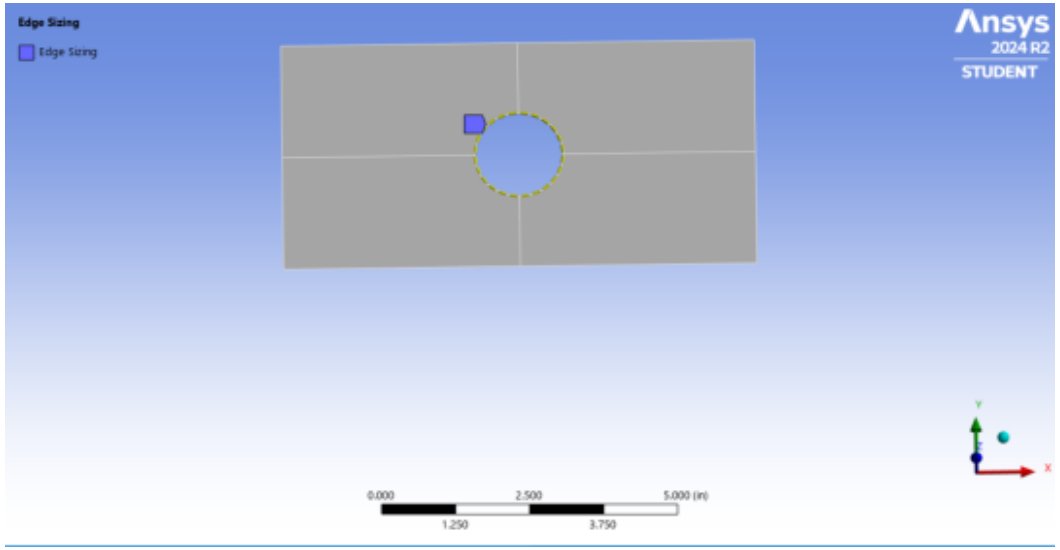


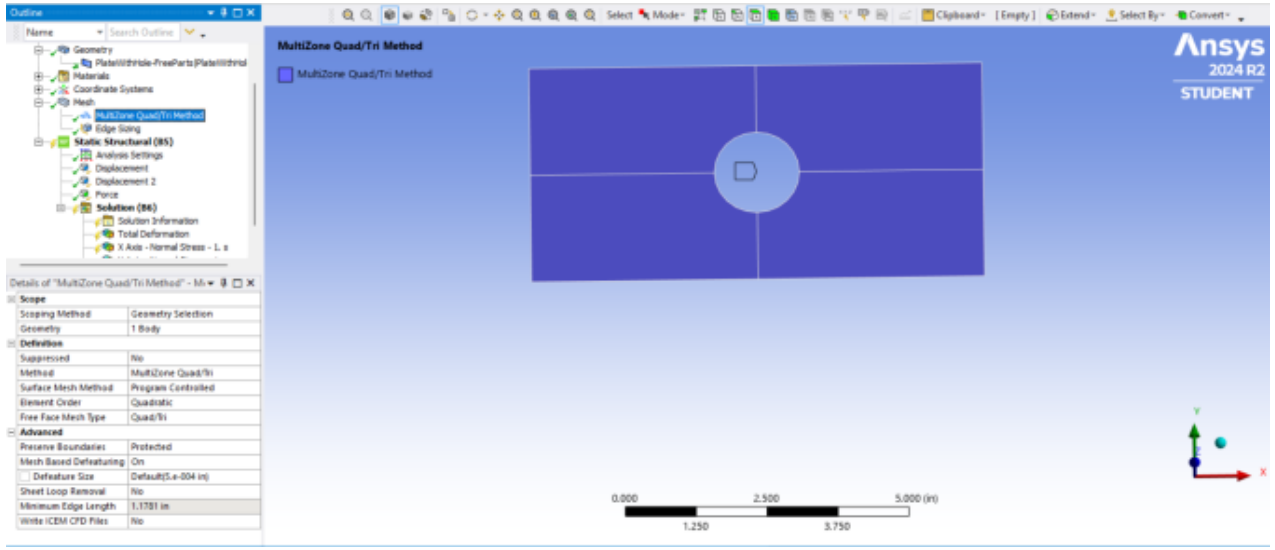


Normal stress
Averaged & unaveraged



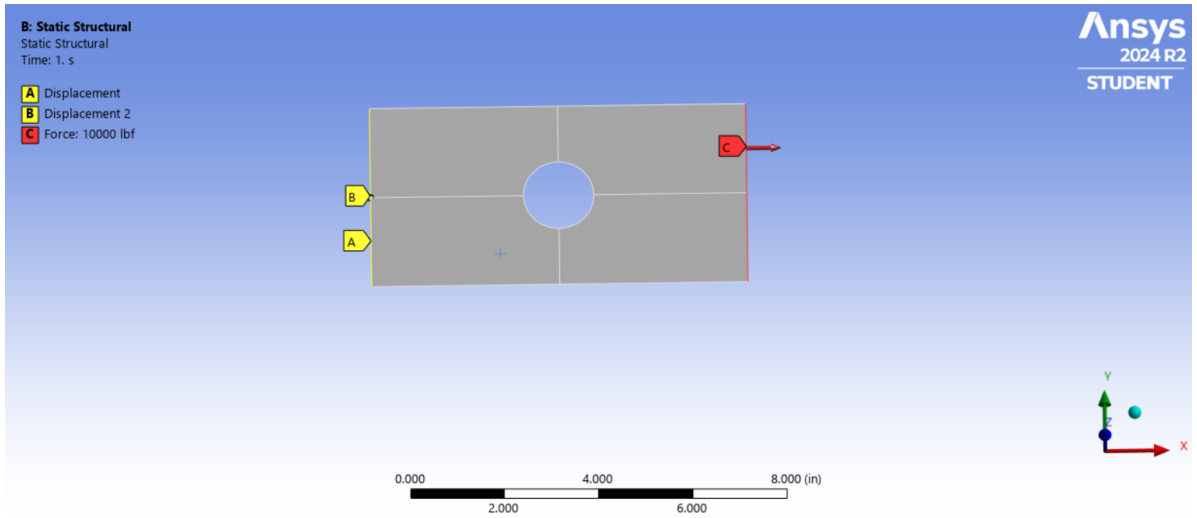
8 Node Quadratic Isoparametrc (fine)

