TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 1

SOLUTION PROPOSAL

Case Study 1 gives a first introduction to Abaqus. A workshop on how to model the different aspects addressed in this case study is uploaded to Blackboard (see "*Workshop1.pdf*" in the folder "Case studies"). We will start by modeling a simple cantilever beam before moving to a more complicated geometry and boundary conditions by introducing a whole in the beam.

Learning outcome:

- Introduction to Abaqus/CAE
- Modelling of plane stress problem
- Convergence study of a plane stress problem
- Visualization and post-processing of results in Abaqus/CAE

Problem description

Figure 1 shows a wooden beam that is fully embedded in a concrete wall at one end and supported by a steel rod on the opposite end. The steel rod supports the beam through a hole at the end of the beam. The beam is loaded by a uniformly distributed pressure (p) of 25 kPa on the top surface (as shown in Figure 1).

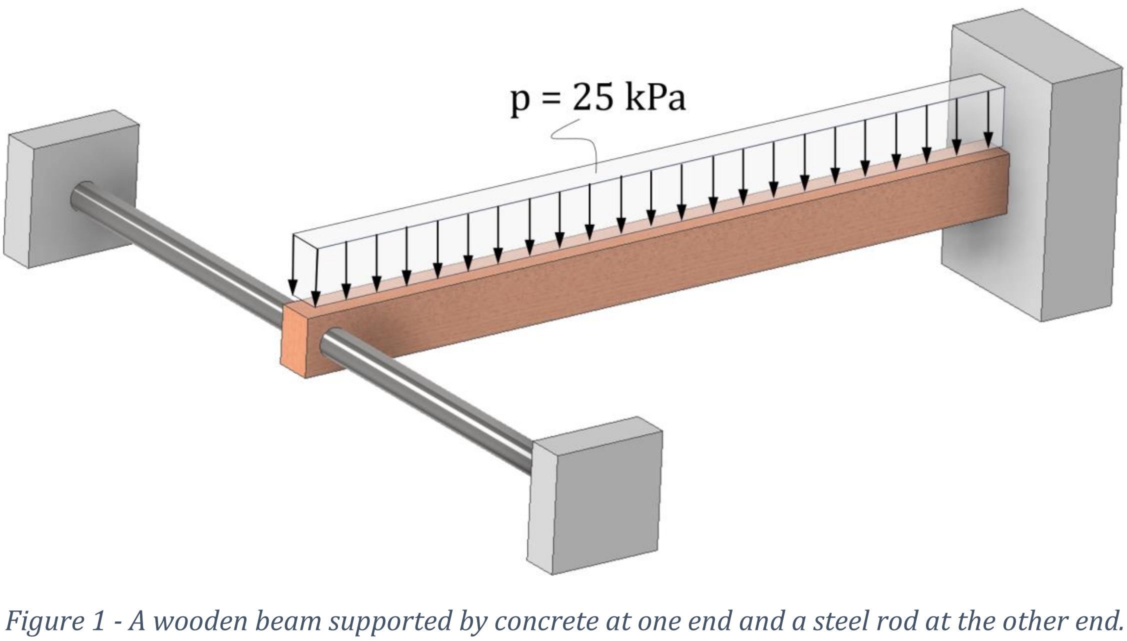


Figure 1 - A wooden beam supported by concrete at one end and a steel rod at the other end.

We will start by modeling the cantilever beam in Figure 2, which is a simplified version of the structural system in Figure 1. The beam is embedded to a concrete wall at the right end and the dimensions are shown in Figure 2. We will assume that the beam is fixed at the concrete wall, and we will assume a state of plane stress.

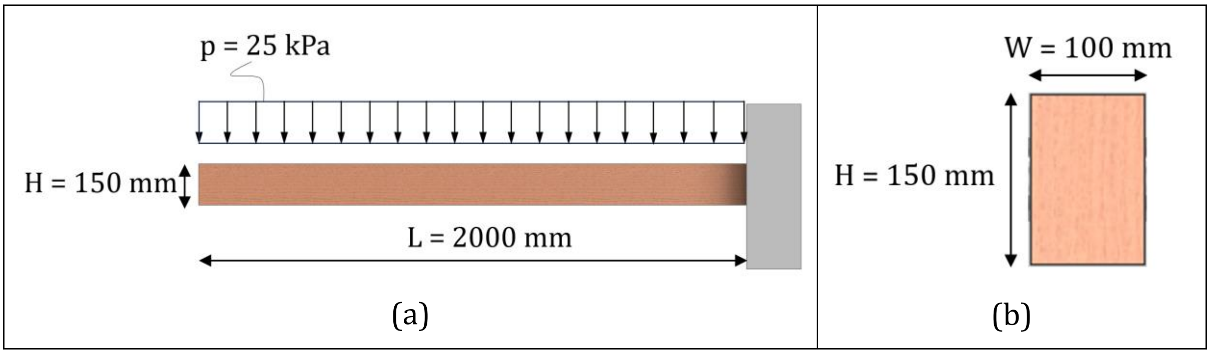


Figure 2 - A cantilever beam representing a simplified version of the structural system.

