Istanbul Technical University
Graduate School of Science Engineering and Technology

MKC525E
Finite Element Analysis in Engineering

Homework 2

Erdem Çalışkan
503191531
01/07/2020

Question 1

Material Properties:
Elastic modulus $E = 210\,\text{GPa}$,
Poisson's ratio $\nu = 0.29$,

Geometric Properties:
Length $L = 1\,\text{m}$,
Height $h = 5\,\text{mm}$,
Width $w = 20\,\text{mm}$,

Boundary Conditions:
Tip load $F = 5\,\text{N}$,
Cantilever at the other end.

Ansys 2020 R1 is used for solution. A 3D solid model of geometry is constructed using Spaceclaim. For all of the different sizes, only 1 element used throughout the height of the beam for hexahedral elements and only 1 element used throughout the width of the beam for tetrahedral elements. Only linear elements are used in the mesh. Boundary conditions (fixed support and force) are applied from the surfaces of the beam. Boundary conditions can be found in Figure 13.

Results

Results can be seen in Table 1 for hexagonal elements and Table 2 for tetrahedral elements. From the results of the analyses we can see that number of elements in the thickness effects the results very dramatically in the tetrahedral mesh, yet hexahedral mesh is not effected from number of elements in the thickness. To understand this, we have to look at the element formulations of Ansys.

Linear hexahedral element (SOLID185) schematic is given in Figure 1. Even though this element is linear, it has differing integration points and integration points are determined using key options in Ansys. Corresponding key option is KEYOPT(2):

KEYOPT(2)
Element technology:
0 – Full integration with method (default)
1 – Uniform reduced integration with hourglass control
2 – Enhanced strain formulation
3 – Simplified enhanced strain formulation
From output file of the analysis, we can see that KEYOPT(2)=3 is selected for this model.

Listing 1: Solver output

```
 *** SELECTION OF ELEMENT TECHNOLOGIES FOR APPLICABLE ELEMENTS ***
      --- GIVE SUGGESTIONS AND RESET THE KEY OPTIONS ---

ELEMENT TYPE      1 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
  KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0
```

This means that simplified enhanced strain formulation is selected for our analysis. Which prevents shear locking in bending-dominated problems. It adds 9 additional internal DOFs to the element. This is why we don't see the effect of shear locking in different mesh sizes.

For tetrahedral elements, even though element is still SOLID185 (same as hexahedral), three nodes of the hexahedral element (N, O, P) is reduced to one node (M), see Figure 1. Because of this, simplified enhanced strain formulation can not prevent shear locking. Results are given in 2. Maximum deformation is similar to hexahedral solution in refinements 3 and 4, yet there is still some difference. So using of tetrahedral linear elements is not recommended.

Figures for boundary conditions, mesh, deformation and stress is given in Appendix 1.

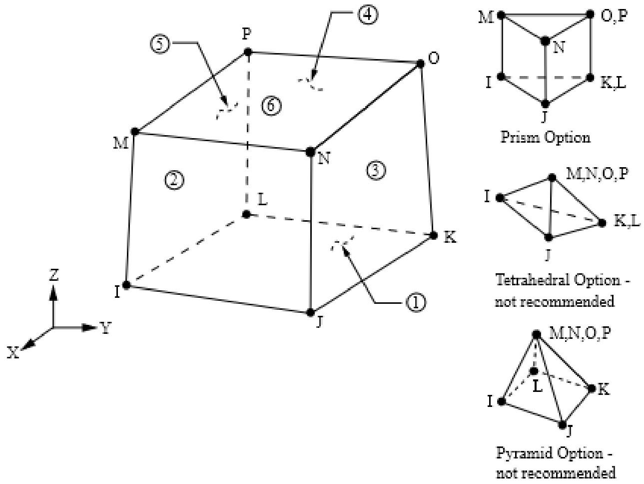


Figure 1: SOLID185 3-D 8-Node Structural Solid element.

Mesh size	Maximum Deflection [mm]	Maximum von-Mises Stress [MPa]
$1x6$	36.75	51.31
$2x12$	37.14	55.56
$4x12$	37.08	56.34

Table 1: Maximum deflection and von-Mises stress for hexahadral elements.

Mesh size (coarsest to finest)	Maximum Deflection [mm]	Maximum von-Mises Stress [MPa]
1	7.57	12.2
2	23.06	23.8
3	35.81	35.8
4	33.47	33.47

Table 2: Maximum deflection and von-Mises stress for tetrahedral elements.

Question 2

A plate fixed at the outer corner having outer dimension of 1 m^2 and inner hole of 0.5 m^2, thickness of 5 mm, elastic modulus of $E = 210$ GPa, Poisson's ratio $\nu = 0.3$ and fillet radius of 10 mm is applied an inner pressure of 50 MPa.
Number of elements at the right bottom corner of the hole is changed by using different element sizes and curve for maximum von-Mises stress versus total number of elements is obtained. Also, more data points (mesh sizes) are used to obtain a smooth curve.

Results

Ansys 2020 R1 is used for analyses. 2-D 4-node structural solid linear shell elements (PLANE182) are used in the analyses with plane stress formulation (see Figure 2). Element size at the corner is defined as parameter and maximum von-Mises stress at the corner defined as output. See Figure 4 for the graph. Also

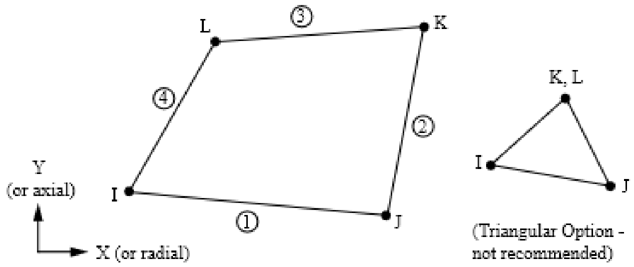


Figure 2: PLANE182 2-D 4-node structural solid element.

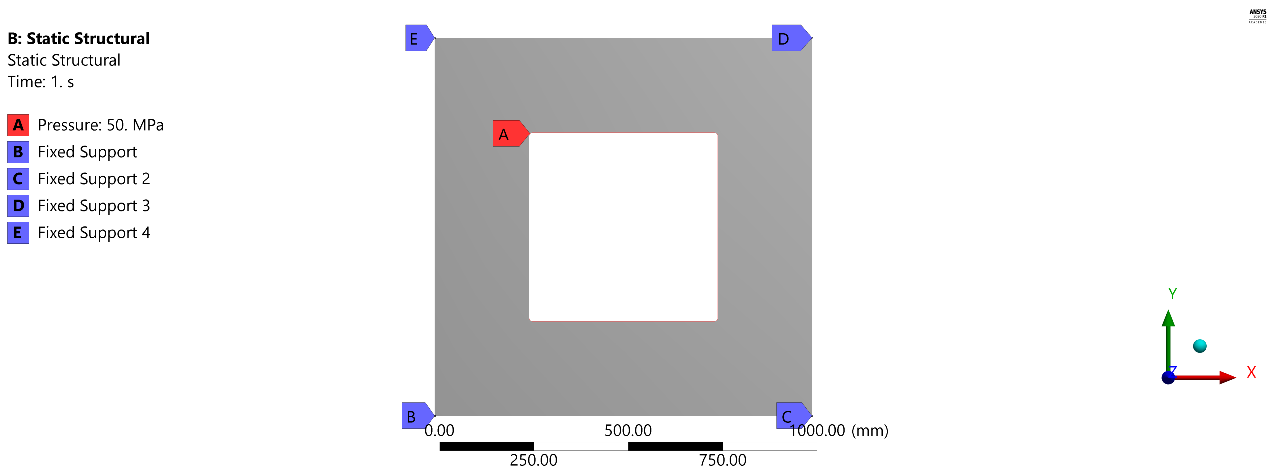


Figure 3: Boundary conditions.

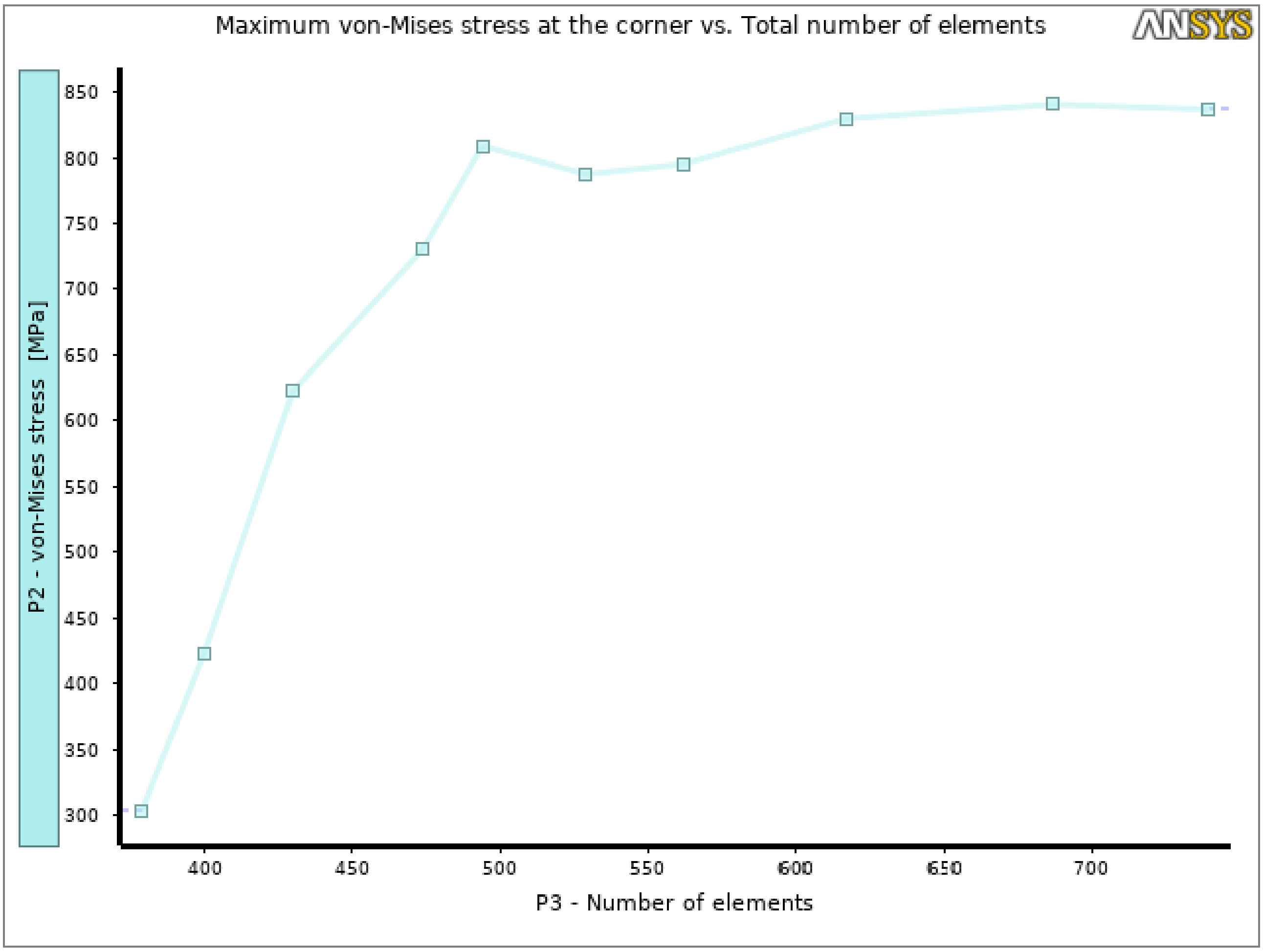
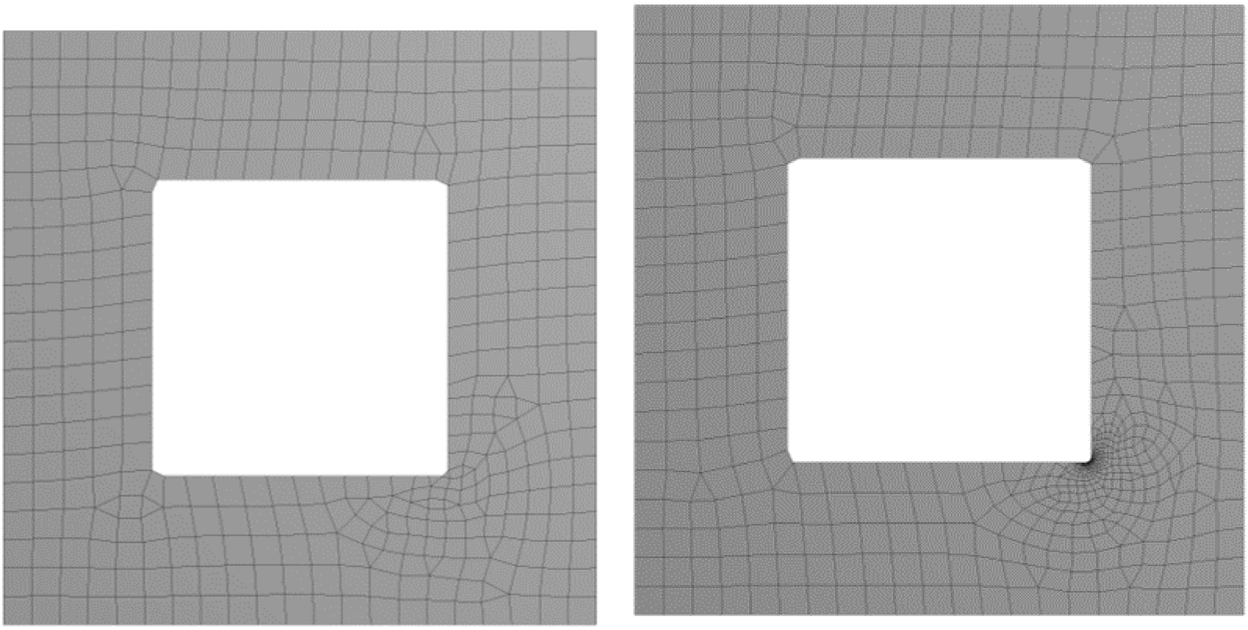
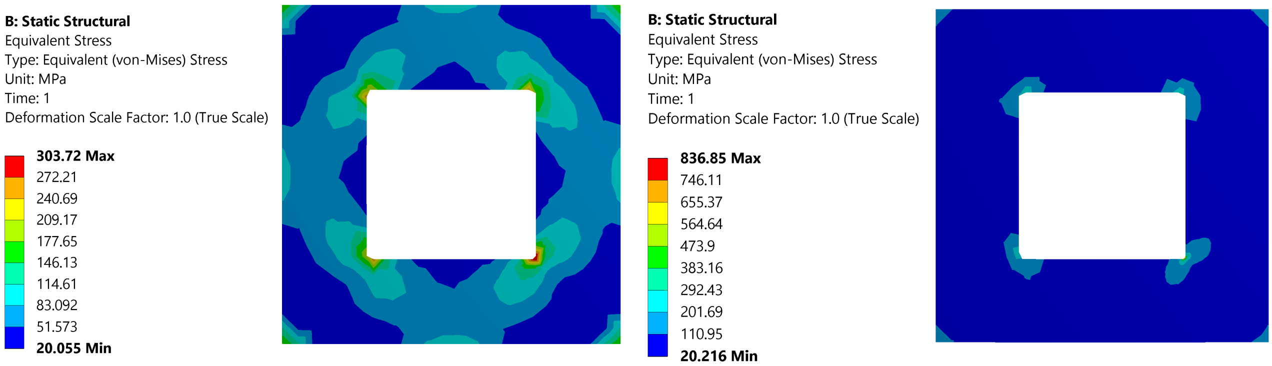


