TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 5

SOLUTION PROPOSAL

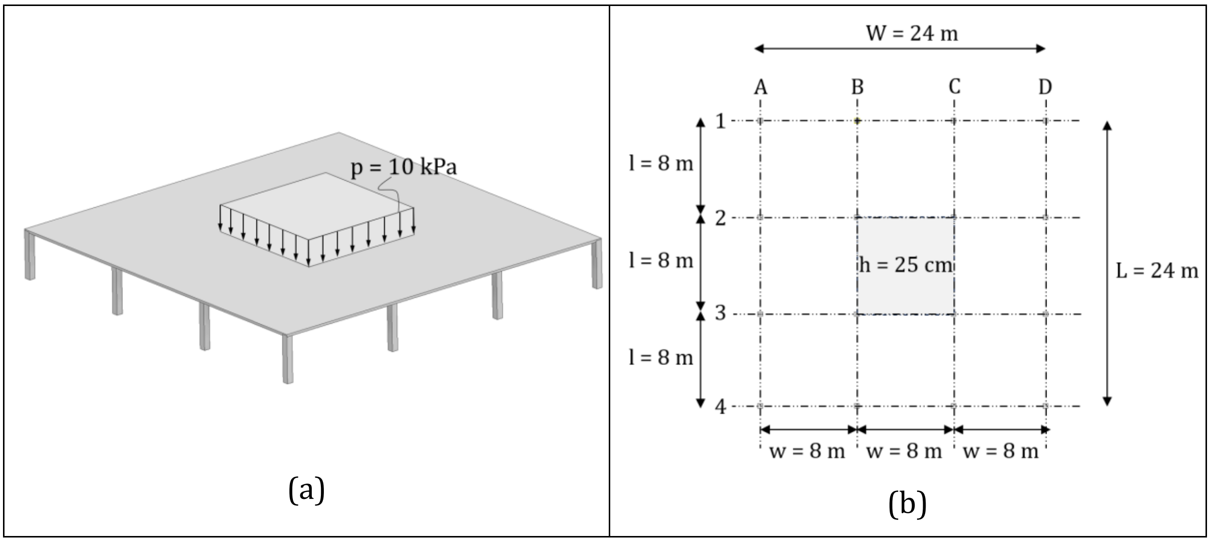
Case Study 5 will use a concrete slab to address the modelling of plate problems. Slabs are plane structures and plate elements are representative of pure bending problems. In plate problems, the thickness t is small compared to its other length dimensions L (e.g., $1/10 < h/L < 1/3$). A workshop on how to model the different aspects addressed in this case study is uploaded to Blackboard (see "*Workshop5.pdf*" in the folder "Case studies"). In Task 1, we will start by modeling a slab with uniformly distributed loading before representing this uniform loading as several point loads distributed over the surface area of the slab in Task 2.

Learning outcome:

- Modelling of plate problems
- Convergence studies
- Visualization and post-processing of results in Abaqus/CAE

Problem description

Figure 1 shows the rooftop of a car park. The rooftop is to be considered as a concrete slab with thickness h = 25 cm. The entire rooftop should be evaluated for a load case where the part of the rooftop between axes B, C, 2, and 3 is loaded with a total weight of 6 cars (including passengers). The loaded area is illustrated as the shaded area in Figure 1b. Assume that the rooftop is supported by the columns in the vertical direction. Also assume that the slab is loaded by a uniformly distributed pressure (p) of 10 kPa on the top surface. The material properties correspond to a concrete of class B30. Assume that the concrete material remains uncracked during deformation.



(Figure 1) – A rooftop supported by 16 columns and exposed to a uniform load p = 10 kPa between axes B, C, 2, and 3.

Load: $p = -0.010\ \text{N/mm}^2$ (downwards)
Material data: $E = 32\,000\ \text{N/mm}^2, \nu = 0.20\ , \rho = 2500\ \text{kg/m}^3, \sigma_y = 20\ \text{N/mm}^2$

Task 1

a) The concrete slab should be modeled using 4-node shell elements (S4 in Abaqus) and 800 mm characteristic size. Report your model in Abaqus by generating a figure of the model.

Solution:

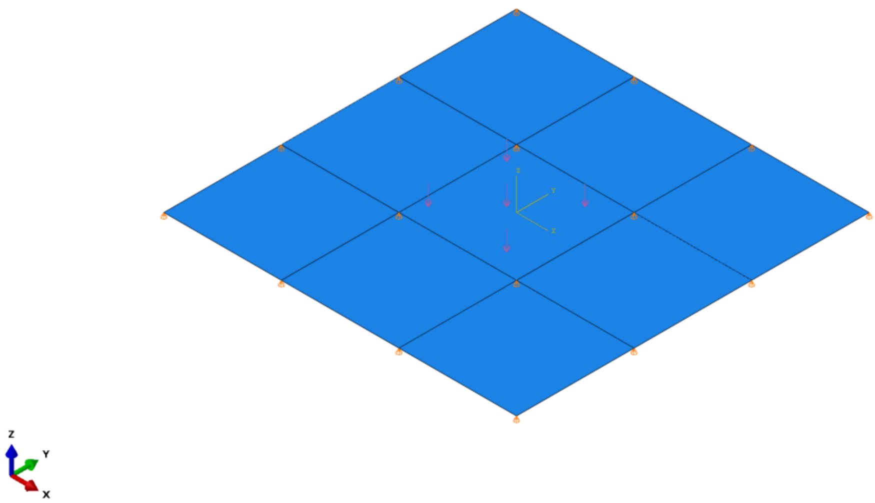


Figure 1: A plot of the model in the Load Module.

b) Run a simulation in Abaqus using the file established in a). View the analysis results in the visualization module. From the simulation, take out and report the following information:

1) The deformed shape of the rooftop on top of the undeformed shape.
2) Contour plot of the vertical displacement on the deformed shape.
3) Contour plot of the von Mises stress on the deformed shape.
4) Contour plot of the maximum and minimum principal stresses on the deformed shape.
5) What is the critical element(s) of the structure? How do the stresses compare to the elastic limit of the material ($\sigma_y = 20\ \mathrm{N/mm^2}$)?
6) Show the stress distribution over the cross section of the critical element(s).

Solution:

Note: all plots of the concrete slabs are seen from under the structure.

