DEPARTMENT OF MECHANICAL ENGINEERING

INDIAN INSTITUTE OF TECHNOLOGY ROPAR

RUPNAGAR-140001, INDIA

MACHINE DESIGN (ME205) LABORATORY REPORT

For

FINITE ELEMENT METHODS (FEM) ASSIGNMENT

Submitted by

YGSV ASHISH (2022MEB1367)

Lab-Group E

Supervised By

Dr. Prabath K Agnihotri

Report Submitted On: 09-03-2024

PROBLEM 1: SIMPLY SUPPORTED BEAM ANALYSIS:

Introduction:

The objective of this analysis is to perform finite element simulations using ABAQUS to analyze a simply supported beam subjected to a distributed load. The beam's properties, including Poisson's ratio, are provided along with the geometry of the beam. The analysis involves computing the deflection of the beam at the center and the maximum bending stress at the top and bottom layers of the beam. Additionally, a mesh convergence study is conducted to determine the computational efficiency of linear and quadratic quadrilateral elements.

Problem Statement:

Consider a simply supported beam as shown in Figure 1. Poisson's ratio of the beam is 0.29, and additional data is provided in the attached file. The force is applied at the free end of the beam.

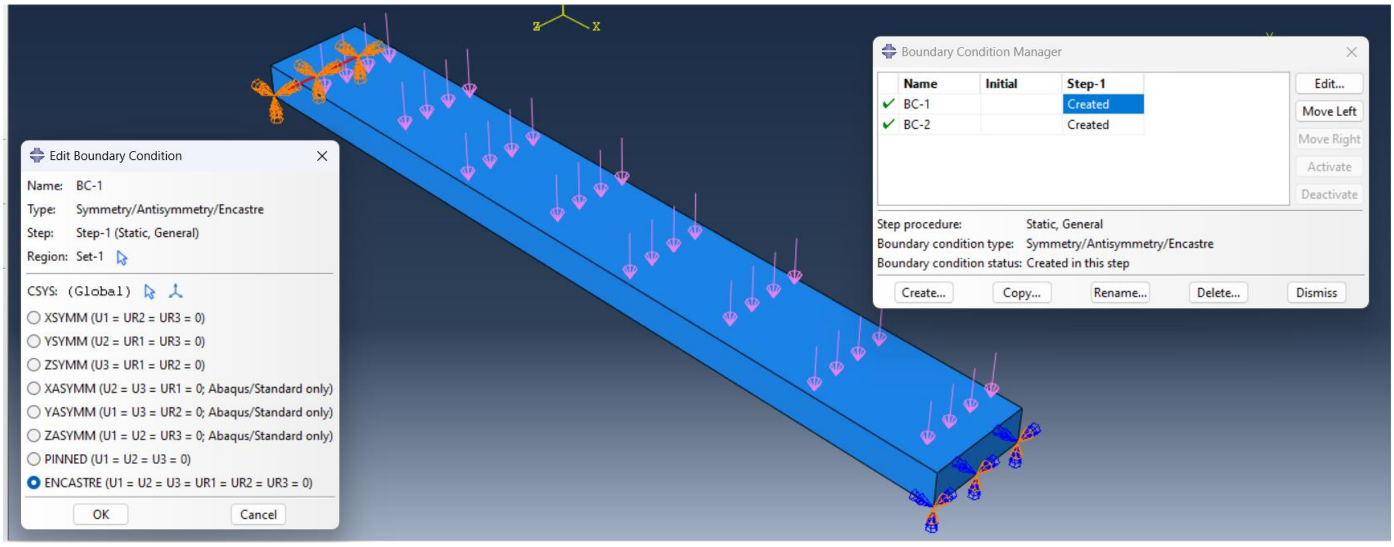
Analysis Procedure:

1. Using 2-dimensional finite element analysis in ABAQUS, compute the deflection of the beam at the center and the maximum bending stress at the top and bottom layers.
2. Compare the FEM results with the beam theory formula, assuming linear elastic behavior and homogeneity.
3. Conduct a mesh convergence study using at least four different mesh sizes for linear and quadratic quadrilateral elements.

4. Plot the center deflection of the beam against mesh sizes to assess computational efficiency.

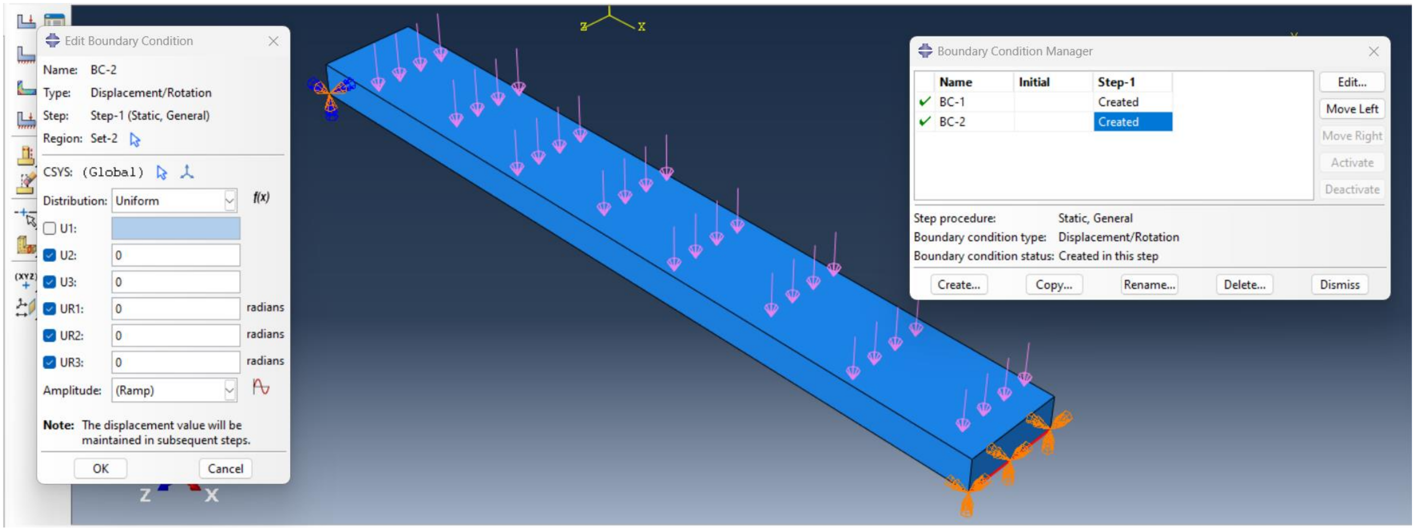
<u>BOUNDARY CONDITIONS FOR Q1</u>

Boundary conditions for the fixed end : (U1=U2=U3=UR1=UR2=UR3=0)



Boundary conditions for the roller end:

(U1!=0)(U1=U2=U3=UR1=UR2=UR3=0)



Calculations:

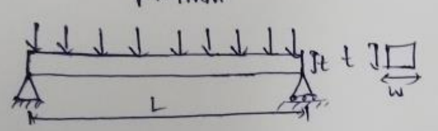
Q1 Length (L) = 94mm

Width (w) = 12mm

Thickness (t) = 4mm

Load (P) = 1293 N/mm^2

E (Young's modulus) = 211×10^9 Pa = 211 GPa



$$W = P \times w = 1293 \times 12 \text{ N/mm} = 15516 \text{ N/mm}$$

$$I = \frac{w \times t^3}{12} = \frac{12 \times (4)^3}{12} = 64 \text{ mm}^4$$

Deflection at center of beam (δ);

$$\delta = \frac{5wL^4}{384EI} = \frac{5 \times 15516 \times 94^4}{384 \times 211 \times 10^9 \times 10^{-6} \times 64} = 1168.0664\text{mm}$$

$$\boxed{\delta = 1168.0664\text{mm}}$$

Maximum bending stress (σ) under w.

$$\sigma = \frac{M \times y}{I} \quad \left(M = \frac{wL^2}{8}\right) \quad y = t/2$$

$$\sigma = \frac{wL^2}{8} \times \frac{t}{2I} = \frac{wL^2 t}{16I} = \frac{15516 \times 94^2 \times 4}{16 \times 64} = 535544.43 \text{ MPa}$$