Figure 4: Maximum von-Mises stress at the corner versus total number of elements.



(a) Coarse mesh (b) Fine Mesh

Figure 5: Two different mesh sizes.



(a) Coarse mesh

(b) Fine Mesh

Figure 6: von-Mises stress.

Question 3

A plate having dimension of 1 mx2 m, thickness of 2 mm, central hole with radius 25 cm, elastic modulus of $E = 210\,\text{GPa}$ and Poisson's ratio $\nu = 0.3$ is applied distributed total in plane tensile load of 1000 N.
In this question, linear triangular, linear quadrilateral, quadratic triangular and quadratic quadrilateral elements are used with 150, 100, 75, 50, 25, 10, 5 mm mesh sizes.

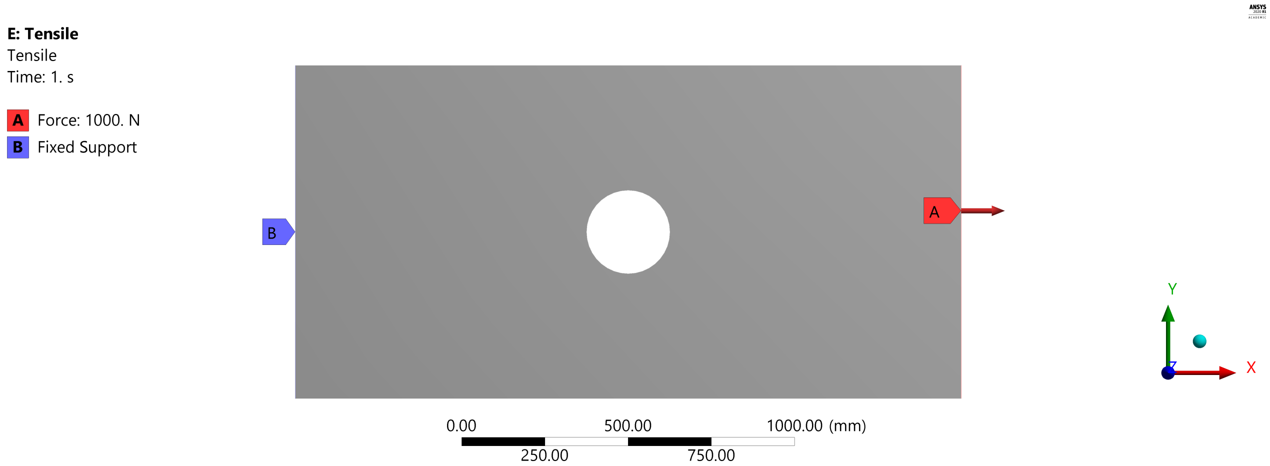


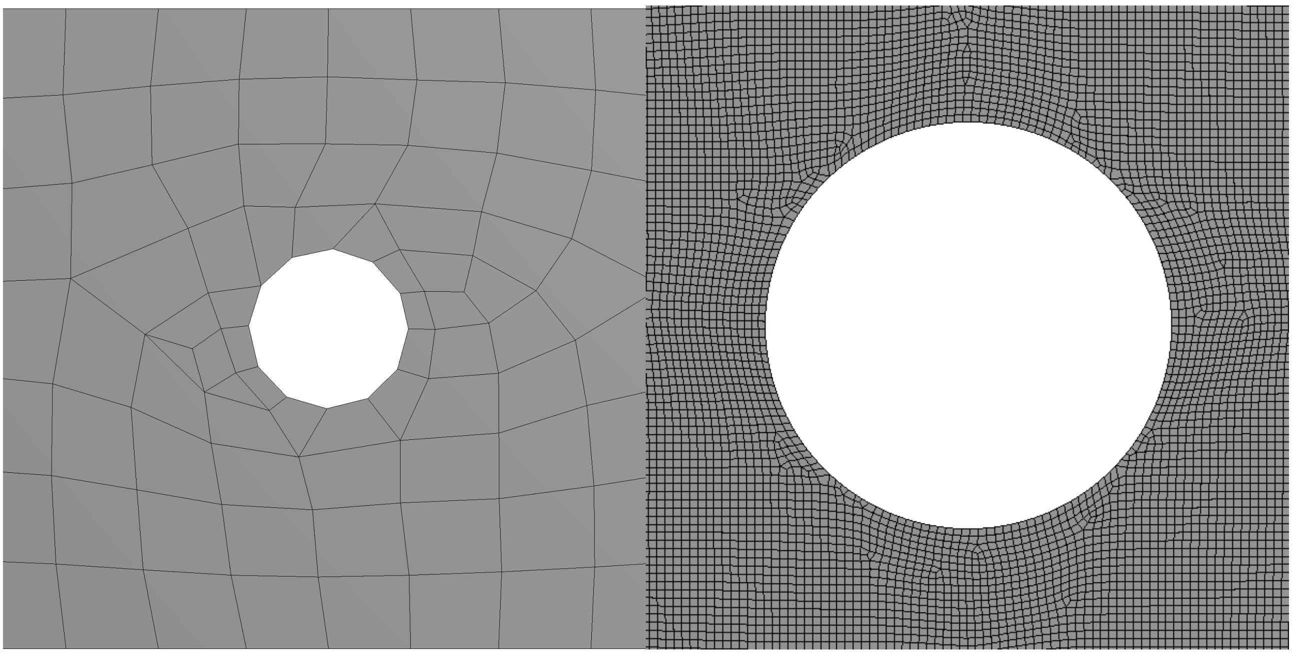
Figure 7: Boundary conditions for tensile loading.

Results

Result for 10 natural frequencies are given in Figure 9. As we can see, first 2 modes of the no boundary condition is 0 because they are rigid motion modes. After that they are similar to the tensile modes, yet there is some difference. However there is no difference between 1000 N and 0.01 N tensile loading, so loading magnitude has no effect on frequencies but supports have some effect.

Figures for comparing the effect of element type and size are given as, Figure 10 for maximum displacement, Figure 11 for maximum von-Mises stress, Figure 12 for 5th natural frequency (chosen arbitrarily). It can be seem that linear elements are struggling to converge when quadratic elements are converging quickly. The fastest one to converge is quadratic quadrilateral and the slowest one is linear triangular. Element size should be at least 25 mm for 2nd order quadrilateral elements; 10 mm for 1st order quadrilateral and 2nd order triangular. First order triangular elements should be avoided.

Displacement plots for first 5 modes of linear quadrilateral are given in Appendix 2.



(a) Coarse mesh (b) Fine Mesh

Figure 8: Two different mesh sizes (fine mesh is zoomed).

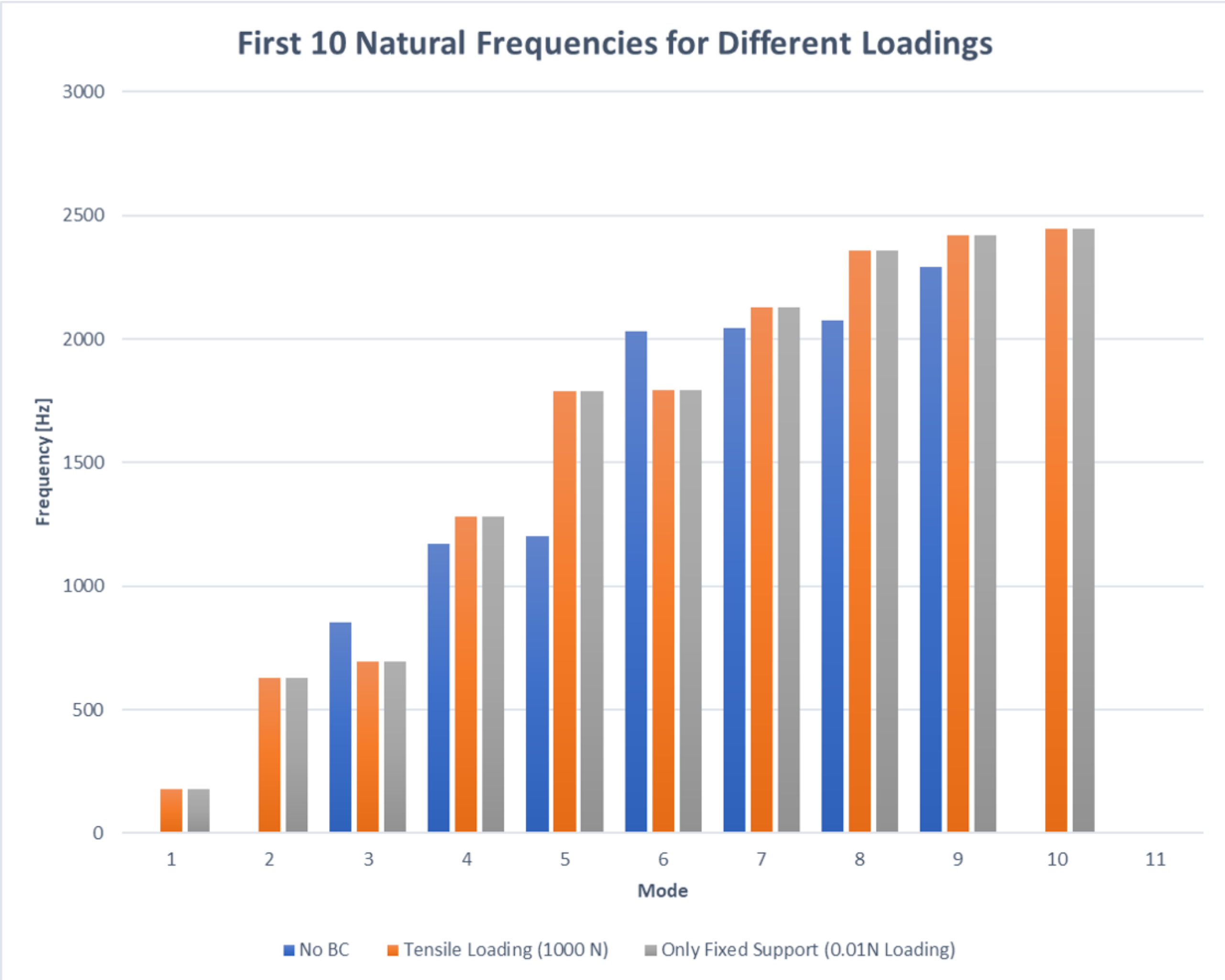


Figure 9: First 10 natural frequencies for different loading conditions (quadratic quadrilateral 5 mm element size).