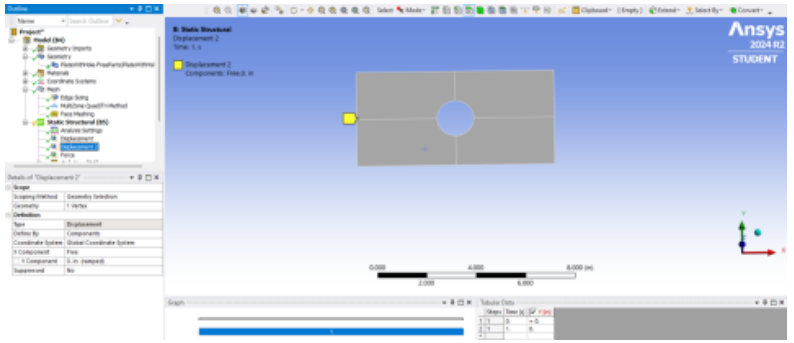


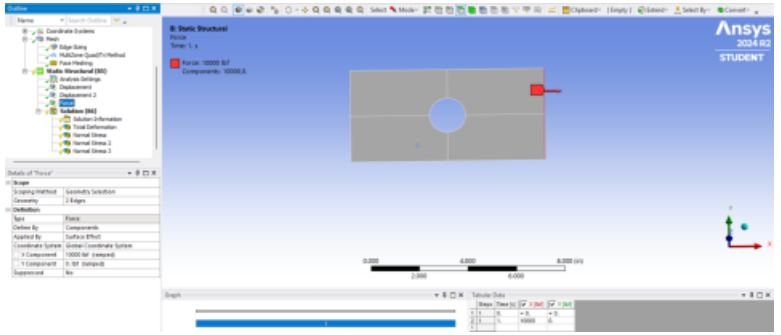


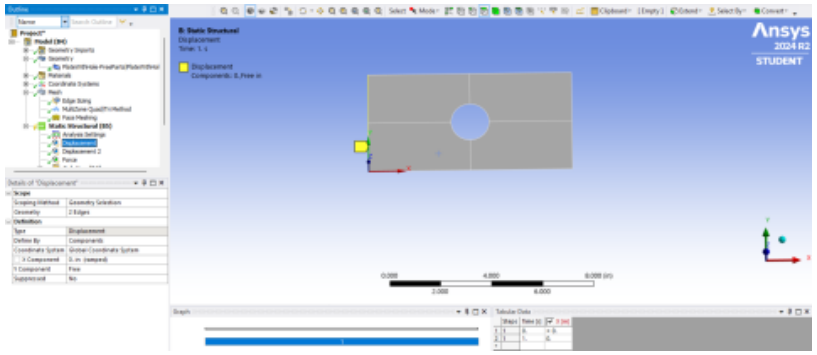
Mesh controls

These are the boundary conditions
Applied the displacement boundary condition on left edge to constrain the plate in x direction
Applied the displacement boundary condition y constraint only at the middle left vertex

