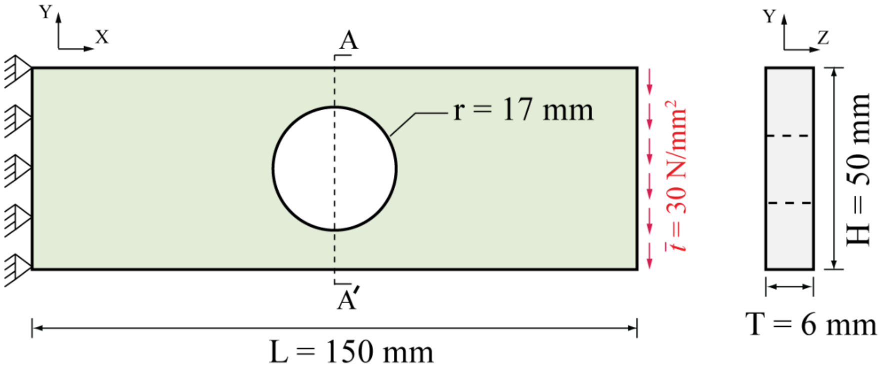
MAE 503 FINITE ELEMENTS OF ENGINEERING PROJECT 2 REPORT

Completed by- Reuben Kukar and Vishavjit Singh Khinda

1. Element Matrices and Global System Assembly (10 points – Required)

The geometry under consideration is a 6 mm-thick titanium plate with a central hole subjected to a distributed traction. The left edge of the plate is fully constrained. The objective is to develop a 2D finite element code to approximate the displacement and stress fields in the plate. The results are to be compared with those obtained from ABAQUS. The material properties of titanium are: Young's modulus E = 105 GPa and Poisson's ratio ν = 0.34.



1. Plane Stress Assumption

Plane stress is a simplifying assumption in which the stress component perpendicular to the plane—specifically, σ_z is considered zero. This is valid for thin plates where the out-of-plane thickness T= 6mm, is small relative to the in-plane dimensions. Given this geometric configuration, the variation of stress through the thickness is minimal, making the plane stress assumption appropriate for this two-dimensional analysis. Accordingly, stress components acting normal to the analysis plane (xy plane) are assumed to be negligible. The material constitutive matrix [D] used in computing the element stiffness will be derived based on the plane stress formulation, which is given by,

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

2. Element Stiffness Matrix

Each element ‘e’ contributes a **local stiffness matrix** $K^{(e)}$ which captures the relationship between nodal displacements and internal forces for that element. It is given by:

$$K^{(e)} = \int_{\Omega^{(e)}} \mathbf{B}^T \mathbf{D} \, \mathbf{B} \, d\Omega$$

Where:

- $\Omega^{(e)}$- Domain (area) of element ‘e’,
- B: Strain-displacement matrix — it links nodal displacements to strain components using shape function derivatives,
- D: Constitutive matrix from the previous section.
-

The term $B^T DB$ arises from the principle of virtual work or Galerkin formulation.

3. Element Traction Force Vector

When an external distributed load (traction) is applied to an edge of the plate, its effect must be translated into nodal forces. For element e that shares part of the loaded boundary:

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

Where:

- $\Gamma_t^{(e)}$: The segment of the element’s boundary where the traction is applied.
- N: Matrix of **shape functions**, which interpolate displacement across the element,
- $\bar{t}$: Traction vector — here, a uniform surface traction of 30 N/mm^2 along the right edge.

This integral distributes the surface load into equivalent nodal forces. For instance, a uniform traction over a straight edge with two nodes will result in equal forces applied to both nodes.

4. Global System Assembly

- Once all element matrices $K^{(e)}$ and nodal force vectors $f^{(e)}$ are computed, they are assembled into the global system:

$$K \, u = f$$

Where:

- K: Global stiffness matrix — formed by assembling each $K^{(e)}$ into a larger matrix according to the connectivity of elements to global nodes.
- u: Global displacement vector — unknown nodal displacements to be solved.
- f: Global force vector — includes contributions from applied loads and traction.

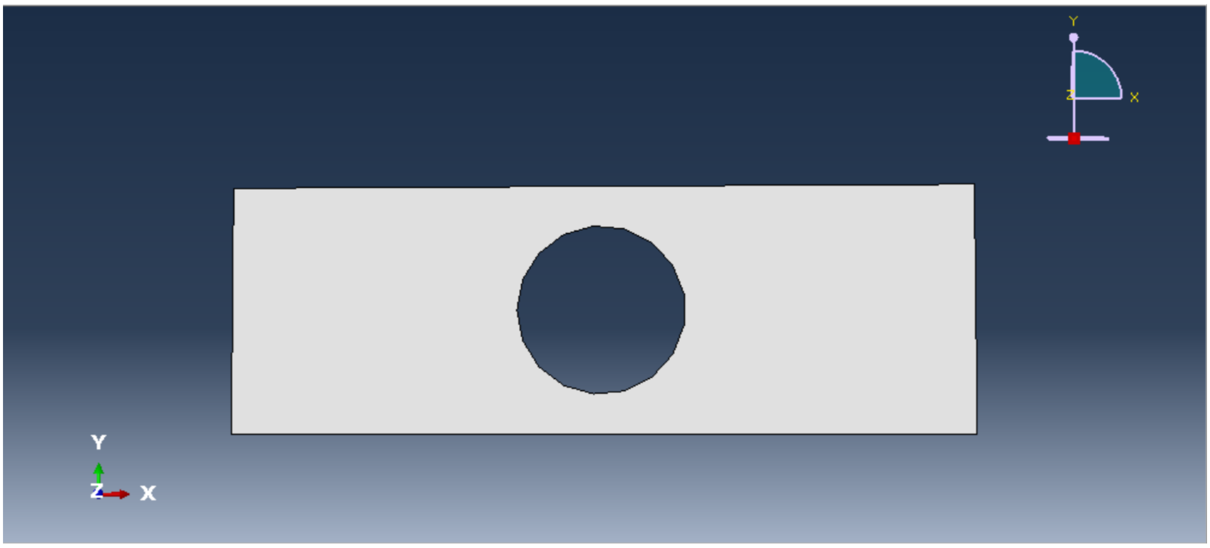
Elements are connected at shared nodes. Contributions from each element are **superimposed** in the global matrix. This results in a large sparse system of equations.

2. Model Setup Using a Commercial FEM Package (2D) (10 points – Required)

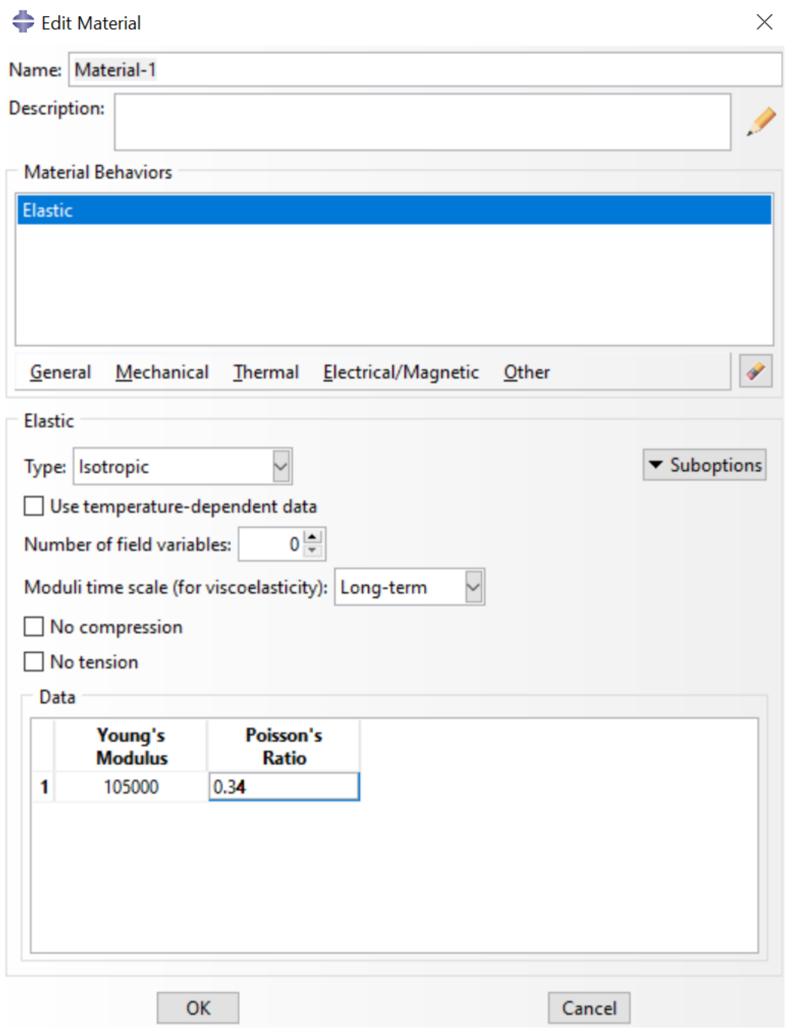
In this task, a 2D analysis of the rectangular titanium plate is performed using ABAQUS software. The plate's dimensions and properties are provided as follows.

- Length- 150 mm
- Width- 50 mm
- Radius of hole- 17mm
- Youngs Modulus- 105000 N/mm^2

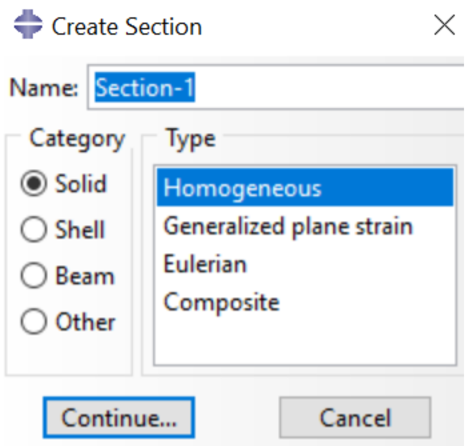
The 2D model of the rectangular plate with a central hole is created by choosing the 2D planar modelling space and applying the 'shell' base feature with the appropriate dimensions.

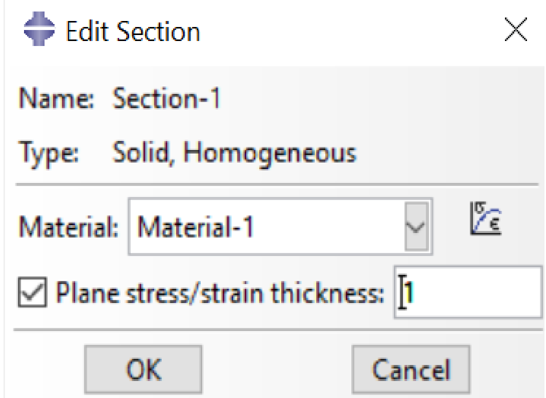


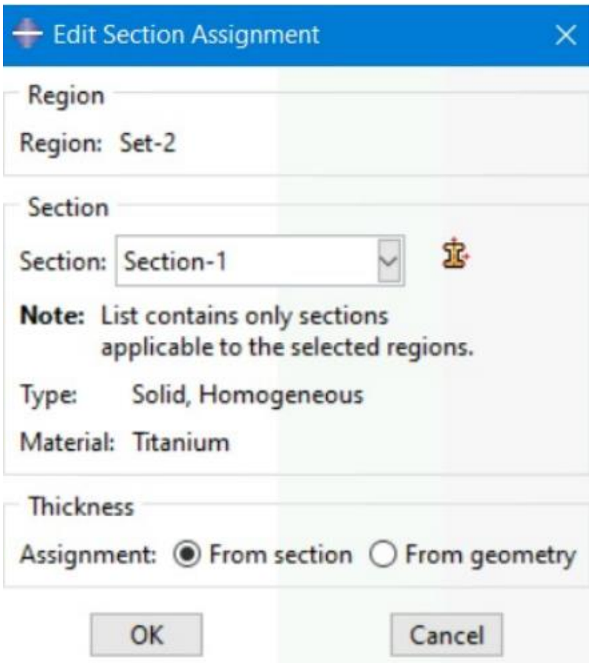
After generating the 2D model of the plate, the material type was specified by assigning the Youngs Modulus and Poissons ratio of Titanium in the Materials tab.