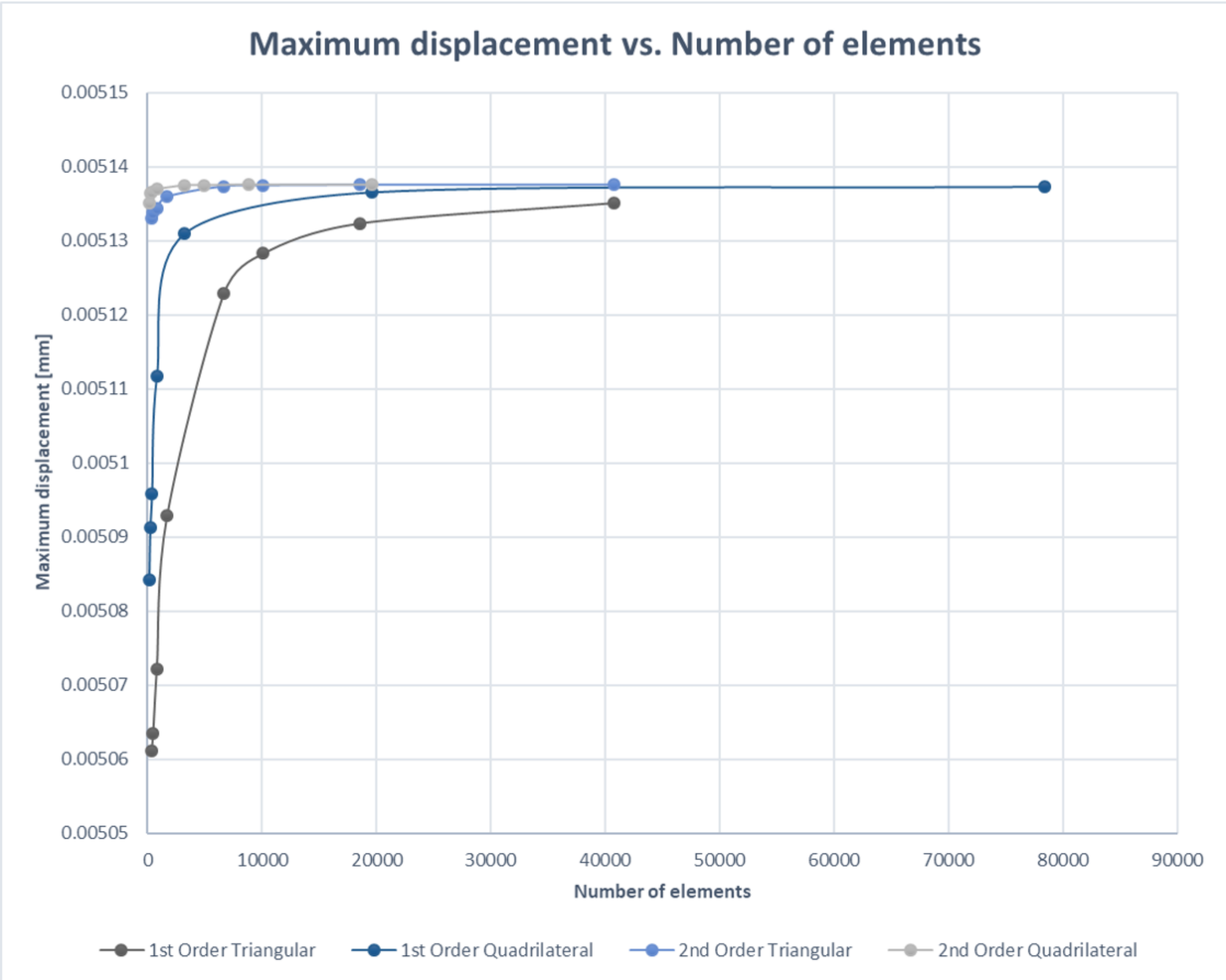


Figure 10: Maximum displacement for different mesh types and sizes.

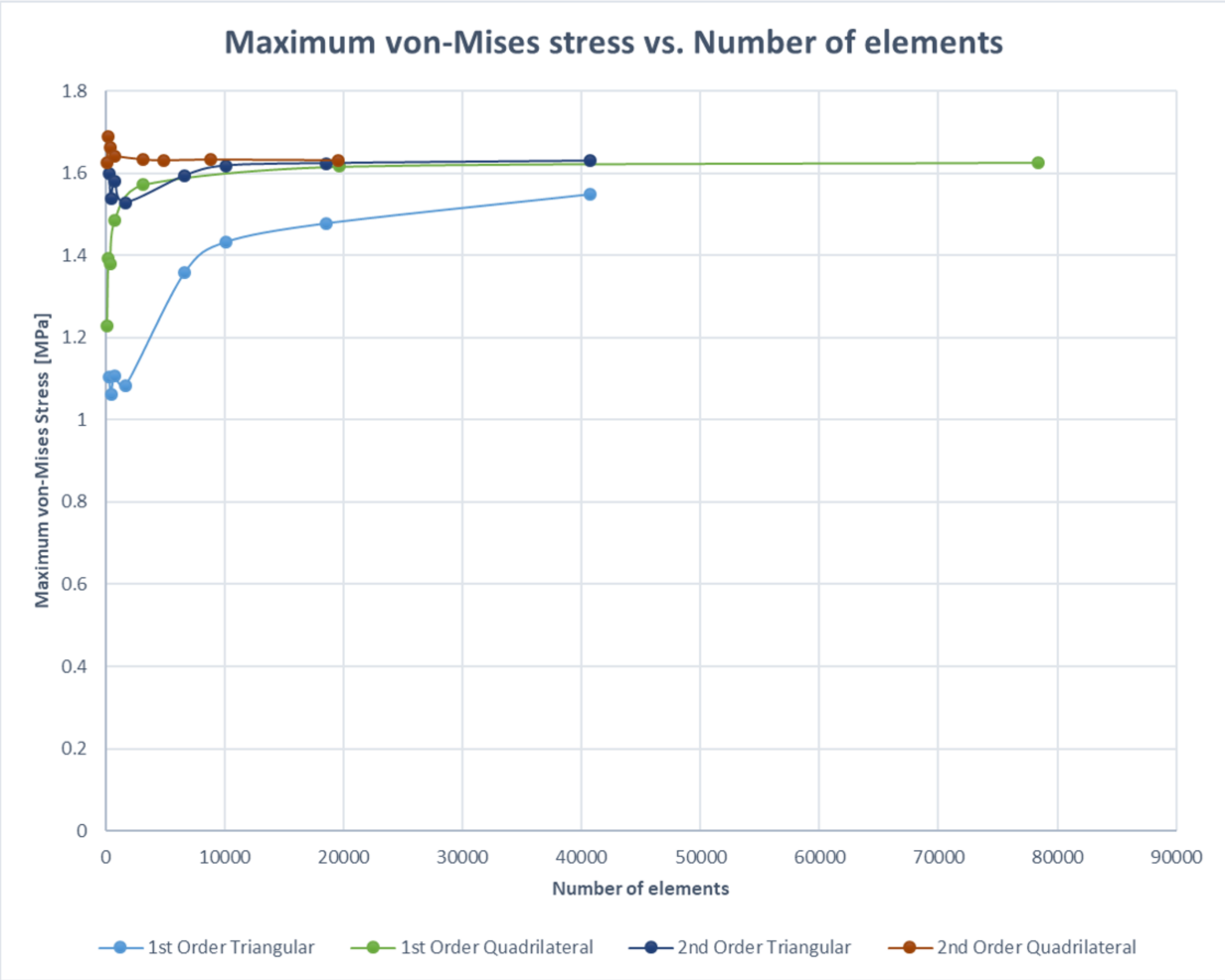


Figure 11: Maximum von-Mises stress for different mesh types and sizes.

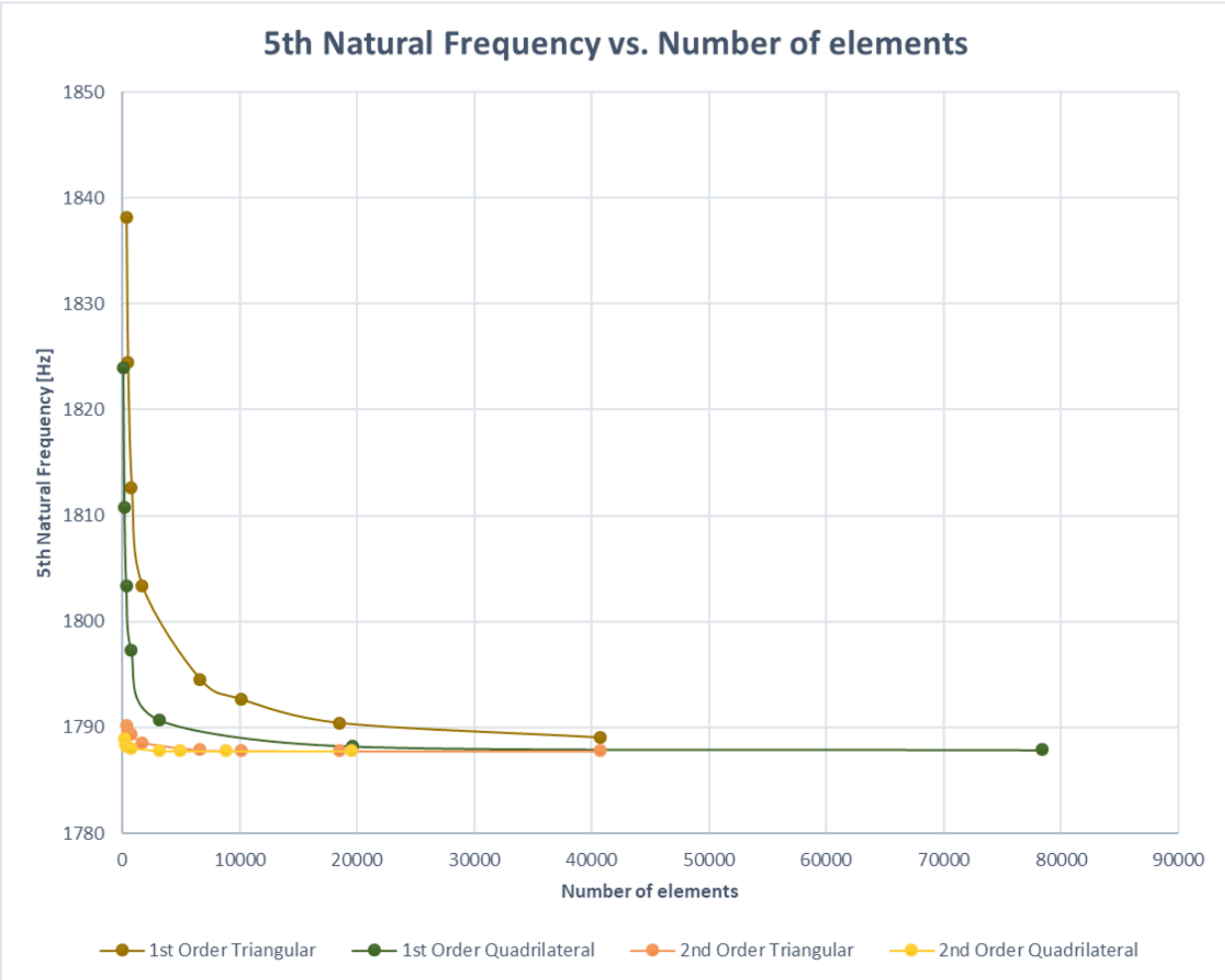


Figure 12: 5th mode natural frequencies for different mesh types and sizes.