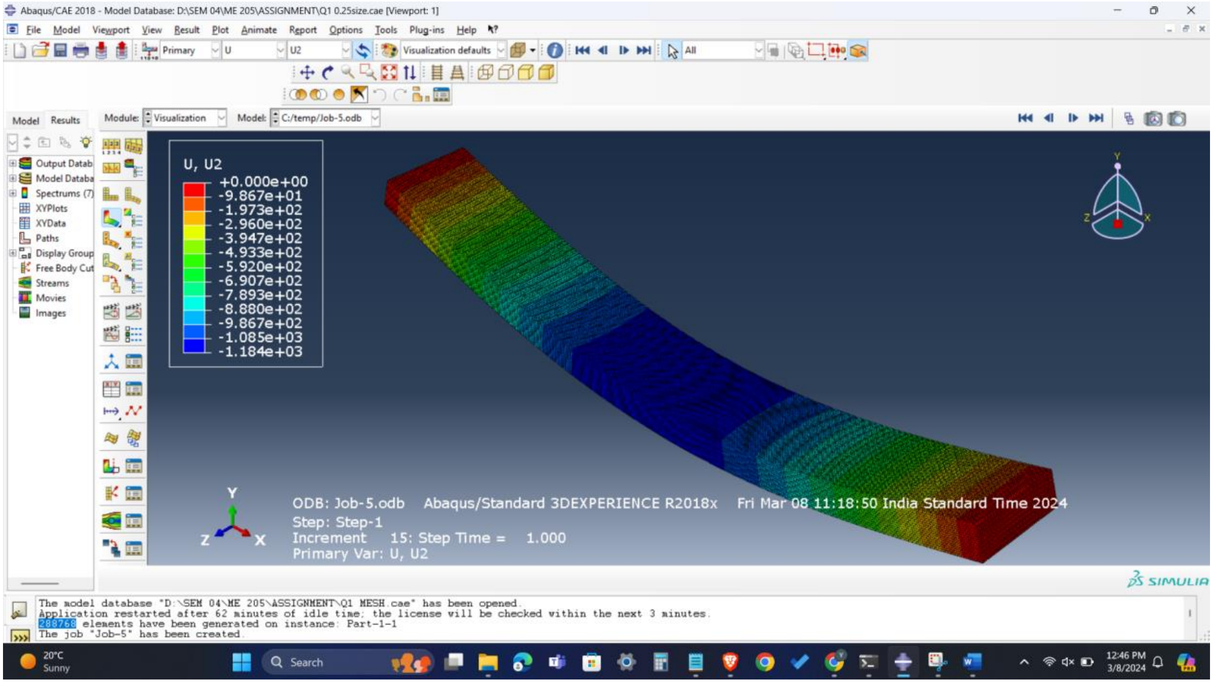
Comparison with theory :

For 288768 cubical shape elements and mesh size of 0.25 ,

Theoretical Deflection at the center of the beam = 1168.0664 mm

Simulation Deflection at the center of the beam = 1184 mm

Error = 1.364 %



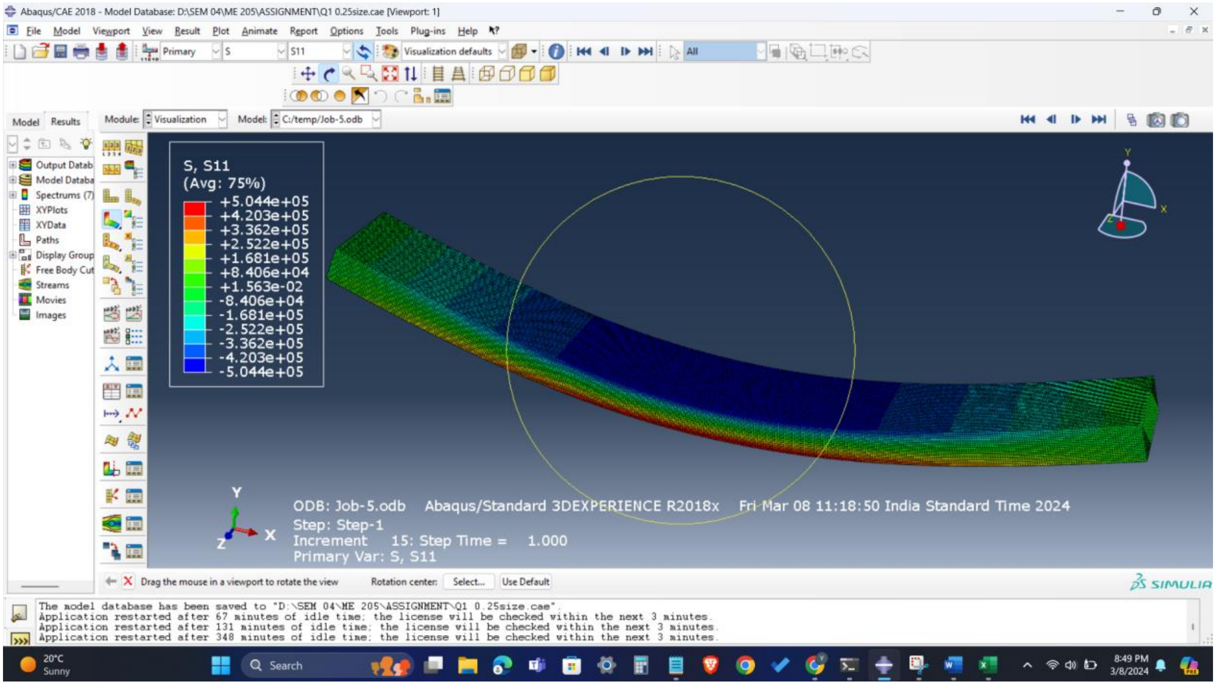
Maximum bending stress on top layer ;

For 288768 cubical shape elements and mesh size of 0.25 ,

Theoretical maximum stress at the center of the beam = 535544.43Mpa

Simulation maximum stress at the center of the beam = 504400Mpa

Error = 5.81%



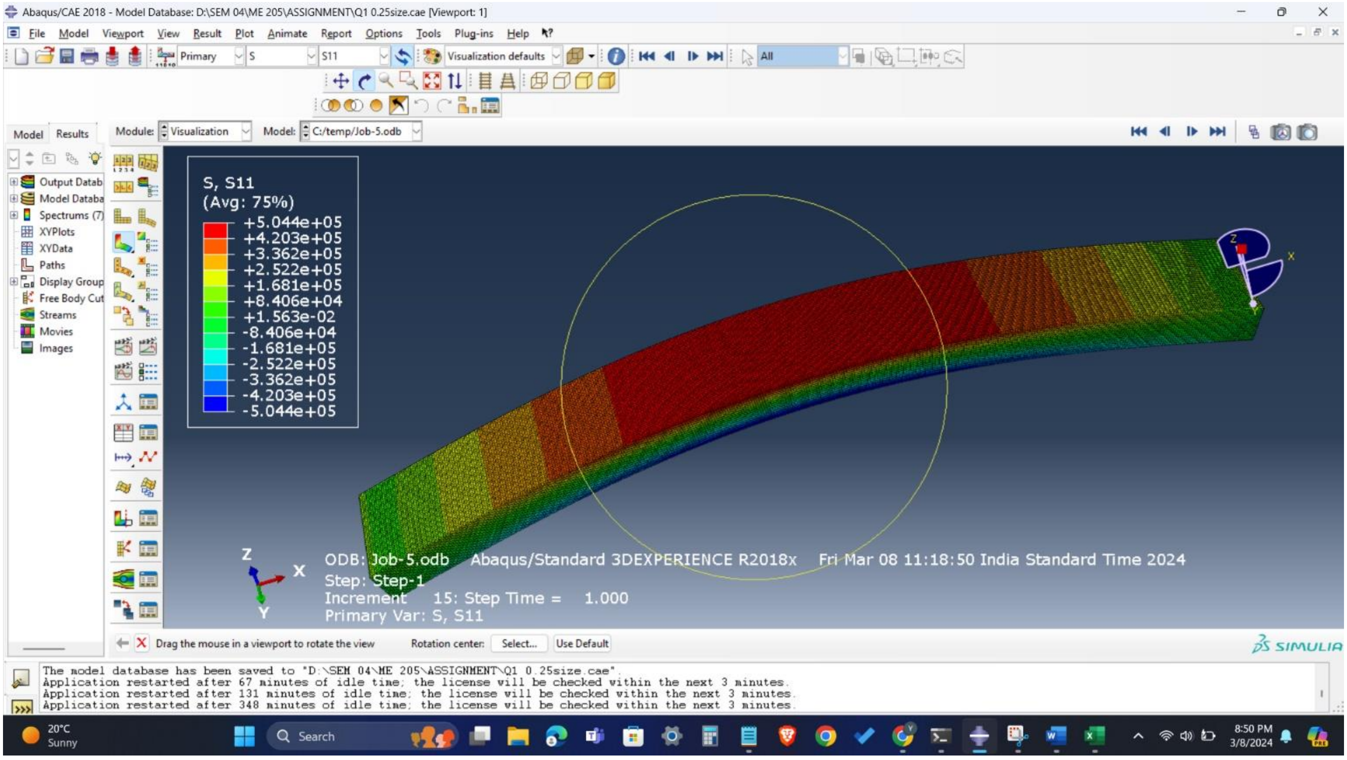
Maximum bending stress on bottom layer;

