

ISTANBUL TECHNICAL UNIVERSITY  
GRADUATE SCHOOL OF SCIENCE ENGINEERING AND  
TECHNOLOGY



MKC525E  
FINITE ELEMENT ANALYSIS IN ENGINEERING

---

## Homework 4

---

*Erdem Çalışkan*  
503191531  
01/07/2020

# 1 Question 1

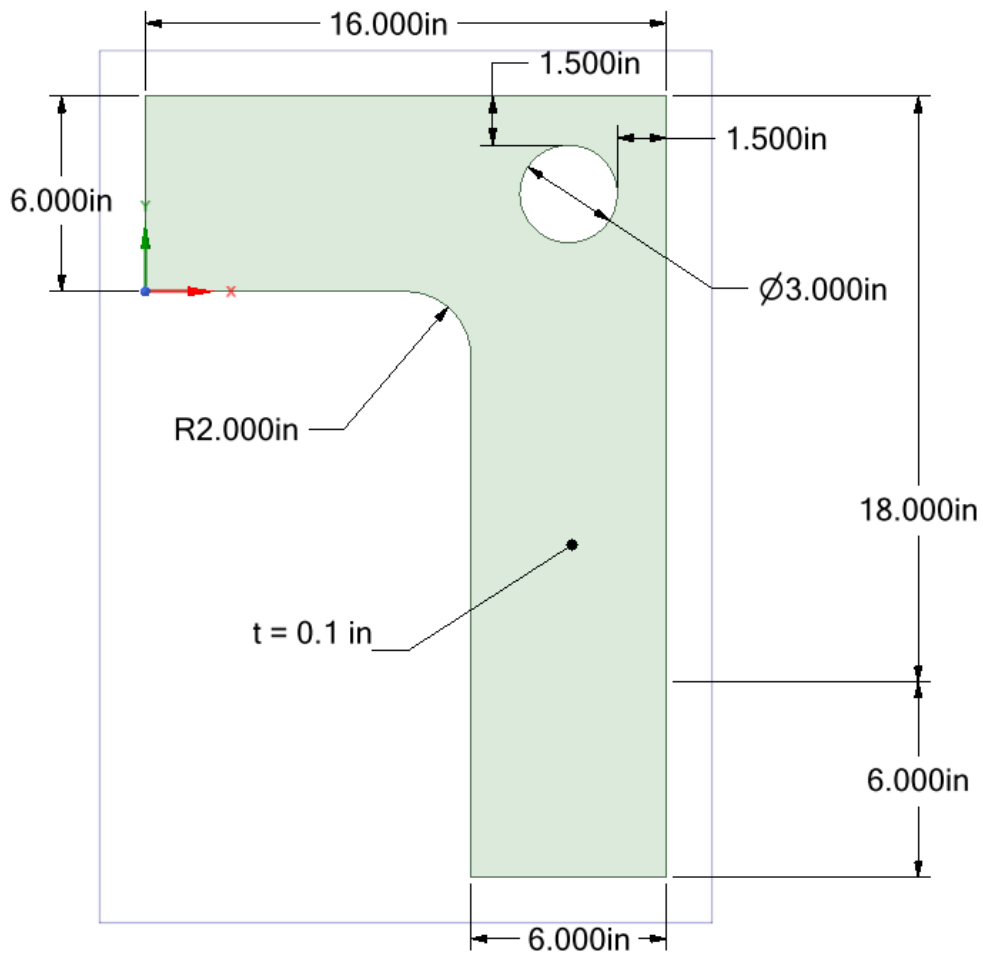


Figure 1: Geometry used in Question 1.

A: Q1  
Static Structural  
Time: 100. s

A Pressure: -1000. psi  
B Fixed Support

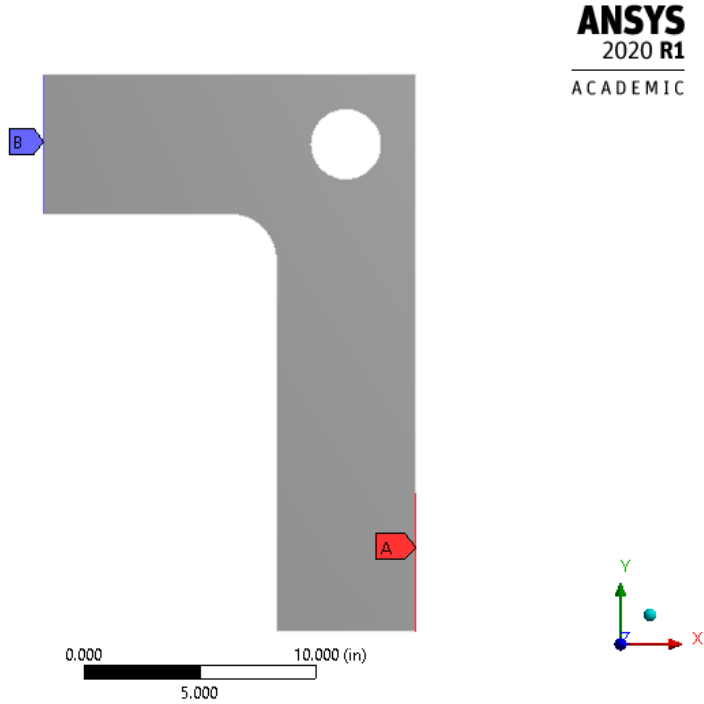
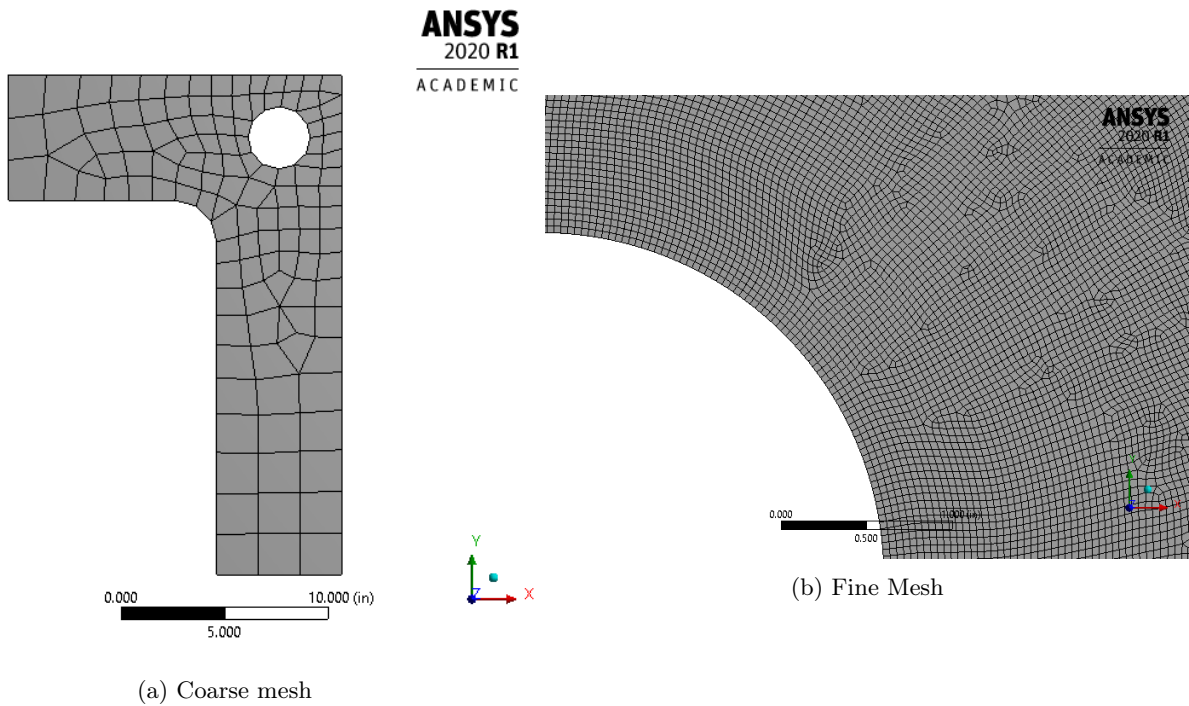


Figure 2: Boundary conditions in Question 1.



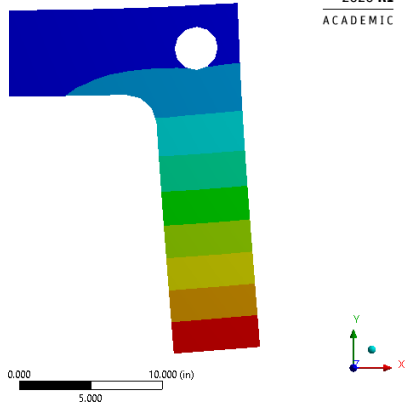
(a) Coarse mesh

(b) Fine Mesh

Figure 3: Example for the different mesh sizes used in Question 1. 2D, Plane stress, linear quadrilateral (PLANE182).

**A: Q1**  
 Directional Deformation  
 Type: Directional Deformation(X,Axis)  
 Unit: in  
 Global Coordinate System  
 Time: 100  
 Deformation Scale Factor: 6.7 (Auto Scale)

**2.16623e-1 Max**  
 1.90372e-1  
 1.64121e-1  
 1.37870e-1  
 1.11619e-1  
 8.53676e-2  
 5.91166e-2  
 3.28655e-2  
 6.61447e-3  
**-1.96366e-2 Min**

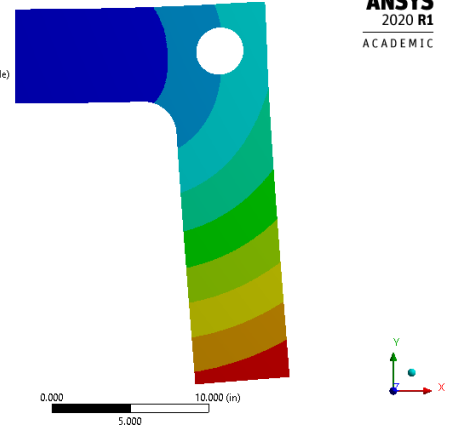


(a) Coarse mesh

**ANSYS**  
 2020 R1  
 ACADEMIC

**A: Q1**  
 Total Deformation  
 Type: Total Deformation  
 Unit: in  
 Time: 100  
 Custom  
 Deformation Scale Factor: 6.1 (Auto Scale)

**0.25433 Max**  
 0.22607  
 0.19781  
 0.16955  
 0.14129  
 0.11304  
 0.084777  
 0.056518  
 0.028259  
**0 Min**

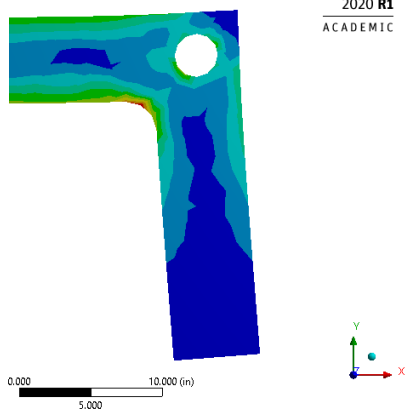


(b) Fine Mesh

Figure 4: Deformation for the mesh sizes given in figure above.

**A: Q1**  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress  
 Unit: psi  
 Time: 100  
 Deformation Scale Factor: 6.7 (Auto Scale)

**30196 Max**  
 26866  
 23536  
 20206  
 16877  
 13547  
 10217  
 6886.9  
 3557  
**227.09 Min**

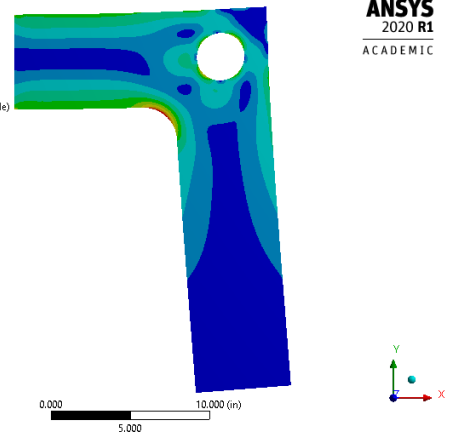


(a) Coarse mesh

**ANSYS**  
 2020 R1  
 ACADEMIC

**A: Q1**  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress  
 Unit: psi  
 Time: 100  
 Custom  
 Max: 35567  
 Min: 0.14988  
 Deformation Scale Factor: 6.1 (Auto Scale)

**35567 Max**  
 31615  
 27664  
 23712  
 19760  
 15808  
 11856  
 7904  
 3952.1  
**0.14988 Min**



(b) Fine Mesh

Figure 5: von-Mises stress for the mesh sizes given in figure above.