1 Appendix for Question 1

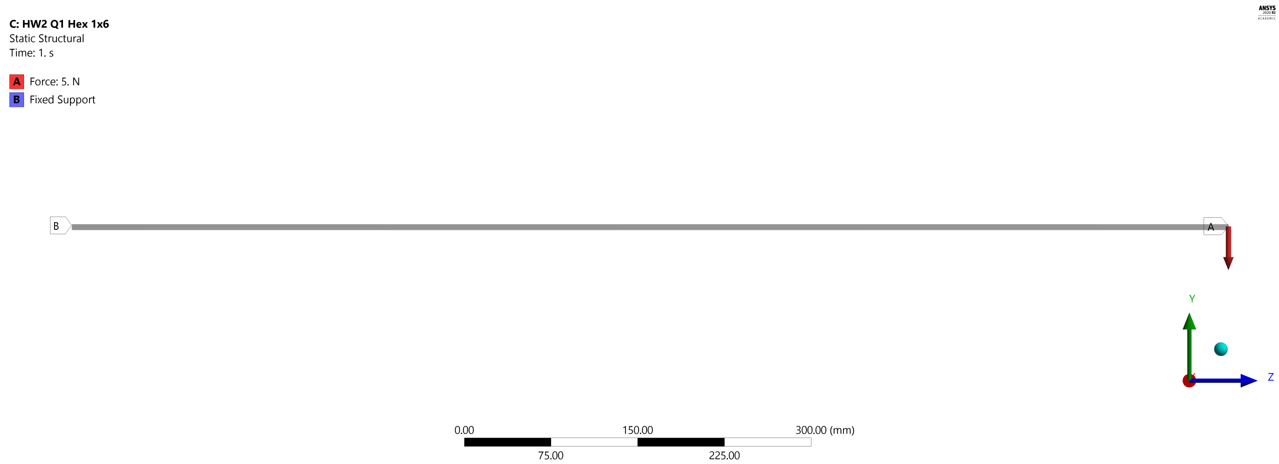


Figure 13: Boundary conditions.

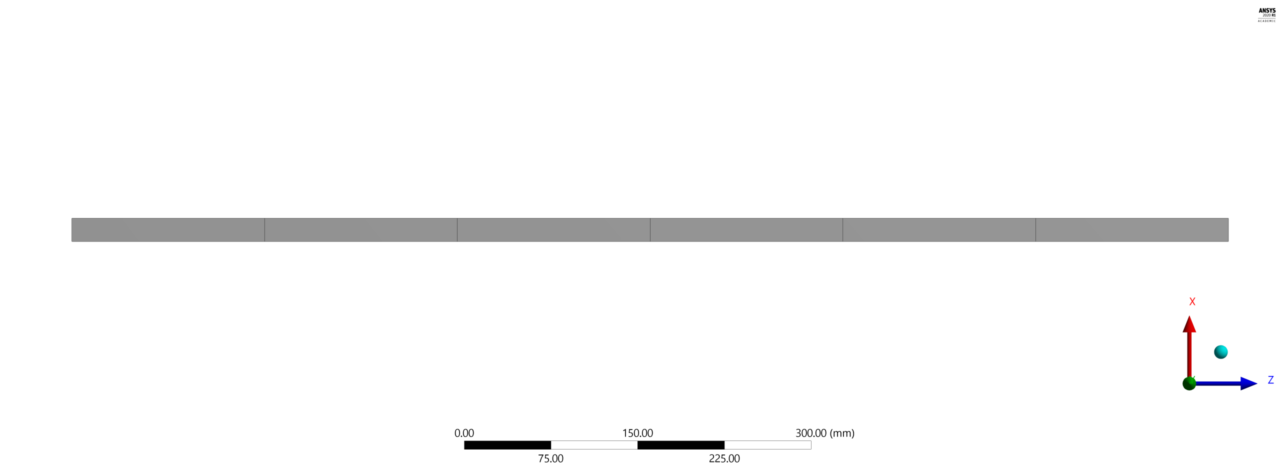


Figure 14: Mesh for 1x6 elements (top view, 1 element through height).

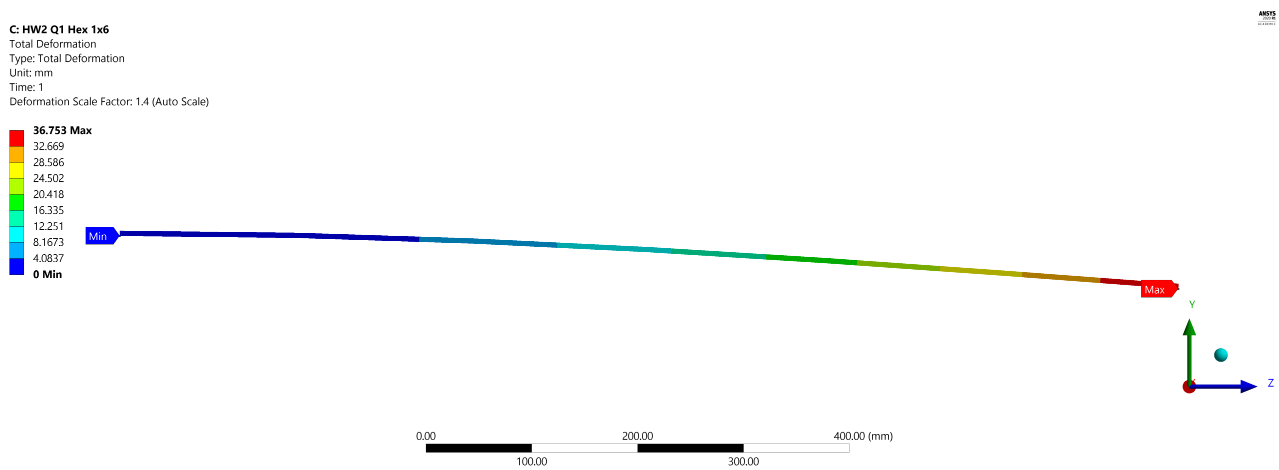


Figure 15: Deformation for 1x6 elements.

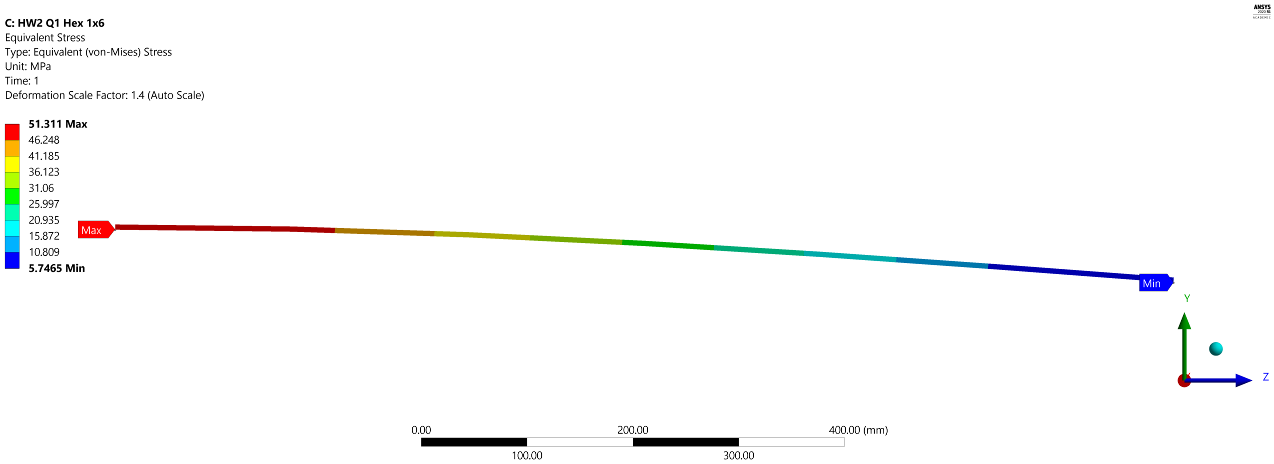


Figure 16: von-Mises stress for 1x6 elements.

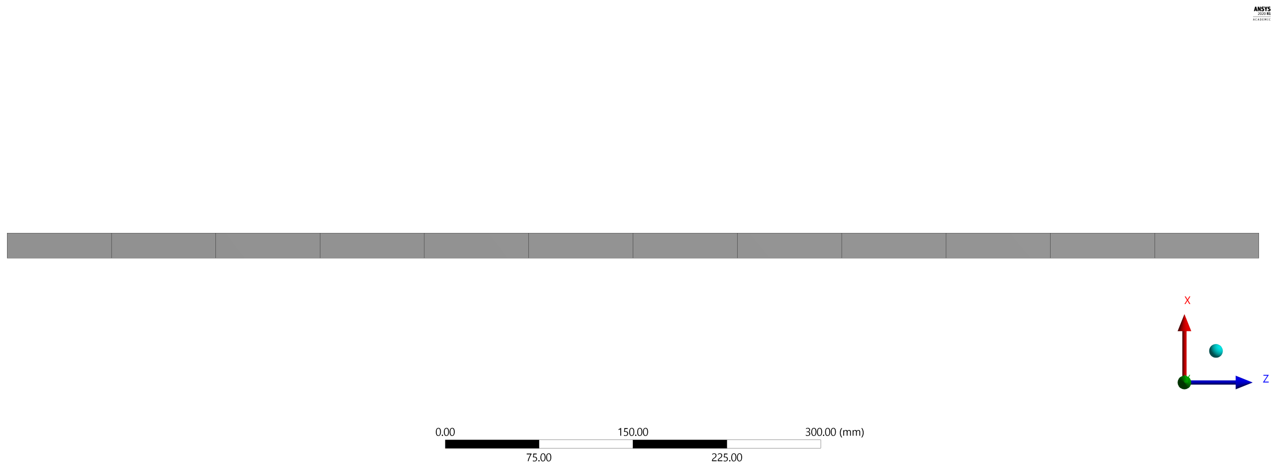


Figure 17: Mesh for 2x12 elements (top view, 1 element through height).

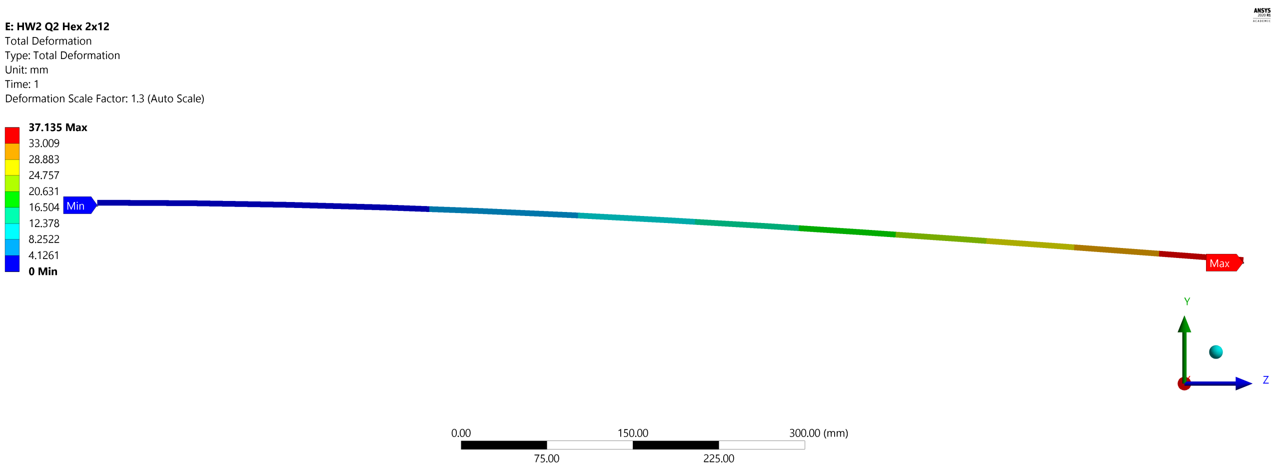


Figure 18: Deformation for 2x12 elements.