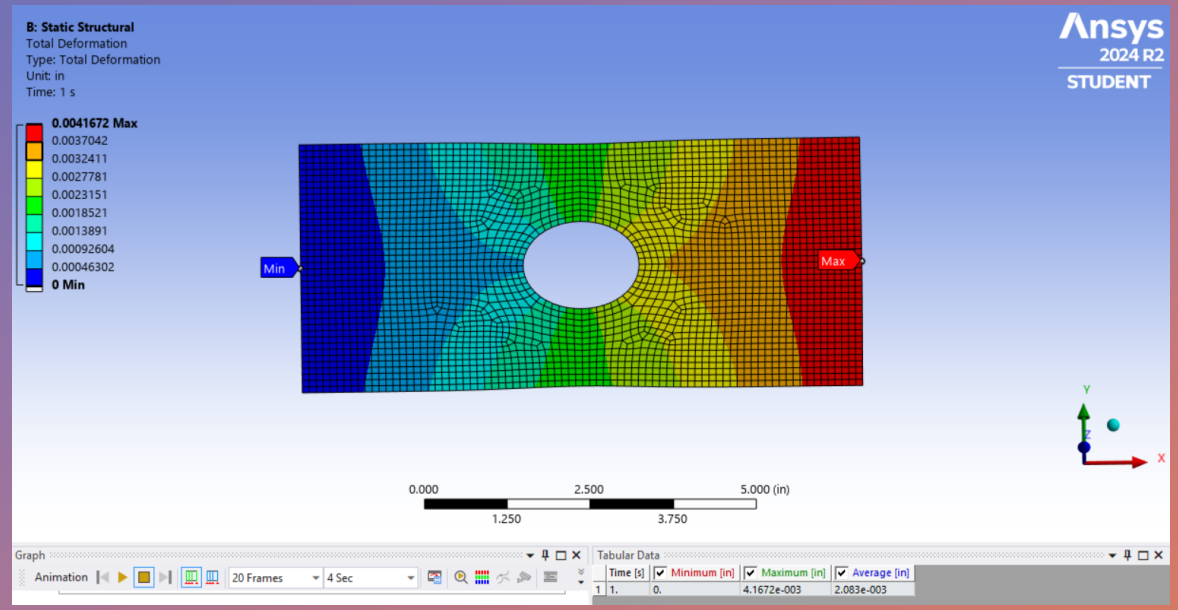


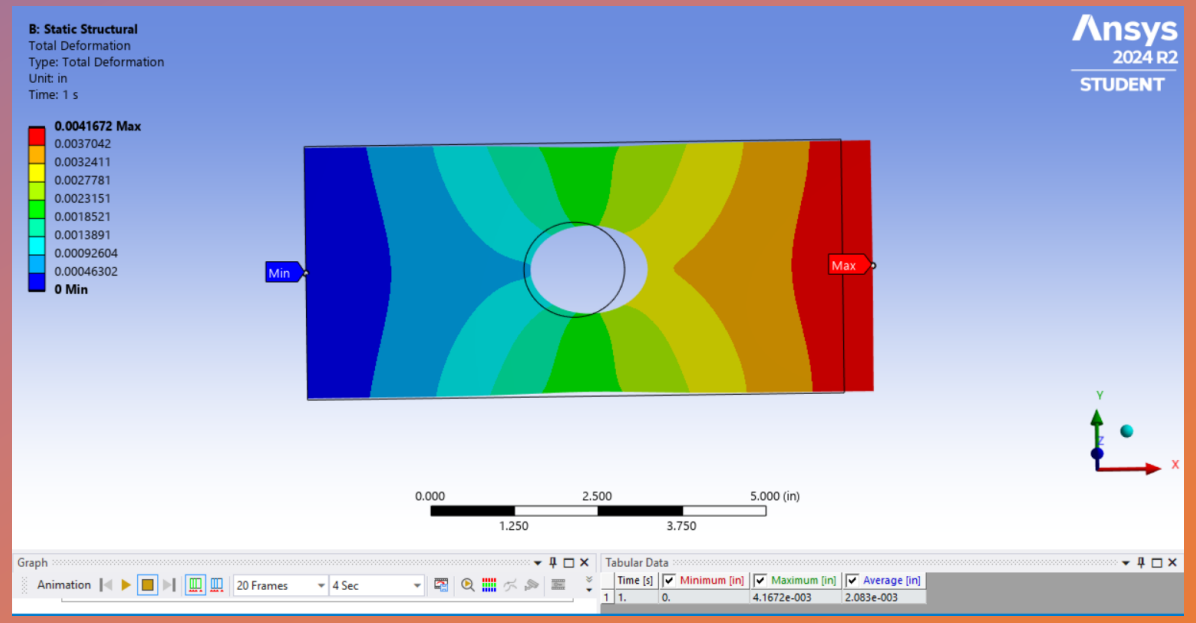




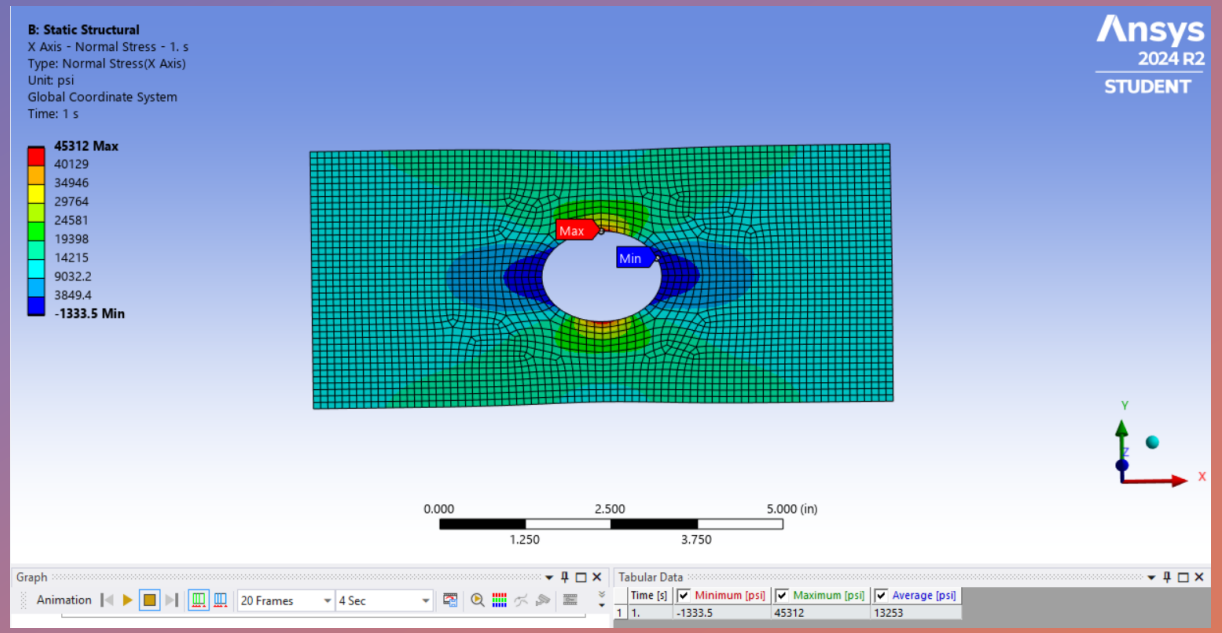


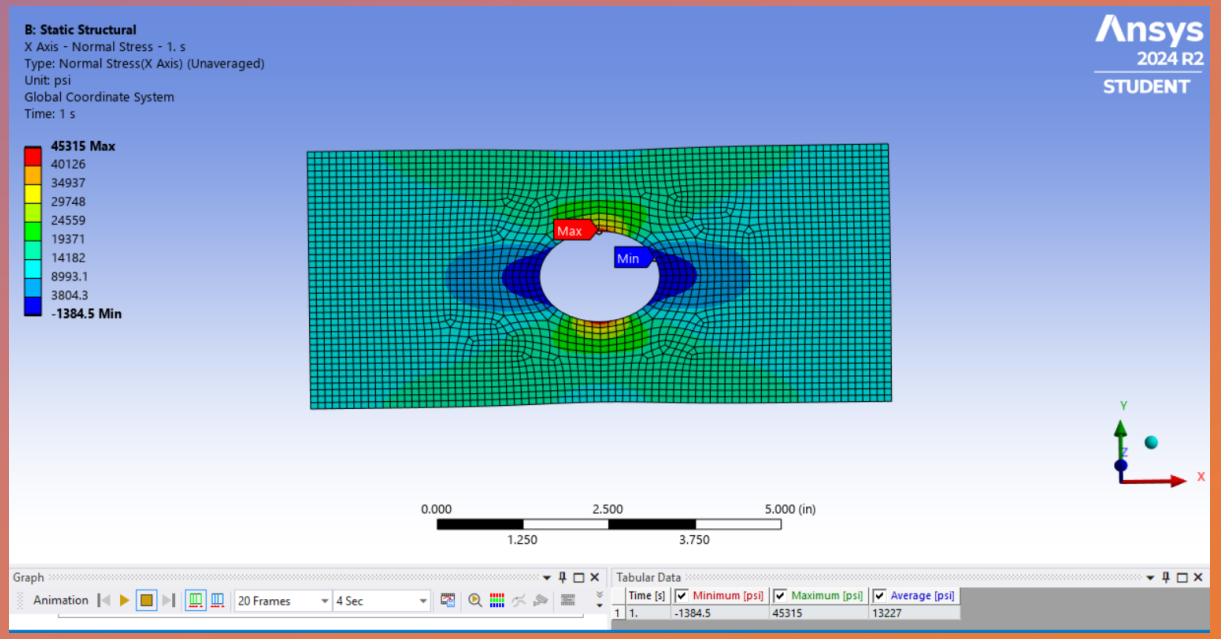
Total deformation



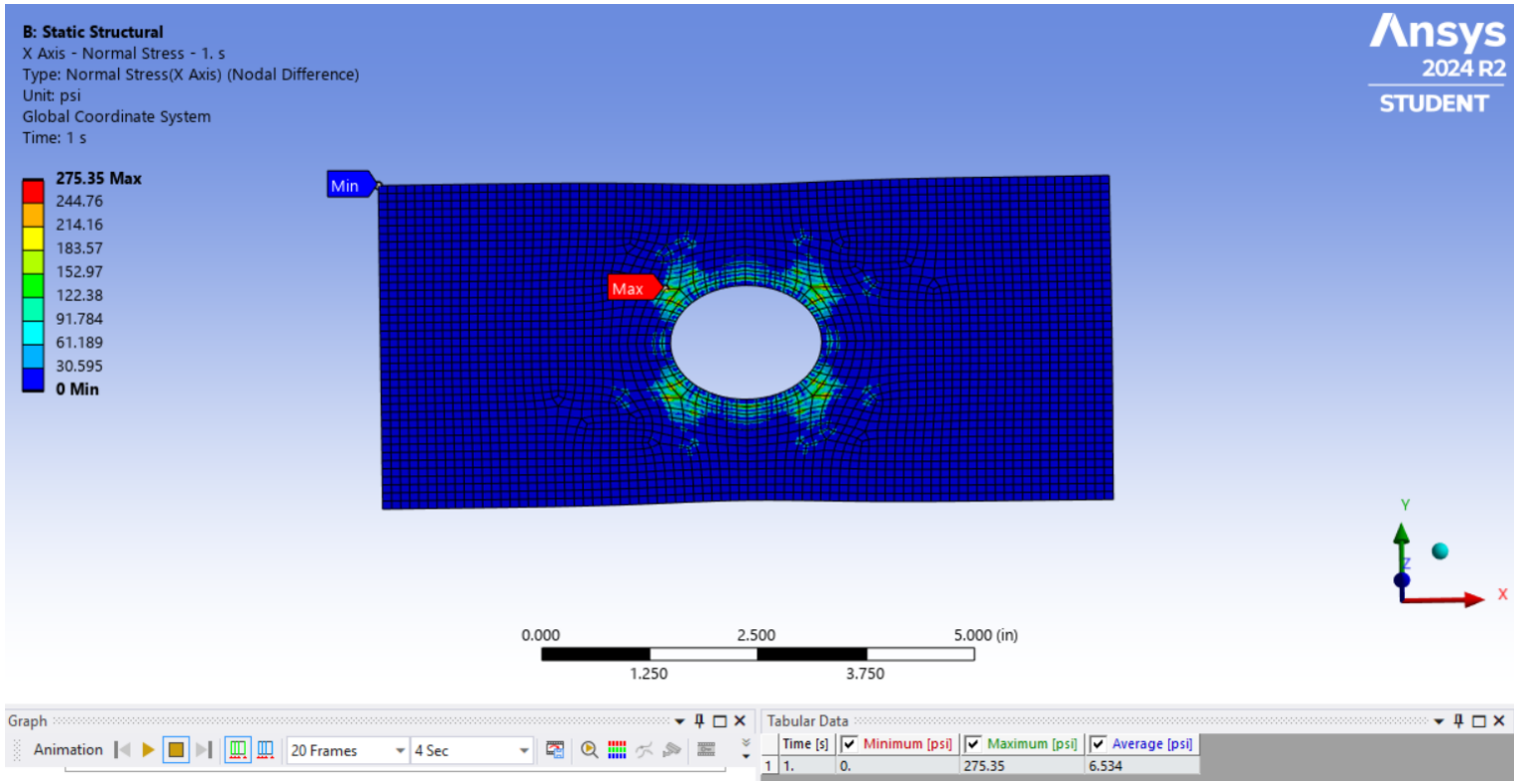


Normal stress averaged and unaveraged