For 288768 cubical shape elements and mesh size of 0.25,

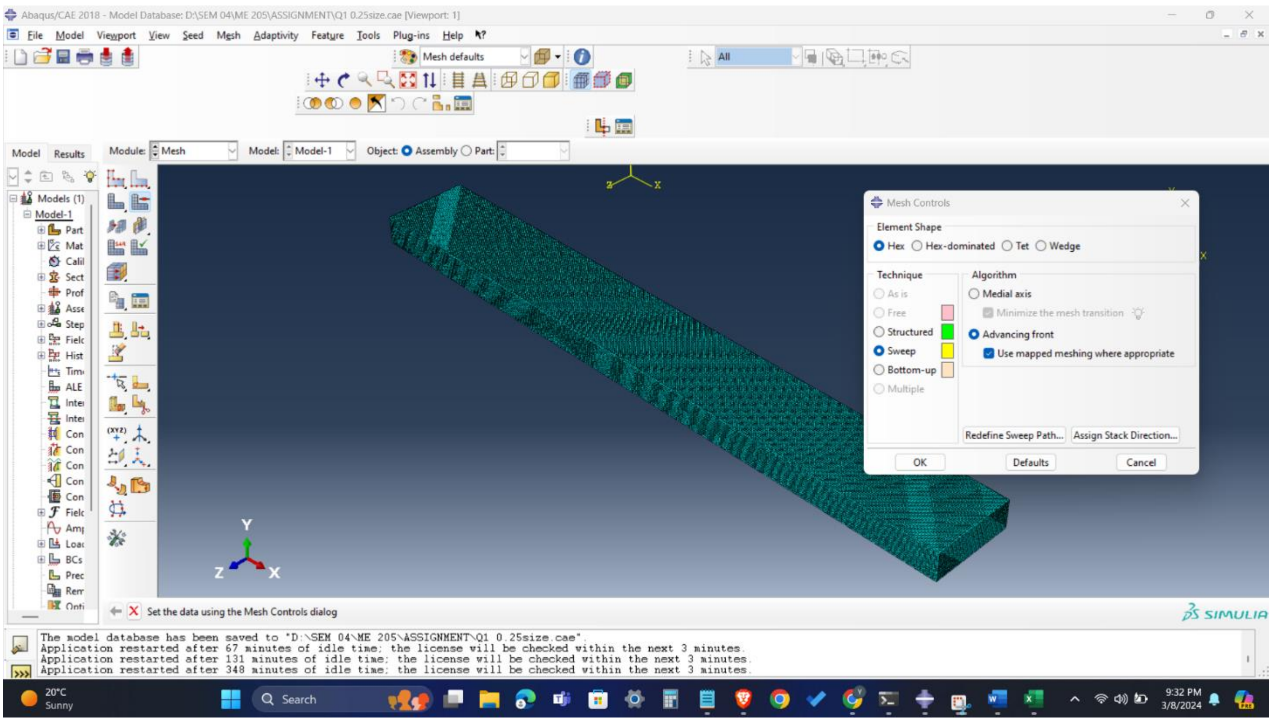
Theoretical maximum stress at the center of the beam = 535544.43Mpa

Simulation maximum stress at the center of the beam = 504400Mpa

Error = 5.81%

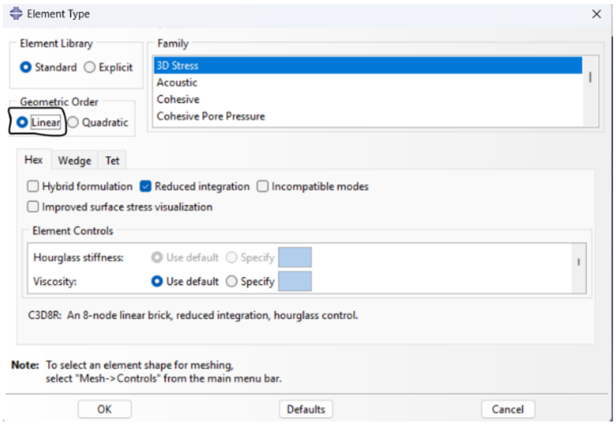


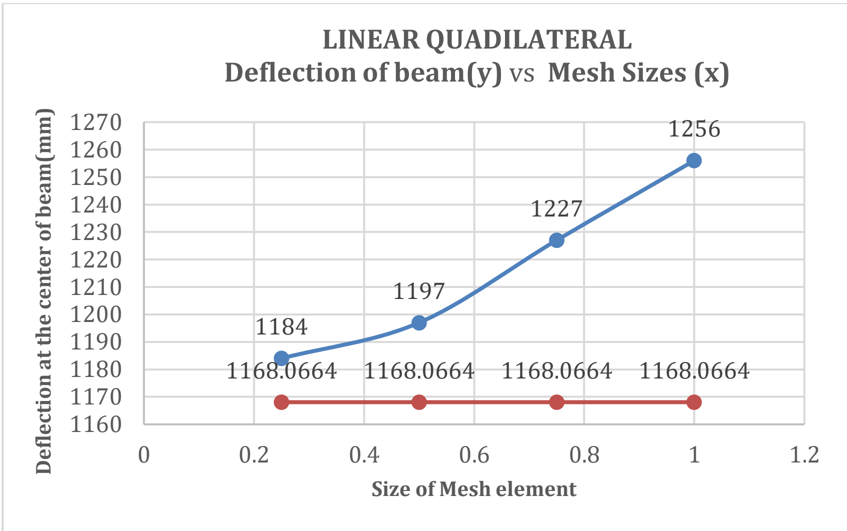
Q1B - ELEMENT SHAPE: HEX



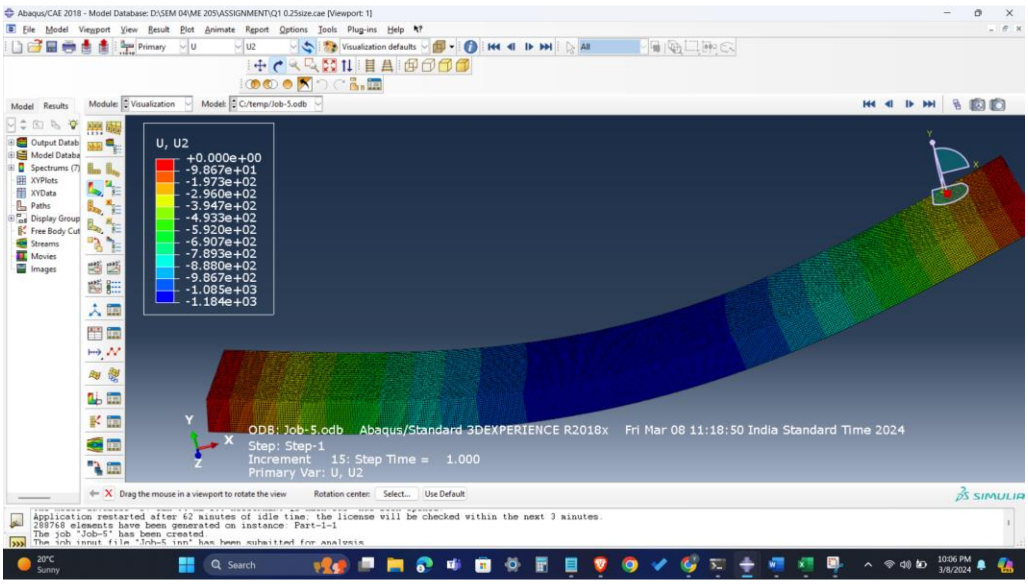
ELEMENT TYPE : C3D8R (Linear Reduced Integration Hexahedral Element):

1. C3D8R is a linear hexahedral element with reduced integration. It has eight nodes, six faces, and twelve edges.
2. The "linear" aspect indicates that this element uses linear shape functions for interpolation, meaning that the displacement within the element varies linearly between nodes.
3. This element type is commonly used for modeling solid structures with relatively simple geometry and moderate deformation behavior.