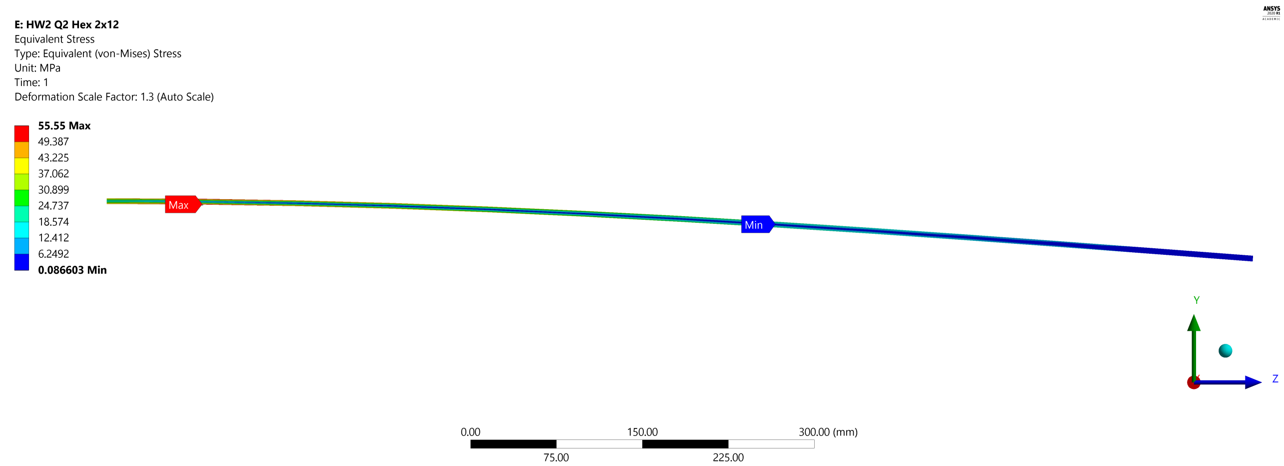


Figure 19: von-Mises stress for 2x12 elements.

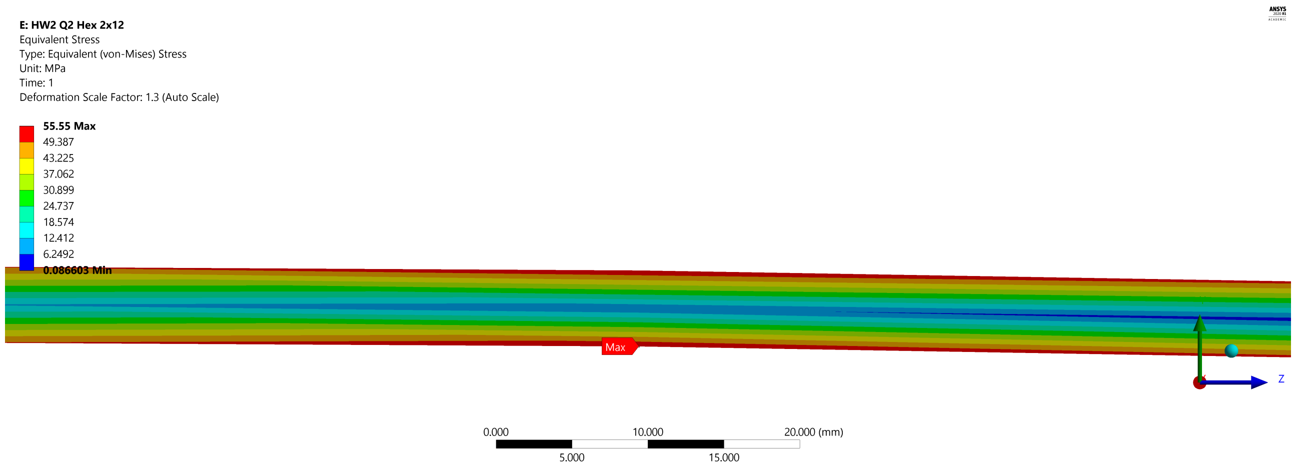


Figure 20: von-Mises stress for 2x12 elements.

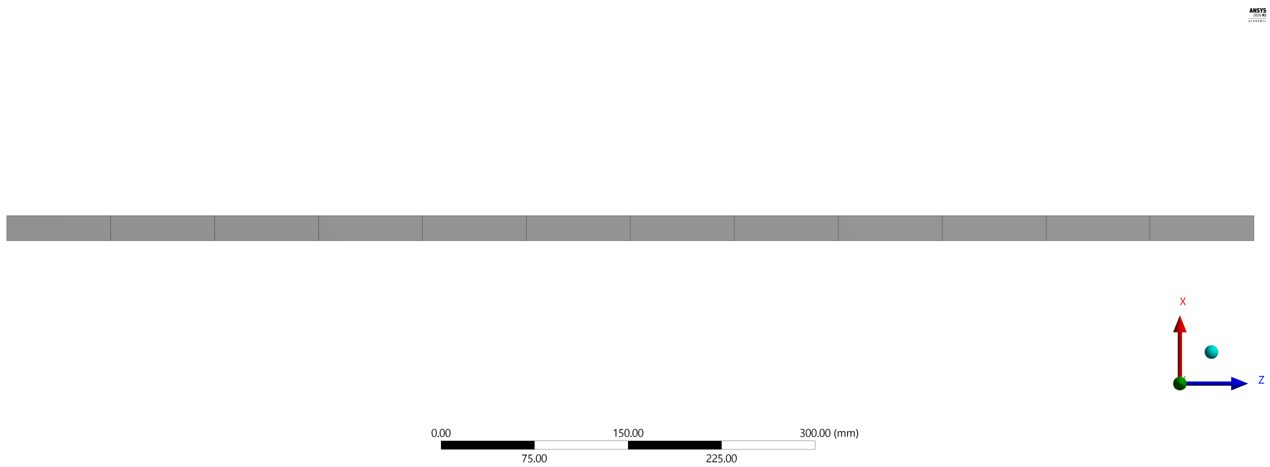


Figure 21: Mesh for 4x12 elements (top view, 1 element through height).

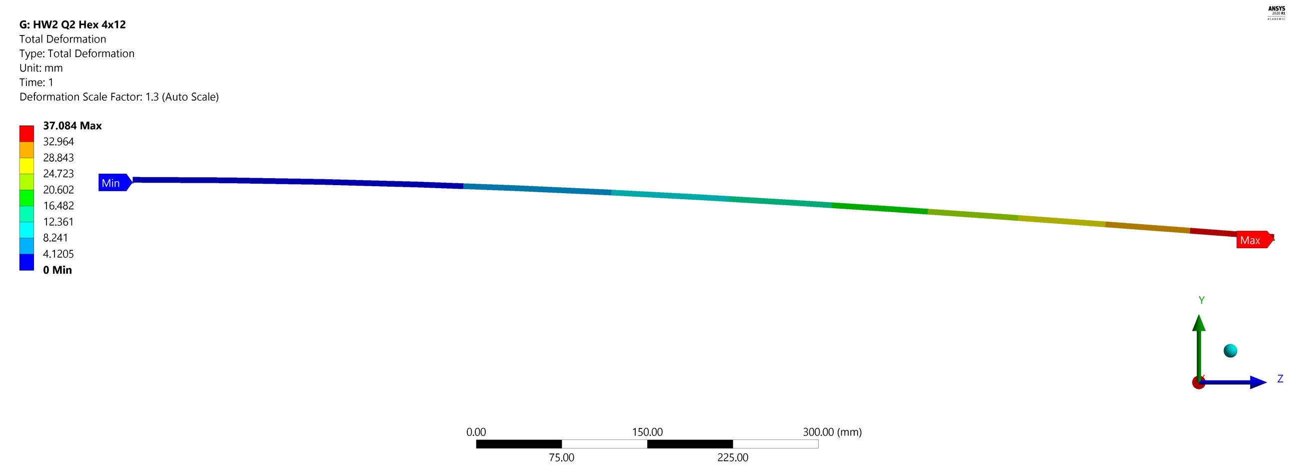


Figure 22: Deformation for 4x12 elements.

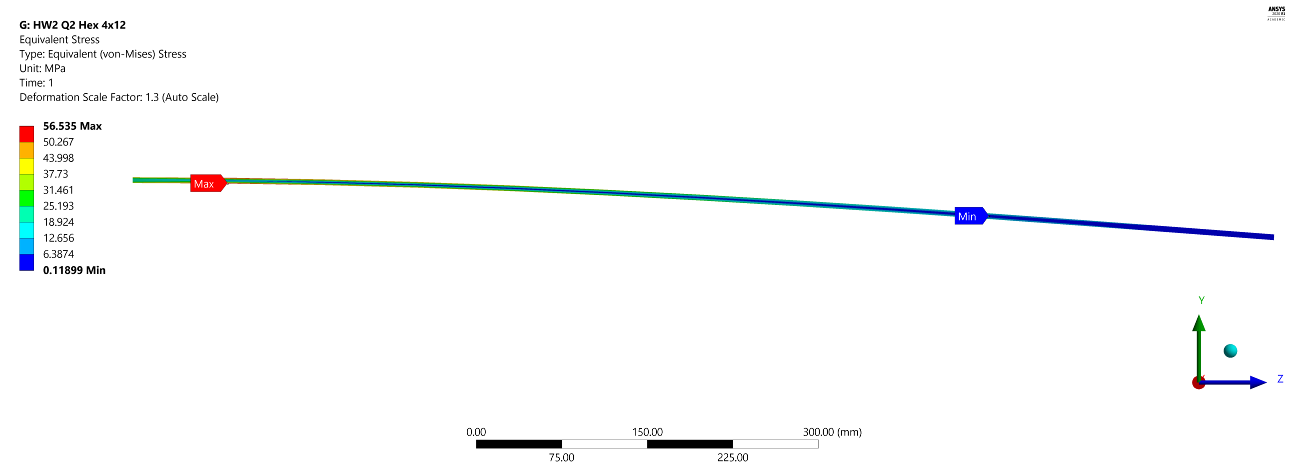


Figure 23: von-Mises stress for 4x12 elements.

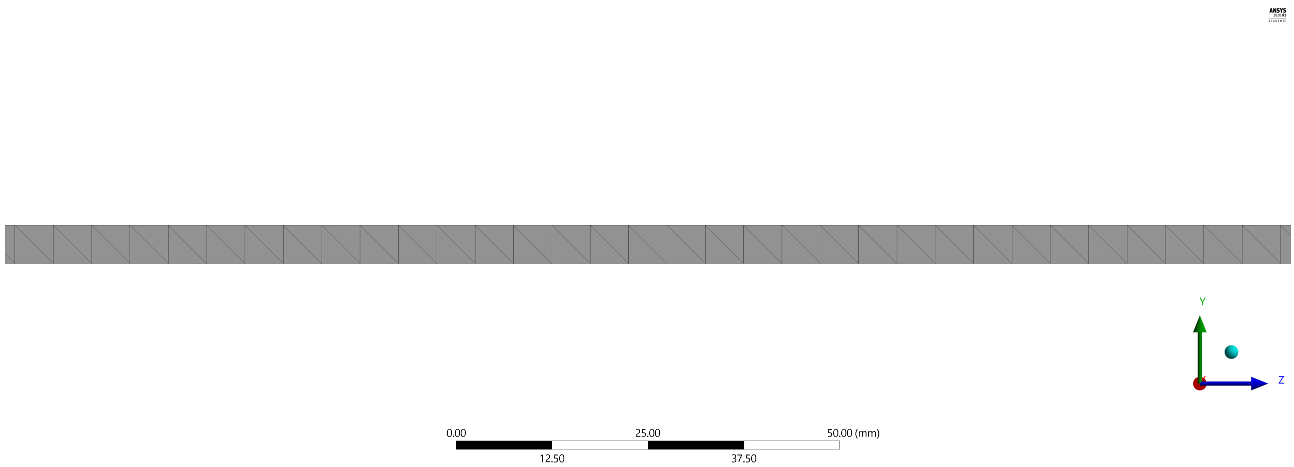


Figure 24: Mesh for 1. tetrahedral model (side view, 1 element through width).