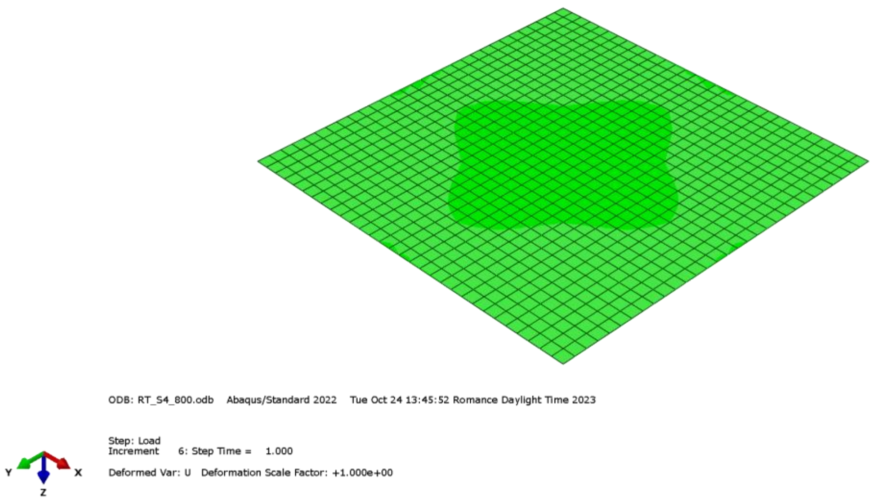


Figure 2: Deformed shape on top of the undeformed shape. Note: The displacement scale factor is set to 1.

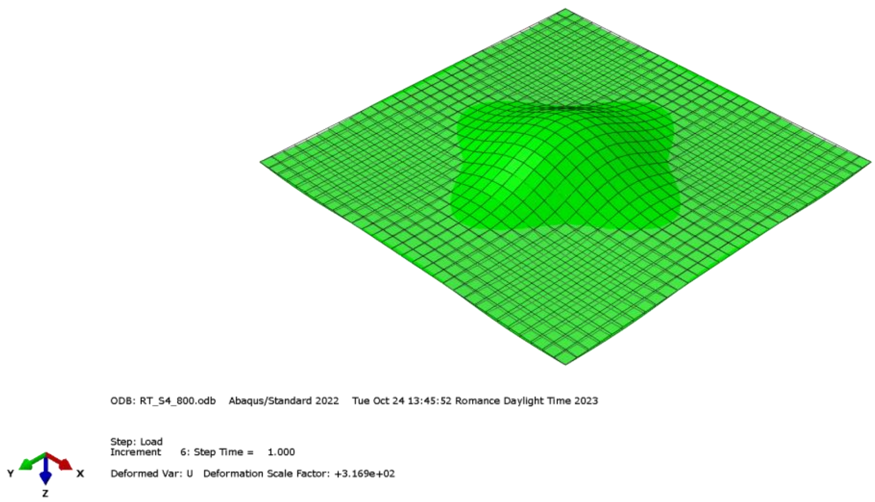


Figure 3: Deformed shape on top of the undeformed shape. Note: The displacement scale factor is set to 317.

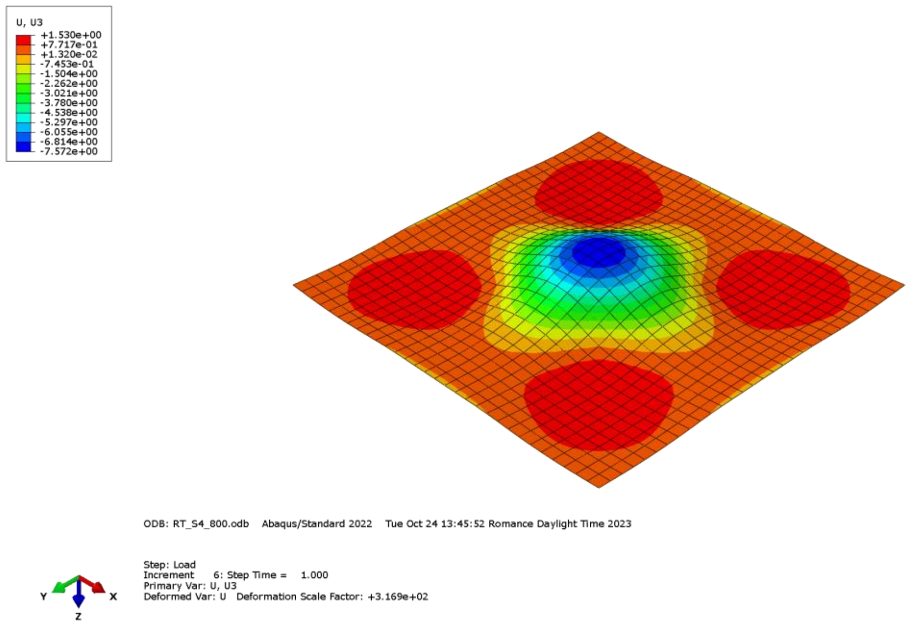


Figure 4: Vertical displacement U3 of the concrete slab. Note: The displacement scale factor is set to 317.

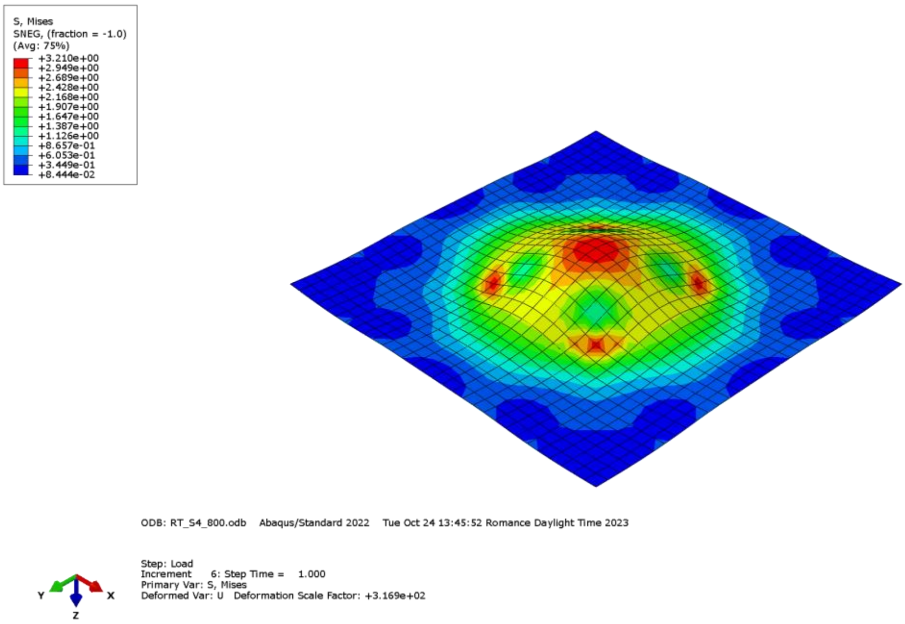


Figure 5: Averaged (75%) von Mises stress on the concrete slab. Note: The displacement scale factor is set to 317.