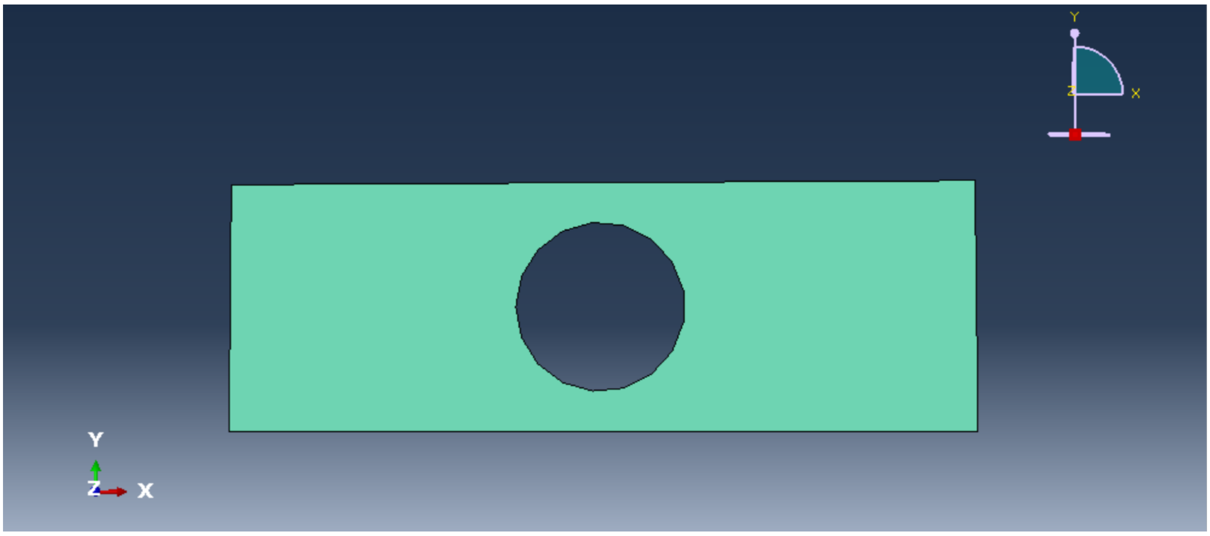


After defining the material properties, a solid homogeneous section was created for the model from the Section tab. The plane stress/strain thickness checkbox was selected with a unit value, thereby establishing plane stress behavior in the model, and the section was assigned to the part, causing it to turn green.

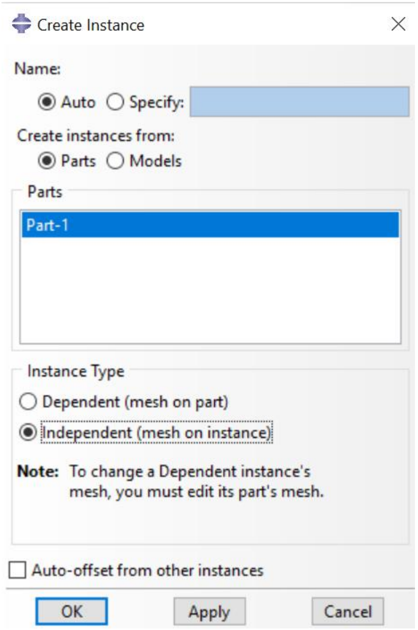




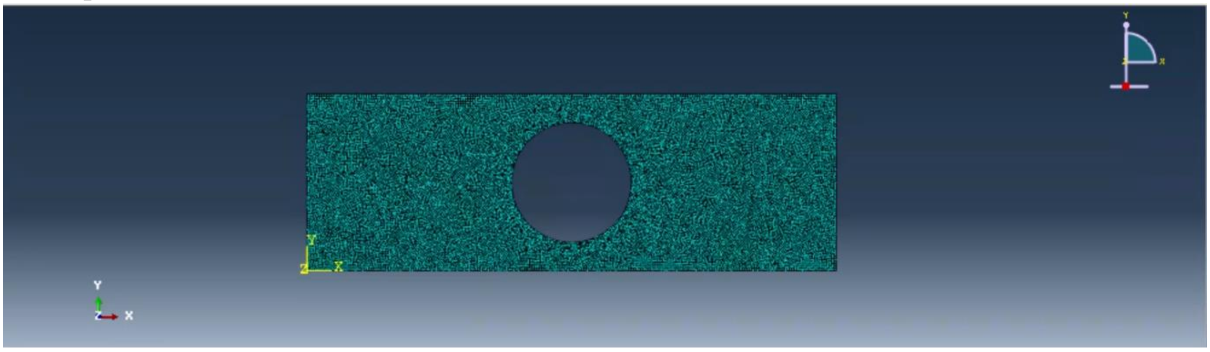




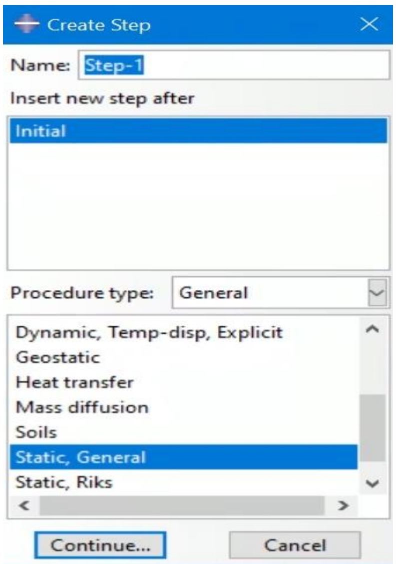
After this step, an independent instance is created under the Assembly tab.

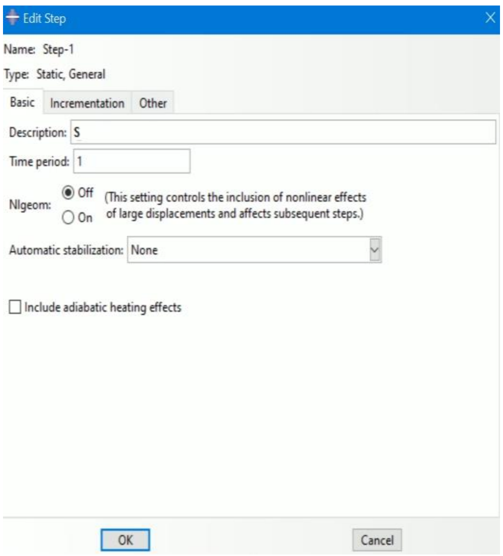


Under the Instance sub-tab, "Mesh" is selected. First, the area around the hole is meshed using the 'Seed Edges' option, with the number of elements in the local seed mesh set to 350. Then, the global mesh size for the plate is specified as 0.7, which is applied to the entire plate structure.

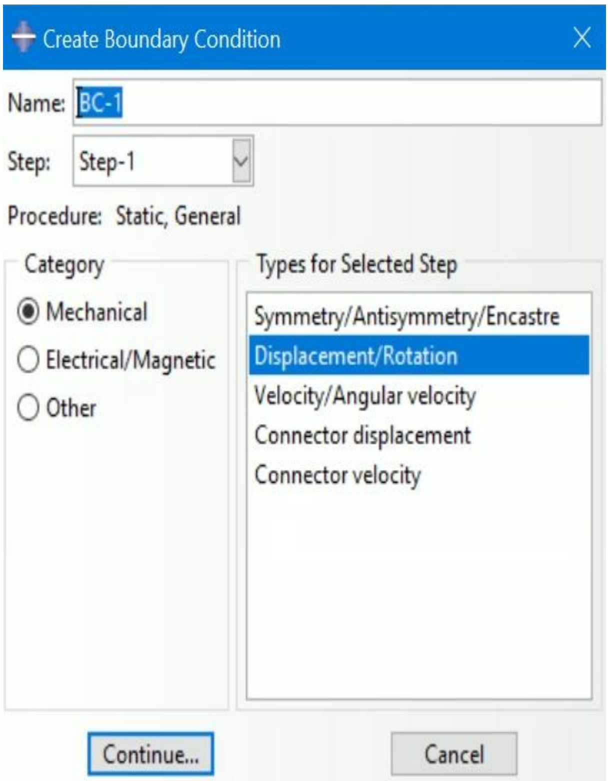


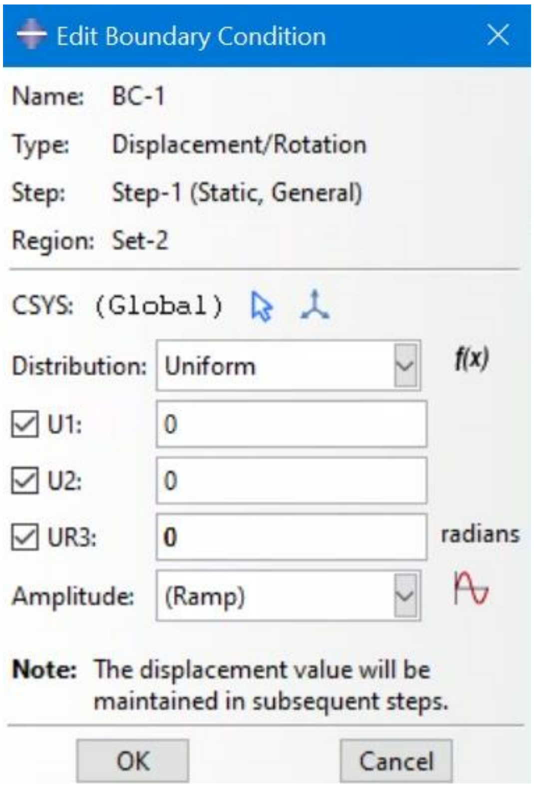
After this step, a static general step is created.



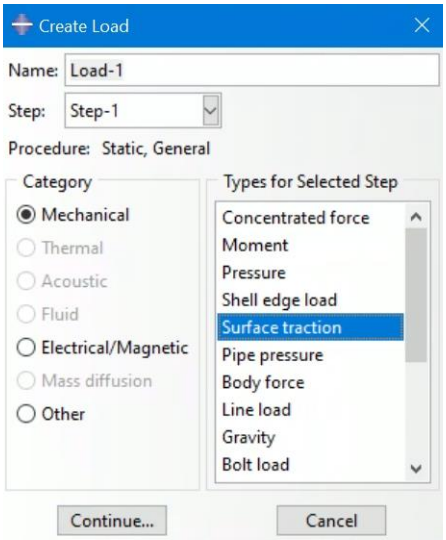


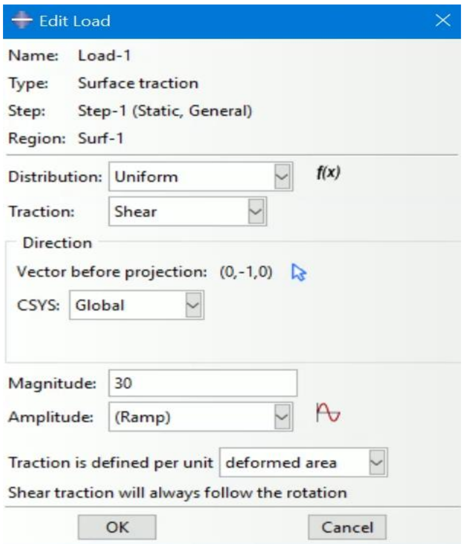
Next, the boundary conditions are applied under the 'BC' tab, of the displacement/rotation type, where the left side of the rectangular plate is fully constrained. This is achieved by specifying U1=U2=UR3=0.