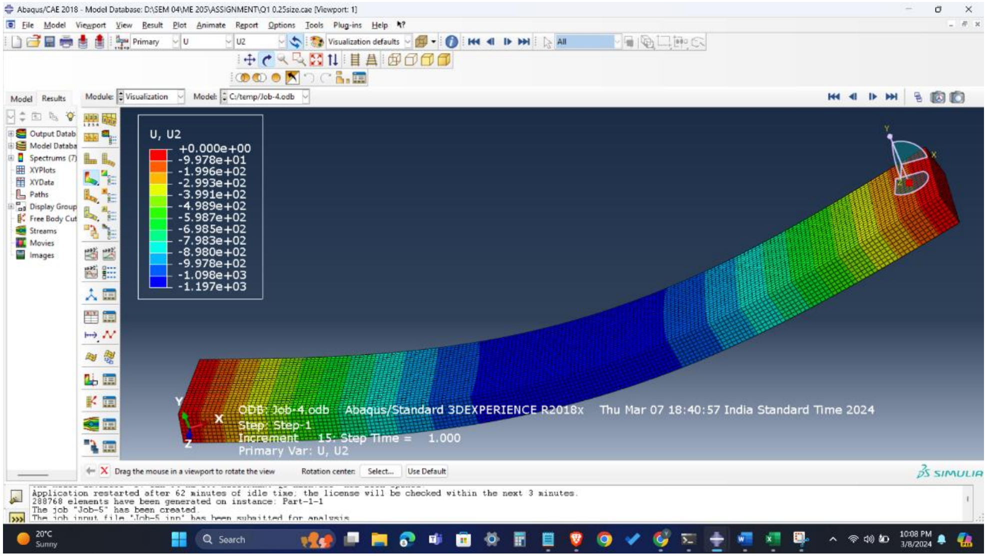




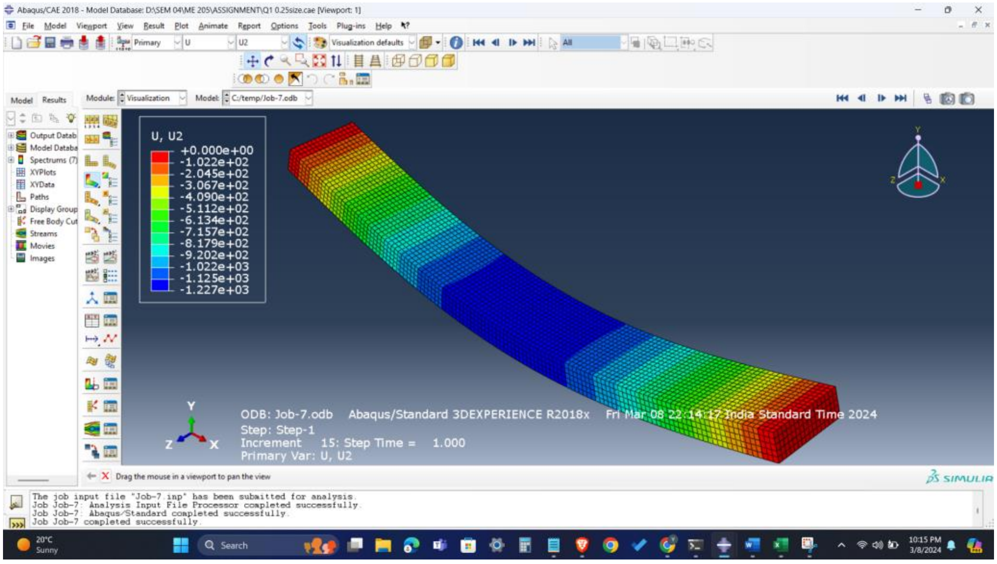
Mesh element size : 0.25 ; Deflection at center : 1184 mm



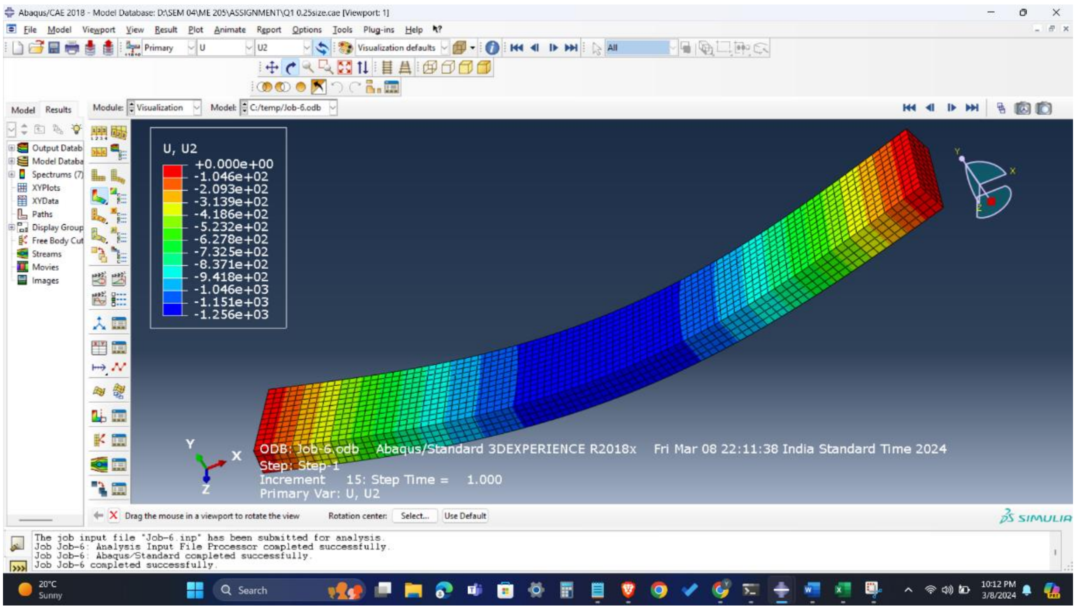
Mesh element size : 0.5 ; Deflection at center : 1197 mm



Mesh element size : 0.75 ; Deflection at center : 1227 mm

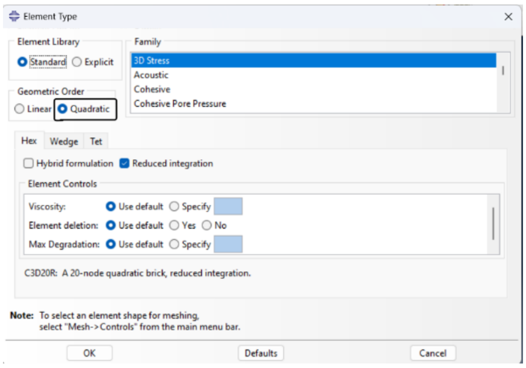


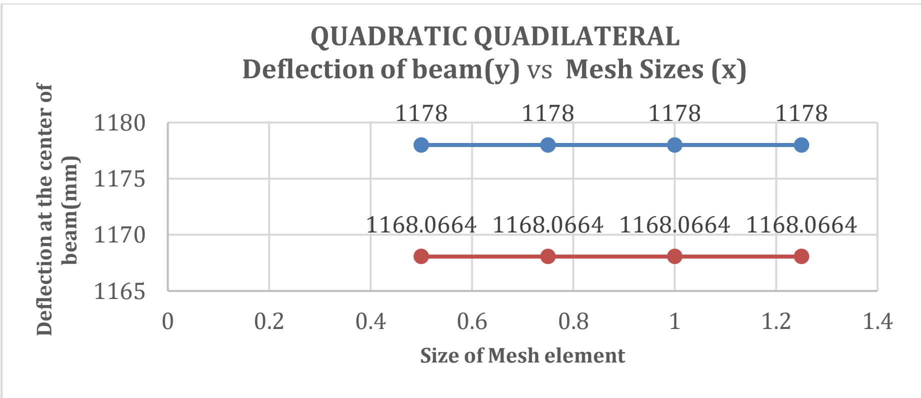
Mesh element size : 1 ; Deflection at center : 1256 mm