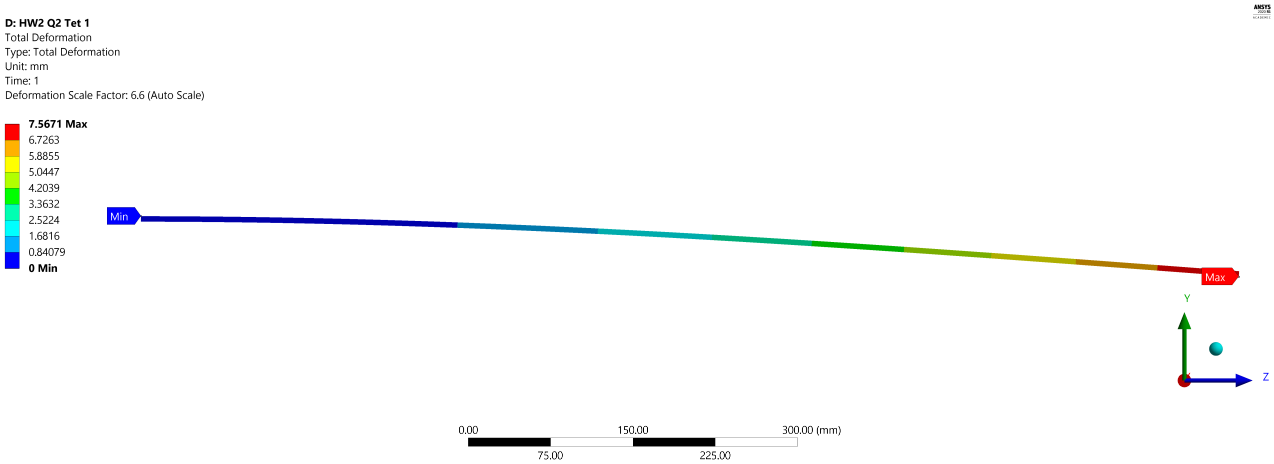


Figure 25: Deformation for 1. tetrahedral model

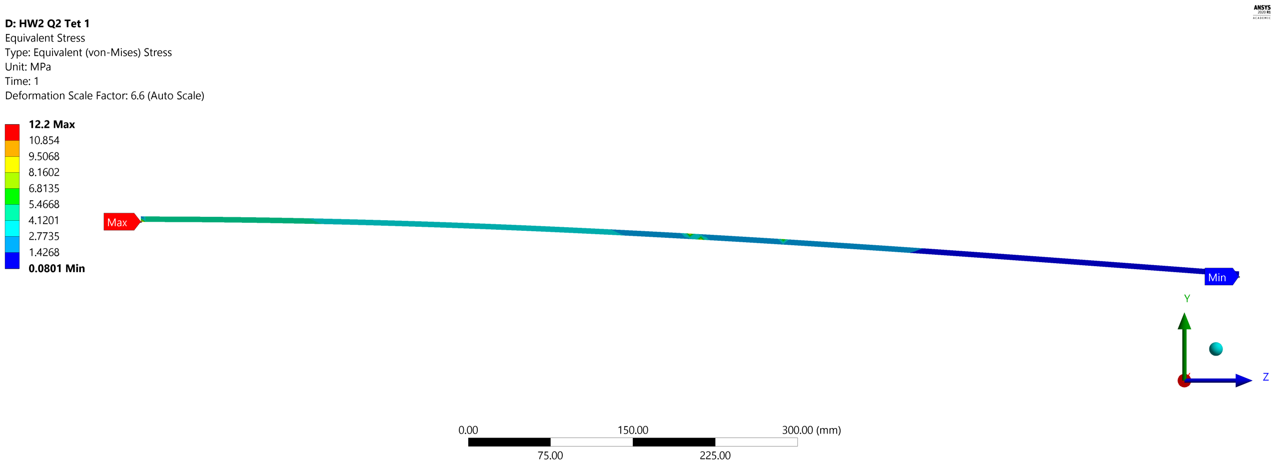


Figure 26: von-Mises stress for 1. tetrahedral model

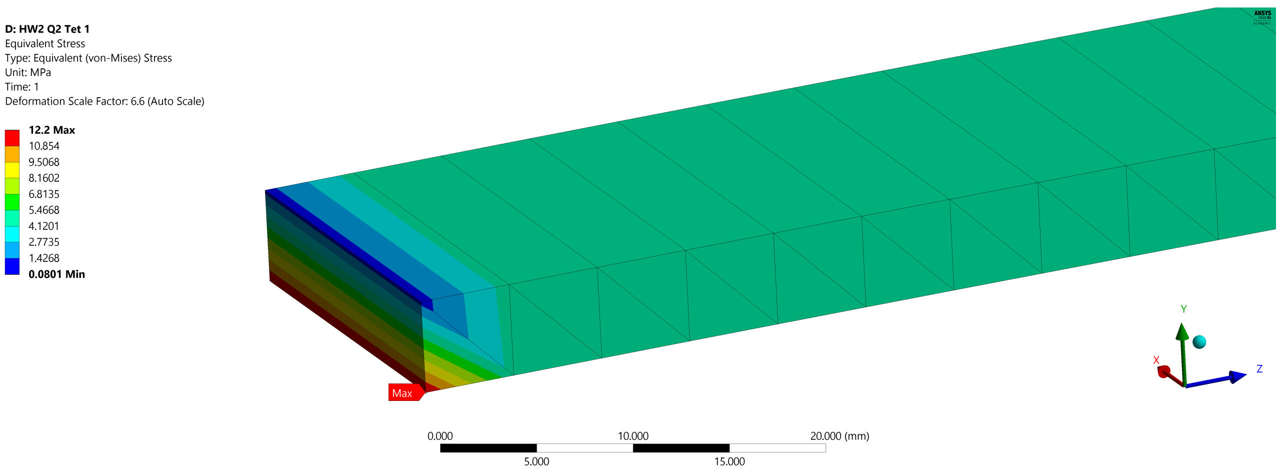


Figure 27: von-Mises stress for 1. tetrahedral model

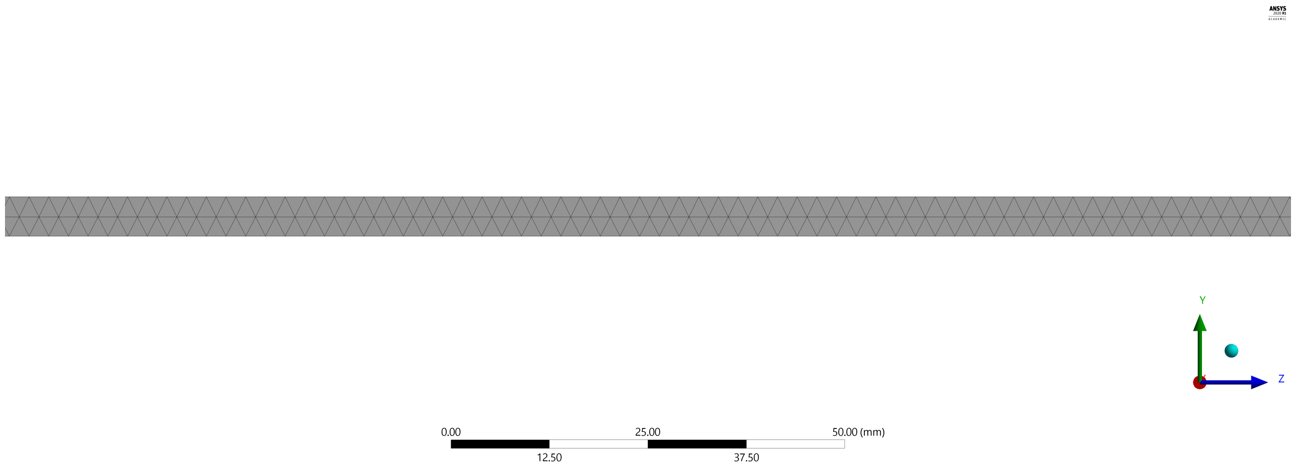


Figure 28: Mesh for 2. tetrahedral model (side view, 1 element through width).