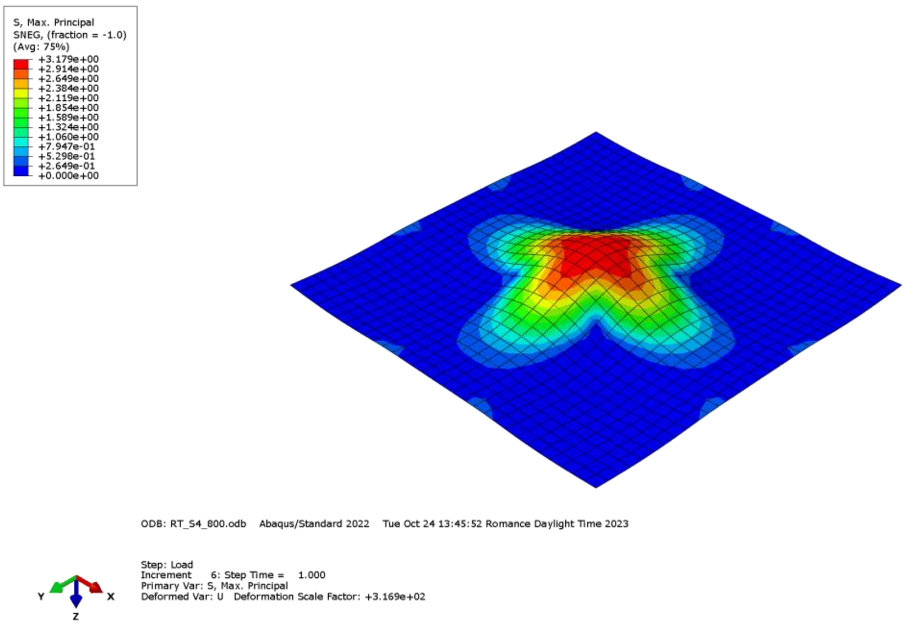


Figure 6: Maximum principal stress on the concrete slab. Note: The displacement scale factor is set to 317.

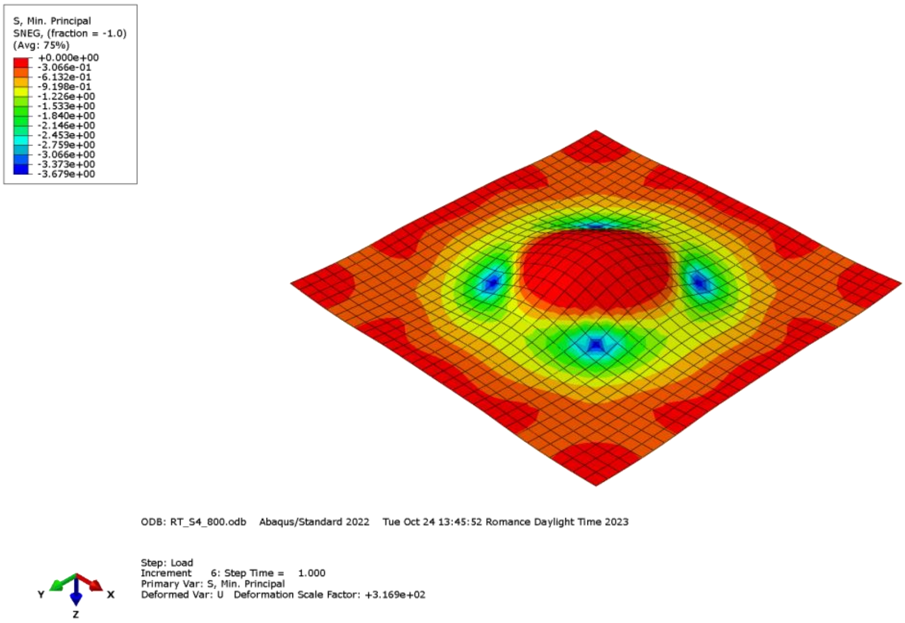


Figure 7: Minimum principal stress on the concrete slab. Note: The displacement scale factor is set to 317.

We will now examine the critical elements in the model.

We can examine the maximum and minimum principal stresses of the concrete to find the critical elements. From the contour plots in Figure 6 and Figure 7, we see that there are two zones we should investigate; the maximum positive stress (from maximum principal stress) and the maximum negative stress (minimum principal stress). By using the Contour Plot Options, we can indeed locate the critical elements.

Maximum principal stress (tensile stress).

We can use the probe tool to find the calculated values in the four elements in the centre, and we find that the stress value is about 3.18 MPa, which is well below the elastic limit of the concrete (20 MPa). However, we know that concrete's tensile capacity is about 1/10 of it's compressive strength. Thus, if we assume that the tensile capacity is 2 MPa, we have exceeded the tensile capacity of the material. In reality, this is taken care of by the reinforcement. A common assumption when modelling concrete in the elastic area is that all tensile strains and stresses occur in the reinforcement.

Minimum principal stress (compressive stress).

We can use the probe tool to find the calculated values in the four elements above the supports, and we find that the stress value is about 3.70 MPa, which is well below the elastic limit of the concrete (20 MPa).

Stress distribution over the critical elements.

Due to symmetry in the model, we only need to plot the principal stress over the thickness for two elements; one in the centre and one over the supports.

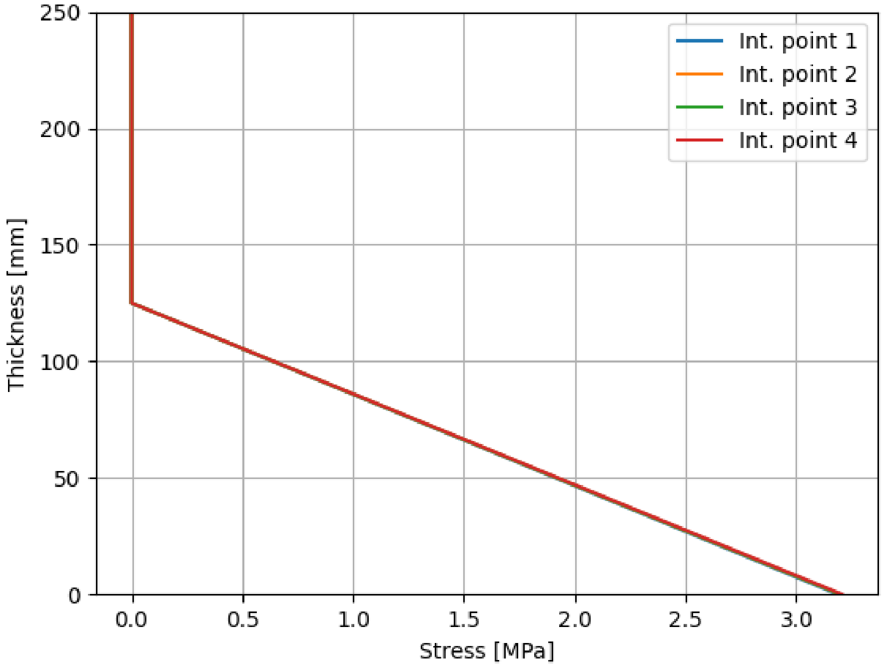


Figure 8: Maximum principal stress over the thickness for one of the four centre elements.