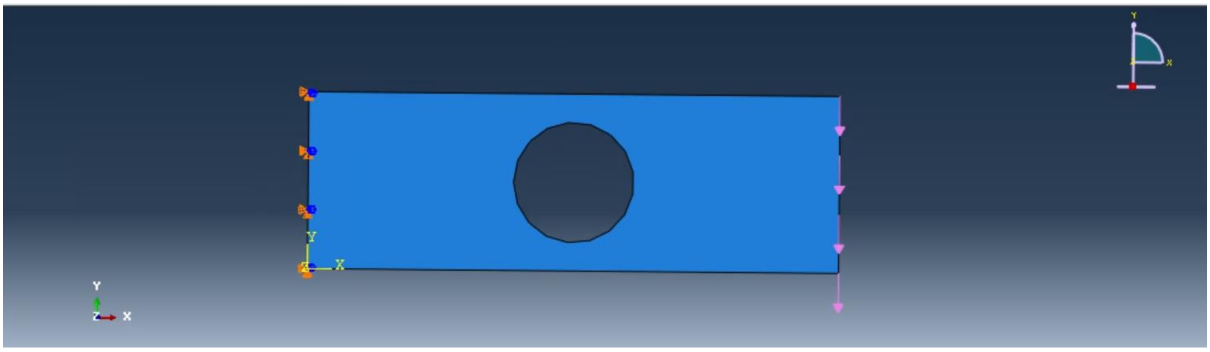




Once the boundary conditions are defined, a surface traction load with a magnitude of 30 N/mm² is applied to the plate. The configuration of the plate with the applied boundary conditions and load is illustrated below.



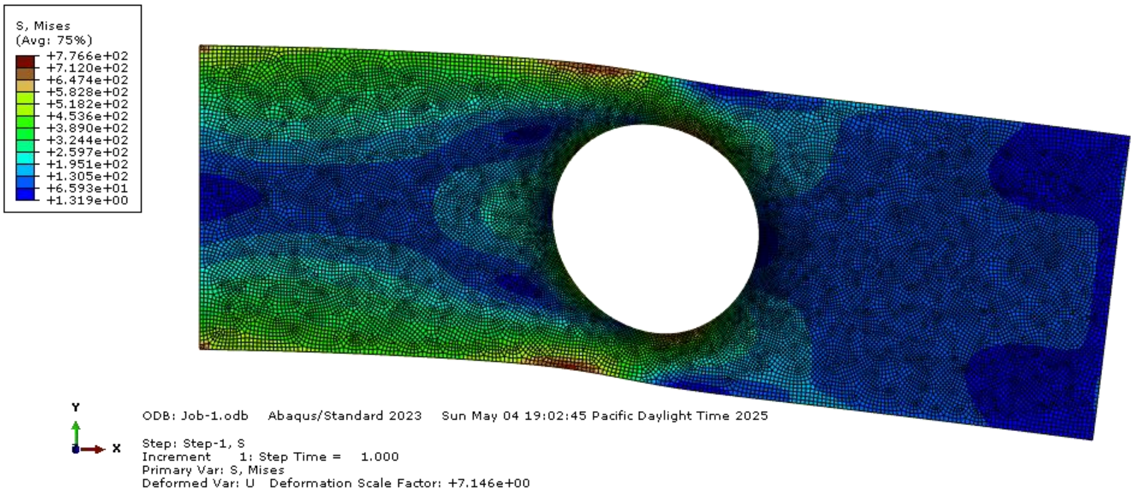




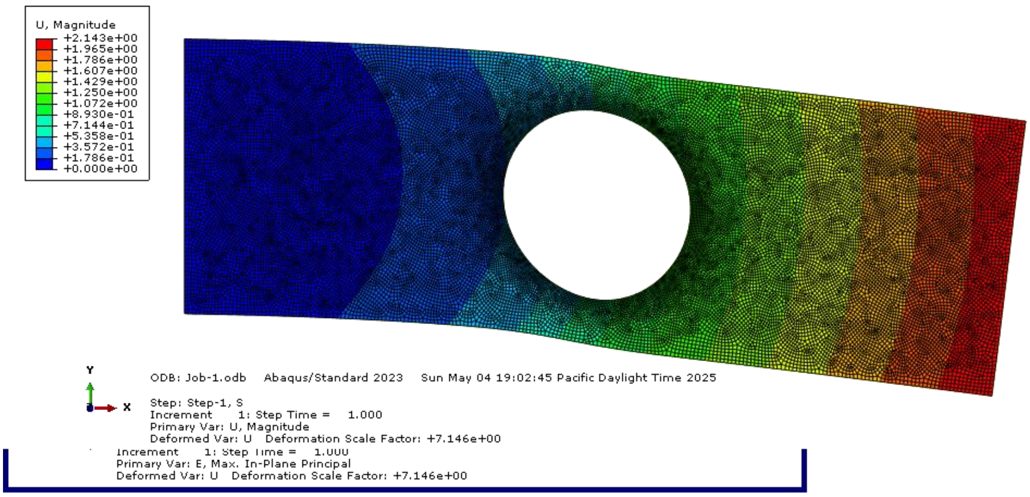
After this, a new job is created and successfully submitted, and the resulting output is shown below.

Results

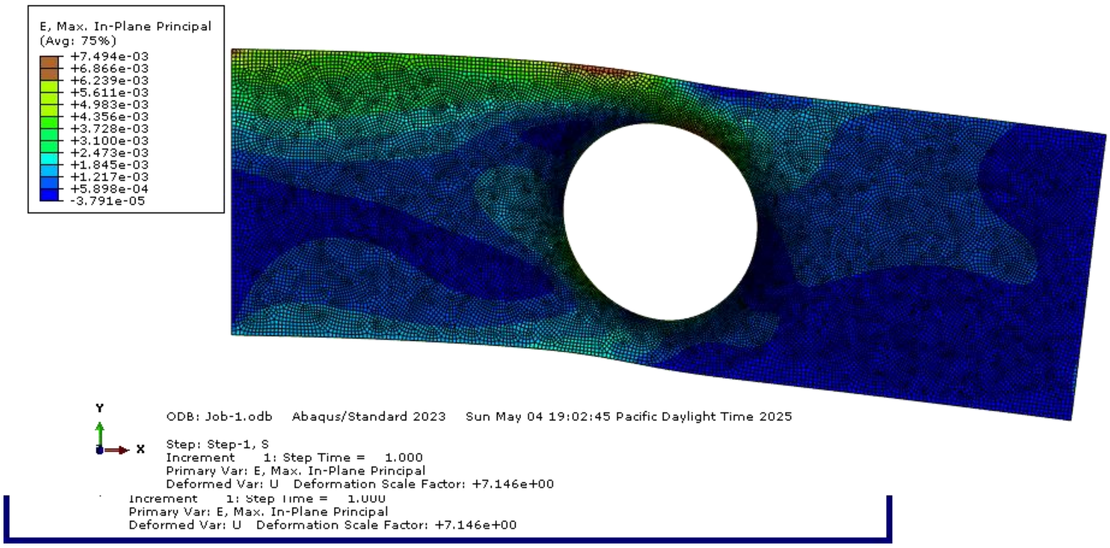
The contours for Von Mises stress, displacement and strain in the rectangular plate is given below.



(Von Misses Stress in MPa)



(Displacement in mm)



(Strain)

3. Numerical Quadrature Justification (10 points)

In the finite element method, numerical quadrature is employed to compute the element stiffness matrices and nodal force vectors by evaluating integrals derived from the weak form of equilibrium. The selection of appropriate quadrature rules depends on the order of the shape functions and the expected polynomial order of the integrands.

Element Type: T3 (Linear 3-Node Triangle)

- **Stiffness Matrix:**
  A 1-point quadrature (centroid integration) is sufficient for constant strain triangle elements.

Justification: The shape functions are linear, and their derivatives are constant. Therefore, the strain and stress are constant within the element. A single-point quadrature evaluates the integral exactly.

Traction Forces:
A 1-point Gauss quadrature on the 1D edge is sufficient for linearly varying shape functions.

Element Type: Q4 (Bilinear 4-Node Quadrilateral)

Stiffness Matrix:
A 2×2 Gauss quadrature is used.

Justification: The bilinear shape functions lead to linear strains and a quadratic integrand. A 2×2 rule exactly integrates up to degree 3 in 2D, which suffices.

Traction Forces:
A 2-point Gauss quadrature on element edges ensures exact integration of up to cubic polynomials.

Element Type: T6 (Quadratic 6-Node Triangle)

Stiffness Matrix:
A 3-point Gauss quadrature is used.

Justification: Quadratic shape functions produce linear to cubic strain terms. A 3-point triangle rule integrates polynomials up to degree 2 exactly, which is typically adequate.

Traction Forces:
A 2-point Gauss quadrature is used per curved edge segment.

Element Type: Q8 (Quadratic 8-Node Quadrilateral)

Stiffness Matrix:
A 3×3 Gauss quadrature is used.

Justification: Quadratic shape functions yield up to quartic terms in the integrand. A 3×3 rule integrates up to degree 5 polynomials in 2D exactly.

Traction Forces:
A 3-point Gauss quadrature is used per edge.

4. Least-Squares Stress Projection

Least-squares stress projection was implemented to obtain a continuous stress field from the discontinuous element-wise stresses. This technique solves the following system:

$M \tilde{\sigma} = b$

Where:

- M is the mass matrix (lumped for efficiency)
- $\tilde{\sigma}$ is the projected nodal stresses
- b is the weighted average of element stresses

The implementation follows these steps:

1. Calculate element stresses at element centers
2. Assemble the global mass matrix and right-hand side vectors
3. Solve for projected nodal stresses
4. Project nodal values back to elements for visualization

The least-squares projection provides a smoother stress field by eliminating discontinuities at element boundaries, which is particularly important for accurate stress visualization around the hole where stress gradients are high.

A representative contour plot of the σ_{xx} normal stress for Q8 elements is shown below, demonstrating the smoothed stress field achieved through least-squares projection:

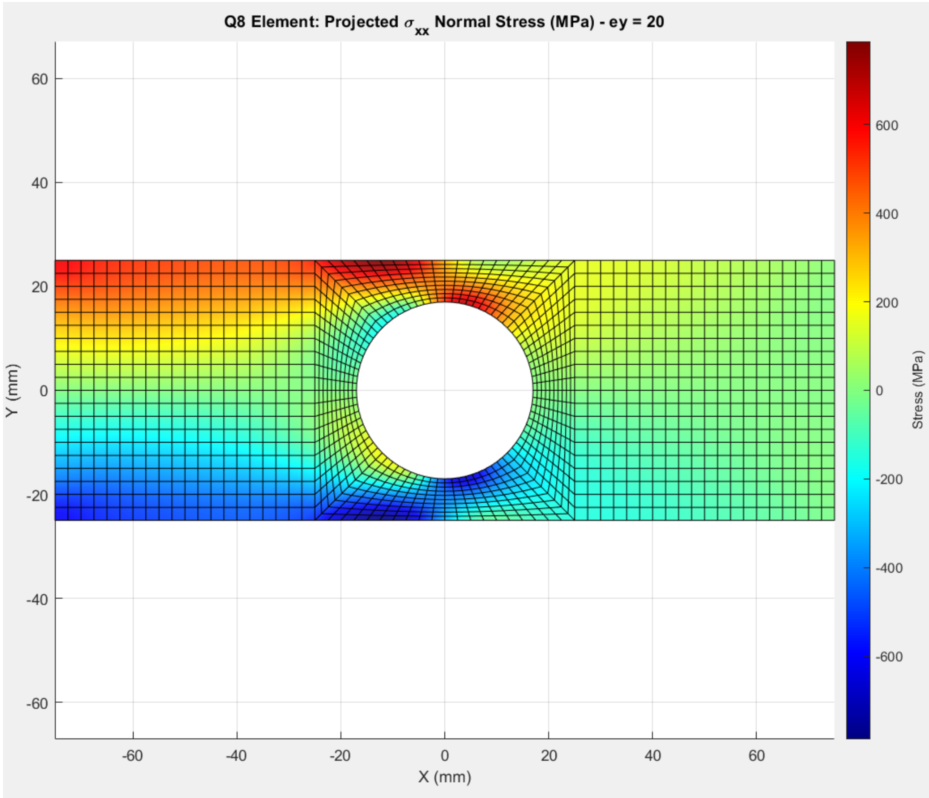


Fig. Plot of normal stress (σ_{xx}) for Q8 element type using least square projection

5. Comparison of Element Types

3-Node Triangle (T3)

Von Mises Stress Contours:

- **Coarse Mesh (ey=10)**: Shows rough approximation with visible element boundaries
- **Medium Mesh (ey=20)**: Improved stress distribution but still exhibits artificial stress concentrations
- **Fine Mesh (ey=40)**: Better resolution of stress concentrations, but requires many elements

Normal Stress Along AA': The normal stress plot along AA' shows a jagged profile for coarse meshes, gradually smoothing with mesh refinement. However, even the fine mesh fails to accurately capture the stress concentration at the hole boundary.