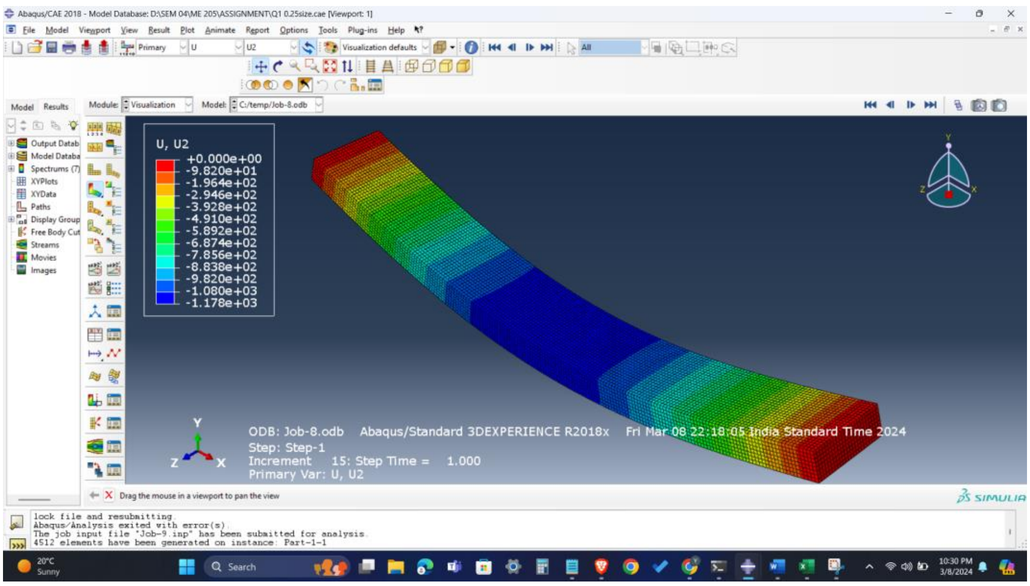
ELEMENT TYPE : C3D20R (Quadratic Reduced Integration Hexahedral Element):

1. C3D20R is a quadratic hexahedral element with reduced integration. It has twenty nodes, six faces, and twelve edges.
2. The "quadratic" aspect indicates that this element uses quadratic shape functions for interpolation, resulting in a higher-order approximation of the displacement field within the element compared to linear elements.
3. **C3D20R** elements are suitable for modeling solid structures with more complex geometries, curved surfaces, and higher deformation gradients. They provide better accuracy in capturing stress concentrations and nonlinear effects compared to linear elements.

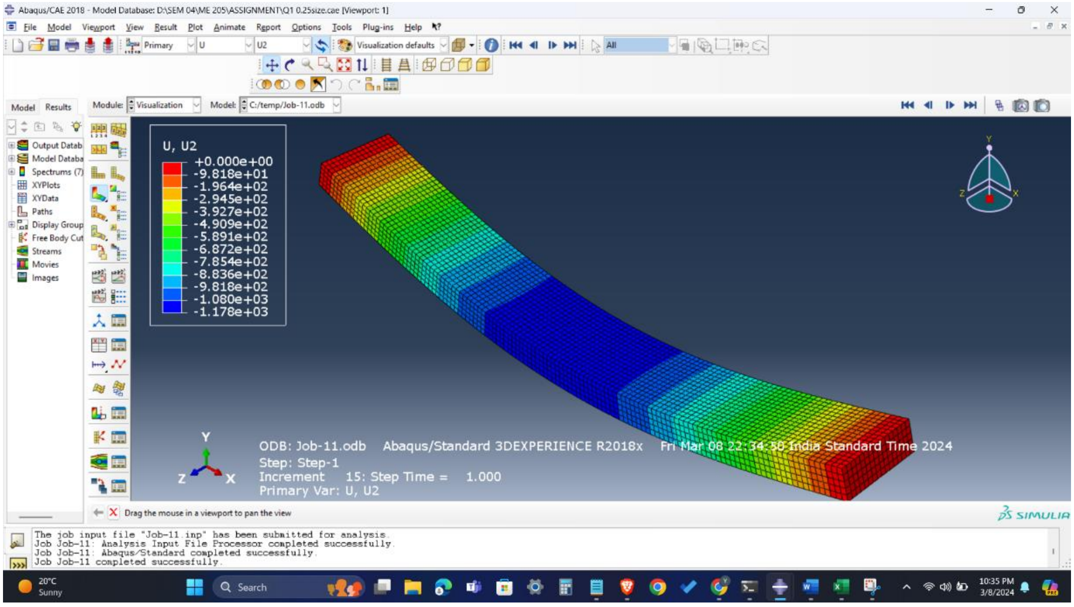




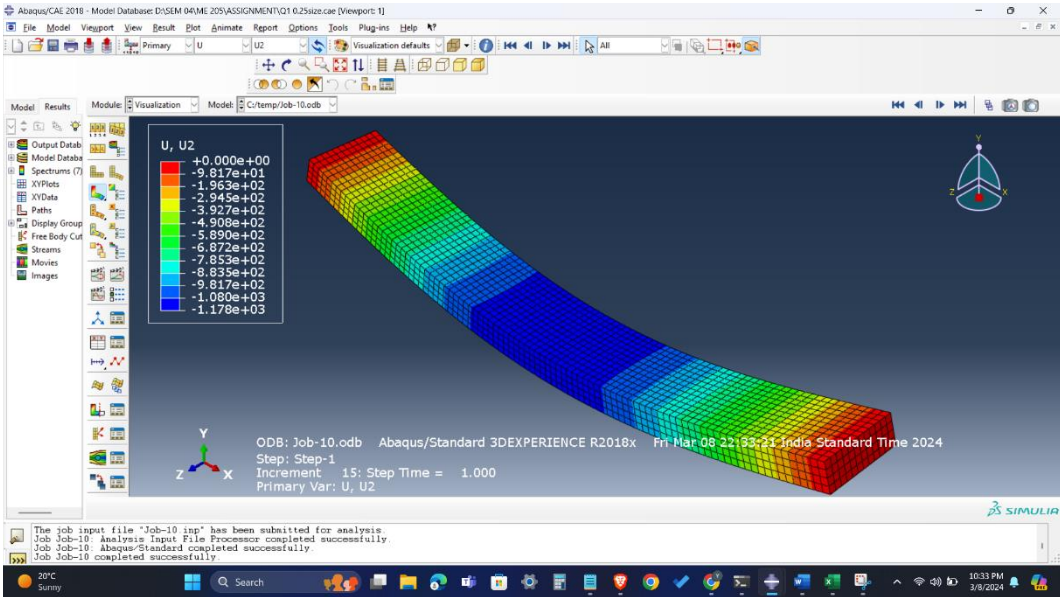
1. Mesh element size : 0.5 ; Deflection at center : 1178 mm (36096 elements)



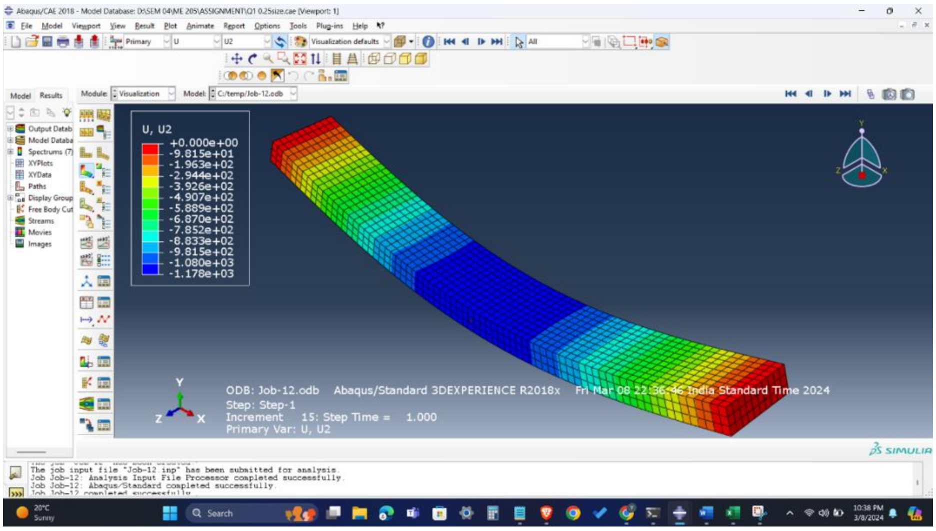
2. Mesh element size : 0.75 ; Deflection at center : 1178 mm (10000 elements)