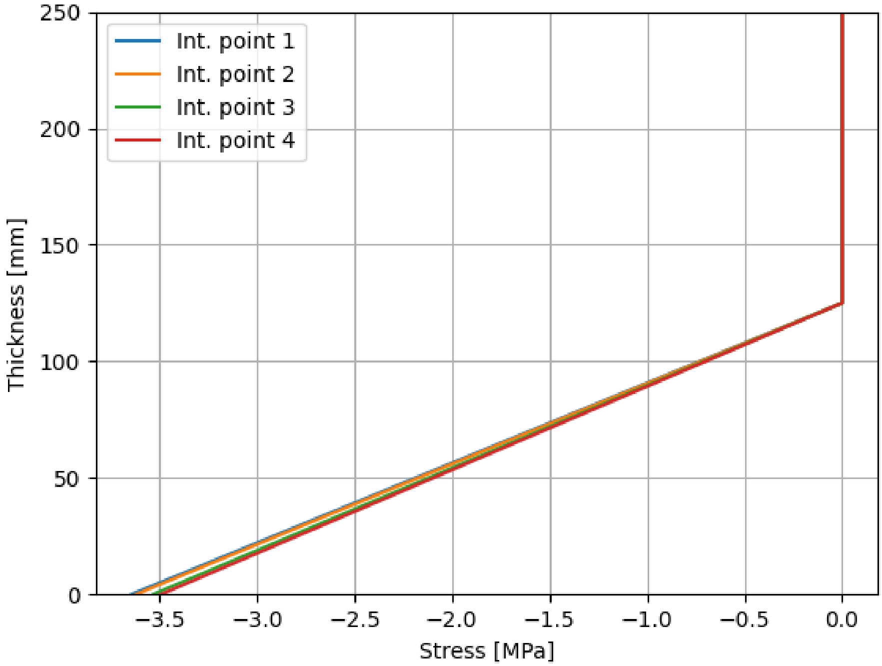


Figure 9: Minimum principal stress over the thickness for one of the four support elements.

We see that we have a "jump" in both stress distributions when the stress is equal to zero. We can explain this using the definition of the maximum and minimum principal stresses. For shell elements, $\sigma_3 = 0$ by definition. Thus, the maximum principal stress is equal to

$$\sigma_{\max} = \max(\sigma_1, \sigma_2, \sigma_3) = \max(\sigma_1, \sigma_2, 0)$$

This explains why the stresses cannot be negative in Figure 8. The same logic can be applied to the element over the support; the minimum principal stress is equal to

$$\sigma_{\min} = \min(\sigma_1, \sigma_2, \sigma_3) = \min(\sigma_1, \sigma_2, 0)$$

The minimum principal stress can thus not be higher than 0 for shell elements.

We can also plot the absolute value of the maximum principal stress to get the bending stress distribution over the entire thickness.

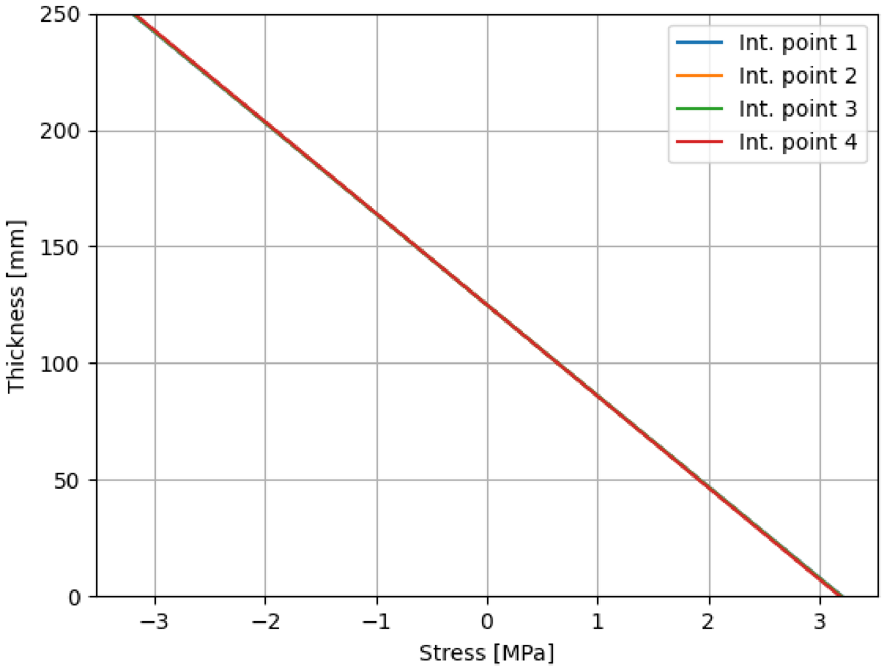


Figure 10: Maximum principal stress (absolute value) over the thickness for one of the four centre elements.

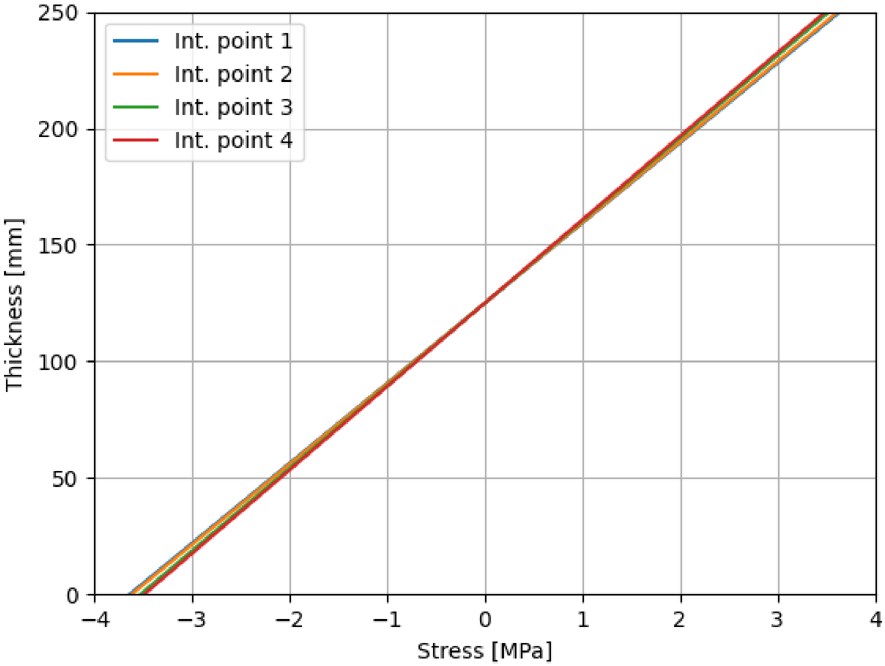


Figure 11: Maximum principal stress (absolute value) over the thickness for one of the four support elements.

c) Perform a convergence study and evaluate the maximum bending stress and the maximum displacement of the rooftop based on finite element analysis (FEA) of both linear and quadratic, triangular and quadrilateral shell elements. Explain how the convergence study has been conducted.