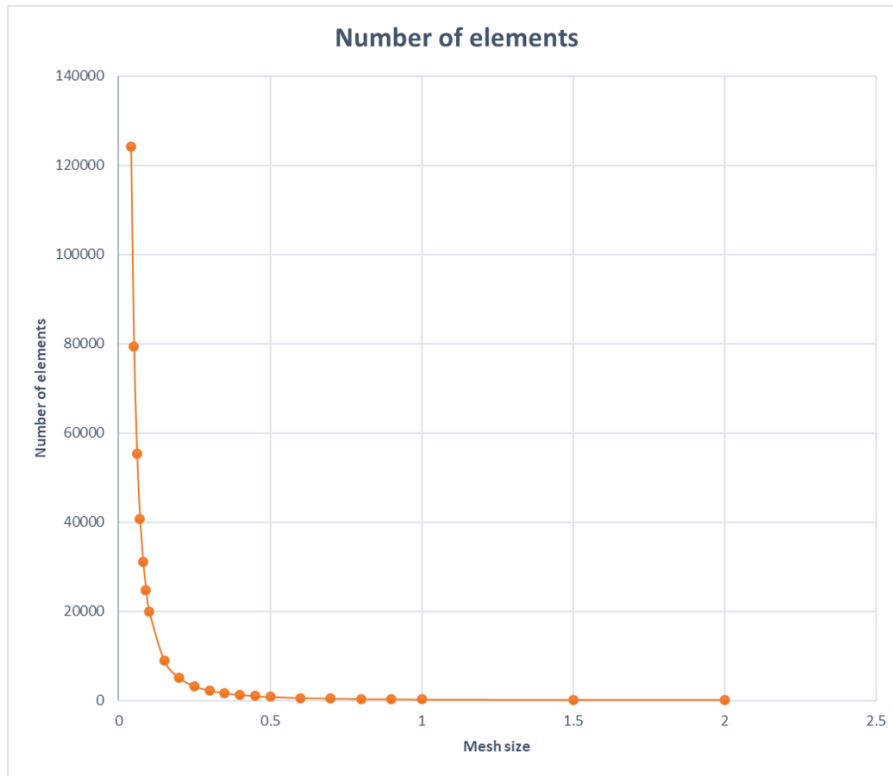


Figure 6: Number of elements as a function of element size.

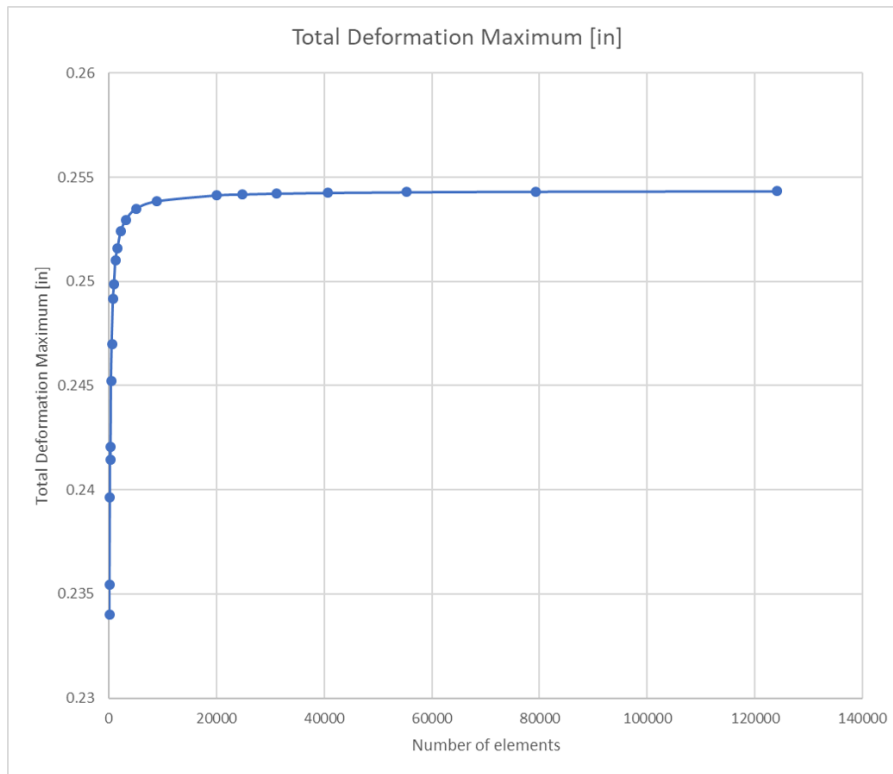


Figure 7: Deformation as a function of number of elements.

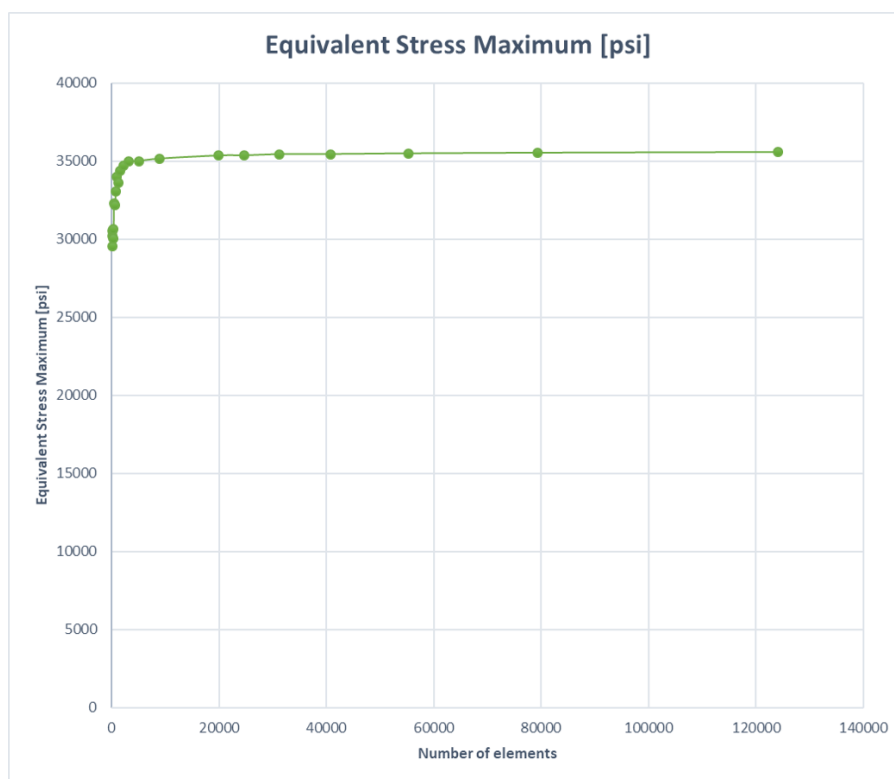


Figure 8: von-Mises stress as a function of number of elements.

## 2 Question 2

Linear beam, shell and solid elements are compared in this question. Figures for deformation and stress are given in Appendix.

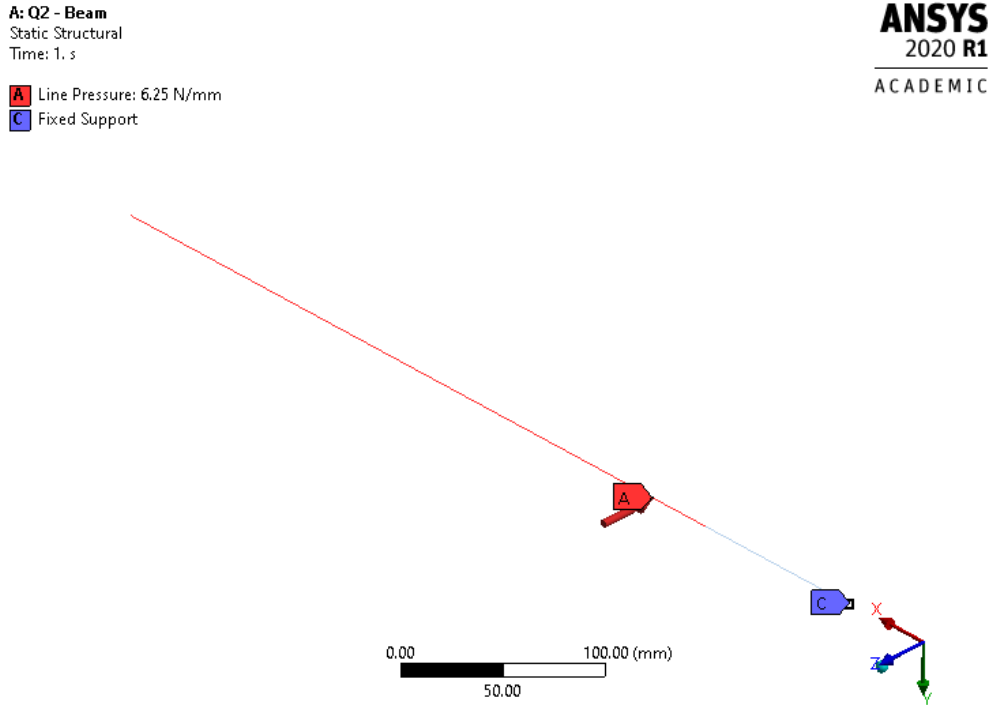


Figure 9: Boundary conditions in Question 2.

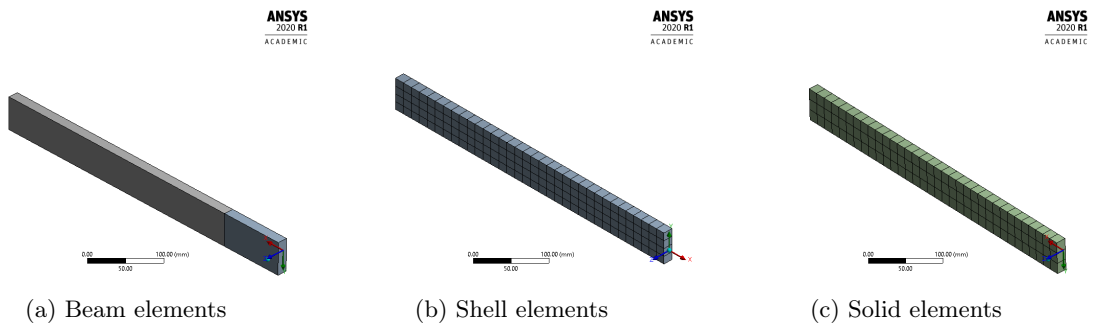


Figure 10: Example for types of mesh used in Question 2 (biggest mesh size).

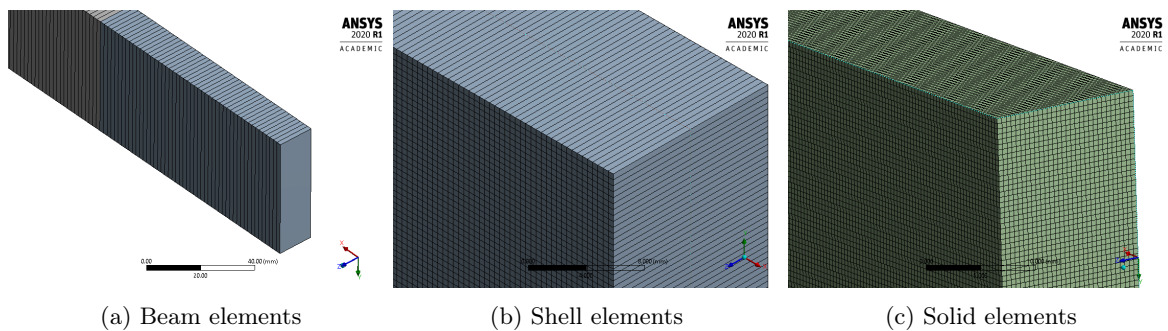


Figure 11: Example for types of mesh used in Question 2 (smallest mesh size).