3.Mesh element size : 1.0 ; Deflection at center : 1178 mm (4512 elements)

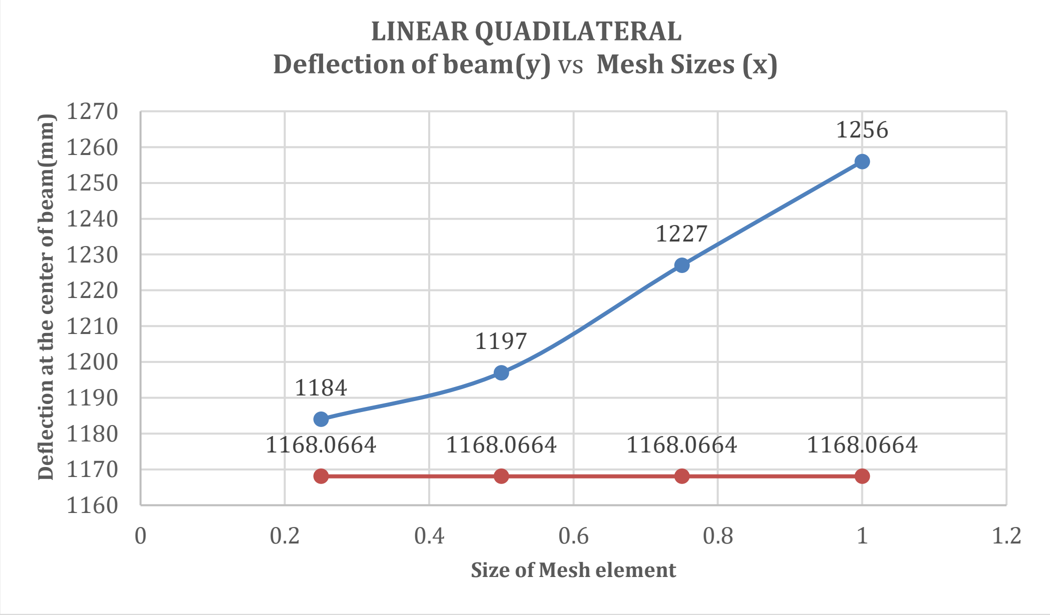


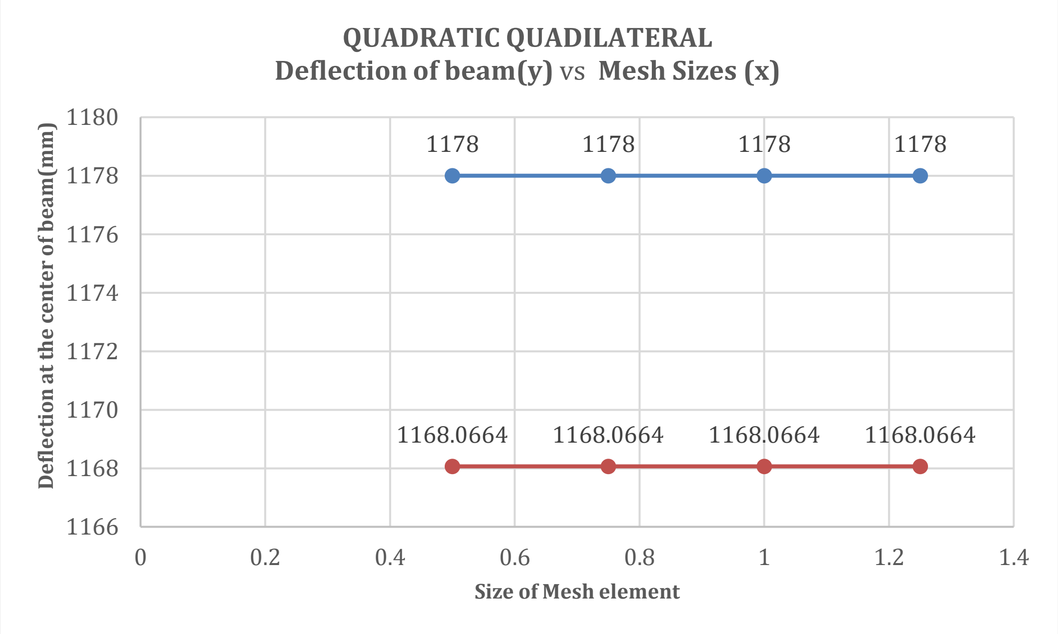
Mesh element size : 1.25 ; Deflection at center : 1178 mm (2250 elements)



OBSERVATIONS :

ELEMENT TYPE (GEOMETRIC ORDER)	Size of Mesh element	Deflection at the center of beam(mm)	%ERROR	THEORITICAL DEFLECTION
LINEAR	0.25	1184	1.364101	1168.066
	0.5	1197	2.477051	1168.066
	0.75	1227	5.045398	1168.066
	1	1256	7.528134	1168.066
QUADRATIC	0.5	1178	0.850431	1168.066
	0.75	1178	0.850431	1168.066
	1	1178	0.850431	1168.066
	1.25	1178	0.850431	1168.066





Results and Discussion:

1. The finite element analysis results are compared with the theoretical beam theory formula to validate the accuracy of the simulation. Additionally, the mesh convergence study is performed to determine the most computationally efficient element type.

2.Comparing results between quadratic and linear quadrilateral elements, the deflection at the center of the beam remained consistent with increased mesh refinement for quadratic quadrilaterals.

3.Conversely, simulation converged to theoretical values as the number of elements increased for linear quadrilaterals.

4.The phenomenon suggests that quadratic quadrilaterals maintain accuracy with coarser meshes, while linear quadrilaterals require finer meshes for convergence, indicating the computational efficiency of quadratic elements over linear counterparts.

5.Quadratic elements can be computationally efficient in certain scenarios due to their ability to accurately capture complex deformations with fewer elements compared to linear elements.

Problem 2: Rectangular Plate with Elliptical Hole Analysis

Introduction:

This analysis aims to conduct finite element analysis using ABAQUS for a rectangular plate containing an elliptical hole. The plate's properties, including Young's modulus and Poisson's ratio, are provided. The analysis involves predicting the maximum stress in the plate and comparing it with the theoretical value.

Problem Statement:

Consider a rectangular plate containing an elliptical hole with specified dimensions and boundary conditions, as shown in Figure 2. The plate has given material properties, including Young's modulus and Poisson's ratio.

Analysis Procedure:

1. Use 2-dimensional finite elements in ABAQUS to analyze the rectangular plate with the given loading conditions and material properties.
2. Predict the maximum stress in the plate using finite element analysis.
3. Compare the FEM predicted maximum stress results with the theoretical value derived from equations.