Normal stress (nodal differnece)



Therotical calculation

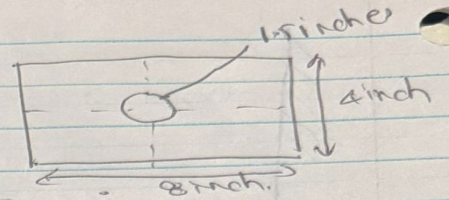
Theoretical calculation

$\sigma_{max} = K_{tq} \times \sigma_{normal}$

$d = 1.5$ inches

$p \Rightarrow 1000$ lbs

$t = 0.2$ inches



$\sigma_{normal} = \frac{P}{A} \Rightarrow \frac{10000}{4 \times 0.2} = 12500 psi$

$K_{tq} = 0.284 + \frac{2}{1-\frac{d}{H}} - 0.600\left(1-\frac{d}{H}\right) + 1.32\left(1-\frac{d}{H}\right)^2$

$0.284 + \frac{2}{1-\frac{1.5}{4}} - 0.6\left(1-\frac{1.5}{4}\right) + 1.32\left(1-\frac{1.5}{4}\right)^2$

$0.284 + \frac{2}{0.621} - 0.6(0.625) + 1.32($

$K_{tq} = 3.62.$

for $K_{tq} = 3.62$

$\sigma_{max} = K_{tq} \times \sigma_{normal}$

$\Rightarrow 3.62 \times 12500 psi$

$\boxed{\sigma_{max} \Rightarrow 45250 psi}$ Theoretical.