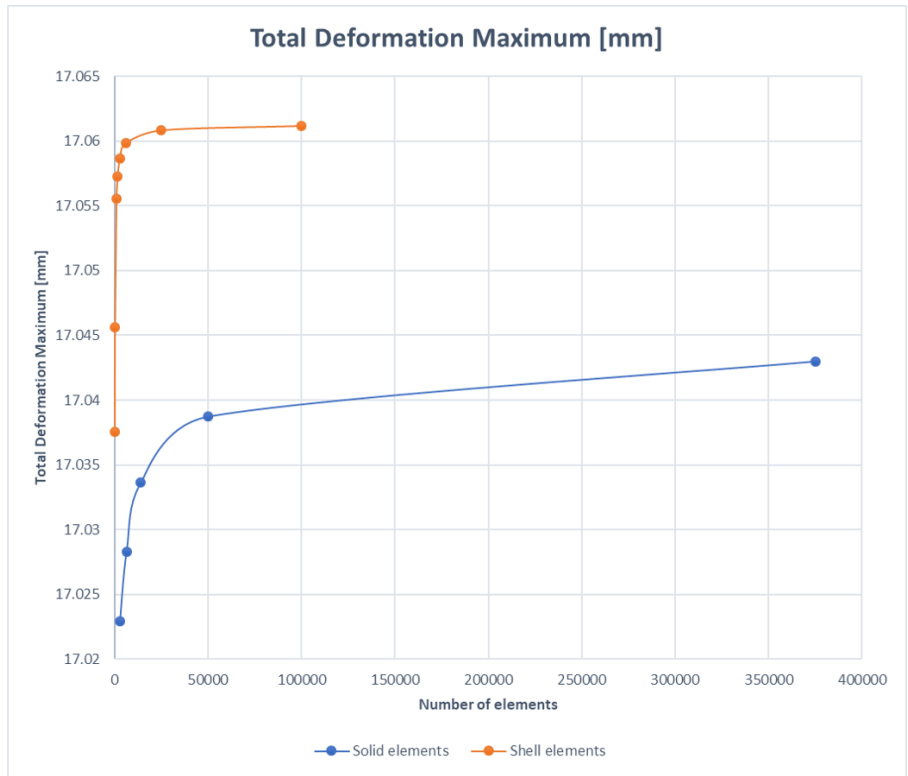


Figure 12: Deformation as a function of number of elements (shell and solid elements).

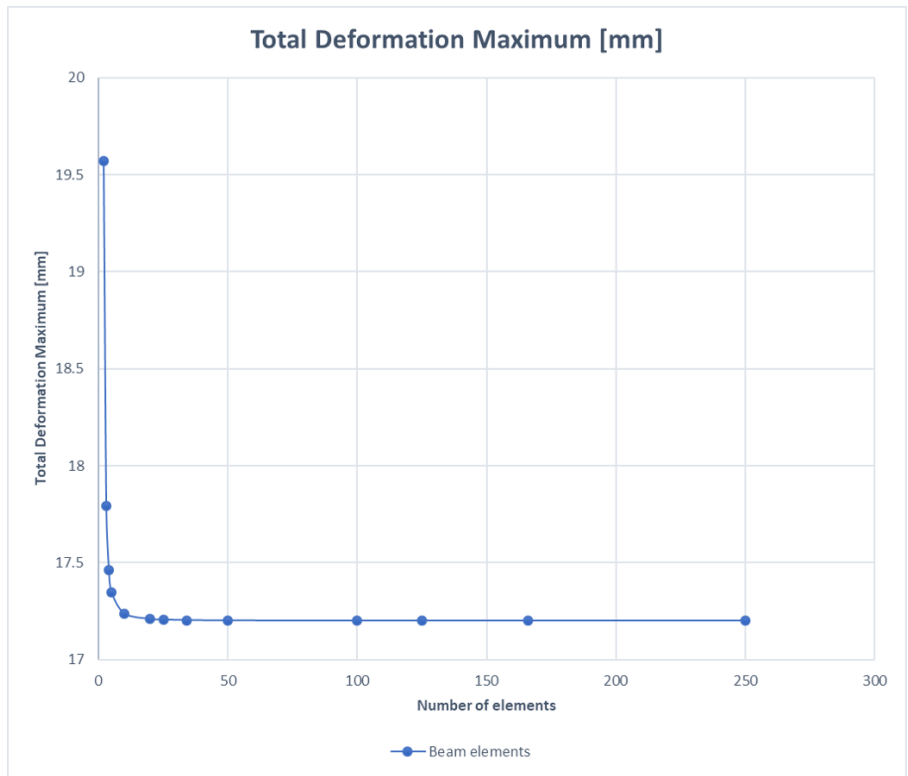


Figure 13: Deformation as a function of number of elements (beam elements).



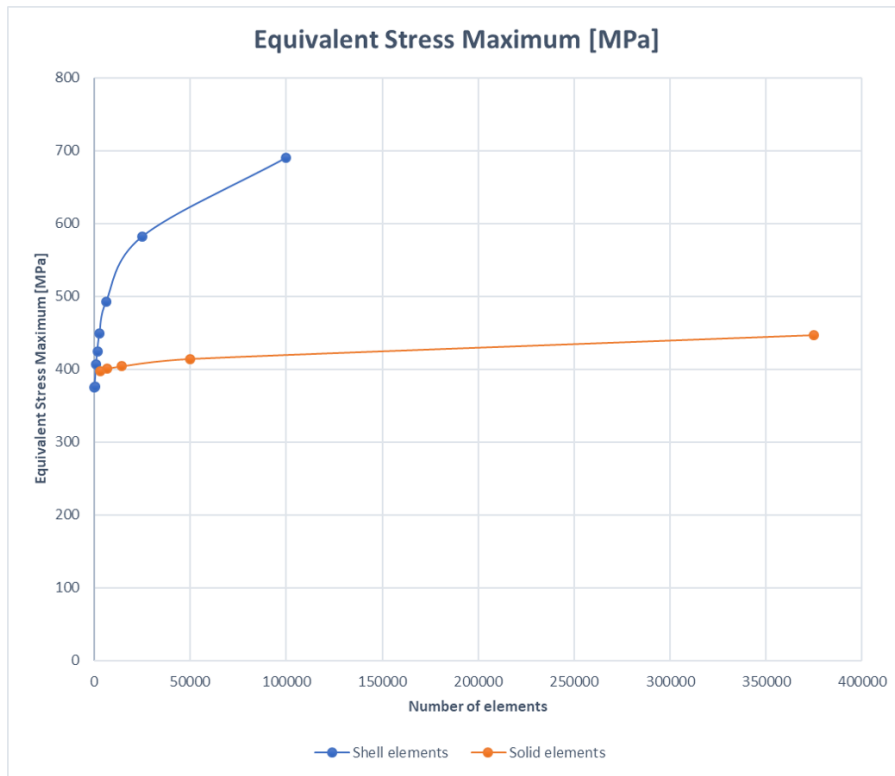
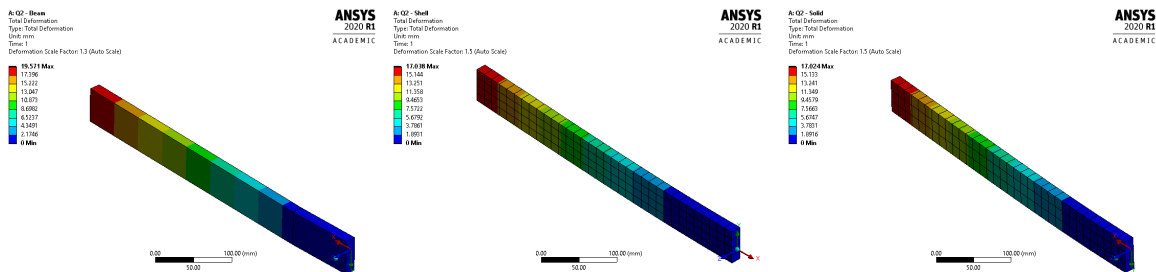
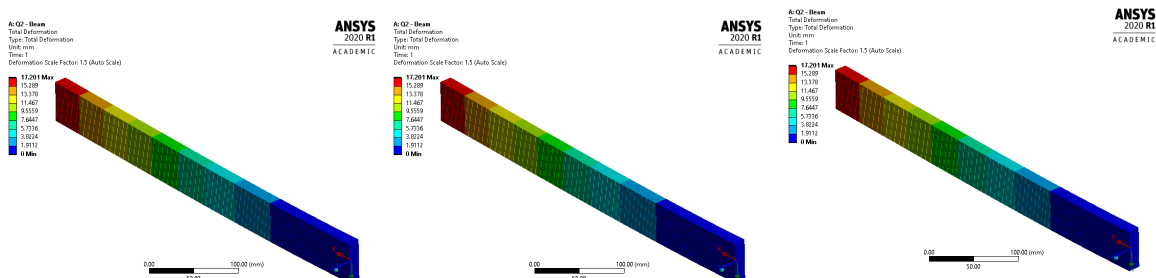


Figure 14: von-Mises stress as a function of number of elements (shell and solid elements).



(a) Beam elements (b) Shell elements (c) Solid elements

Figure 15: Deformation for the mesh sizes given in figure above (biggest mesh size).

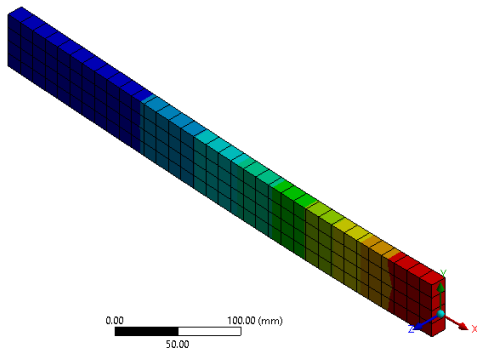


(a) Beam elements (b) Shell elements (c) Solid elements

Figure 16: Deformation for the mesh sizes given in figure above (smallest mesh size).

A: Q2 - Shell  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: MPa  
 Time: 1  
 Deformation Scale Factor: 1.5 (Auto Scale)

374.74 Max  
 333.13  
 291.52  
 249.92  
 208.31  
 166.7  
 125.09  
 83.489  
 41.875  
 0.26629 Min

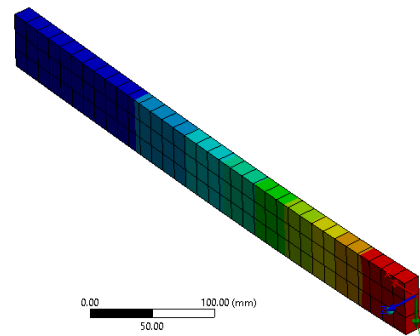


(a) Shell elements

ANSYS  
 2020 R1  
 ACADEMIC

A: Q2 - Solid  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress  
 Unit: MPa  
 Time: 1  
 Deformation Scale Factor: 1.5 (Auto Scale)

368.27 Max  
 327.39  
 286.52  
 245.64  
 204.76  
 163.89  
 123.01  
 82.13  
 41.259  
 0.37517 Min



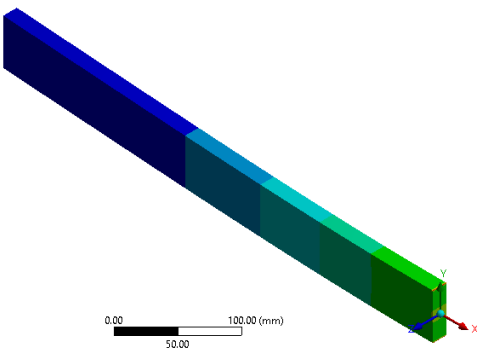
(b) Solid elements

ANSYS  
 2020 R1  
 ACADEMIC

Figure 17: von-Mises stress for the mesh sizes given in figure above (biggest mesh size).