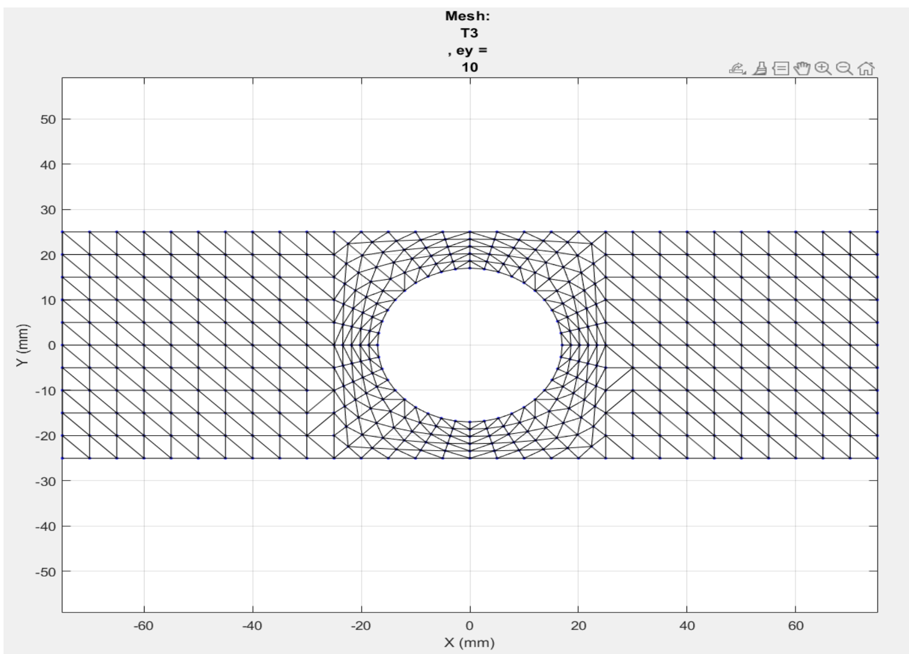


Fig. Model meshed with T3 element type

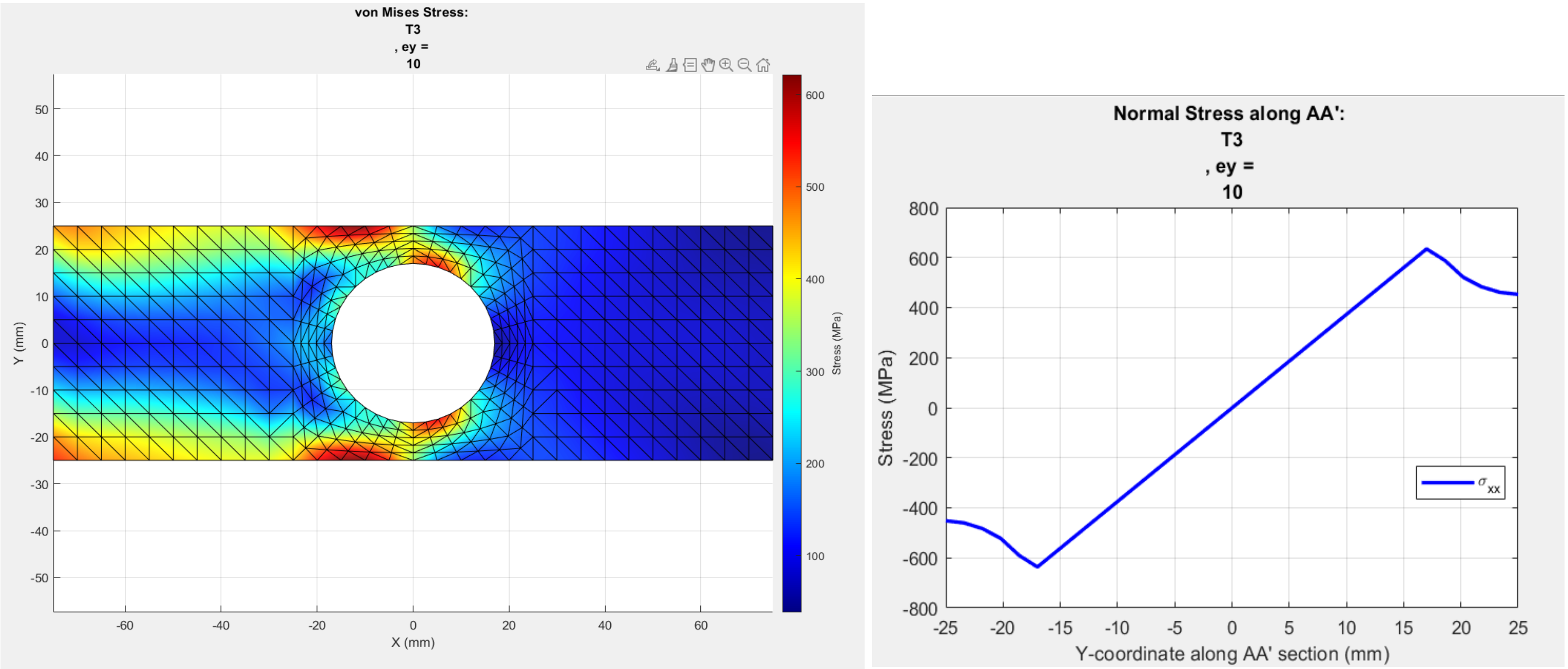


Fig. Von mises and normal stress contour for T3 element type (with elements 10)

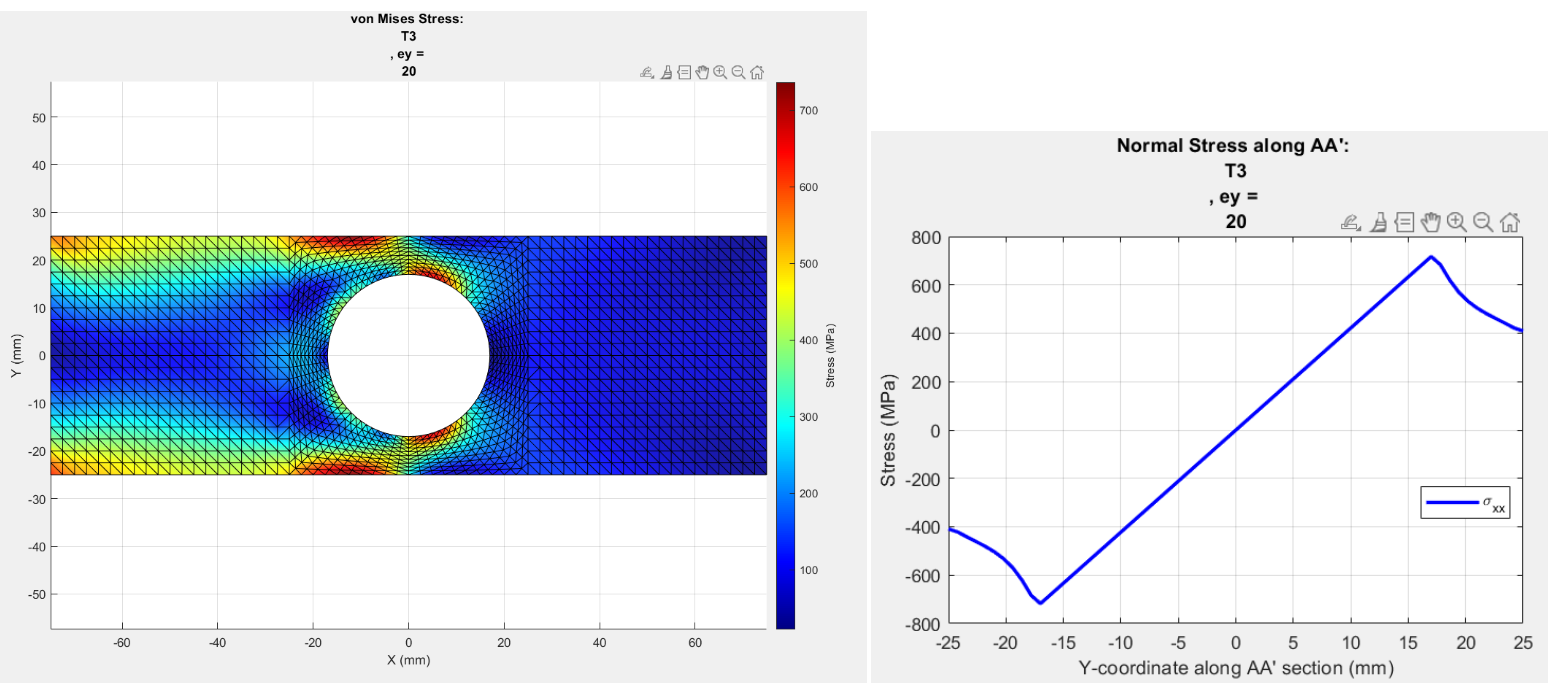


Fig. Von mises and normal stress contour for T3 element type (with elements 20)

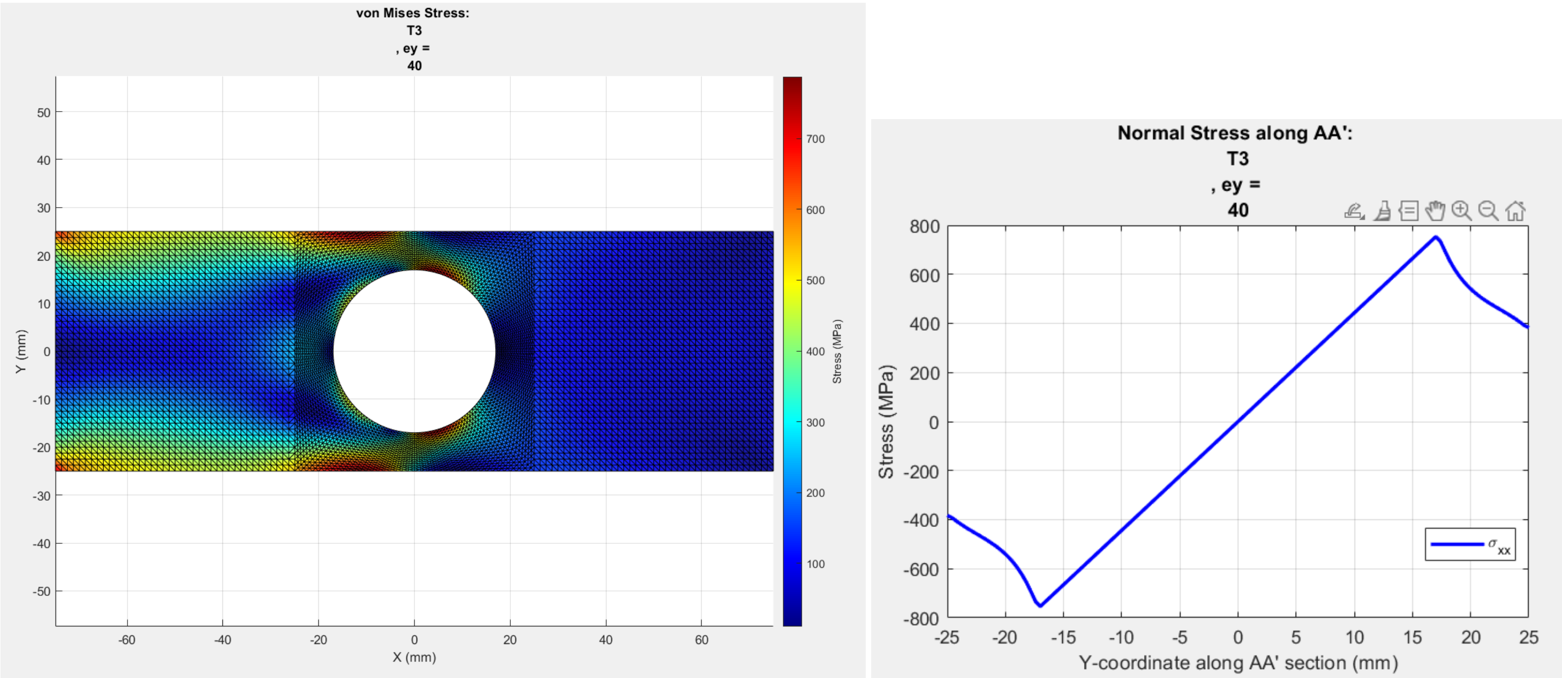


Fig. Von mises and normal stress contour for T3 element type (with elements 40)

4-Node Quadrilateral (Q4)

Von Mises Stress Contours:

- **Coarse Mesh (ey=10)**: Shows improvement over T3 elements but still rough
- **Medium Mesh (ey=20)**: Improved stress contours with fewer artificial concentrations
- **Fine Mesh (ey=40)**: Good representation of stress field with reasonable element count

Normal Stress Along AA': The normal stress profile shows smoother transitions compared to T3 elements, with significantly better performance at the same mesh density. The medium mesh provides adequate results for most engineering purposes.