Solution:

We can perform a convergence study by varying the global element size from 800 to 400 and 200. We can also investigate four element types that are commonly used for plates; S3, S4R, S4 and S8R. We report the maximum vertical displacement, as well as the maximum and minimum principal stresses for the 12 analyses.

Maximum vertical displacement (in negative z-direction).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	7.44	7.60	7.57	7.75
400	7.66	7.72	7.71	7.76
200	7.73	7.76	7.75	7.77

Maximum principal stress

We collect the maximum principal stress as an average of the integration points in one plane in each element (same as using Quilt in Abaqus CAE).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	3.20	3.19	3.20	3.24
400	3.26	3.26	3.26	3.27
200	3.28	3.28	3.28	3.28

Minimum principal stress

We collect the minimum principal stress as an average of the integration points in one plane in each element (same as using Quilt in Abaqus CAE).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	3.86	3.73	3.57	3.66
400	5.04	4.97	4.78	4.84
200	6.24	6.19	5.99	6.03

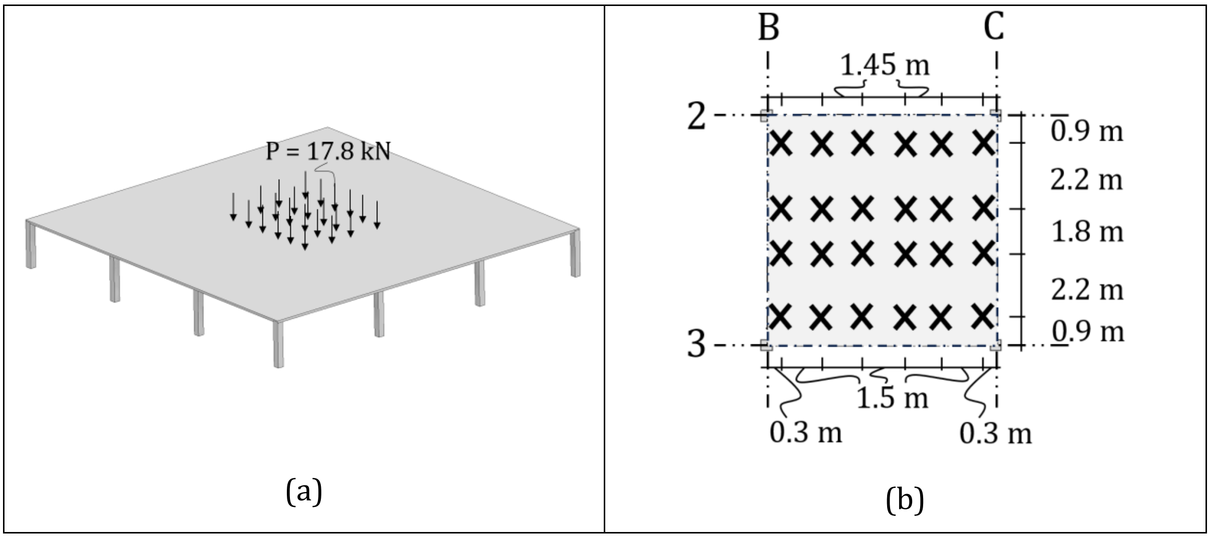
d) Assume now that the concrete material will experience cracks during the deformation. How does this change your modelling strategy for simulation of the displacements?

Solution:

When cracking occurs in a concrete slab, a common approach is to scale the bending stiffness EI of the slab. We can vary the Young's modulus or the thickness for the cracked cross-section and see how this affects our displacements. This cracking is denoted the transition between Stage 1 and 2 for the concrete.

Task 2

We will now replace the uniform load by points loads P = 17.8 kN representing the forces imposed on the rooftop by the car tires (see Figure 2). However, the structural characteristics and material properties remain unchanged.



(Figure 2) – A rooftop supported by 16 columns and exposed to 24 points loads P = 17.8 kN representing the forces imposed on the rooftop by the car tires.

a) The concrete slab should be modeled using 4-node shell elements (S4 in Abaqus) and 800 mm characteristic size. Report your model in Abaqus by generating a figure of the model.

Solution:

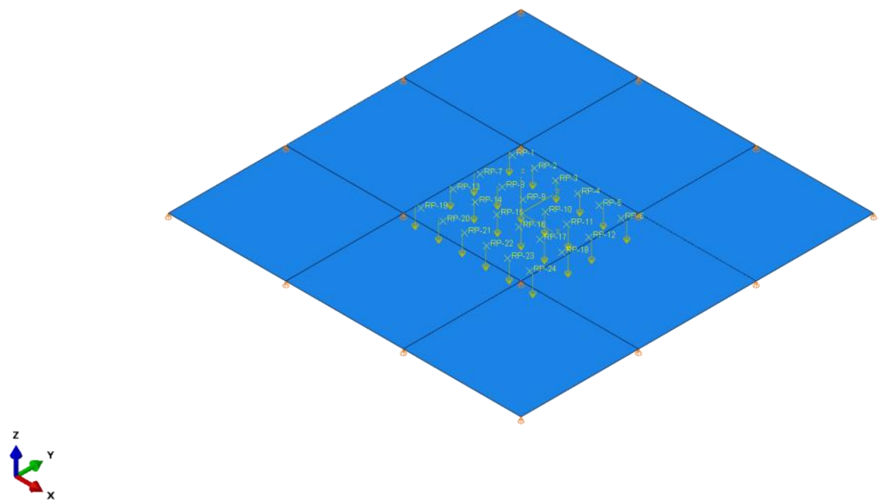


Figure 12: A plot of the model in the Load Module.

b) Run a simulation in Abaqus using the file established in a). View the analysis results in the visualization module. From the simulation, take out and report the following information:

1) The deformed shape of the rooftop on top of the undeformed shape.
2) Contour plot of the vertical displacement on the deformed shape.
3) Contour plot of the von Mises stress on the deformed shape.
4) Contour plot of the maximum and minimum principal stresses on the deformed shape.
5) What are the critical elements of the structure? Do the results change from those in Task 1b? How do the stresses compare to the elastic limit of the material ($\sigma_y = 20\ \mathrm{N/mm^2}$)?
6) Show the stress distribution over the cross section of the critical element(s).