A: Q2 - Shell  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: MPa  
 Time: 1  
 Deformation Scale Factor: 1.5 (Auto Scale)

690.64 Max  
 613.9  
 537.17  
 460.43  
 383.69  
 306.95  
 230.21  
 153.48  
 76.739  
 0.00099119 Min

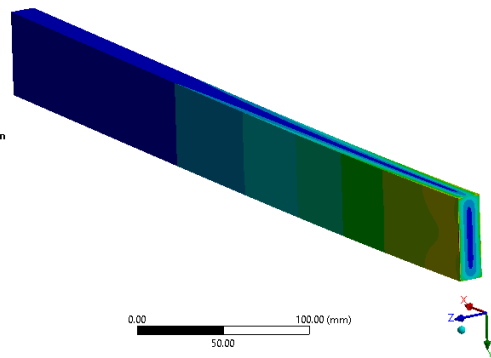


(a) Shell elements

ANSYS  
 2020 R1  
 ACADEMIC

A: Q2 - Solid  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress  
 Unit: MPa  
 Time: 1  
 Deformation Scale Factor: 1.5 (Auto Scale)

527.42 Max  
 468.82  
 410.22  
 351.61  
 293.01  
 234.41  
 175.81  
 117.21  
 58.603  
 0.0013401 Min



(b) Solid elements

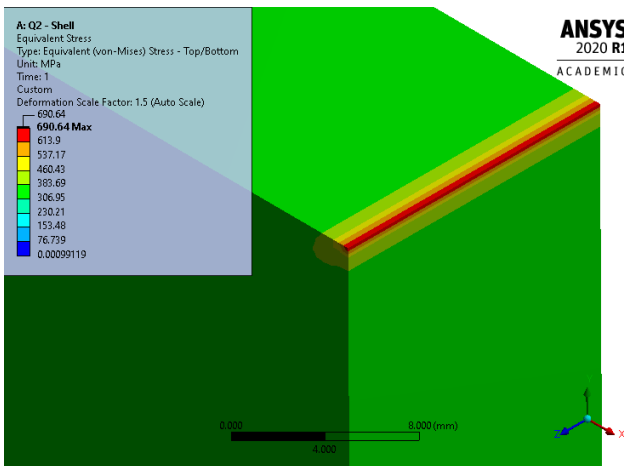
ANSYS  
 2020 R1  
 ACADEMIC

def

Figure 18: von-Mises stress for the mesh sizes given in figure above (smallest mesh size).

A: Q2 - Shell  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: MPa  
 Time: 1  
 Custom  
 Deformation Scale Factor: 1.5 (Auto Scale)

690.64 Max  
 613.9  
 537.17  
 460.43  
 383.69  
 306.95  
 230.21  
 153.48  
 76.739  
 0.00099119

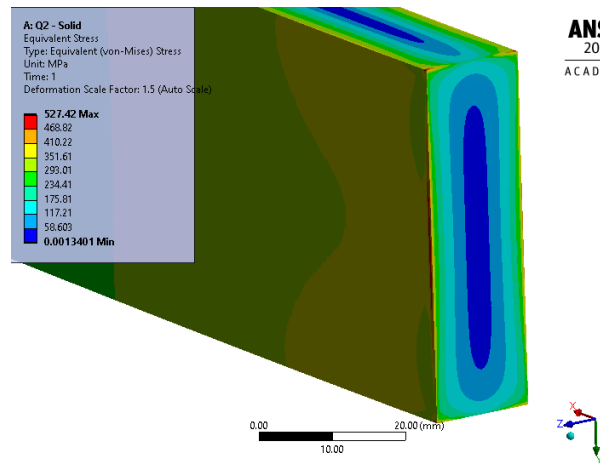


(a) Shell elements

ANSYS  
 2020 R1  
 ACADEMIC

A: Q2 - Solid  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress  
 Unit: MPa  
 Time: 1  
 Deformation Scale Factor: 1.5 (Auto Scale)

527.42 Max  
 468.82  
 410.22  
 351.61  
 293.01  
 234.41  
 175.81  
 117.21  
 58.603  
 0.0013401 Min



(b) Solid elements

ANSYS  
 2020 R1  
 ACADEMIC

Figure 19: Closer look at von-Mises stress for the smallest mesh sizes given in figure above.

### 3 Question 3

There is an ideal geometry in this question. So there is no distorted elements. Temperatures and heat flux in x-direction is same for every element size (200 mm(1 element), 2 mm(130,000 elements)). Heat flux in y and z-directions are changing with the element size but they are considered numerical errors ( $< 10^{-7}$ ).

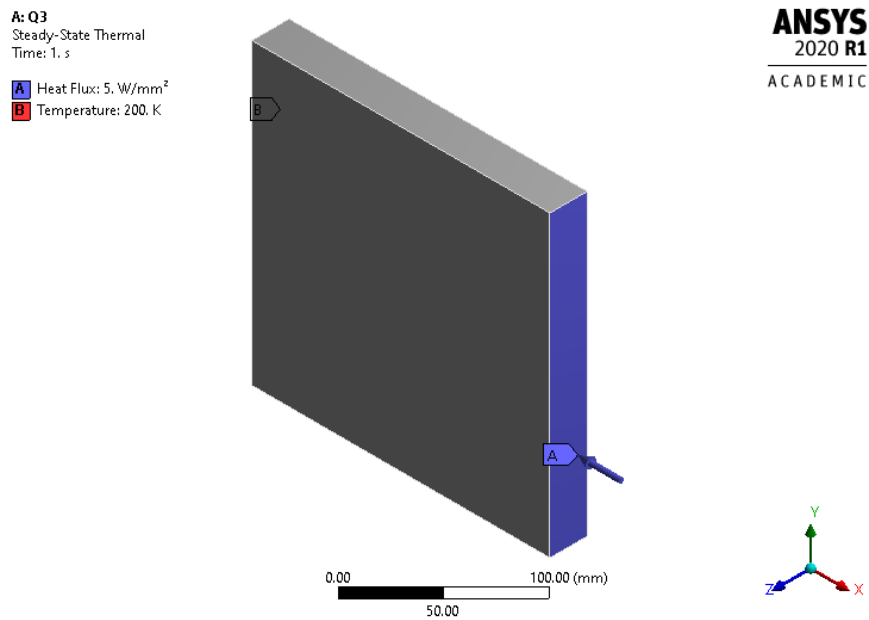


Figure 20: Boundary conditions in Question 3.

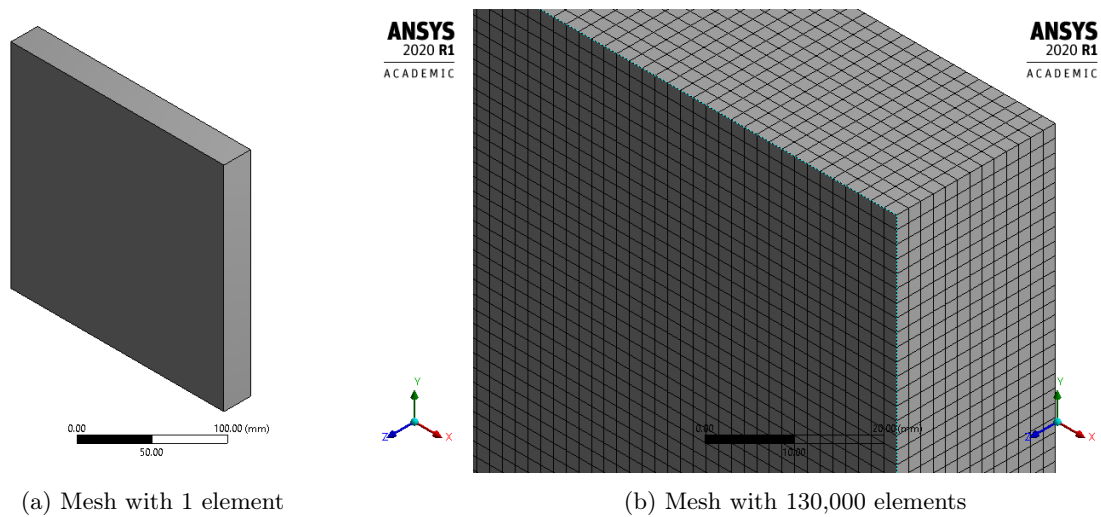
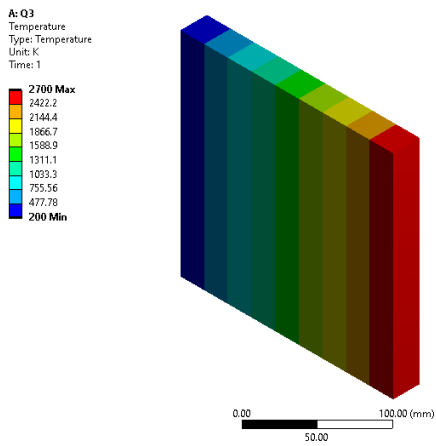
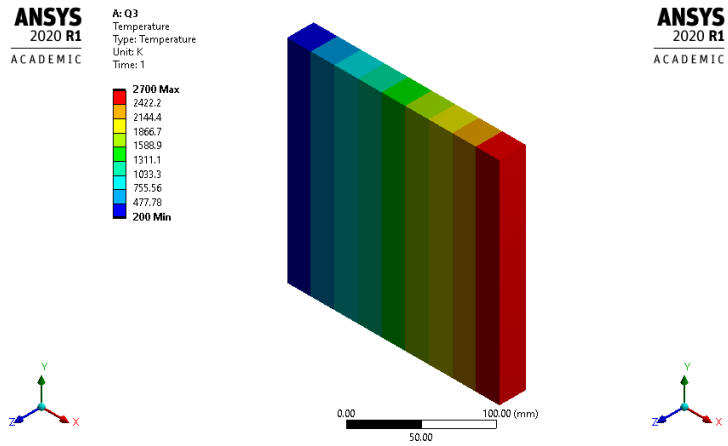


Figure 21: Two different mesh for the problem.

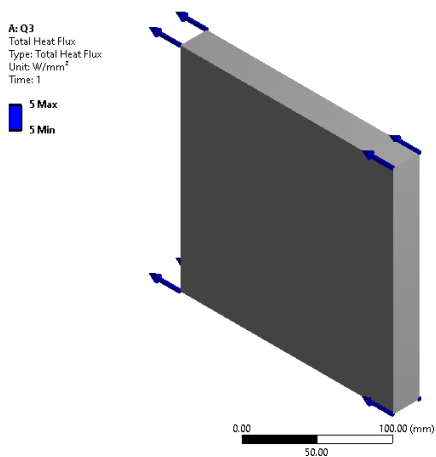


(a) Mesh with 1 element

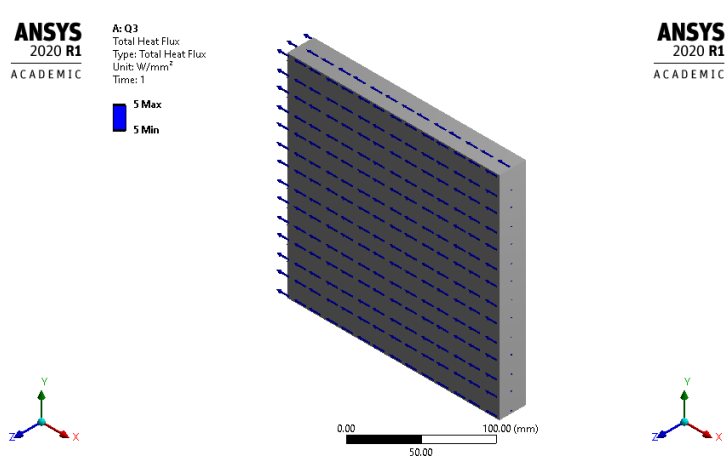


(b) Mesh with 130,000 elements

Figure 22: Temperatures.



(a) Mesh with 1 element



(b) Mesh with 130,000 elements

Figure 23: Heat flux (x-direction).

# 4 Question 4

**A: Static Structural**  
 Static Structural  
 Time: 1. s

- A** Pressure: 10. MPa
- B** Fixed Support

**ANSYS**  
 2020 R1  
 ACADEMIC

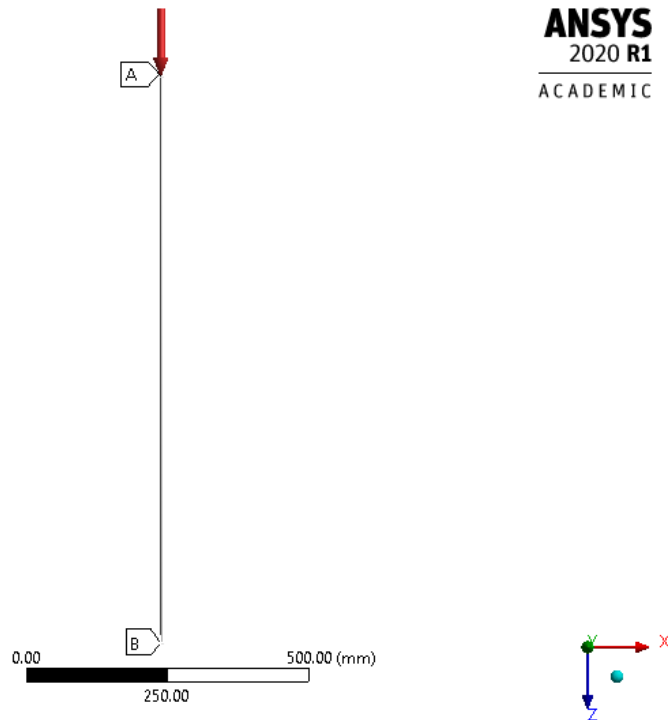


Figure 24: Boundary conditions in Question 4.