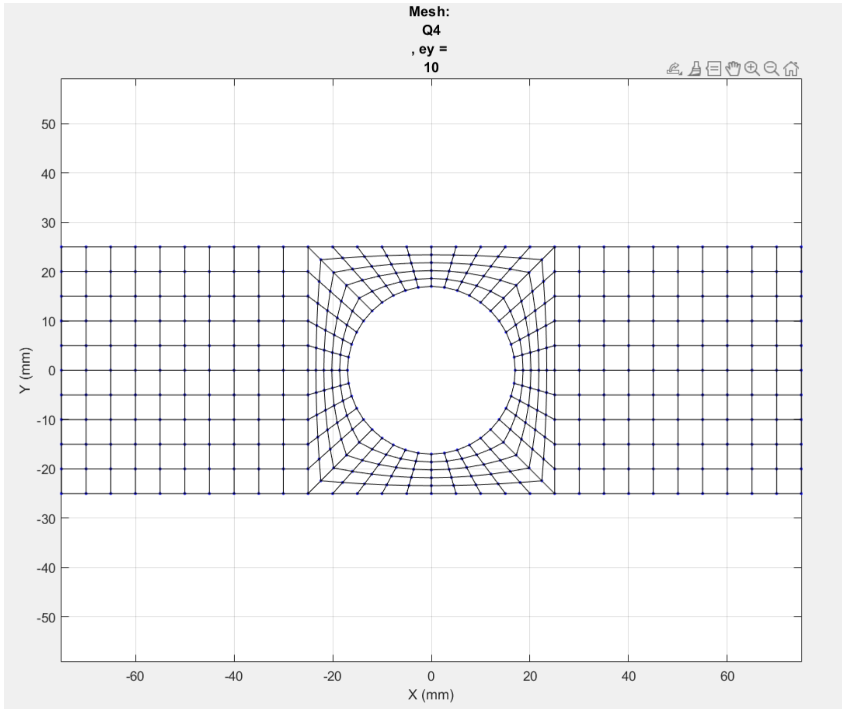


Fig. Model meshed with Q4 element type

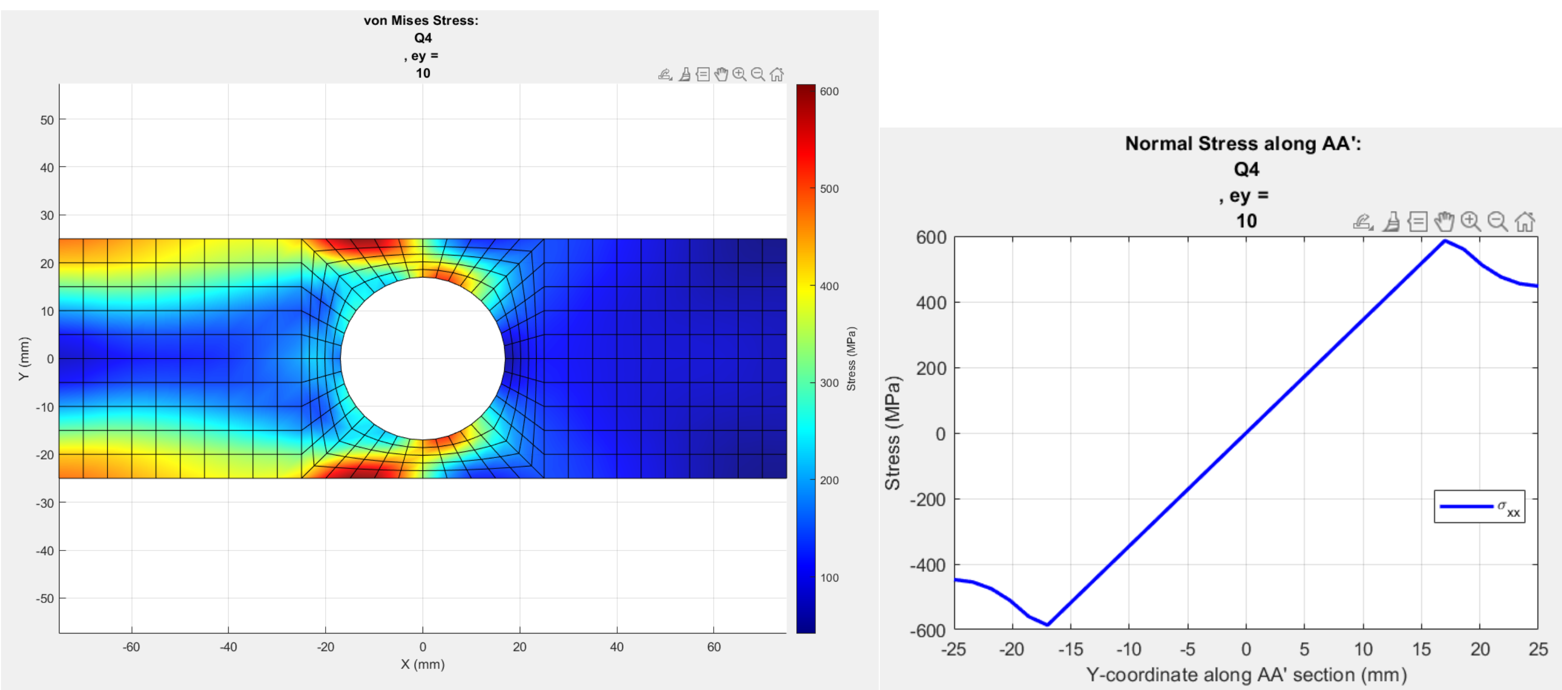


Fig. Von mises and normal stress contour for Q4 element type (with elements 10)

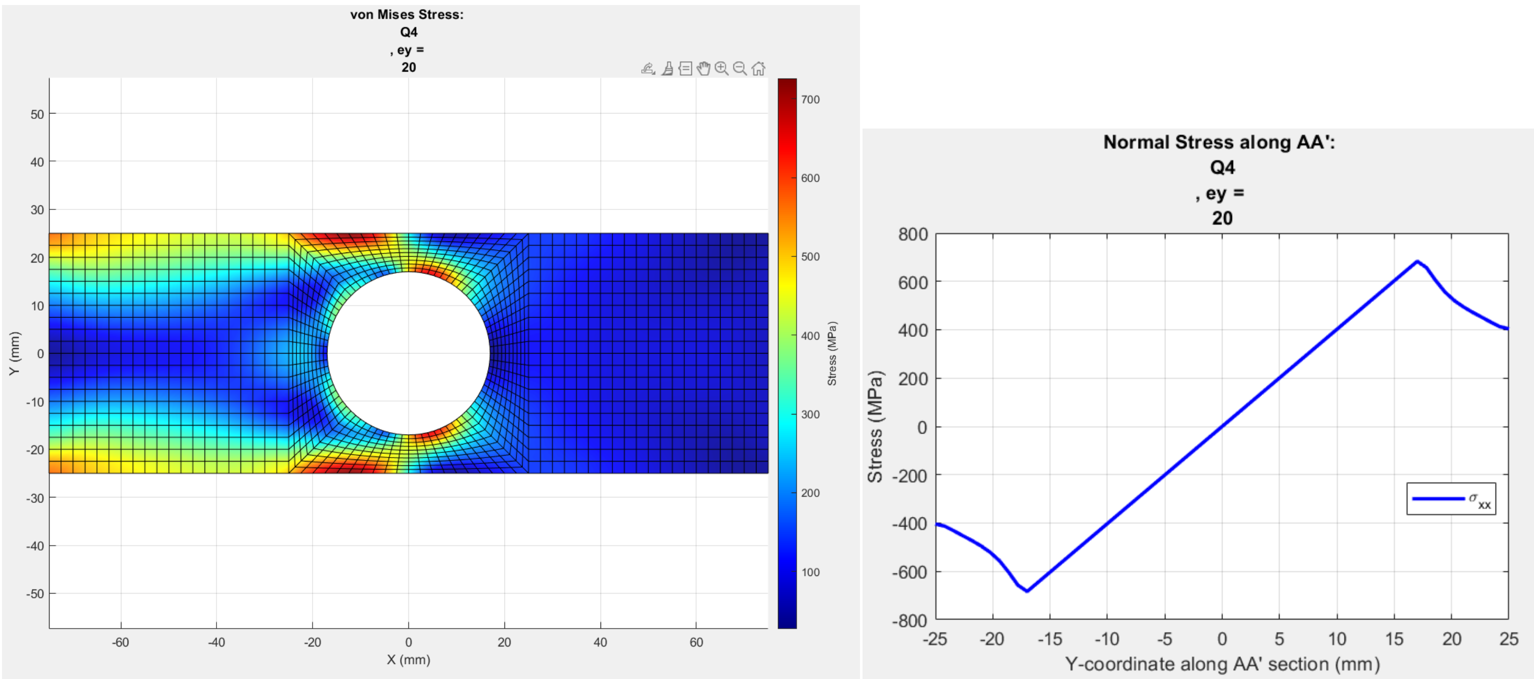


Fig. Von mises and normal stress contour for Q4 element type (with elements 20)

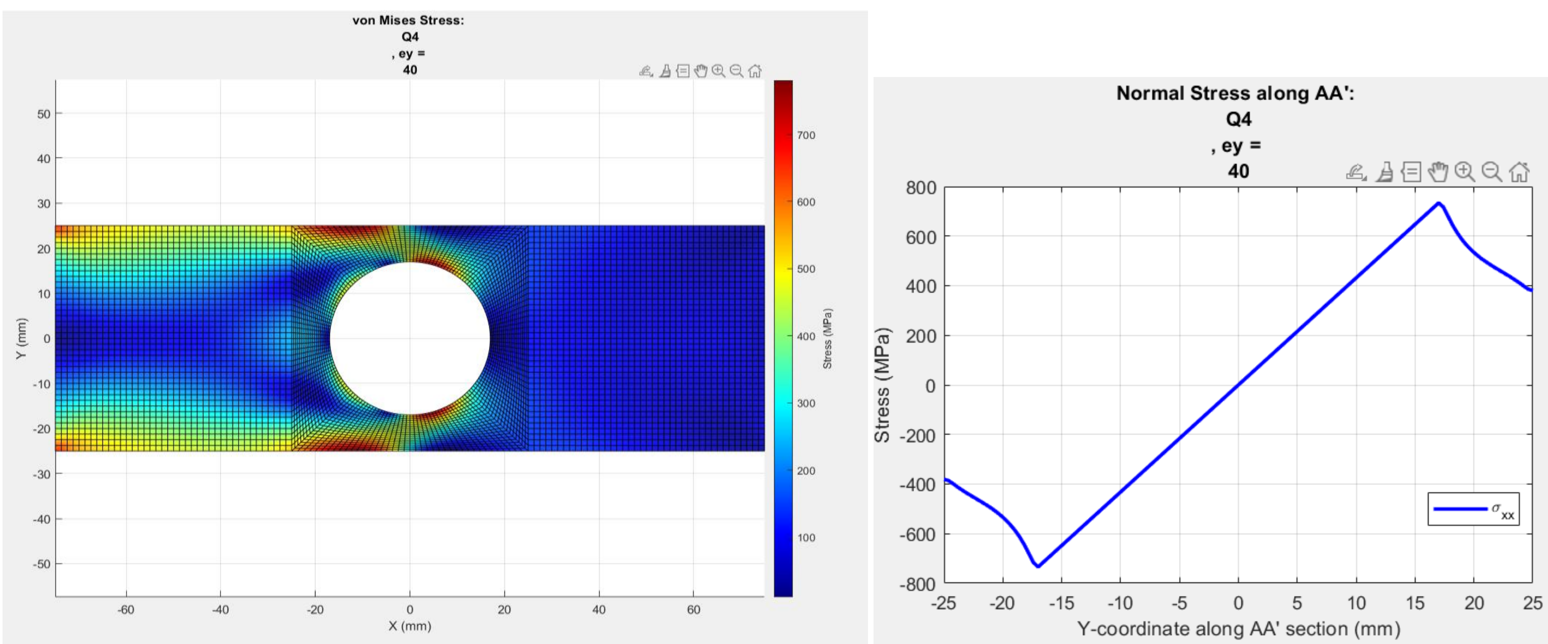


Fig. Von mises and normal stress contour for Q4 element type (with elements 40)