Solution:

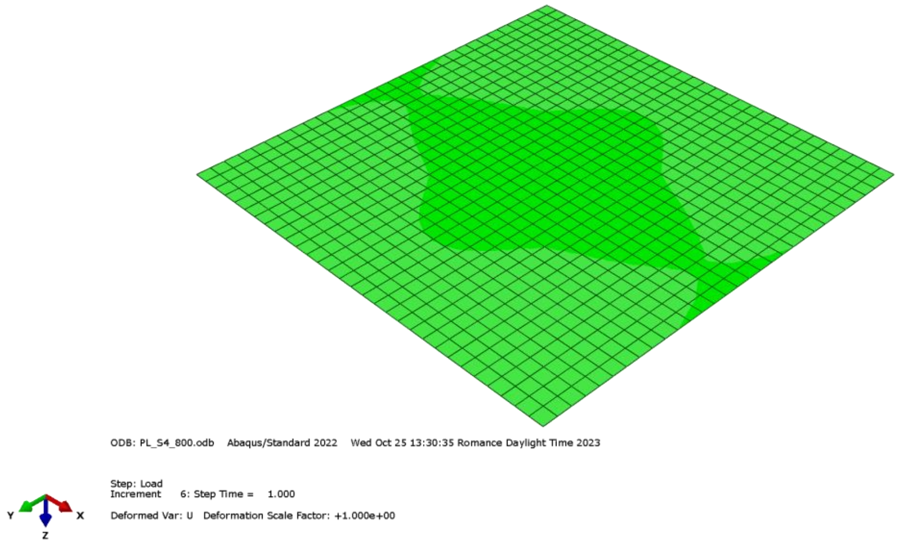


Figure 13: Deformed shape on top of the undeformed shape. Note: The displacement scale factor is set to 1.

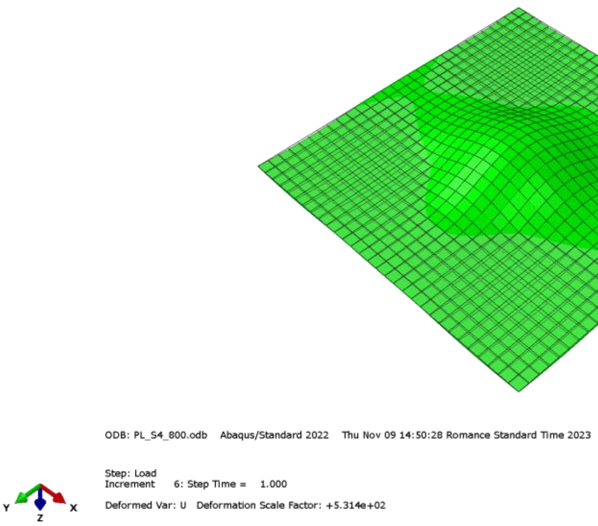


Figure 14: Deformed shape on top of the undeformed shape. Note: The displacement scale factor is set to 53.

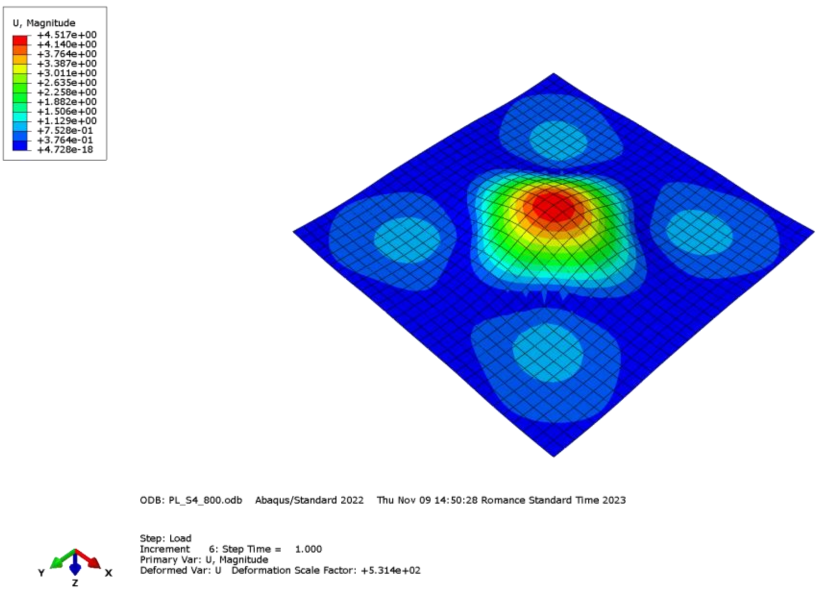


Figure 15: Vertical displacement U3 of the concrete slab. Note: The displacement scale factor is set to 53.

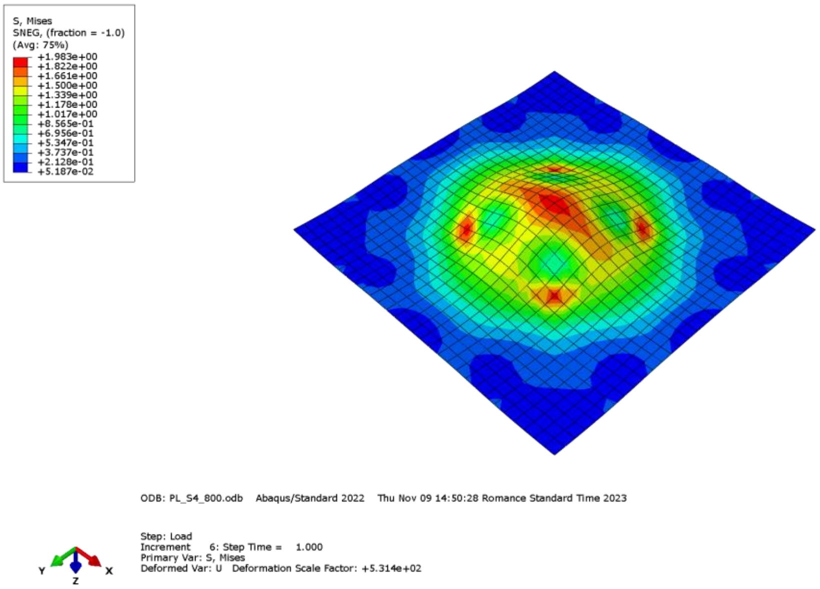


Figure 16: Averaged (75%) von Mises stress on the concrete slab. Note: The displacement scale factor is set to 53.