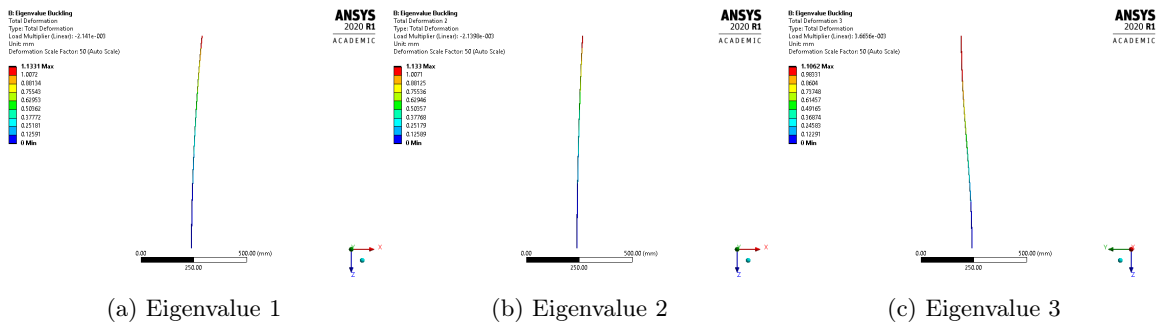


Figure 25: Deformation for first three eigenvalues (Load Multipliers) (Quadratic hexahedral).

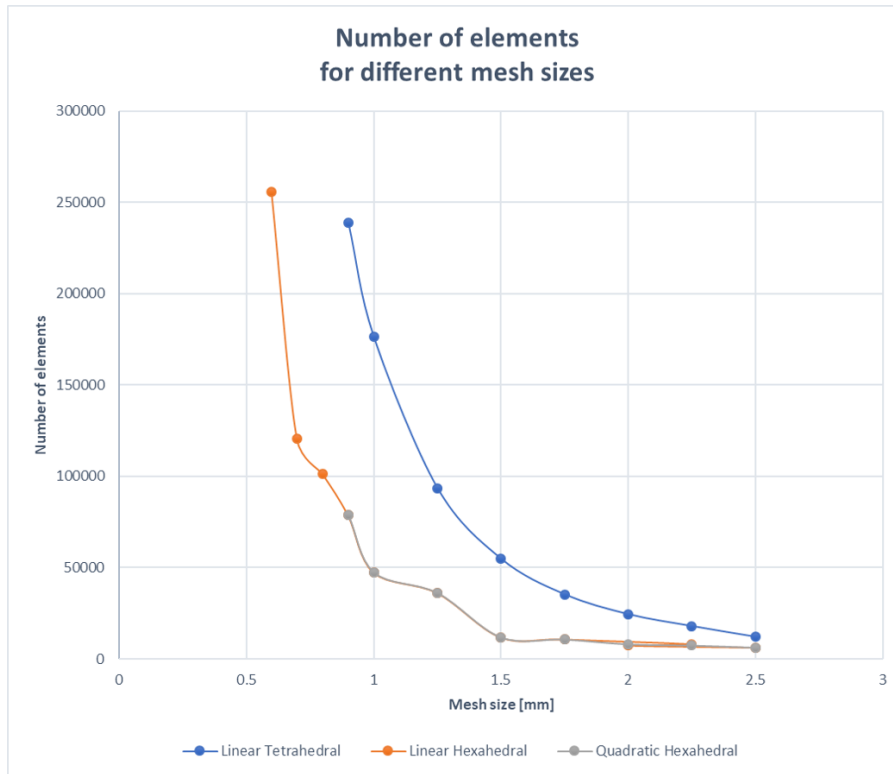


Figure 26: Number of elements as a function of element size.

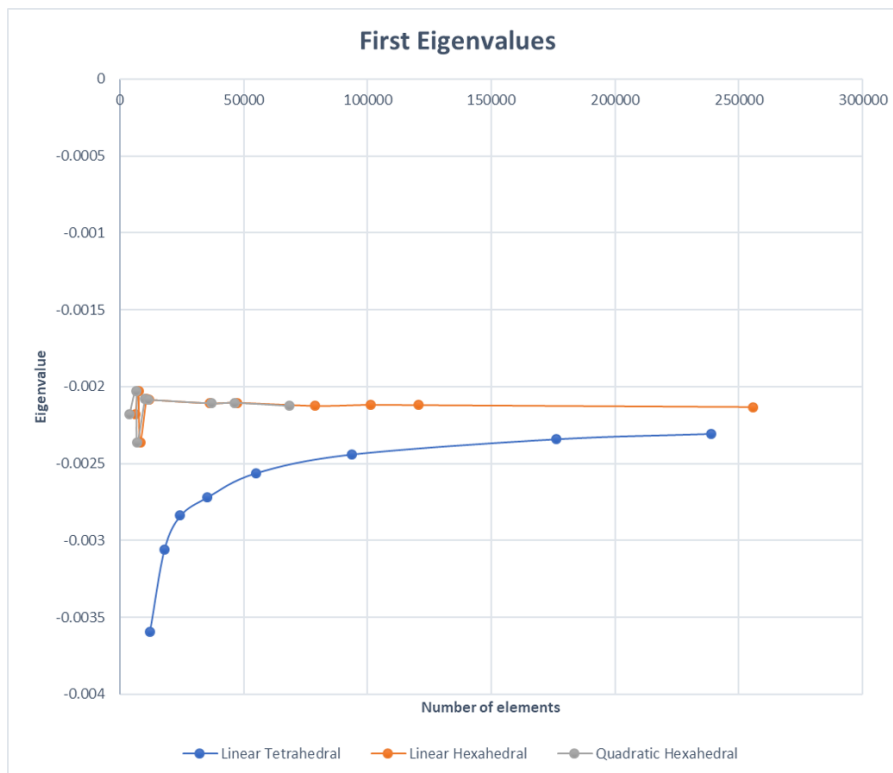


Figure 27: First eigenvalues as a function of number of elements.

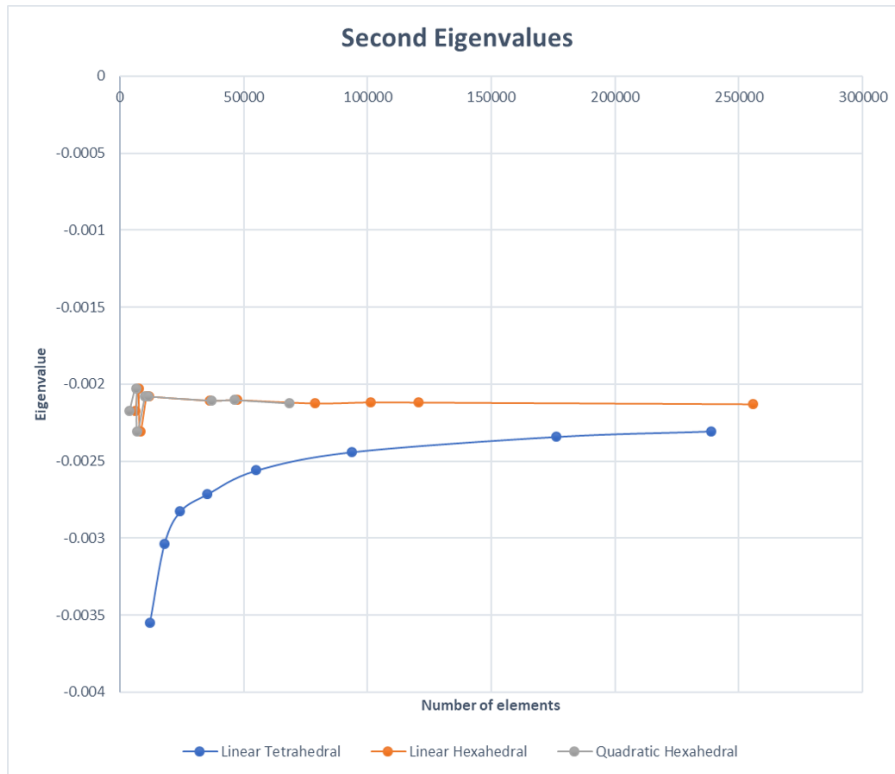


Figure 28: Second eigenvalues as a function of number of elements.

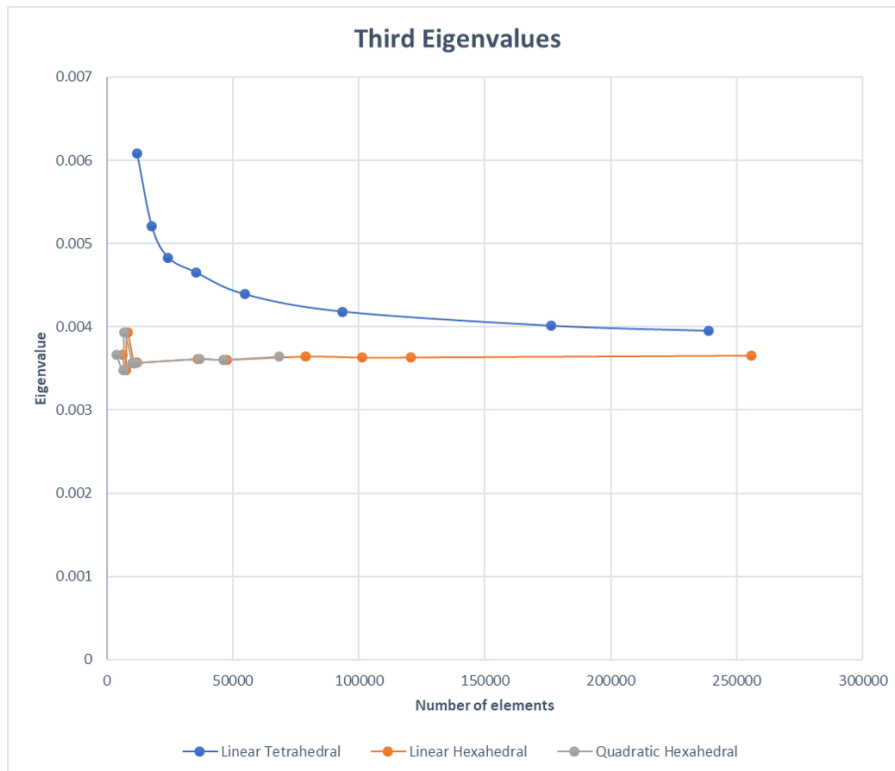


Figure 29: Third eigenvalues as a function of number of elements.

## 5 Question 5

In Ansys Workbench Mechanical, 2D surface geometry can not be divided into layers. So in this section, 3D linear tetrahedral, hexahedral and quadratic hexahedral elements are used to compare two and four layers. Comparisons made in this section:

- **2 layers vs. 4 layers** for quadratic hexahedral (first 5 constrained mode frequencies),
- **Linear hexahedral vs. quadratic hexahedral** for 4 layers (4. and 5. constrained mode frequencies)
- **Linear tetrahedral vs. linear hexahedral** for 4 layers (1., 2., 4. and 5. constrained mode frequencies)

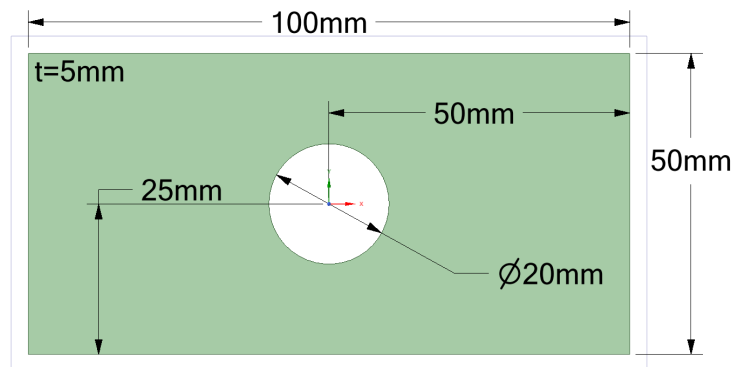


Figure 30: Geometry used in Question 5.