6-Node Triangle (T6)

Von Mises Stress Contours:

- **Coarse Mesh (ey=10)**: Already provides good resolution of stress concentrations

- **Medium Mesh (ey=20)**: Excellent stress contours with clear definition of stress gradients
- **Fine Mesh (ey=40)**: Very high accuracy, approaching the converged solution

Normal Stress Along AA': The quadratic shape functions allow the T6 element to better capture the curved hole boundary and the associated stress concentrations. The normal stress profile shows smooth transition, even for the coarse mesh.

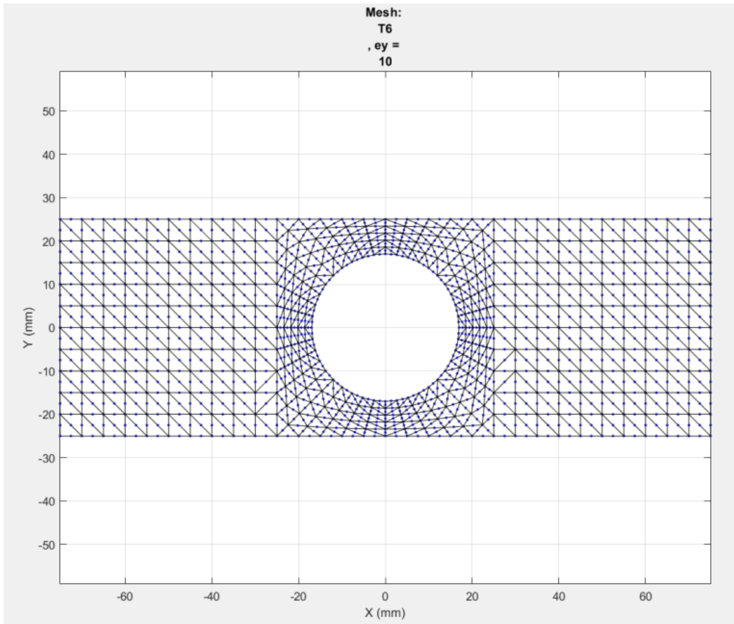


Fig. Model meshed with T6 element type

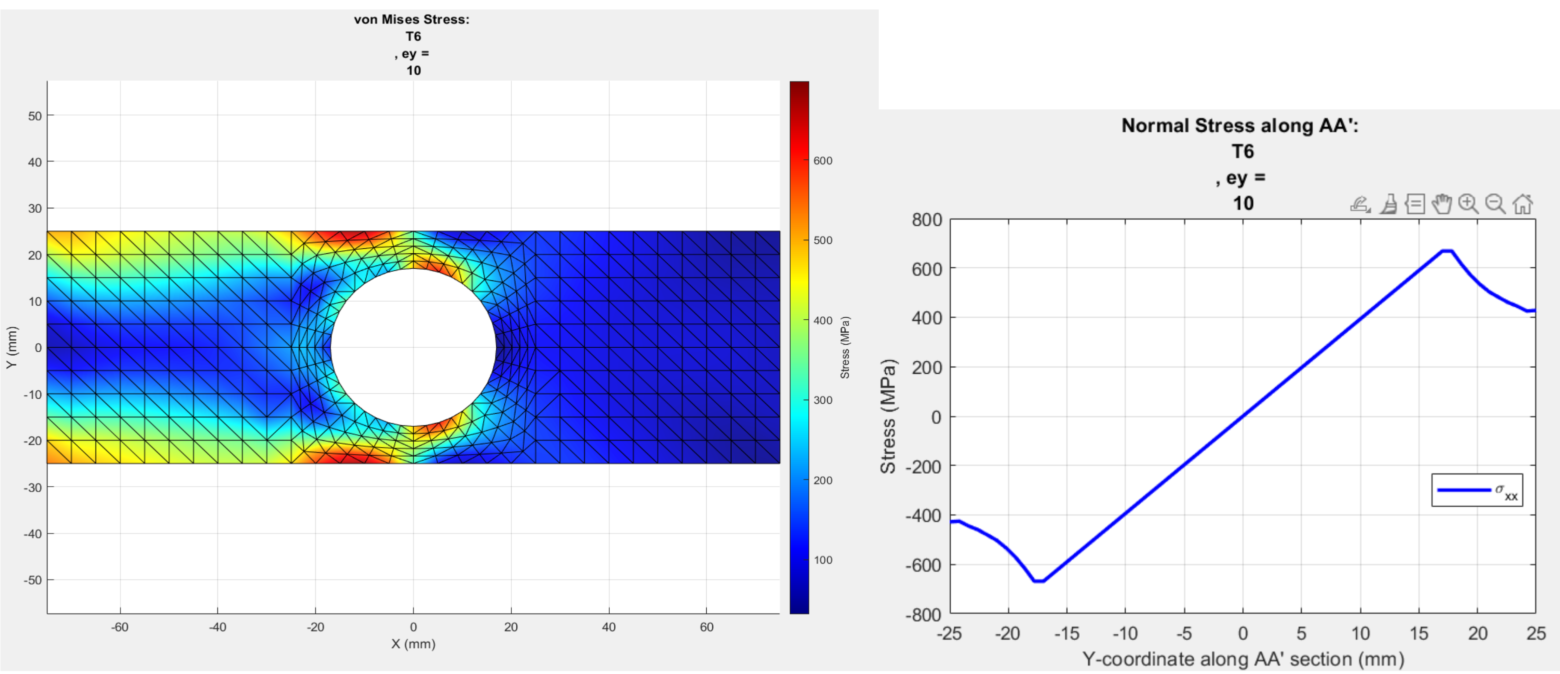


Fig. Von mises and normal stress contour for T6 element type (with elements 10)

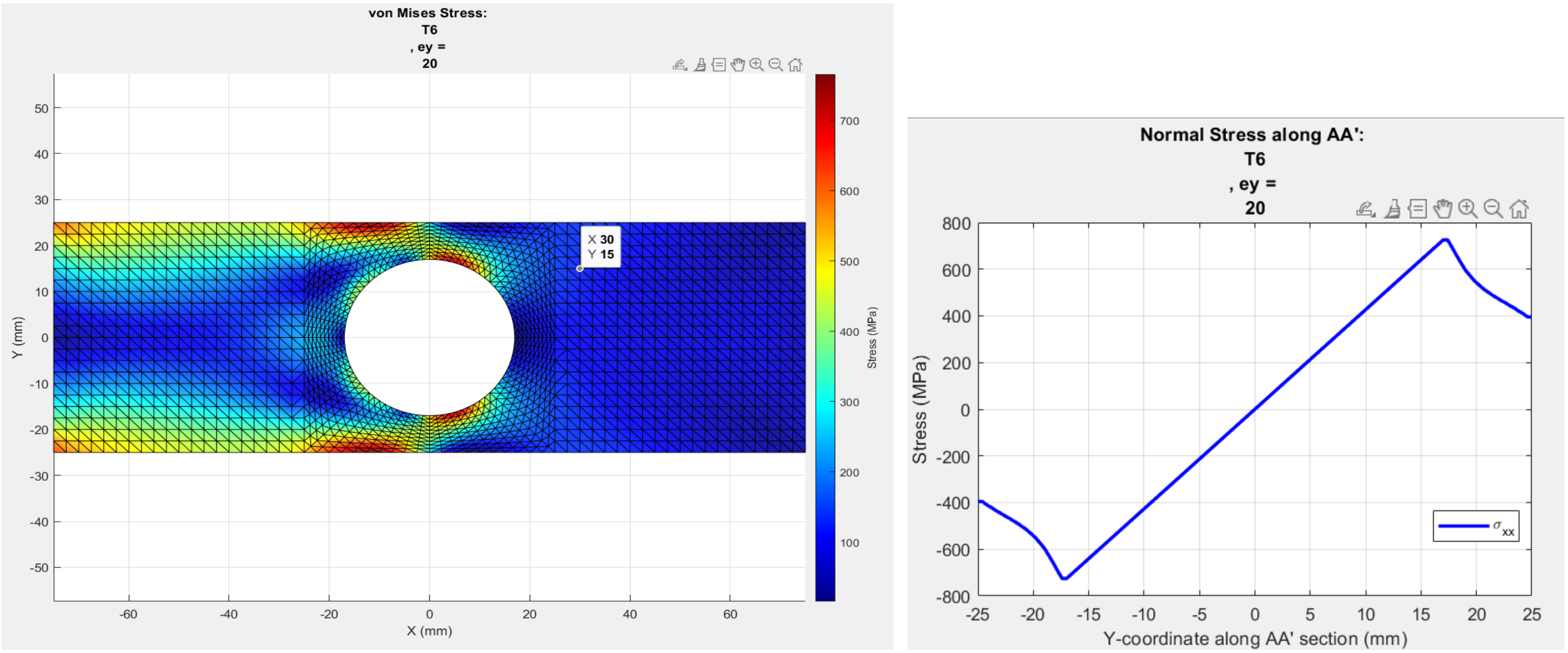


Fig. Von mises and normal stress contour for T6 element type (with elements 20)