Load: $p = -0.025\ \text{N/mm}^2$ (Downwards)
Material data: $E = 10\,000\ \text{N/mm}^2, \nu = 0.30\ , \rho = 500\ \text{kg/m}^3, \sigma_y = 20\ \text{N/mm}^2$

Tasks

a) Follow the detailed instructions given in the file *Workshop1.pdf* and pay special attention to the various modules required for modeling the cantilever beam (see Figure 3). The cantilever beam should be modeled using 3 x 40 (50.0 mm) CPS4 elements.

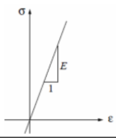
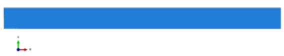
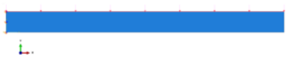
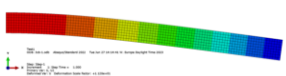
Part	Property	Assembly
Create the part geometry 	Define materials Define and assign sections to parts and regions 	Position the part for initial configuration 
Step	**Interaction**	**Load**
Define analysis steps and output requests	Not applicable for this example	Apply loads and BCs to regions or named sets; and assign them to steps in the analysis history 
Mesh	**Job**	**Visualization**
Mesh the part 	Submit, manage, and monitor analysis jobs	Examine results 

Figure 3 – Abaqus modules required for modeling the cantilever beam.

Solution:

Report your finite element model in Abaqus by generating a figure of the model:

Use **File → Print → Destination** = **File** to make a *.png, *.ps, *.eps, *.tif, or another format to plot the model and include this in your report.

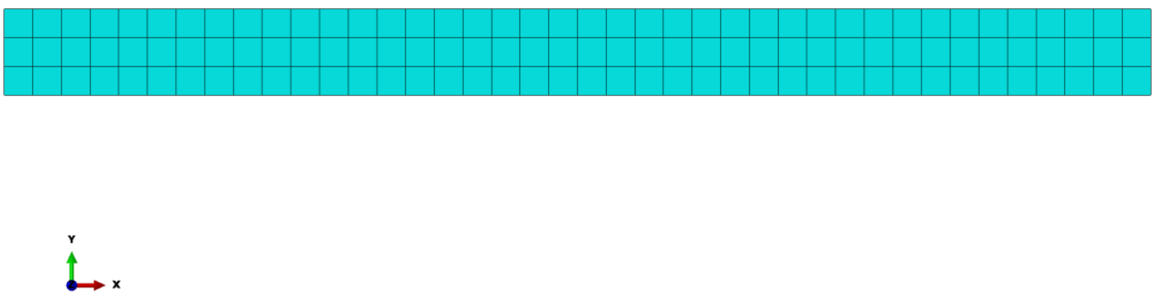


Figure 4 - Finite element model of cantilever beam represented by 2D plane stress elements.

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 cantilever beam on top of the undeformed shape

2) Contours plot of the vertical deformation on the deformed shape

3) Contours plot of the von Mises stress on the deformed shape

4) Default visualization in Abaqus of contours plots uses averaging of the field output between elements. Repeat the contours plot of the von Mises stress on the deformed shape but without averaging between elements, i.e., evaluating results on an element-by-element basis. Discuss the results.

5) The von Mises stress in the two most critical elements and the vertical deformation in the top and bottom nodes at the free end of the beam.

Solution:

1) **Plot → Allow multiple plot states → Plot undeformed shape → Plot deformed shape**

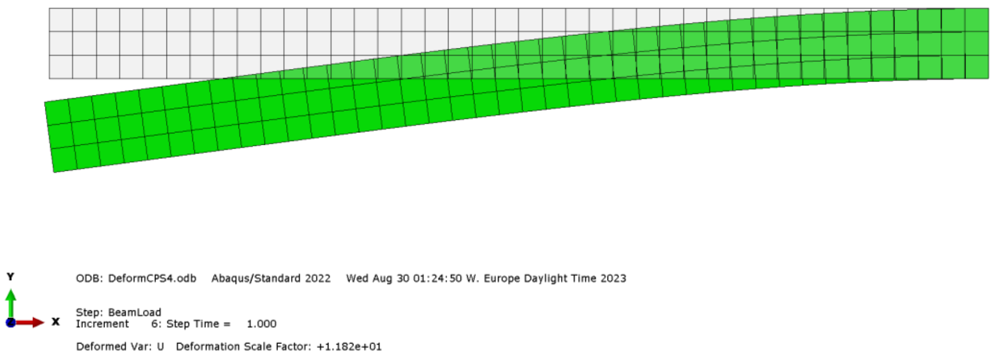


Figure 5 - The deformed shape of the cantilever beam on top of the undeformed shape.