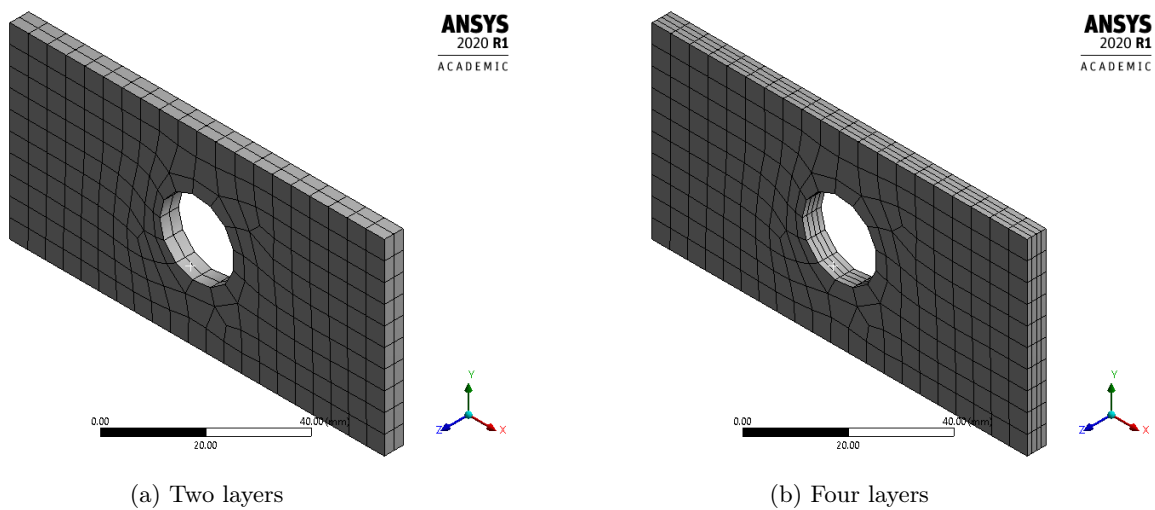


Figure 31: Example for the two and four layers used in Question 5 (5 mm element size) (Linear hexahedral).



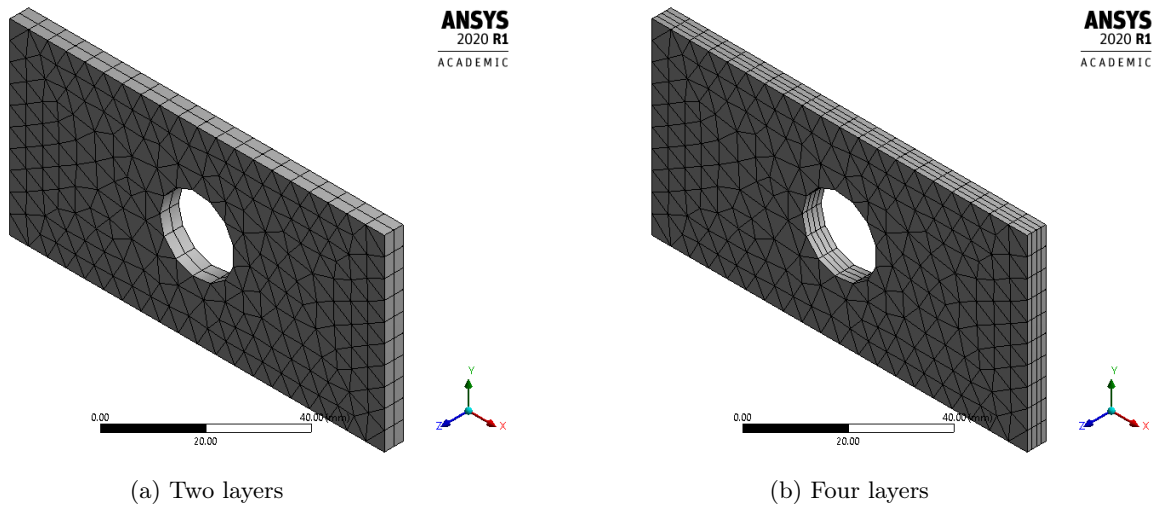


Figure 32: Example for the two and four layers used in Question 5 (5 mm element size) (Linear tetrahedral).

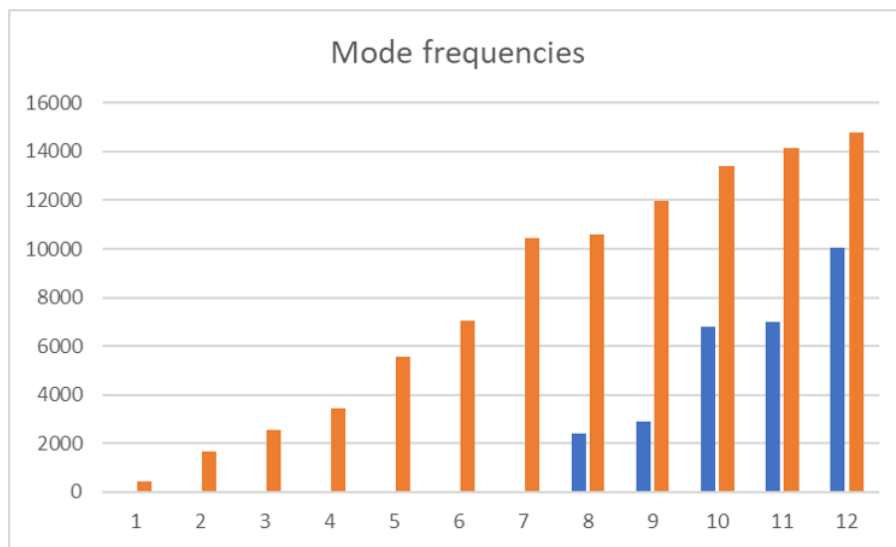


Figure 33: First 12 natural frequencies for normal (blue) and constrained (orange) modes.

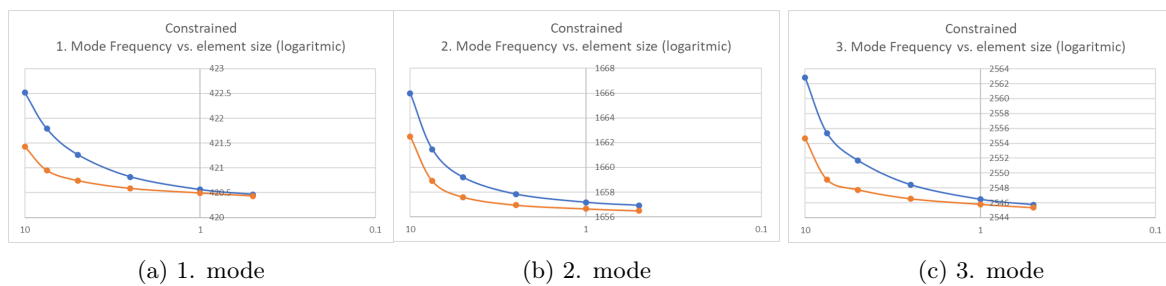
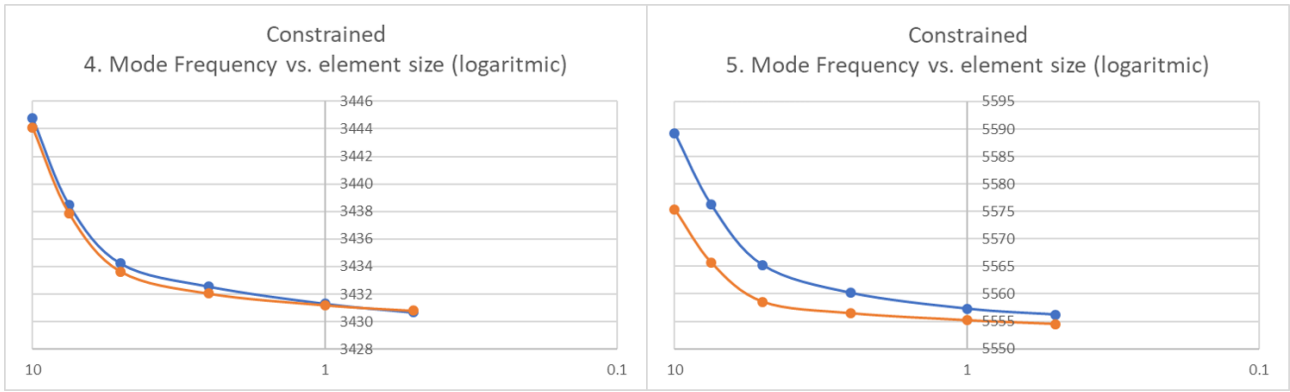


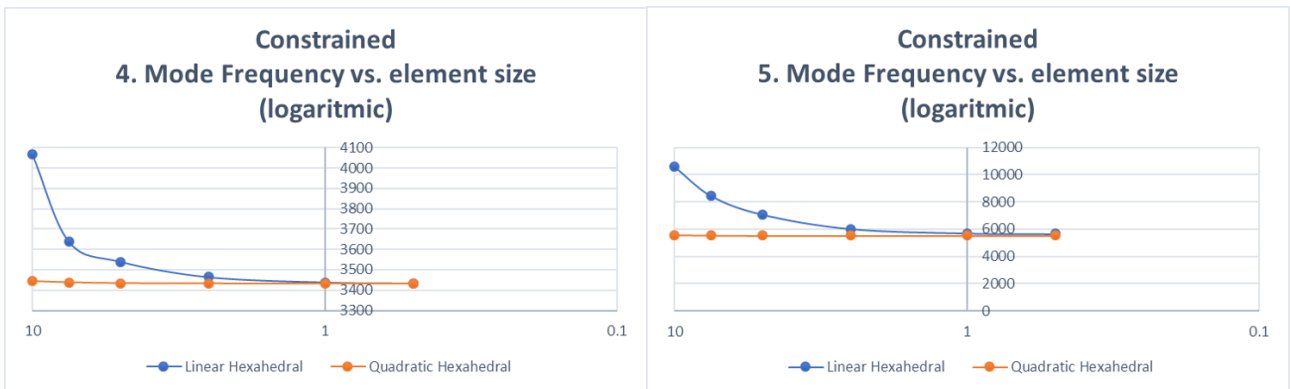
Figure 34: 2 layers (blue) vs. 4 layers (orange) for quadratic hexahedral.



(a) 4. mode

(b) 5. mode

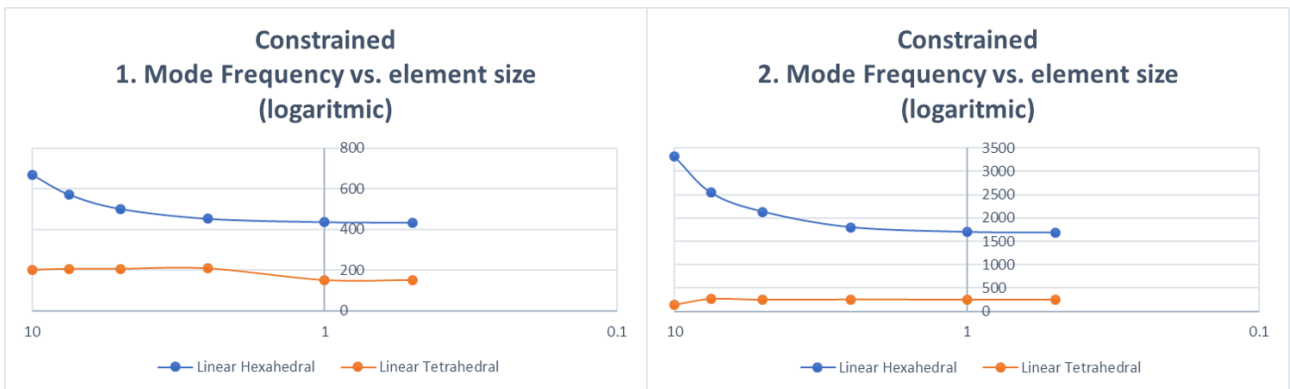
Figure 35: **2 layers (blue) vs. 4 layers (orange)** for quadratic hexahedral.