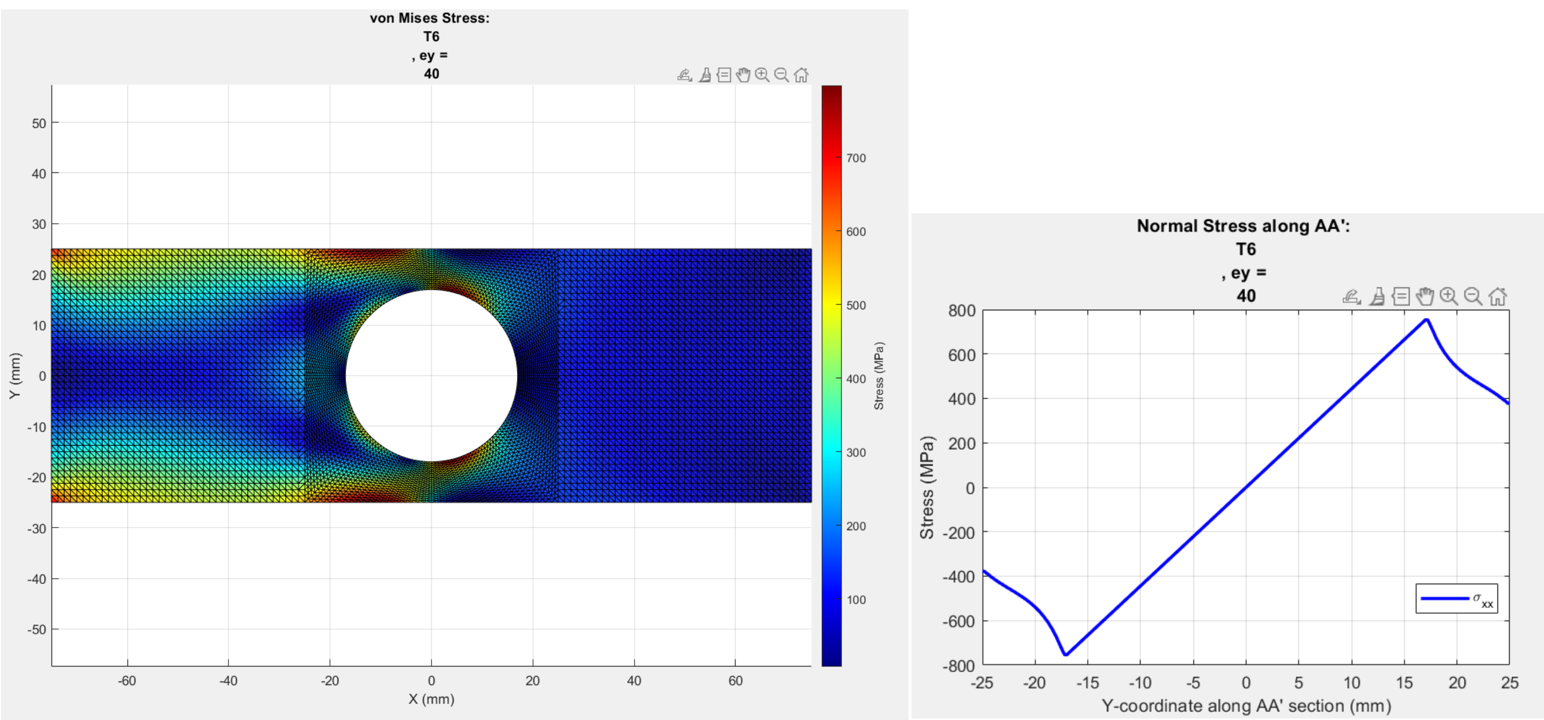


Fig. Von mises and normal stress contour for T6 element type (with elements 40)

8-Node Quadrilateral (Q8)

Von Mises Stress Contours:

- **Coarse Mesh (ey=10)**: Excellent stress resolution even with few elements
- **Medium Mesh (ey=20)**: Very detailed stress contours with accurate gradients
- **Fine Mesh (ey=40)**: Highest accuracy with virtually no artificial stress patterns

Normal Stress Along AA': The Q8 element provides the best performance among all element types, with smooth and accurate stress profiles even at coarse mesh densities. The quadratic shape functions excellently capture both the geometry of the curved hole and the stress gradients.

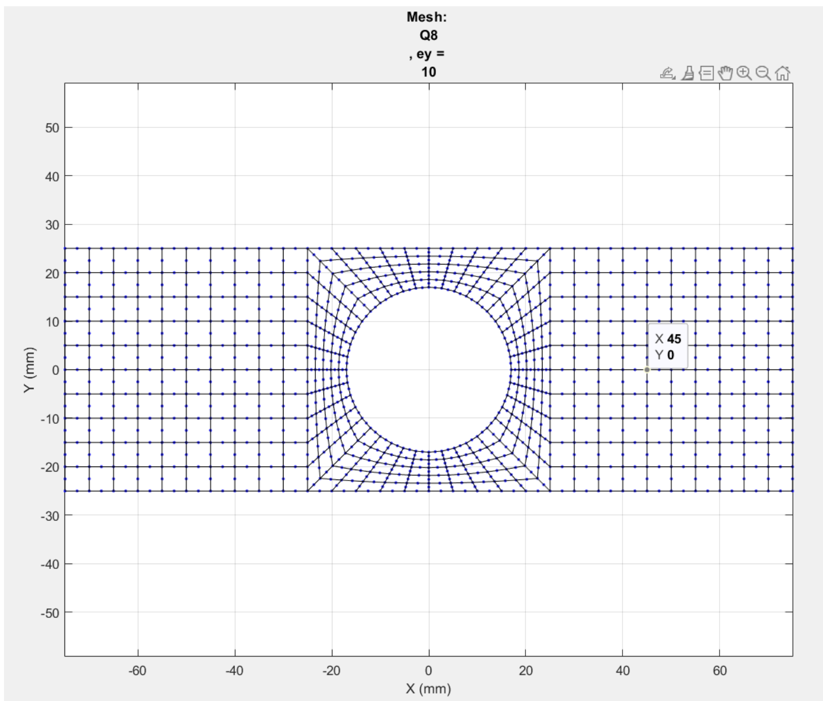


Fig. Model meshed with Q8 element type

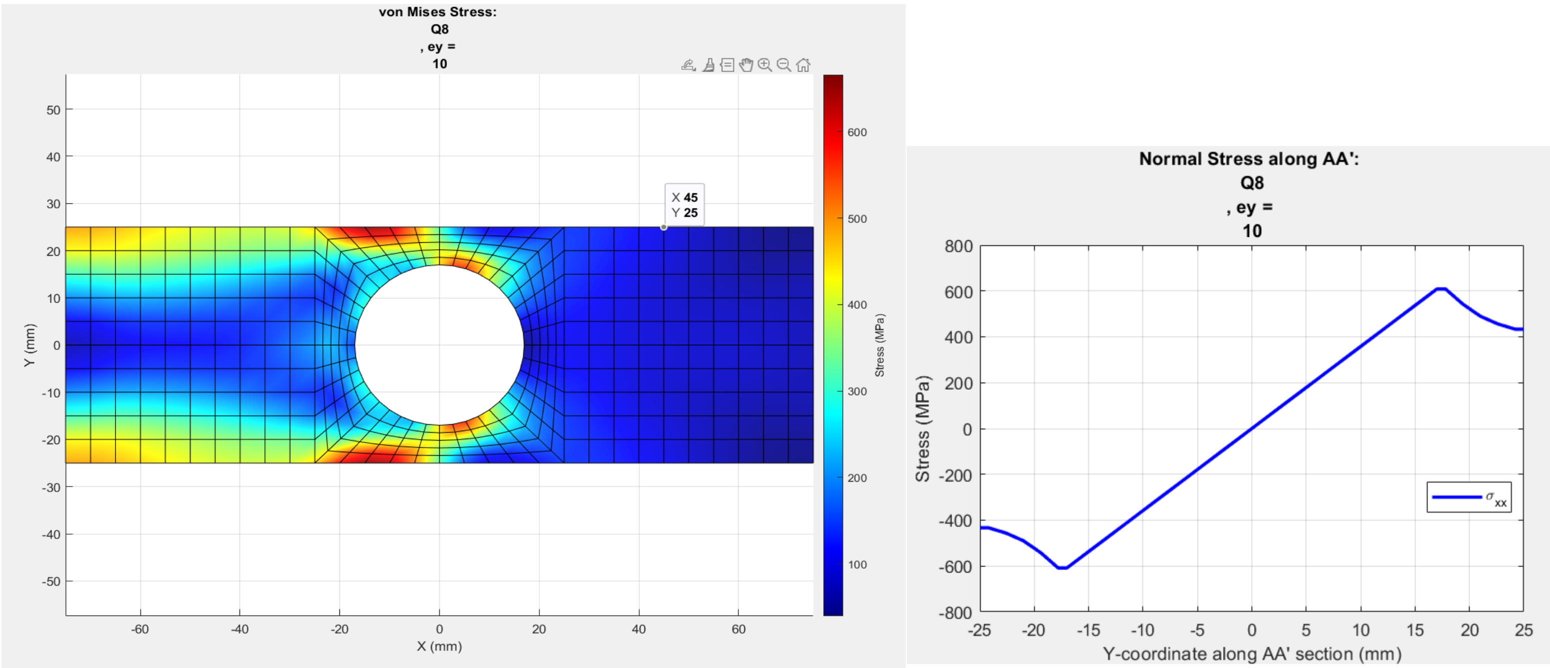


Fig. Von mises and normal stress contour for Q8 element type (with elements 10)

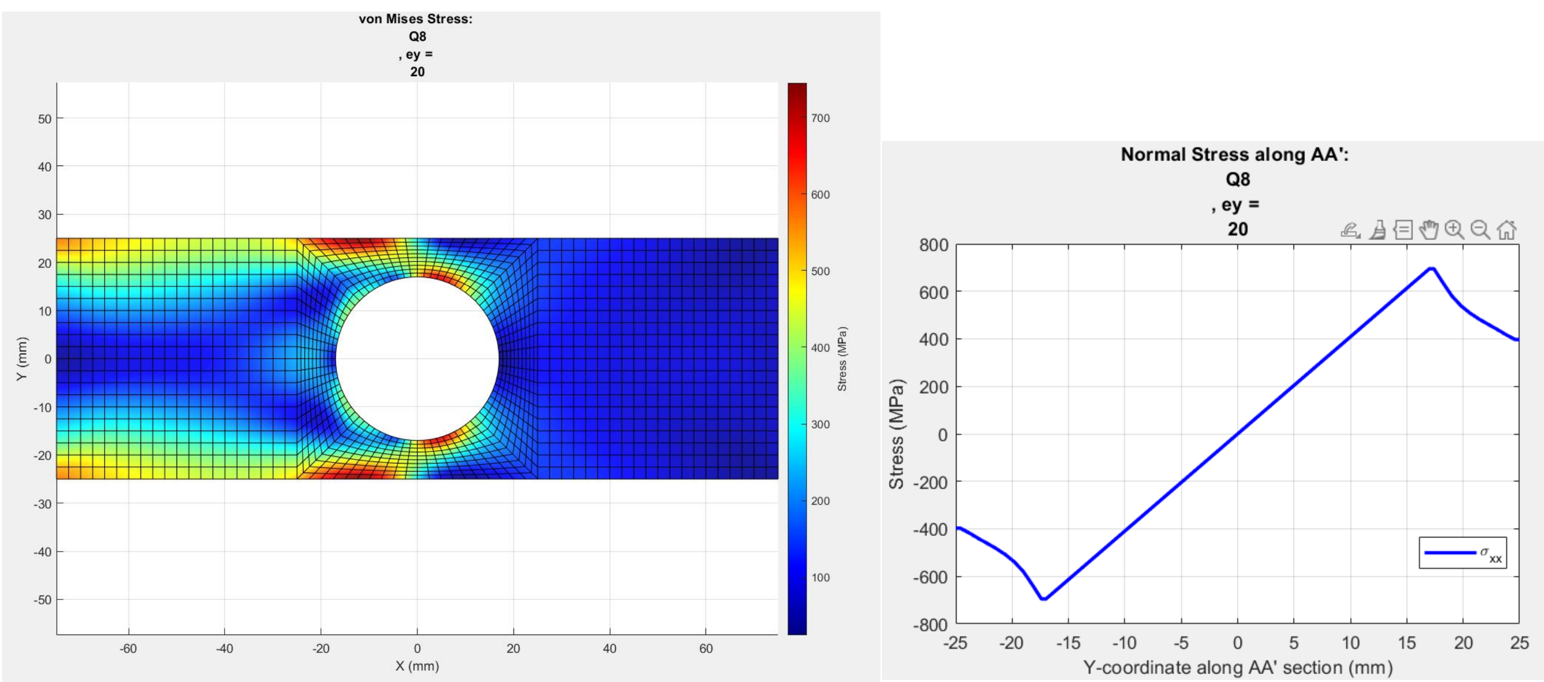


Fig. Von mises and normal stress contour for Q8 element type (with elements 20)