Note: For small-displacement analyses, Abaqus chooses the default scale factor to fit the viewport optimally. To see the actual relative scale of the displacements: Go to **Options** → **Common** → Change the **Deformation Scale Factor** from Auto-compute (...) to Uniform and value **1**.

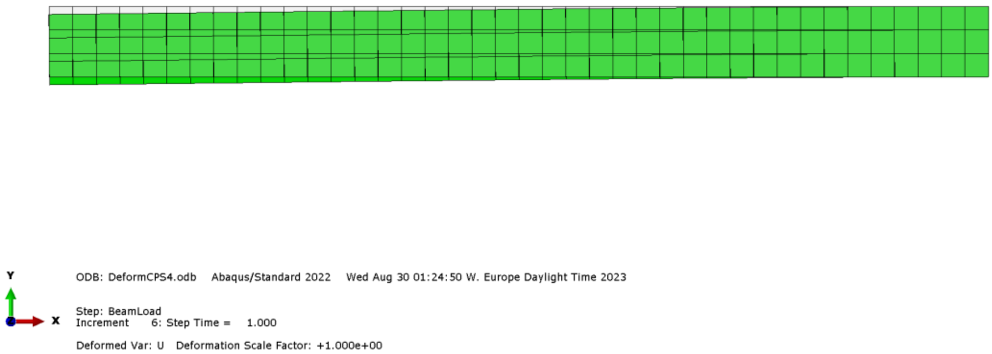


Figure 6 - The deformed shape (no scaling) of the cantilever beam on top of the undeformed shape.

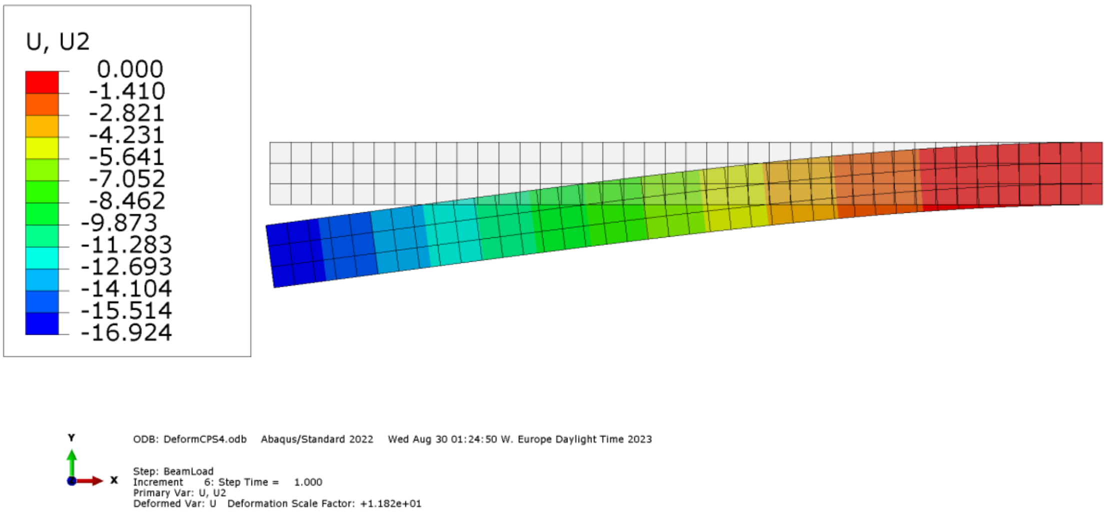


Figure 7 - Contours plot of the vertical deformation (U2 in Abaqus) on the deformed shape.