(a) 4. mode

(b) 5. mode

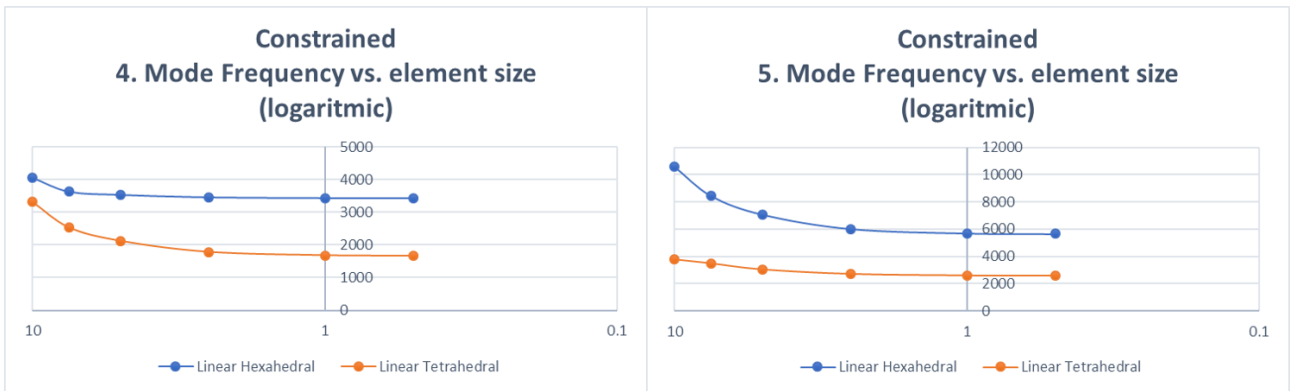
Figure 36: **Linear hexahedral vs. quadratic hexahedral** for 4 layers.



(a) 1. mode

(b) 2. mode

Figure 37: **Linear tetrahedral vs. linear hexahedral** for 4 layers.



(a) 4. mode

(b) 5. mode

Figure 38: Linear tetrahedral vs. linear hexahedral for 4 layers.

## 5.1 Deformations

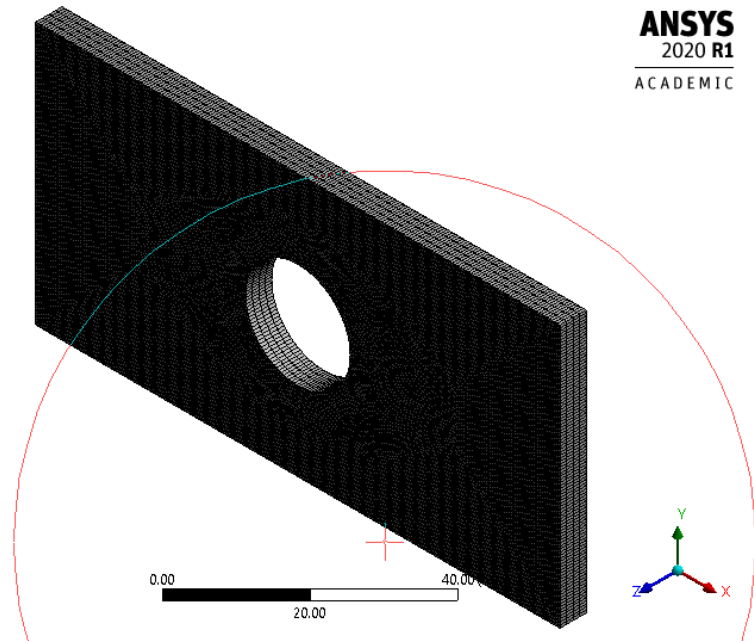


Figure 39: Deformations are given with this mesh (5 mm Linear tetrahedral).

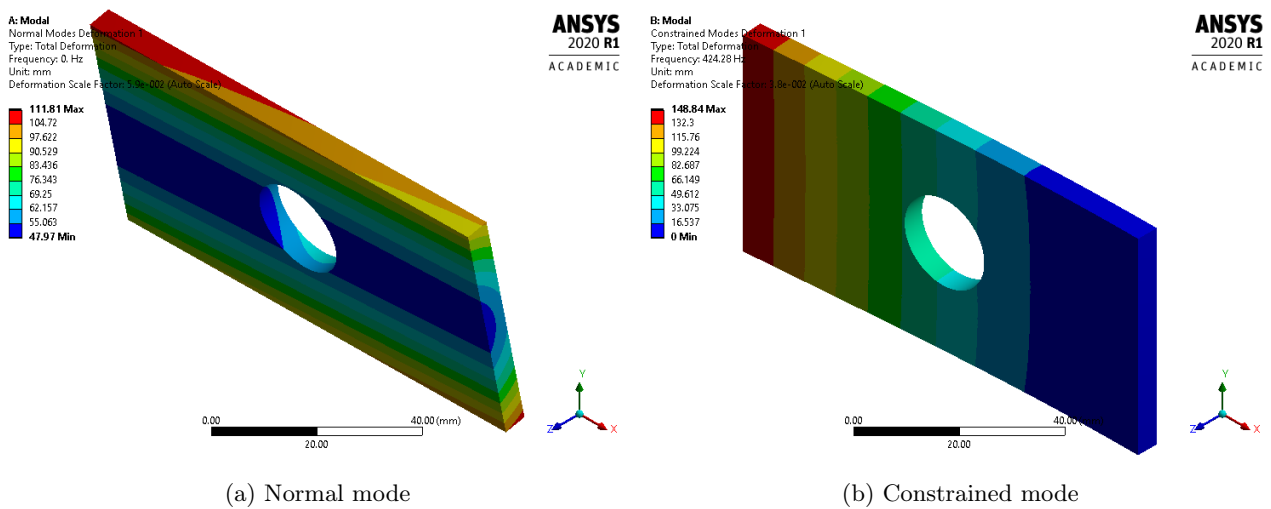
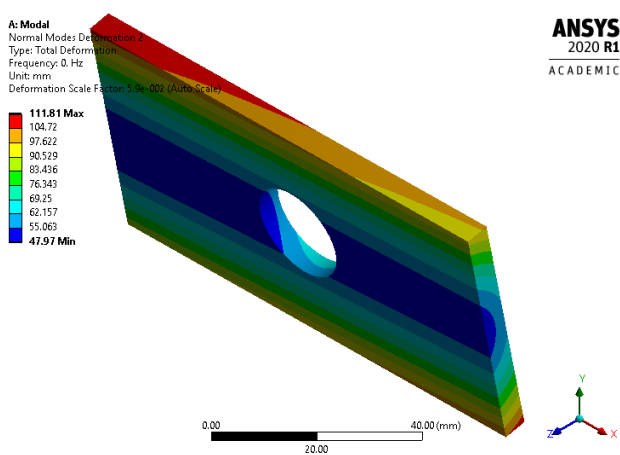
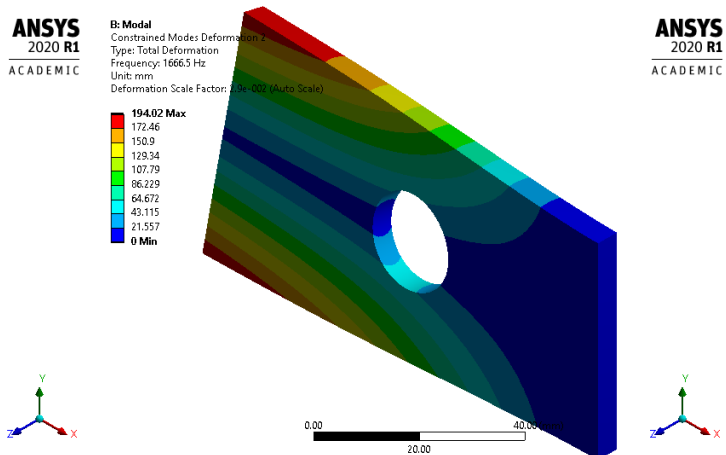


Figure 40: Deformation for 1. modes (Linear tetrahedral - four layer).

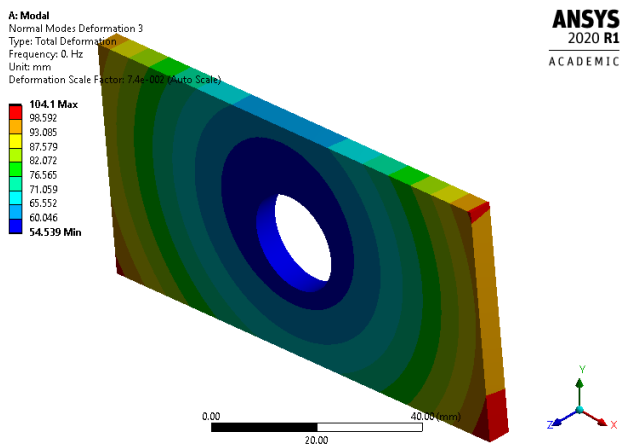


(a) Normal mode

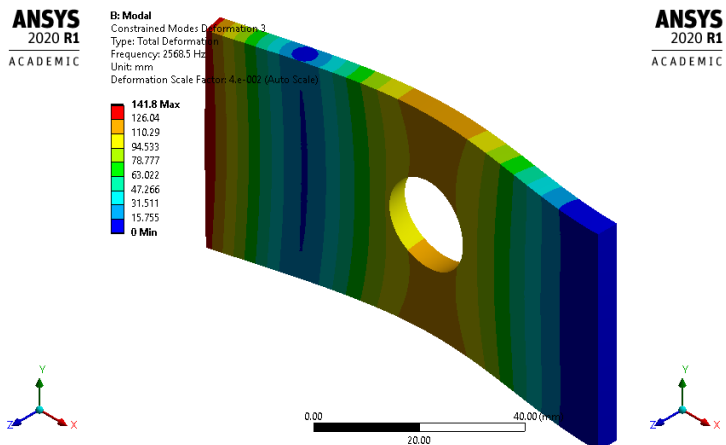


(b) Constrained mode

Figure 41: Deformation for 2. modes (Linear tetrahedral - four layer).

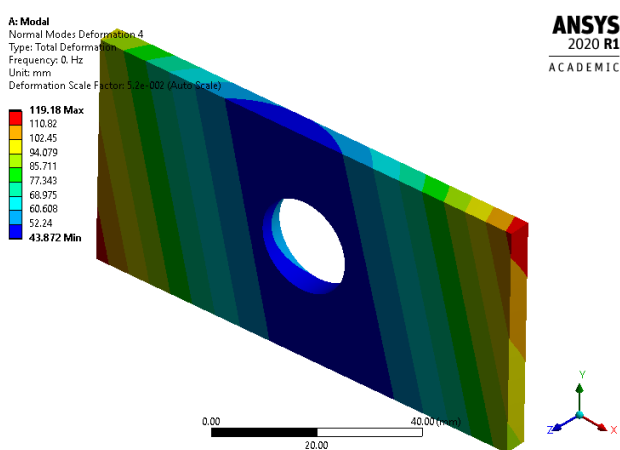


(a) Normal mode

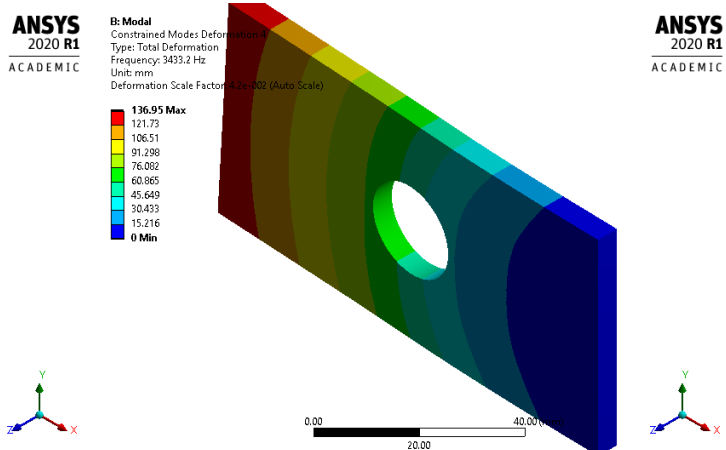


(b) Constrained mode

Figure 42: Deformation for 3. modes (Linear tetrahedral - four layer).