Here I got σ_{max} from Ansys.

$\boxed{\sigma_{max} \Rightarrow 45312 psi}$ Ansys

Error :- $\frac{\sigma_{max(Ansys)} - \sigma_{max(theo)}}{\sigma_{max(theo)}} \times 100$

$\Rightarrow \frac{45312 - 45250}{45250} \times 100 \Rightarrow 0.00137 \times 100$

$\boxed{error \Rightarrow 0.13\%}$

Mesh densitys

Mesh Type	Element Count	Description
Coarse	≤ 30 elements	Fast to compute, but less accurate—may miss stress gradients, especially near discontinuities like holes.
Medium	31–99 elements	Balanced approach with moderate accuracy and reasonable computation time. Captures stress distribution better than coarse.
Fine	> 500 elements	High accuracy due to refined mesh, especially effective in capturing stress concentration. Requires more computation time.

Results summary

Plate with hole model	Number of Element	Max axial stress averaged (psi)	Max axial stress unaveraged (psi)	Max axial stress nodal difference (psi)
Analytical calculation	N/A	45250	N/A	N/A
4 node coarse	28	33476	33480	8430.4
4 node medium	76	42154	41945	10474
4 node fine	2964	45075	45509	6067.9
8 node coarse	28	38428	38429	3344.9
8 node medium	76	45703	45706	2444.8
8 node fine	2964	45312	45315	275.35

DOES IT MAKE SENSE?

The error of 0.13% indicates an excellent match between the FEA results and the theoretical stress calculation.

With a theoretical stress of 45,250 psi and an ANSYS result of 45,312 psi, the difference is minimal and within acceptable engineering limits.

This small error confirms that the chosen mesh density and element type are appropriate and yield highly reliable results .

The strong correlation between FEA and theoretical values validates the accuracy and effectiveness of the finite element model.

Peterson’s Stress Concentration Factors:

- For a circular hole in an axially loaded infinite plate, the typical **stress concentration factor (k_{tg})** ranges from **3.0 to 3.6**, depending on geometry.
- The value of **3.62** used in this analysis falls well within that expected range, confirming its suitability for the given geometry.
- This selection reflects a theoretically **accurate stress magnification**, consistent with Peterson’s guidelines for plates with central holes.
- Therefore, using **k_{tg} = 3.62** supports the **validity of the analytical stress prediction** and enhances confidence in the comparison with FEA results.

Comparison of Theoretical and ANSYS Results

The theoretical stress was calculated as 45,250 psi, while the FEA result from ANSYS yielded 45,312 psi.

This results in a very small error of 0.13%, indicating an excellent agreement between the analytical and simulation results.

Such a minimal deviation demonstrates the accuracy and reliability of the finite element model.

Mesh Comparison Results

COARSE MESH

- 4-Node Elements: Not very accurate. The stress values vary a lot between nodes and are usually lower than expected.
- 8-Node Elements: Slightly better than 4-node elements, but still not very close to the theoretical stress.

MEDIUM MESH

- 4-Node Elements: Accuracy improves, but there's still some noticeable difference in stress between nearby nodes.
- 8-Node Elements: Much better stress prediction. The results are smoother and closer to the theoretical value.

FINE MESH

- 4-Node Elements: Results get very close to the expected values, but there's still a bit of inconsistency between node values.
- 8-Node Elements: Excellent results. The stress is very accurate, smooth across the mesh, and nearly matches the theoretical calculation.

RECOMMENDATIONS

Prefering 8-Node Quadratic Elements
These elements showed superior accuracy and smoother stress results, especially in stress concentration regions.

Use Medium Mesh for Efficiency
A medium-density mesh with 8-node elements offers a good balance between accuracy and computational time.

Fine Mesh for High Precision
Use fine mesh with 8-node elements when very high accuracy is required—e.g., in critical components or for validation.

Avoid Coarse Mesh for Accurate Stress Prediction
Coarse meshes, especially with 4-node elements, underestimated stress and showed high nodal differences.

Validating Theoretical Results
By comparing FEA results with analytical values (e.g., 45,250 psi theoretical vs. 45,312 psi from ANSYS) to confirm model accuracy.