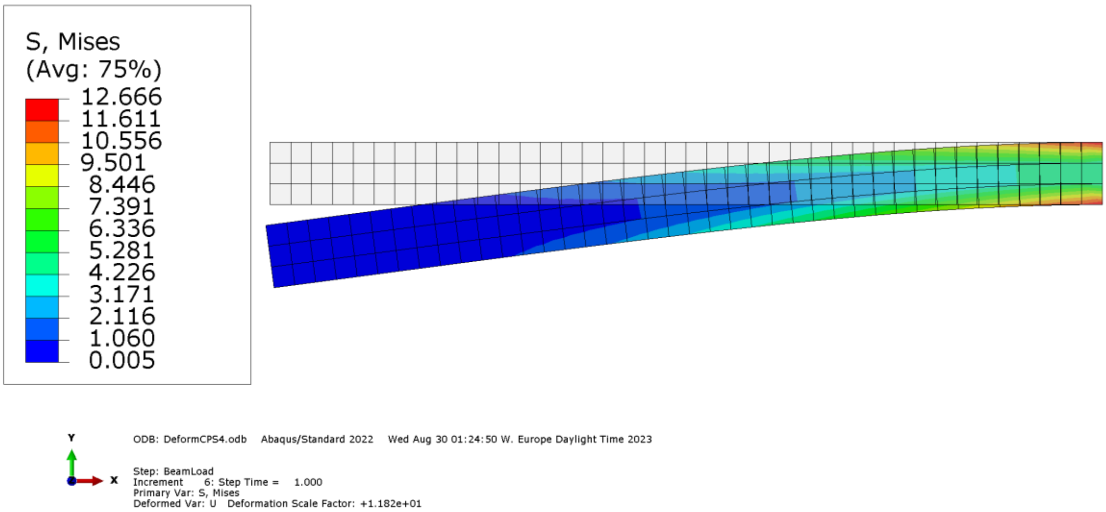


Figure 8 - Contours plot of the von Mises stress on the deformed shape.

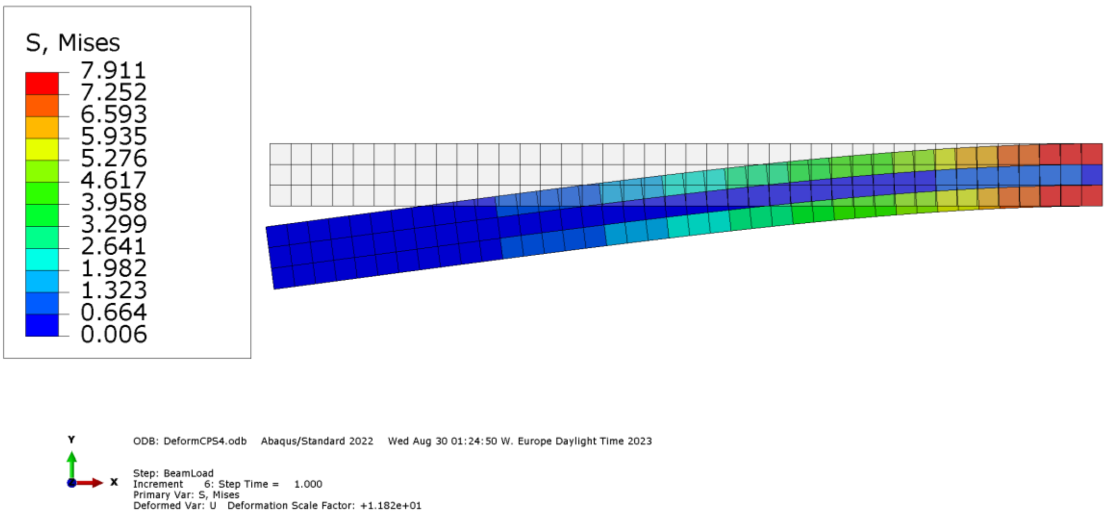


Figure 9 - Contours plot of the von Mises stress on the deformed shape but without averaging between elements.

The maximum and minimum von Mises stress are 12.67 MPa and 0.0054 MPa, respectively, with the default averaging presented in Figure 8. However, when evaluating the von Mises stress on an element-by-element basis the maximum and minimum stress levels are 7.91 MPa and 0.0056 MPa, respectively. Inspecting the critical element (upper right) using the Query on Element quantities and Probe values we get the following stress values in the integration points:

```
Element: BEAM-1.120
  Type: CPS4
  Material: WOOD
  Section: WHOLE.Section-ASSEMBLY_BEAM-1_WHOLE, Homogeneous Solid Section, Thickness = 100
  Connect: 122, 123, 164, 163
  S, Mises (Not averaged): 5.78045, 5.6629, 10.1861, 10.4969
```

These stresses are the actual stresses computed in the integration points of the critical element.