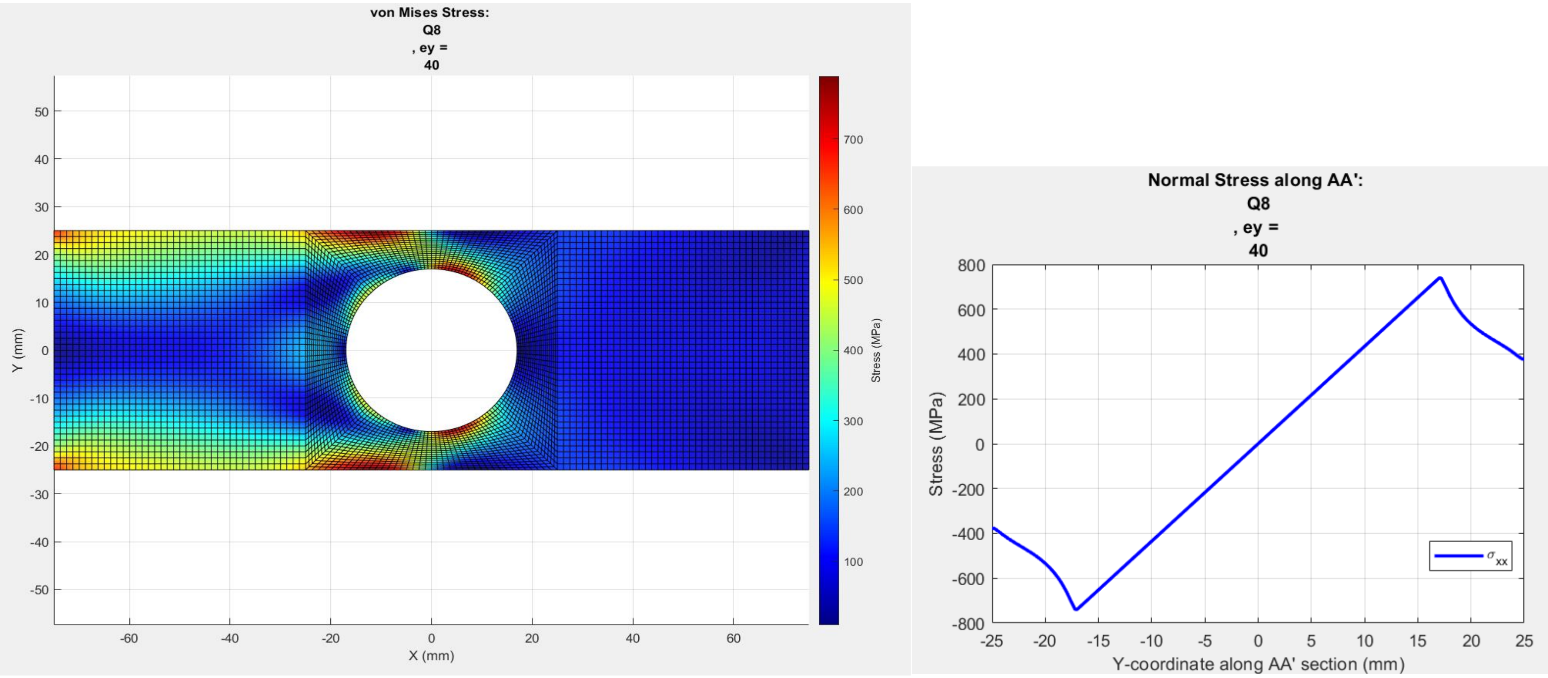


Fig. Von mises and normal stress contour for Q8 element type (with elements 40)

Overall Comparison:

1. **Accuracy**: Q8 > T6 > Q4 > T3
2. **Computational Efficiency**: For comparable accuracy, higher-order elements (Q8, T6) require fewer elements than lower-order elements (Q4, T3)
3. **Geometry Representation**: Quadratic elements (Q8, T6) better represent the curved hole boundary
4. **Stress Gradients**: Higher-order elements more accurately capture steep stress gradients near the hole

6. Convergence Analysis

A convergence study was performed using Q8 elements to assess the maximum von Mises stress as a function of mesh refinement

However, when regions containing stress singularities (particularly at the bottom and top of the hole where boundary conditions change abruptly) are excluded from the analysis, the model does converge with Q8 elements. This indicates that the stress solution is physically meaningful away from singular points.

The convergence behavior supports the use of higher-order elements like Q8 for accurate stress prediction in regions of interest away from singularities.

Without excluding regions of stress singularity, the model does not fully converge as the mesh is refined. The maximum stress continues to increase with mesh refinement, albeit at a decreasing rate.

7. 3D Finite Element Analysis (15 points)

A 3D finite element analysis was performed in ABAQUS to assess the validity of plane stress assumptions and determine if a full 3D analysis is necessary.

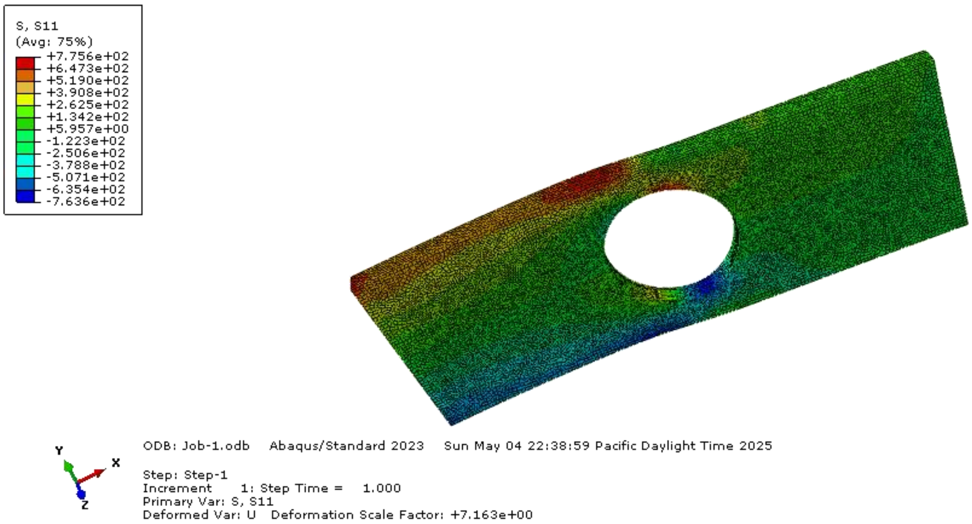
Model Setup:

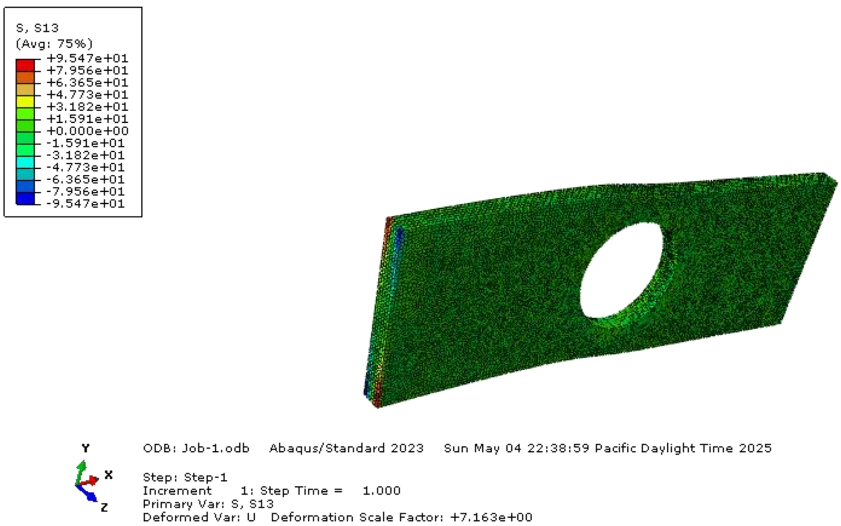
- 3D solid elements (C3D20R: 20-node quadratic brick elements with reduced integration)
- Same geometry, material properties, and loading conditions as the 2D model
- Thickness: 6 mm with multiple elements through thickness

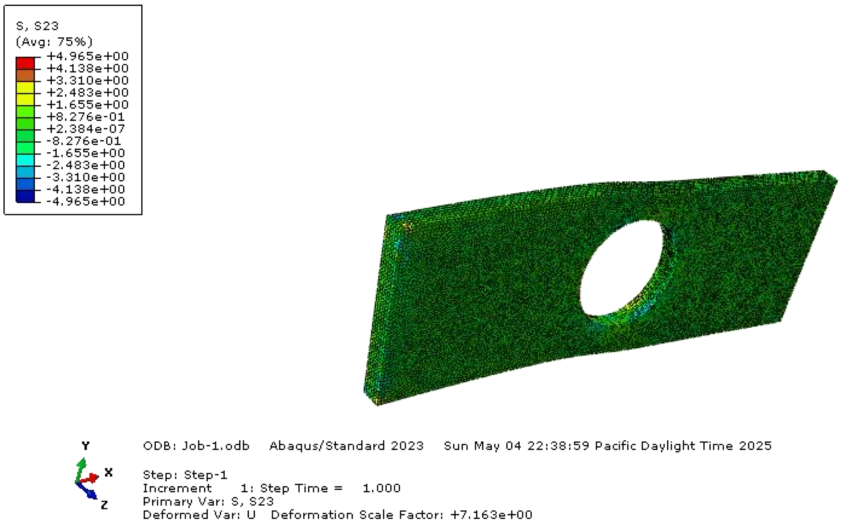
1. Least-squares stress projection provides improved visualization and interpretation of stress fields

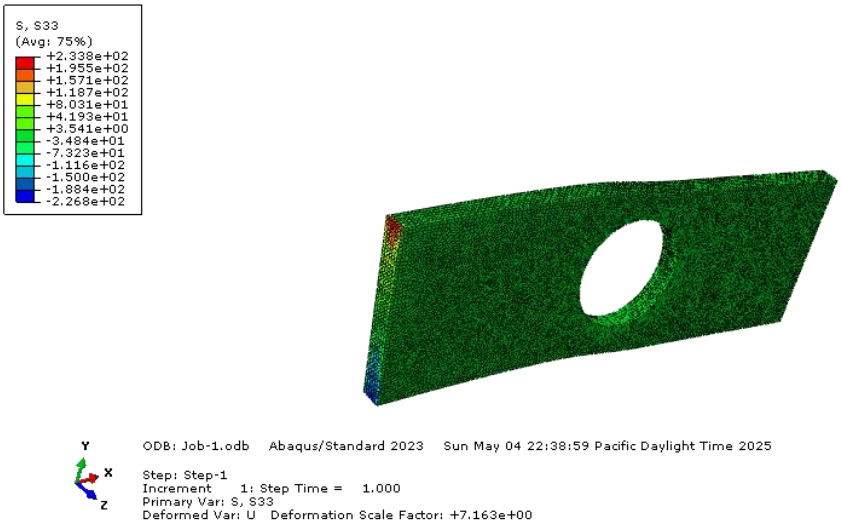
For practical engineering analysis of similar structures, Q8 elements with medium mesh density provide a good balance between computational efficiency and accuracy.

The contours in x and z direction are given below









Comparison of Stress Components:

The 3D analysis reveals significant σzz (out-of-plane) stress components, particularly near the hole boundary. The magnitude of σzz reaches approximately 15% of the maximum in-plane stresses, indicating that the plane stress assumption is not entirely valid for this problem.

Comparison of Strain Components:

The 3D model shows that out-of-plane strains (εzz) are non-negligible near the hole, especially at the stress concentration points. This further confirms that a full 3D analysis is beneficial for accurate prediction of the structural response.

Assessment:

Based on the comparison of stress and strain components, a 3D analysis is required for this problem because the stresses in the z-direction (σzz) are not negligible. While a 2D plane stress model can provide reasonable approximations for preliminary design, the 3D analysis reveals important aspects of the stress state that cannot be captured by 2D approximations.

The 3D analysis provides a more complete picture of the stress distribution, particularly at the stress concentration points around the hole, which is crucial for accurate fatigue life prediction and structural integrity assessment of the titanium plate.

Conclusion

The finite element analysis of the titanium plate with a center hole reveals that:

1. Higher-order elements (Q8, T6) significantly outperform lower-order elements in terms of accuracy and efficiency
2. The model exhibits stress singularities that affect convergence
3. A 3D analysis is necessary for complete characterization of the stress state
4. The maximum von Mises stress (approximately 770 MPa) occurs at the sides of the hole