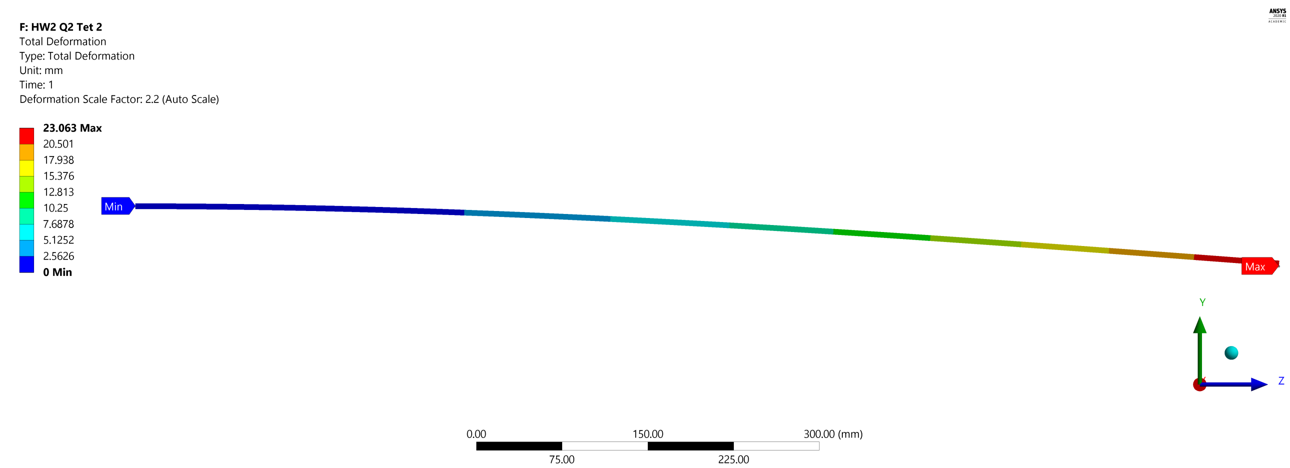


Figure 29: Deformation for 2. tetrahedral model

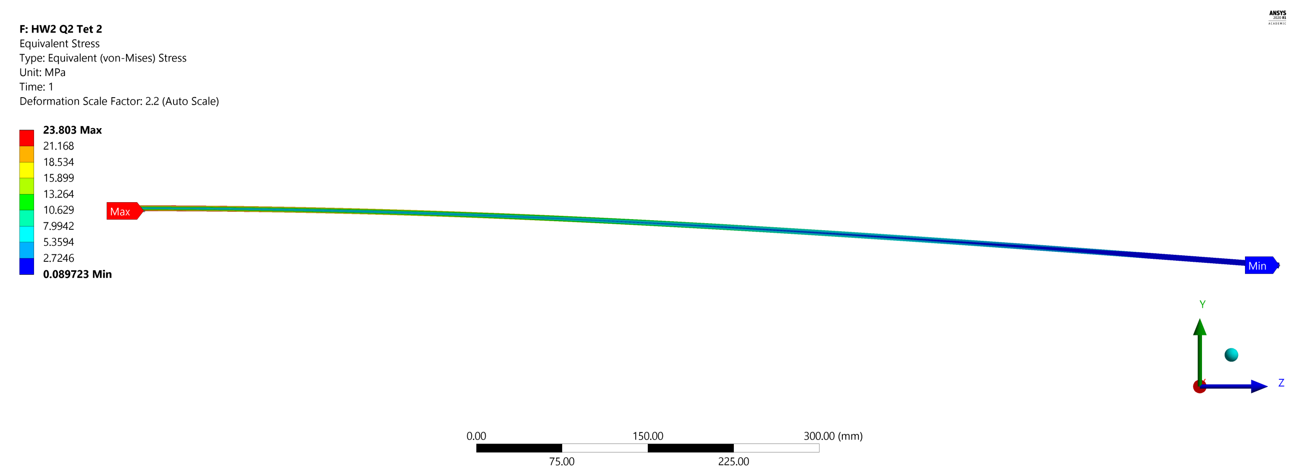


Figure 30: von-Mises stress for 2. tetrahedral model

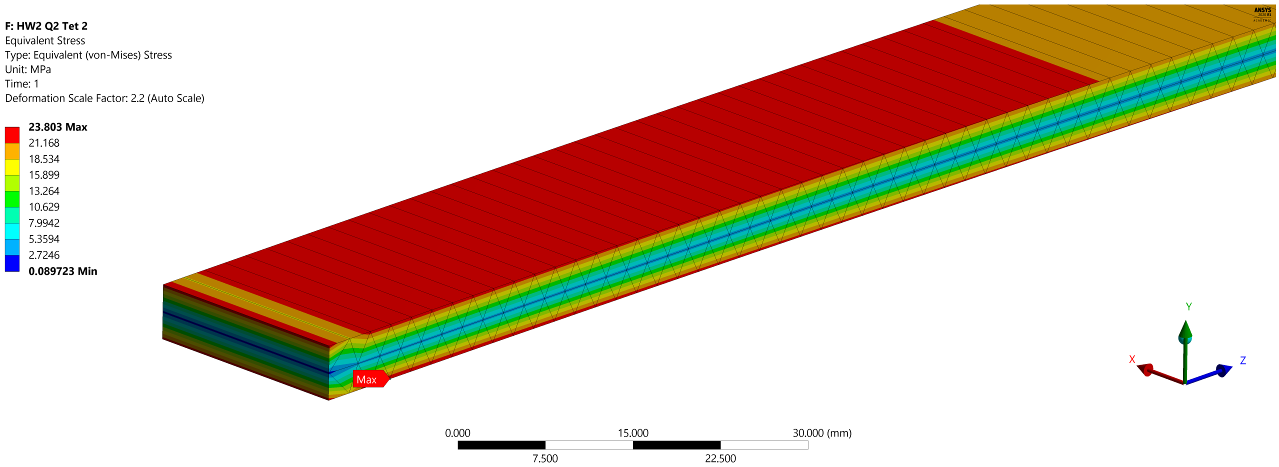


Figure 31: von-Mises stress for 2. tetrahedral model

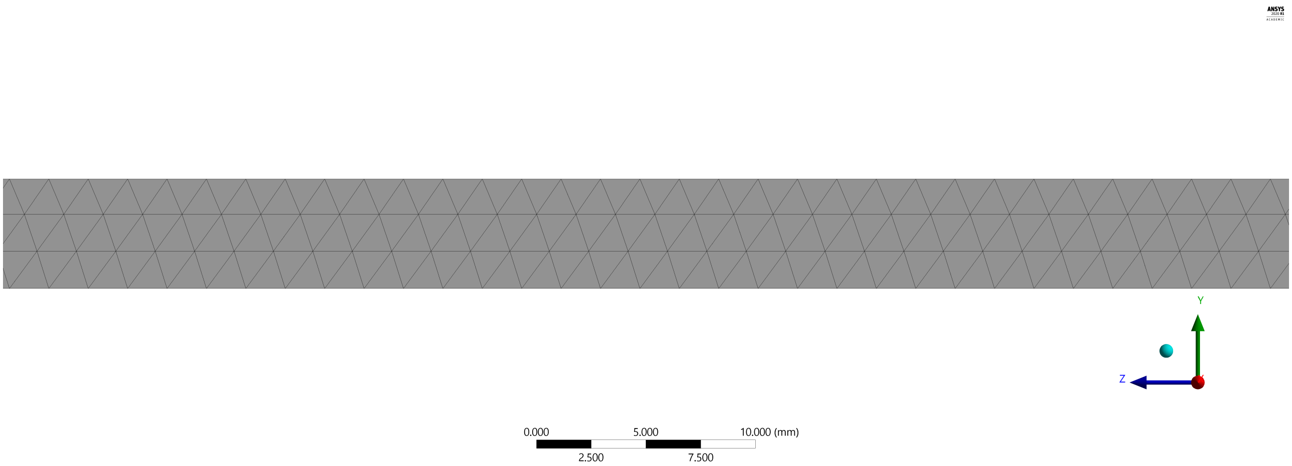


Figure 32: Mesh for 3. tetrahedral model (side view, 1 element through width).

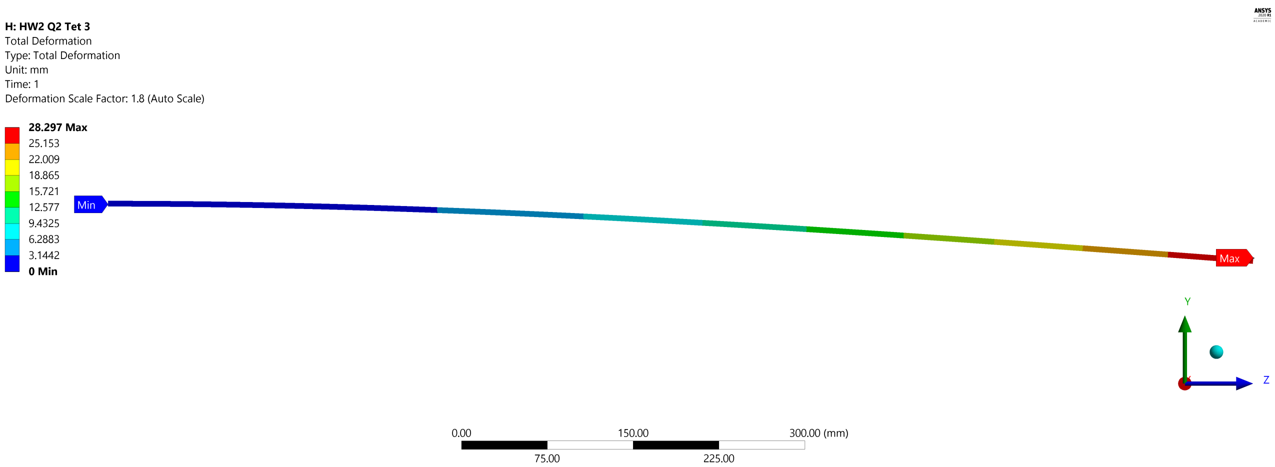


Figure 33: Deformation for 3. tetrahedral model

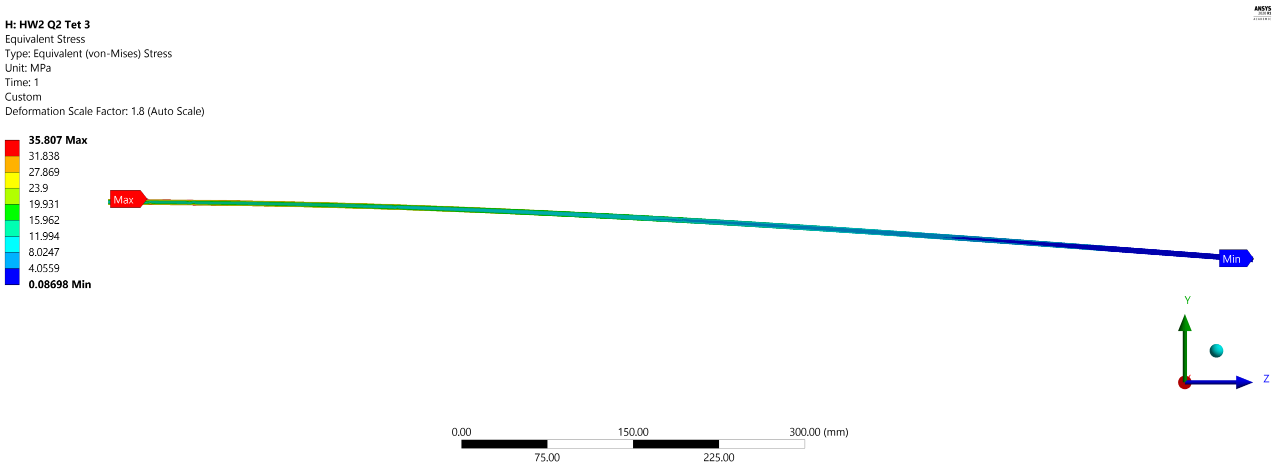


Figure 34: von-Mises stress for 3. tetrahedral model

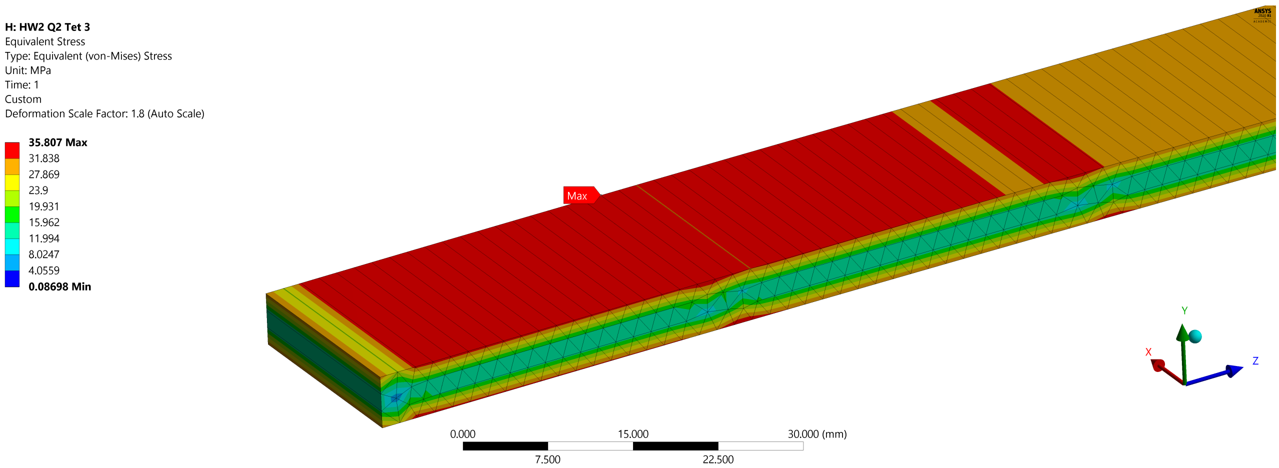


Figure 35: von-Mises stress for 3. tetrahedral model

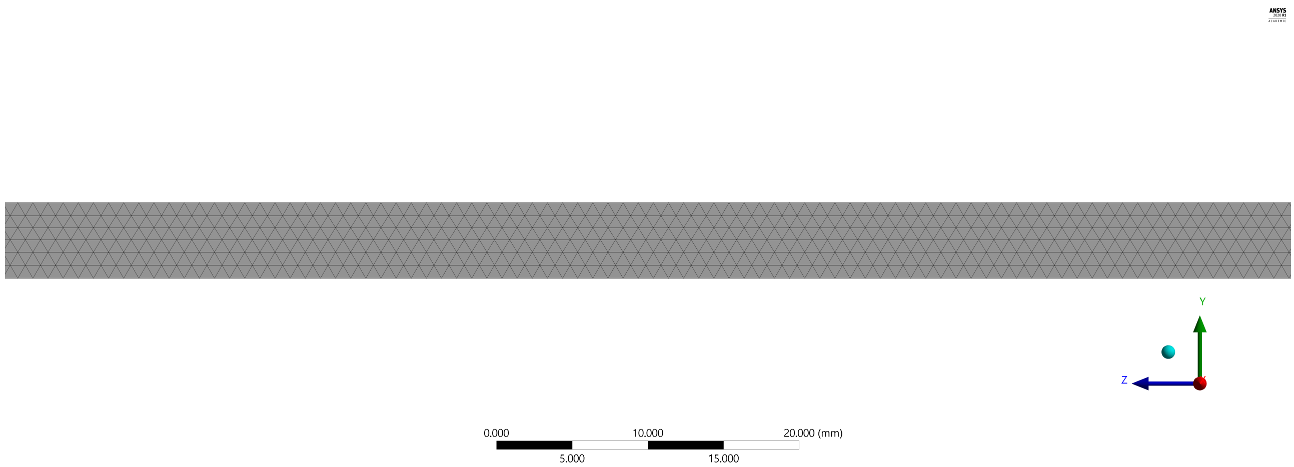


Figure 36: Mesh for 4. tetrahedral model (side view, 1 element through width).