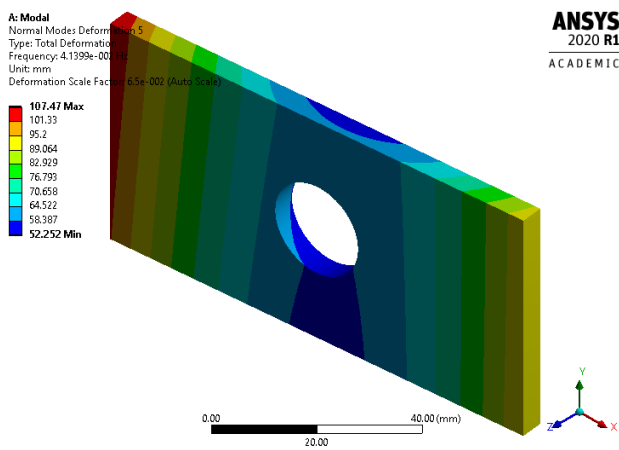


(a) Normal mode

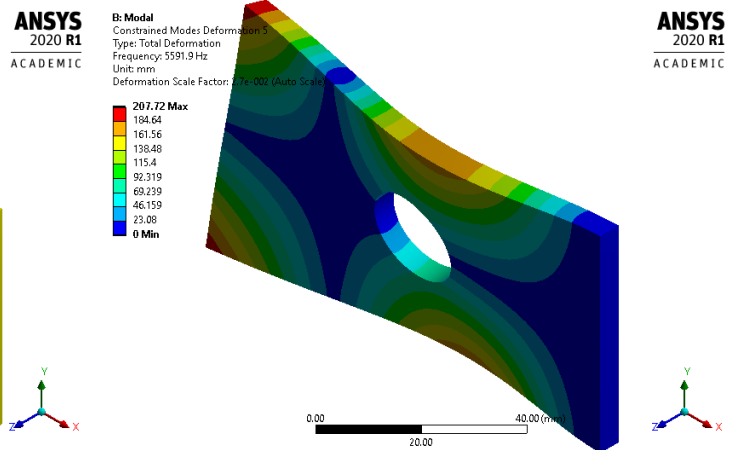


(b) Constrained mode

Figure 43: Deformation for 4. modes (Linear tetrahedral - four layer).

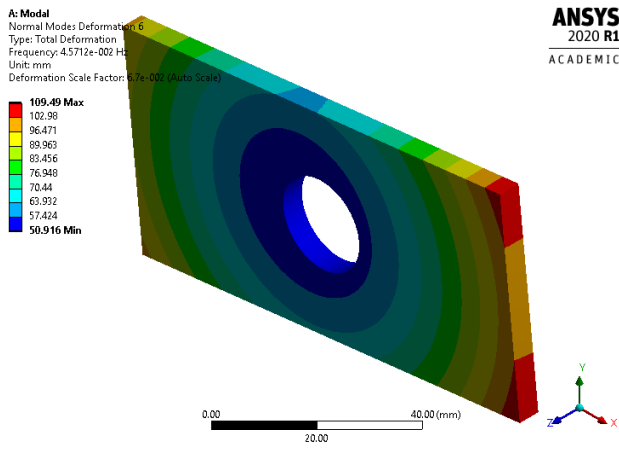


(a) Normal mode

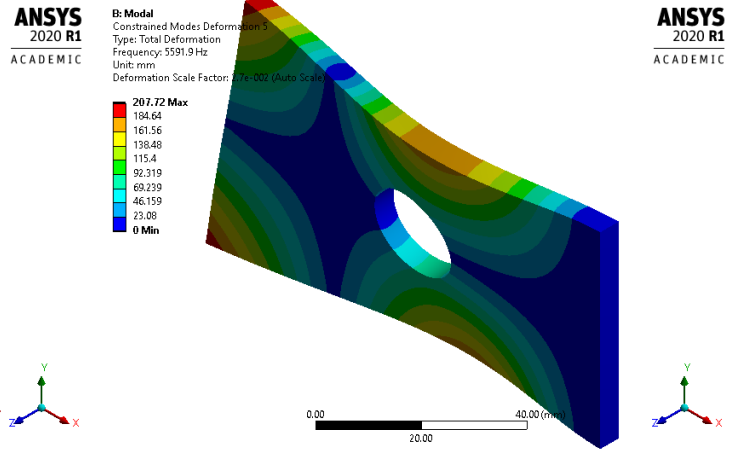


(b) Constrained mode

Figure 44: Deformation for 5. modes (Linear tetrahedral - four layer).

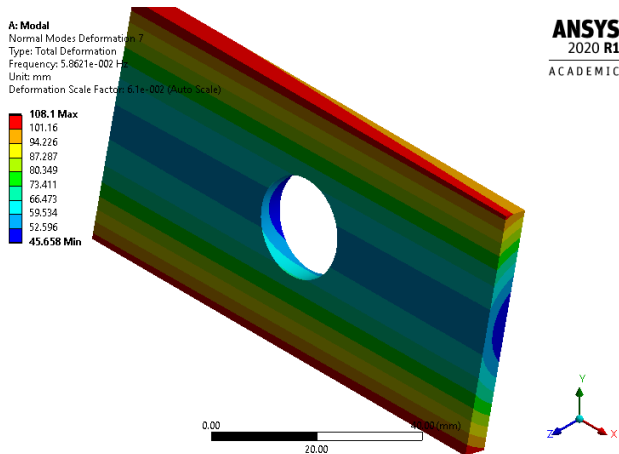


(a) Normal mode

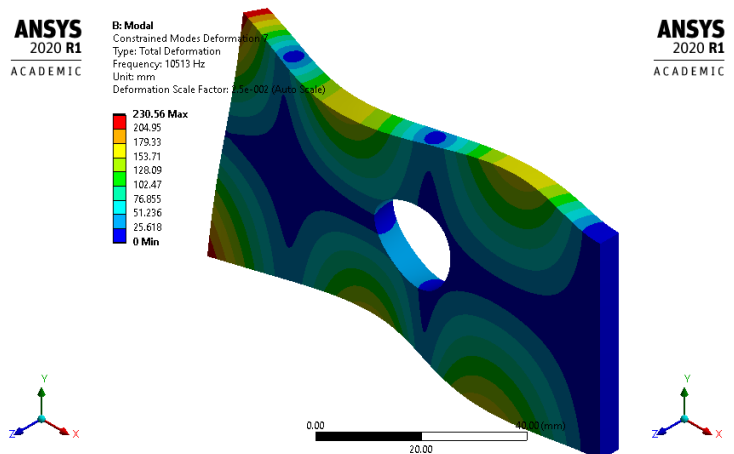


(b) Constrained mode

Figure 45: Deformation for 6. modes (Linear tetrahedral - four layer).

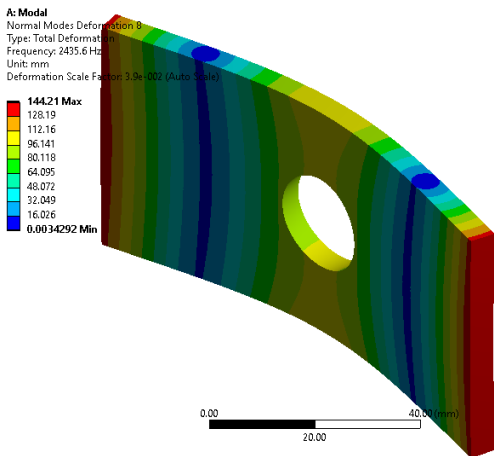


(a) Normal mode

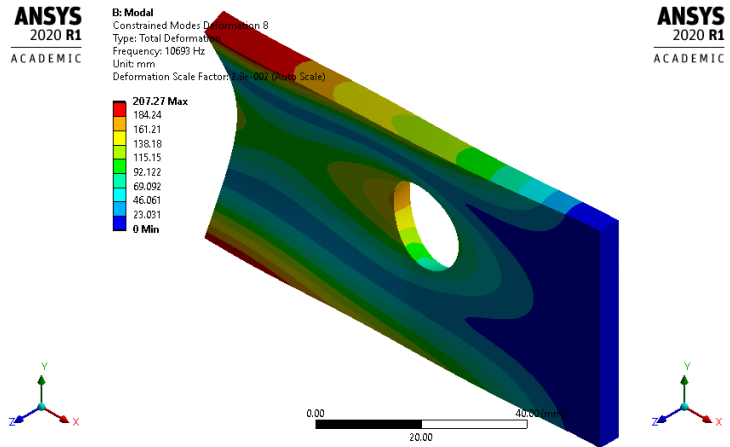


(b) Constrained mode

Figure 46: Deformation for 7. modes (Linear tetrahedral - four layer).

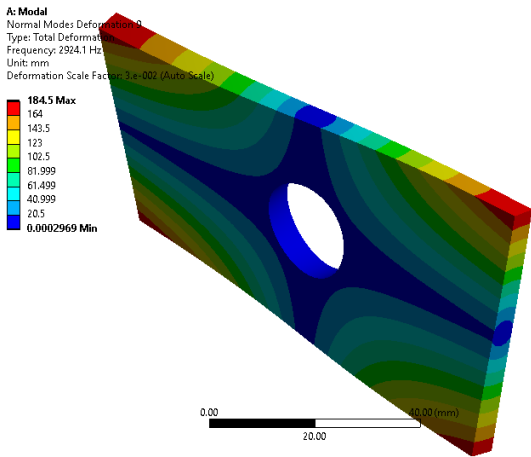


(a) Normal mode

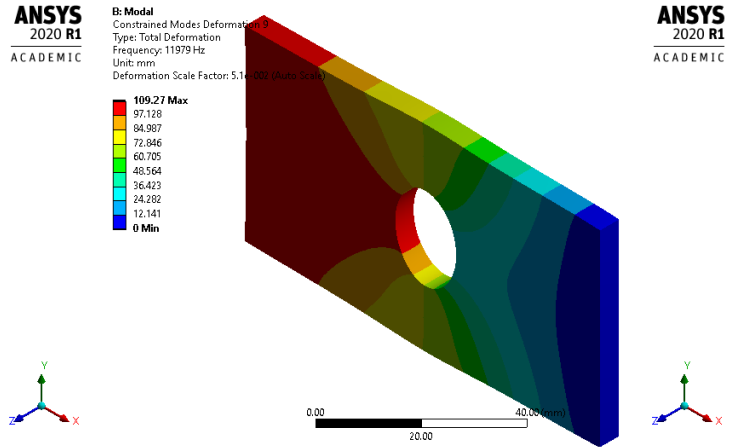


(b) Constrained mode

Figure 47: Deformation for 8. modes (Linear tetrahedral - four layer).

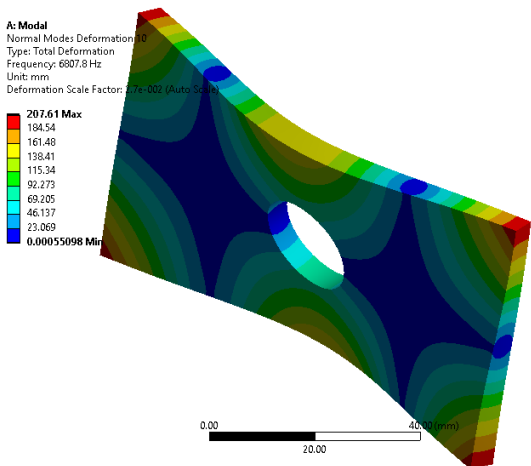


(a) Normal mode

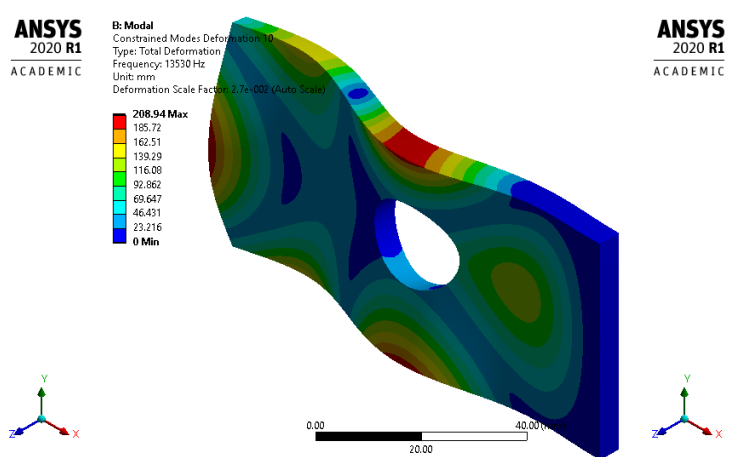


(b) Constrained mode

Figure 48: Deformation for 9. modes (Linear tetrahedral - four layer).



(a) Normal mode



(b) Constrained mode

Figure 49: Deformation for 10. modes (Linear tetrahedral - four layer).