CALCULATIONS:

Q-2

$$\sigma_{max} = K \sigma_{nom}$$

where,

$$\sigma_{normal} = f \times \frac{h}{h-a}$$

from given data,

~~h = 62m~~ $h = 65mm$

$a = 2mm$ $\quad f = 148 N/mm^2$

$b = 5mm$ $\quad E = 72794 MPa$

$l = 118mm$

$$\sigma_{nom} = 148 \times \frac{65}{65-2} = 152.698 N/mm^2$$

Now,

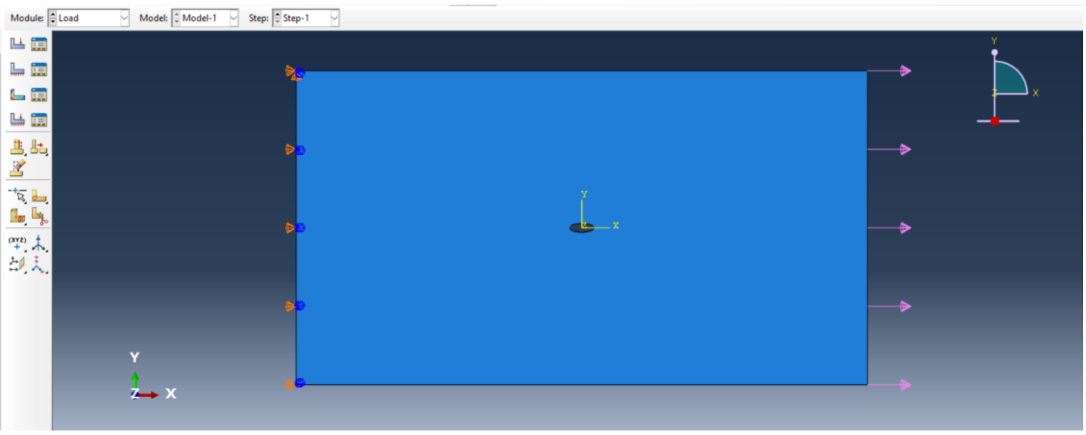
$$K = \left(1 + \frac{2a}{b}\right) = \left(1 + \frac{2(2)}{5}\right) = \left(1 + \frac{4}{5}\right) = \frac{9}{5} = 1.8$$

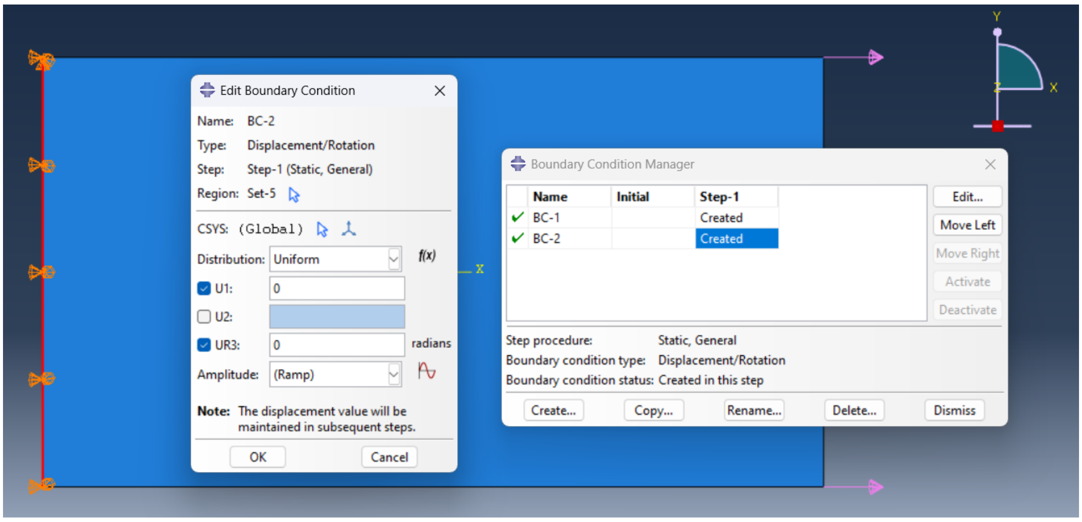
$$\sigma_{max} = \sigma_{nom} \times \left(1 + \frac{2a}{b}\right)$$

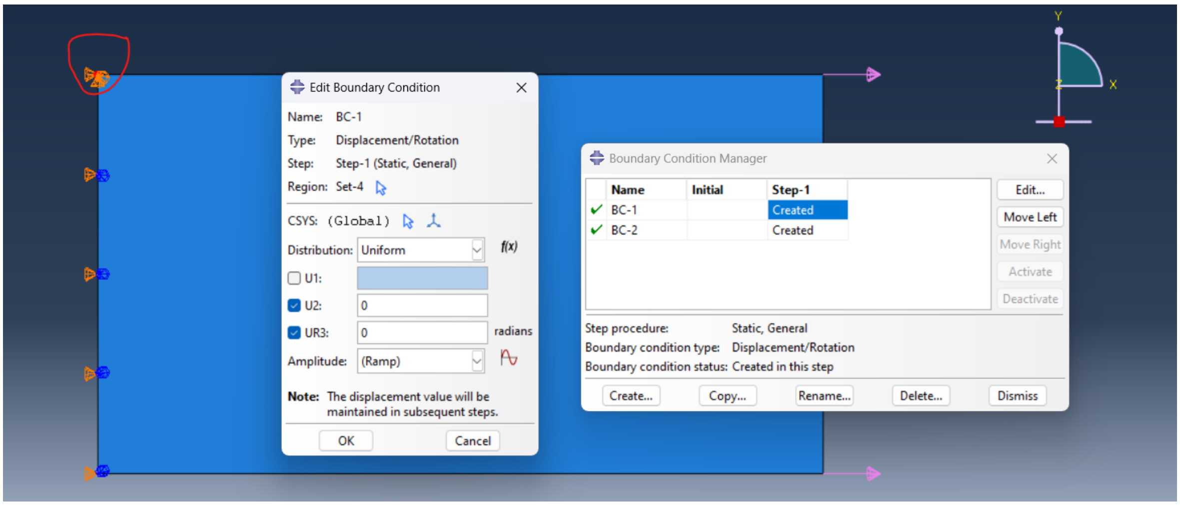
$$= 152.698 \times \left(\frac{9}{5}\right) = 274.856 N/mm^2$$

$$\boxed{\sigma_{max} = 274.856 N/mm^2}$$

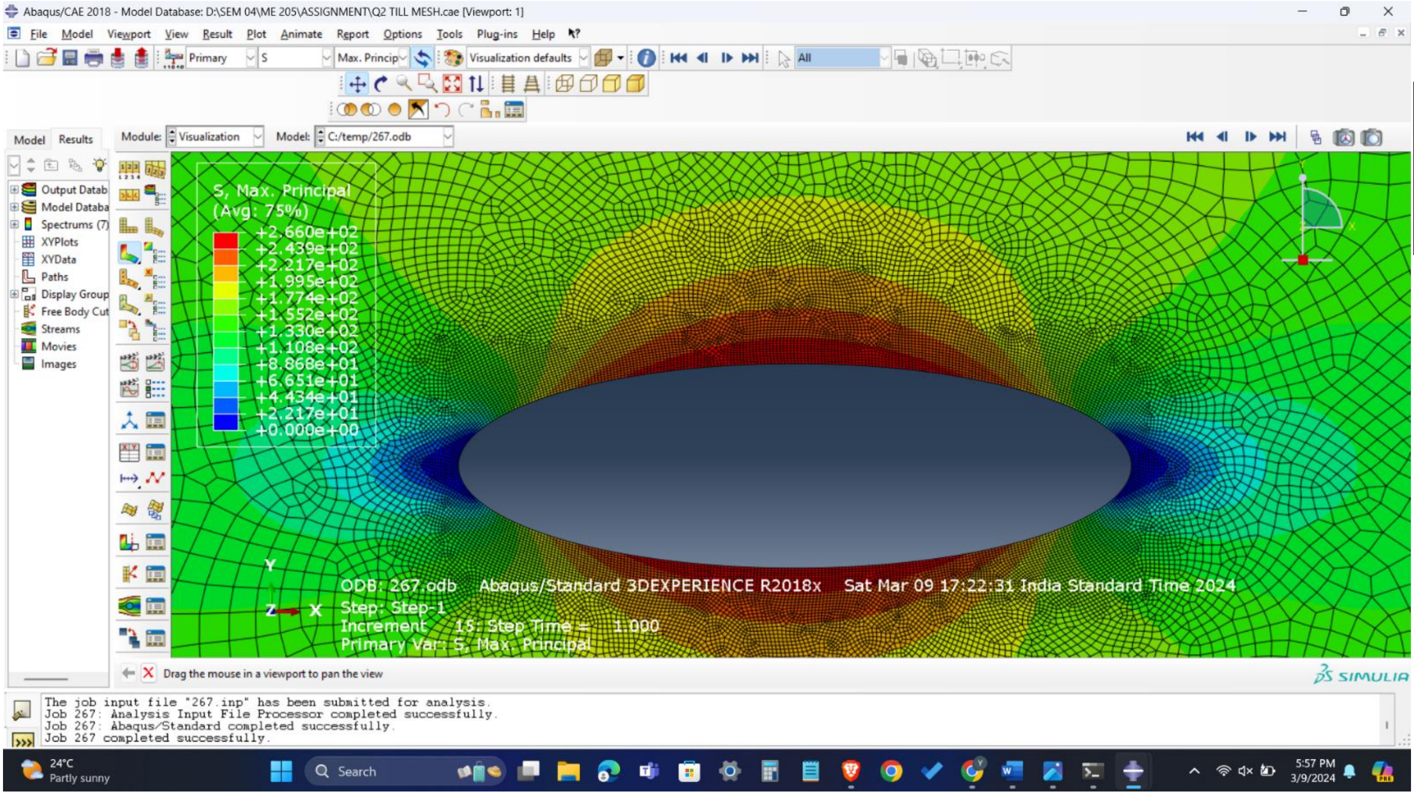
BOUNDARY CONDITIONS FOR Q2 :