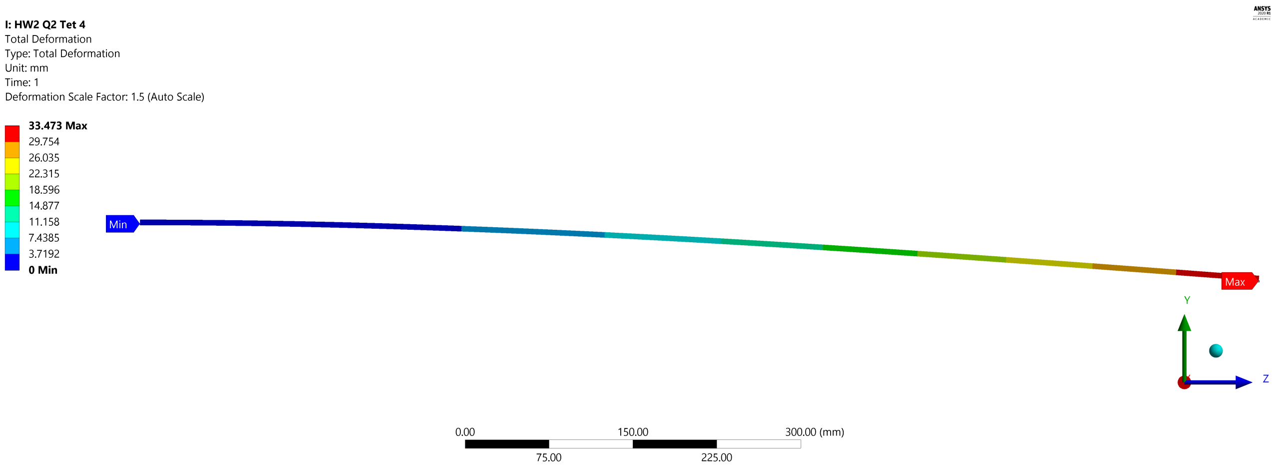


Figure 37: Deformation for 4. tetrahedral model

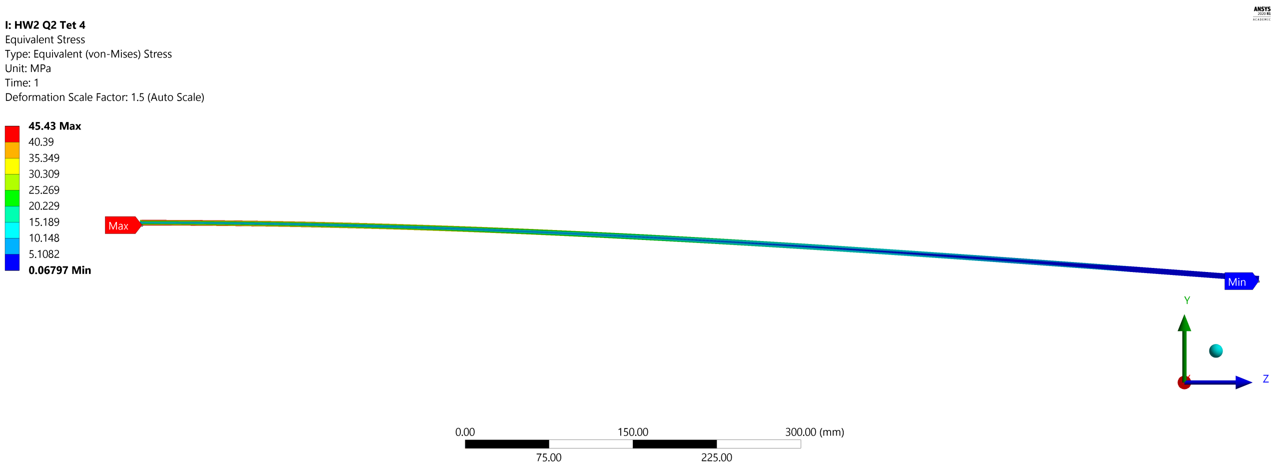


Figure 38: von-Mises stress for 4. tetrahedral model

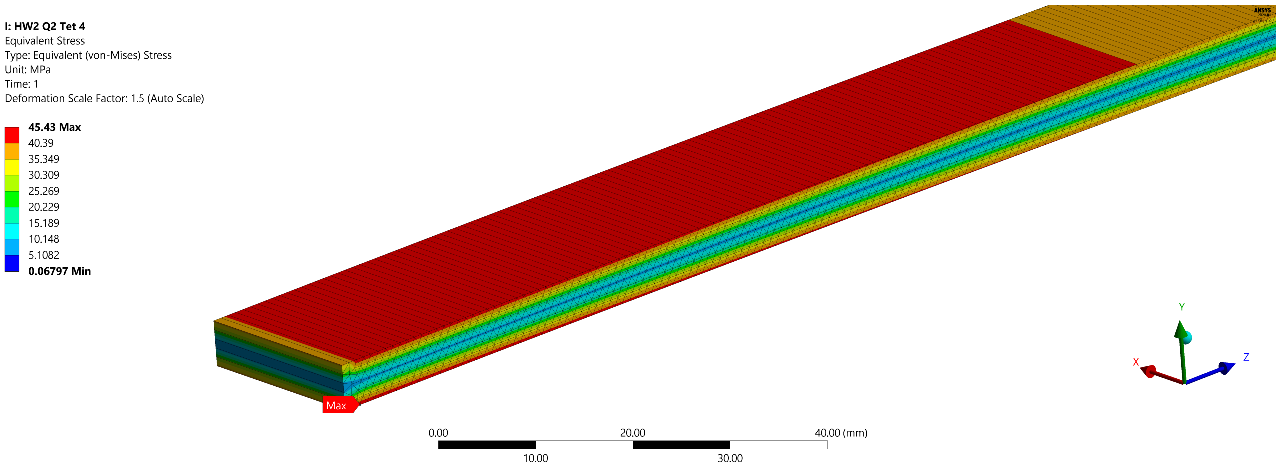
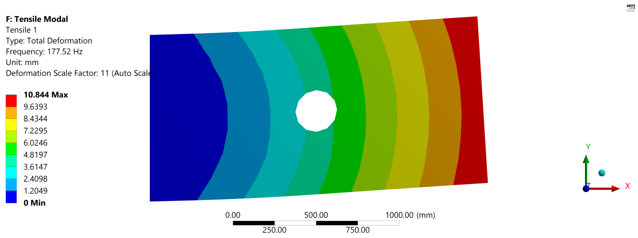
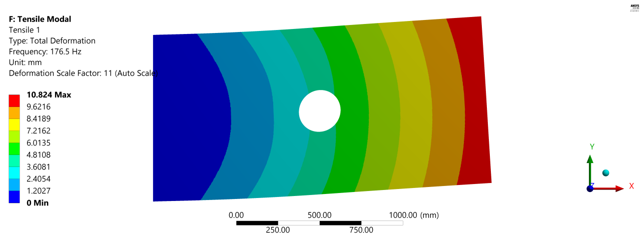


Figure 39: von-Mises stress for 4. tetrahedral model

2 Appendix for Question 3

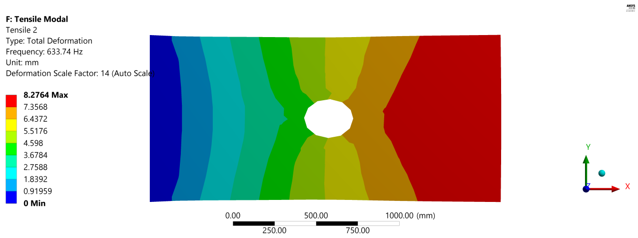


(a) Coarse mesh

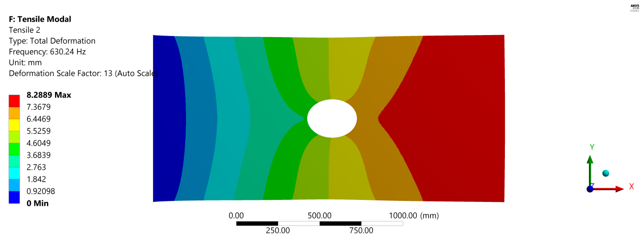


(b) Fine Mesh

Figure 40: 1. mode deflections for 2 different mesh size (linear quadrilateral).

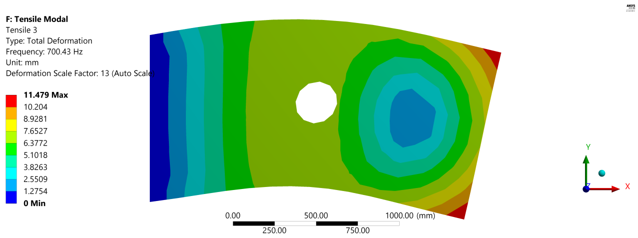


(a) Coarse mesh

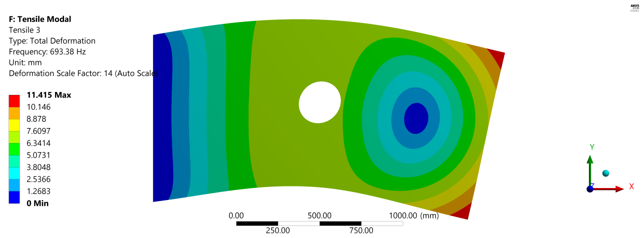


(b) Fine Mesh

Figure 41: 2. mode deflections for 2 different mesh size (linear quadrilateral).

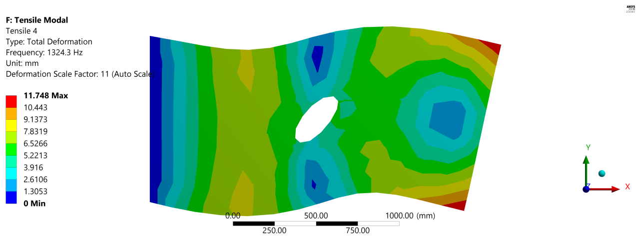


(a) Coarse mesh

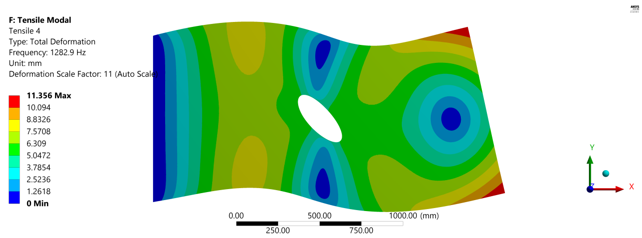


(b) Fine Mesh

Figure 42: 3. mode deflections for 2 different mesh size (linear quadrilateral).

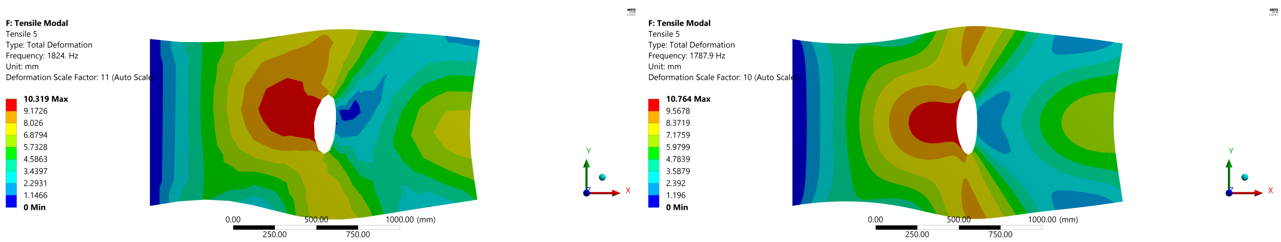


(a) Coarse mesh



(b) Fine Mesh

Figure 43: 4. mode deflections for 2 different mesh size (linear quadrilateral).



(a) Coarse mesh

(b) Fine Mesh

Figure 44: 5. mode deflections for 2 different mesh size (linear quadrilateral).