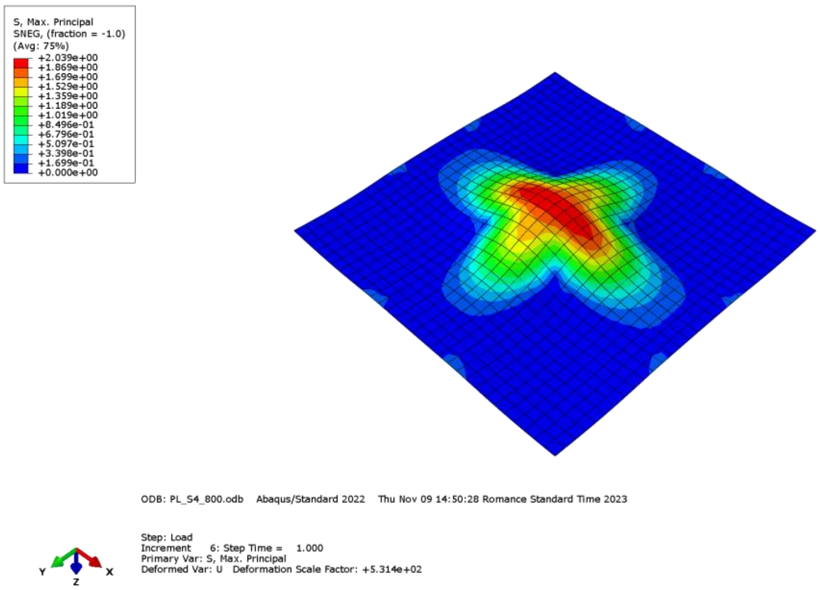


Figure 17: Averaged (75%) Maximum principal stress on the concrete slab. Note: The displacement scale factor is set to 53.

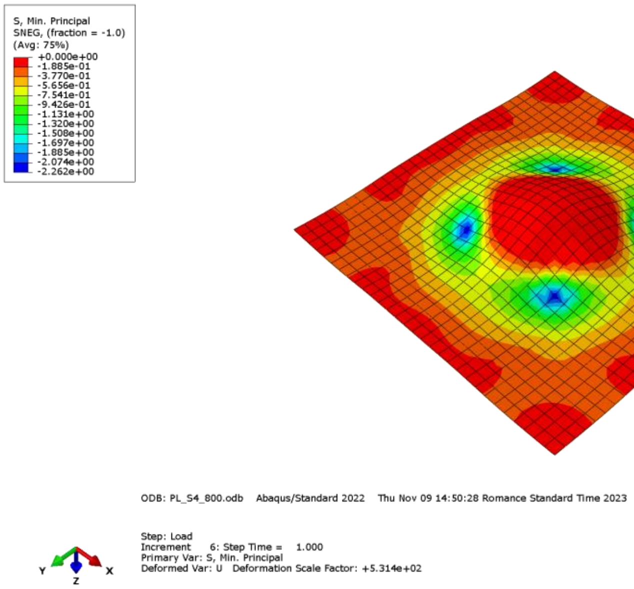


Figure 18: Averaged (75%) Minimum principal stress on the concrete slab. Note: The displacement scale factor is set to 53.

We will now examine the critical elements in the model.

As in Task 1, we can examine the maximum and minimum principal stresses of the concrete to find the critical elements. We investigate the same zones as in task 1. By using the Contour Plot Options, we can indeed locate the critical elements.

Maximum principal stress (tensile stress).

We can use the probe tool to find the calculated values in the four elements in the centre, and we find that the stress value is about 2.05 MPa.

Minimum principal stress (compressive stress).

We can use the probe tool to find the calculated values in the four elements above the supports, and we find that the stress value is about 2.25 MPa.

Stress distribution over the critical elements.

Due to symmetry in the model, we only need to plot the principal stress over the thickness for two elements; one in the centre and one over the supports.

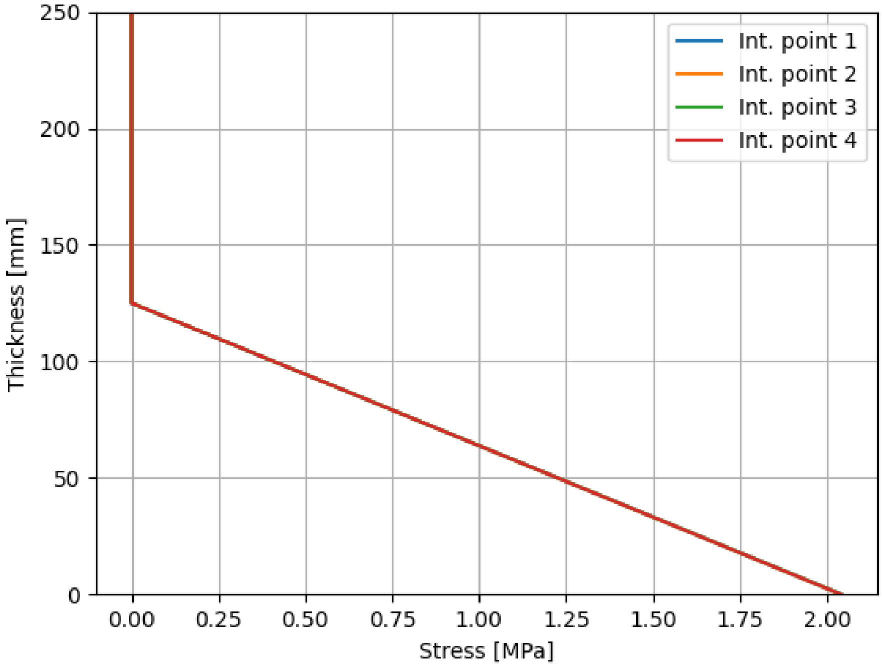


Figure 19: Maximum principal stress over the thickness for one of the four centre elements.

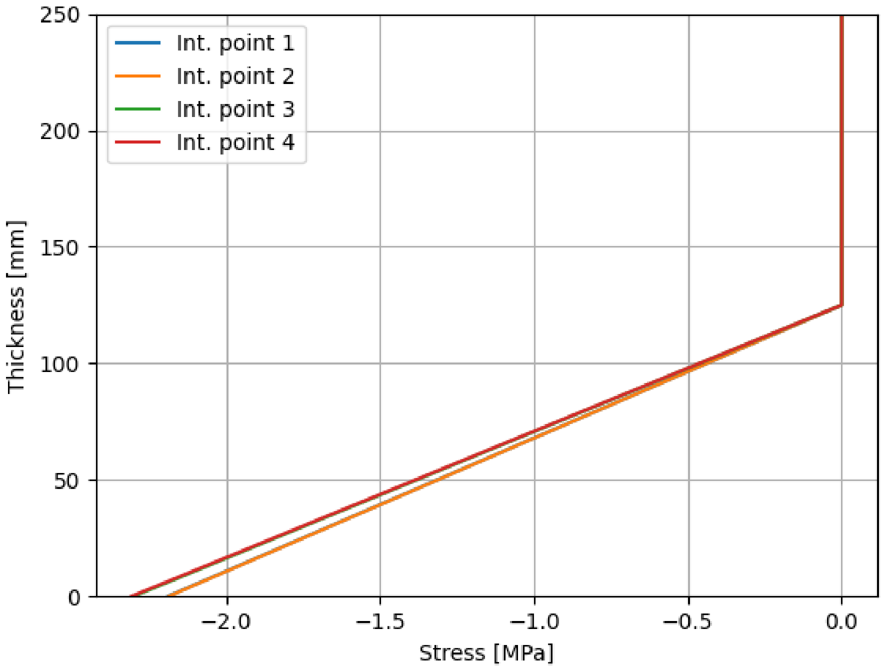


Figure 20: Minimum principal stress over the thickness for one of the four support elements.