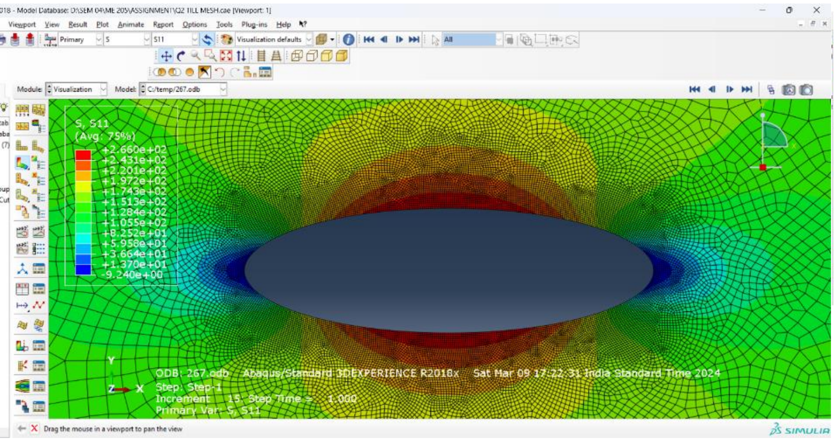


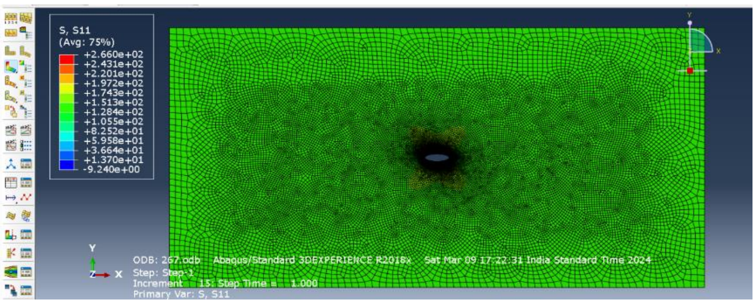


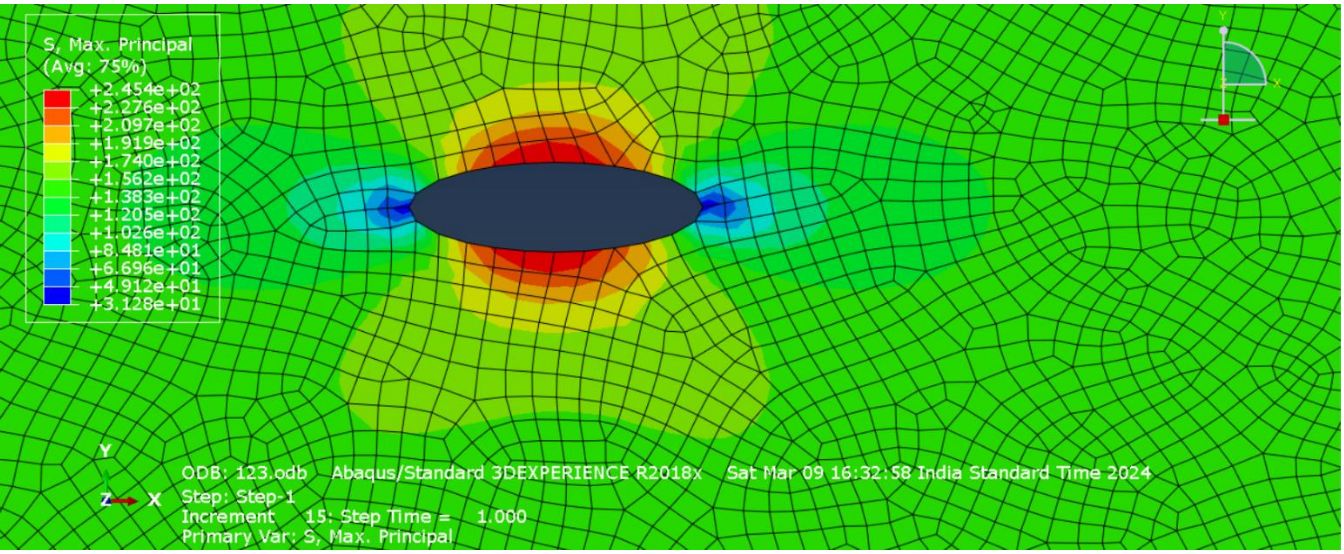
COMPARISION WITH SIMULATION:



Maximum stress = 266MPa







BIGGER ELEMENT SIZE MESH – MORE ERROR

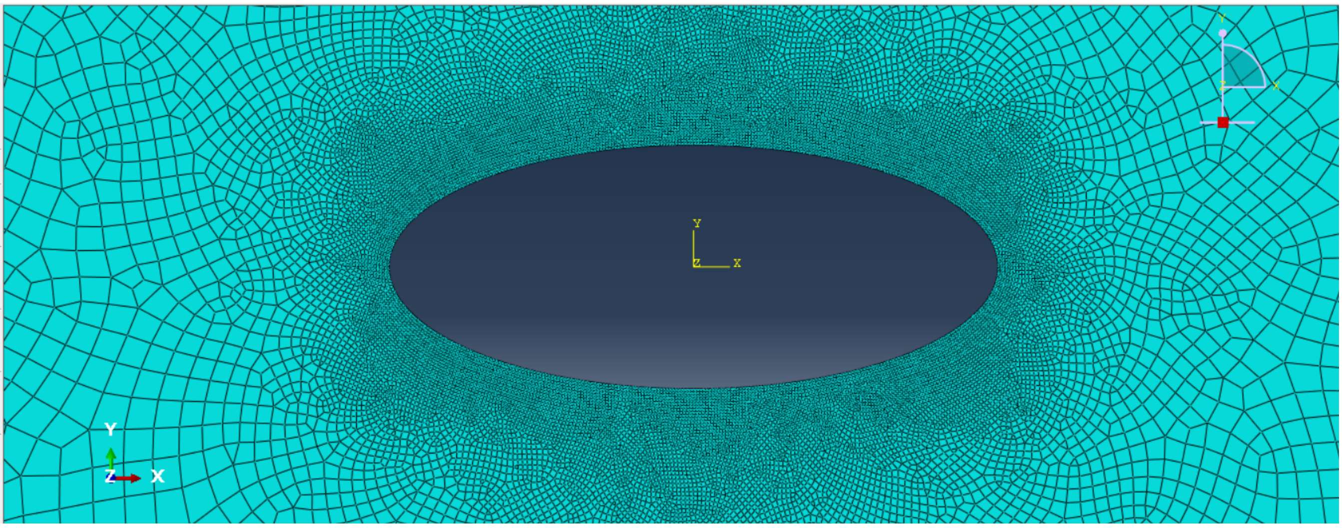
ERROR:

Theoretical maximum stress = 274.856 MPa

Simulation maximum stress = 266 MPa

%Error = | (266-274.856) |*100/274.856= 3.22%

Results and Discussion:



The maximum stress predicted by finite element analysis is compared with the theoretical value and the simulation value is around 3% error to the theoretical value this assess the accuracy of the simulation. Small sizes of meshes brought the simulation values closer to the theoretical value, the mesh near the critical regions is made smaller and mesh size of other less important regions is increased to compensate for the required computational resources.

The assumptions of linear elastic behavior and material homogeneity are considered.