We can also use Max. Principal (Abs) values for these two elements. Then, we can get the bending stress distributions over the entire thickness.

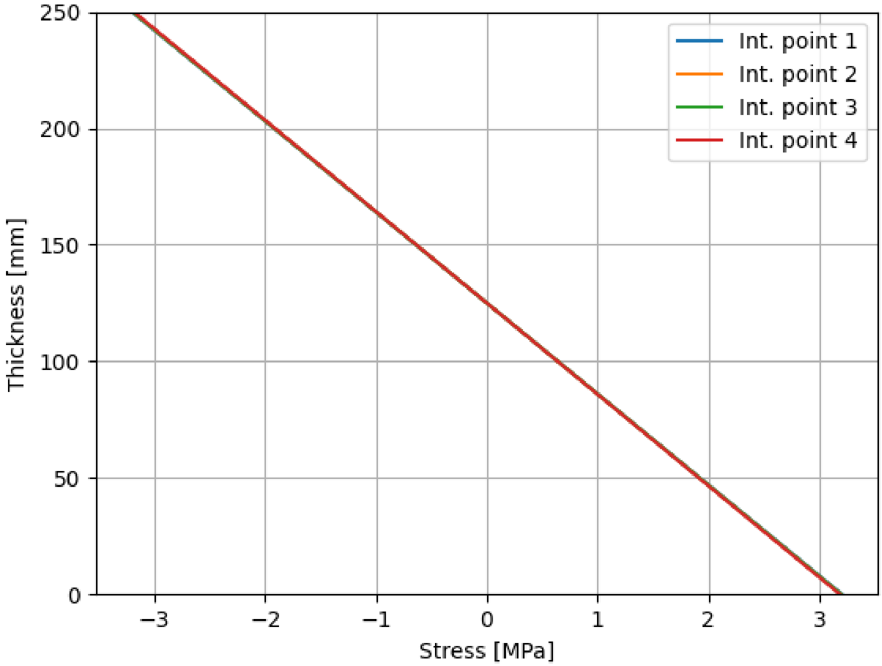


Figure 21: Maximum principal stress (absolute value) over the thickness for one of the four centre elements.

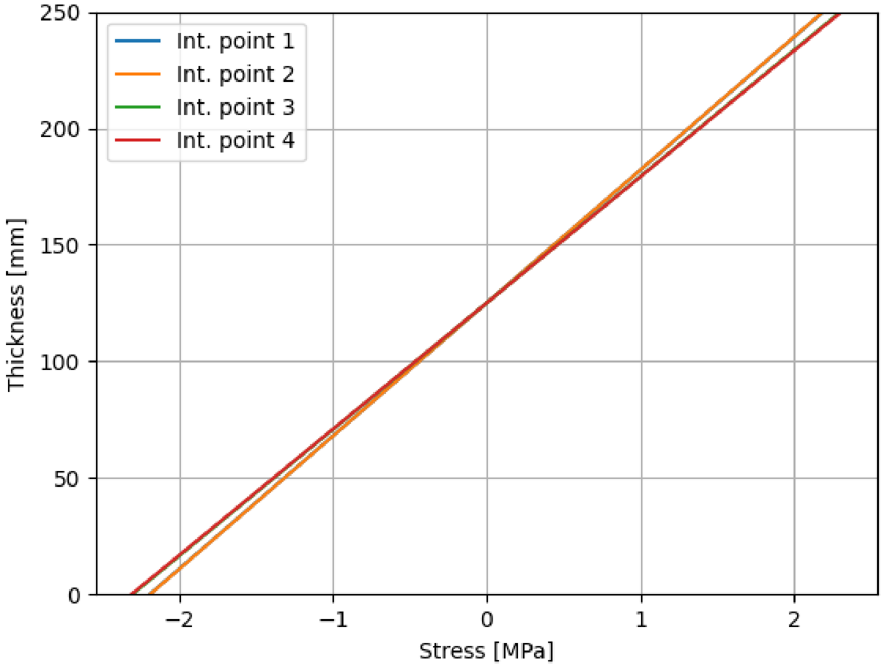


Figure 22: Minimum principal stress (absolute value) over the thickness for one of the four support elements.

Now we see that we are well within the compressive strength of the concrete (20 MPa).

c) Perform a convergence study and evaluate the maximum bending stress and the maximum displacement of the rooftop based on finite element analysis (FEA) of both linear and quadratic, triangular and quadrilateral shell elements. Explain how the convergence study has been conducted.

Solution:

We can perform a convergence study by varying the global element size from 800 to 400 and 200. We can also investigate four element types that are commonly used for plates; S3, S4R, S4 and S8R. We report the maximum vertical displacement, as well as the maximum and minimum principal stresses for the 12 analyses.

When we vary the mesh size, the new nodes that are generated will not have same locations as the previous mesh. Thus, we have to assign our concentrated loads again for 400 mm global element size and 200 mm global element size.

Maximum vertical displacement (in negative z-direction).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	4.43	4.53	4.52	4.59
400	4.96	5.00	4.99	5.01
200	5.05	5.07	5.07	5.08

Maximum principal stress

We collect the maximum principal stress as an average of the integration points in one plane in each element (same as using Quilt in Abaqus CAE).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	2.05	2.04	2.04	2.05
400	2.24	2.24	2.24	2.23
200	2.38	2.38	2.37	2.36

Minimum principal stress

We collect the minimum principal stress as an average of the integration points in one plane in each element (same as using Quilt in Abaqus CAE).

Element size	Linear			Quadratic
	Triangular (S3)	Quadrilateral (S4R)	Quadrilateral (S4)	Quadrilateral (S8R)
800	2.37	2.35	2.25	2.28
400	3.32	3.26	3.14	3.17
200	4.14	4.12	3.98	4.01