Similarly, the actual stresses computed in the integration point of the other critical element (lower left at the fixed boundary) give similar stress values.

```
Element: BEAM-1.40
  Type: CPS4
  Material: WOOD
  Section: WHOLE.Section-ASSEMBLY_BEAM-1_WHOLE, Homogeneous Solid Section, Thickness = 100
  Connect: 40, 41, 82, 81
  S, Mises (Not averaged): 10.1771, 10.4852, 5.77411, 5.65779
```

Following the same procedure, we may find the displacements of the top and bottom nodes at the free end of the beam, respectively:

```
Node: BEAM-1.124
                                         1              2              3          Magnitude
Base coordinates:                -1.10000e+03,   2.00000e+02,   0.00000e+00,        -
Scale:                            1.18172e+01,   1.18172e+01,   1.18172e+01,        -
Deformed coordinates (unscaled): -1.10084e+03,   1.83076e+02,   0.00000e+00,        -
Deformed coordinates (scaled):   -1.10997e+03,   0.00000e+00,   0.00000e+00,        -
Displacement (unscaled):         -8.43399e-01,  -1.69245e+01,   0.00000e+00,   1.69455e+01

Node: BEAM-1.1
                                         1              2              3          Magnitude
Base coordinates:                -1.10000e+03,   5.00000e+01,   0.00000e+00,        -
Scale:                            1.18172e+01,   1.18172e+01,   1.18172e+01,        -
Deformed coordinates (unscaled): -1.09916e+03,   3.30757e+01,   0.00000e+00,        -
Deformed coordinates (scaled):   -1.09005e+03,  -1.49998e+02,   0.00000e+00,
Displacement (unscaled):          8.41920e-01,  -1.69243e+01,   0.00000e+00,   1.69452e+01
```

Note: As an alternative to Figure 8 you can also turn off the averaging of the von Mises stress (75 %) by using **Result – Options** – Uncheck **Average element output at nodes**. If you leave the **Average element output at nodes** checked, you can also play with the level of averaging to see how this influences the contour plots.

c) Evaluate the maximum bending stress and the maximum deformation of the uniformly distributed loaded beam. You are given the following solution based on elementary (Euler-Bernoulli) beam theory:

$$\sigma_{x,\max} = \frac{M}{I}\frac{H}{2} = 13.33\ \text{MPa}, \qquad v_{\max} = \frac{qL^4}{8EI} = -17.78\ \text{mm}$$

Compare the results to the finite element analysis (FEA) predictions. What can be done to increase the accuracy of the FEA results?

Solution:

We observe that both the vertical displacement and the von Mises stress (Figure 8) are slightly lower (approximately 5 %) compared to the elementary beam theory. If one considers the actual stress level in the integration point of the critical element (10.4969 MPa), the stress level in the FE model is 21 % lower than that predicted by the elementary beam theory.

The comparison to the elementary beam (Euler-Bernoulli) theory is a good choice for this particular beam. That is, the length of the beam is large compared to its height (i.e., the beam is slender). This implies that shear deformations are not significant. Moreover, the von Mises stress is a result of both the normal stress and the transverse shear stress. Actually, by only inspecting the normal stress (S11 in Abaqus) the stress level is very close to elementary beam theory (see Figure 10).

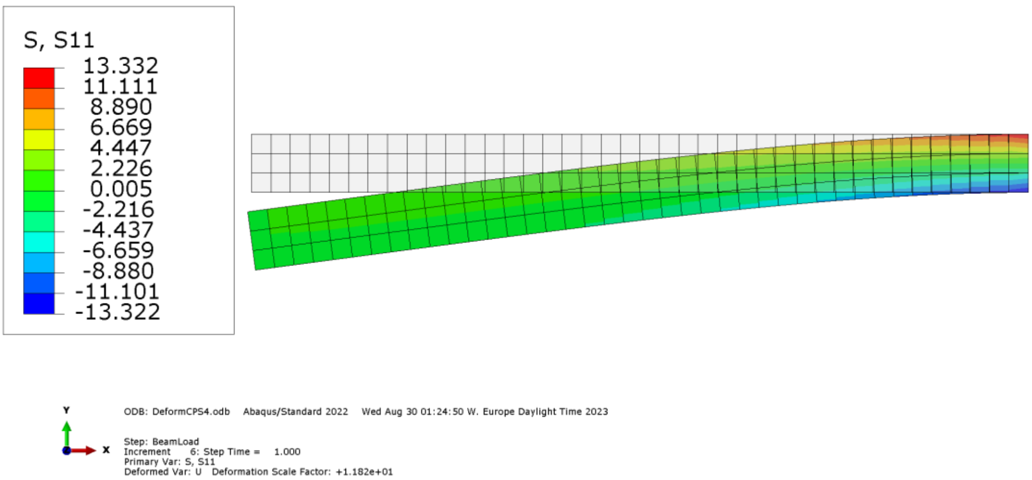


Figure 10 - Contours plot of the longitudinal normal stress (S11) on the deformed shape.

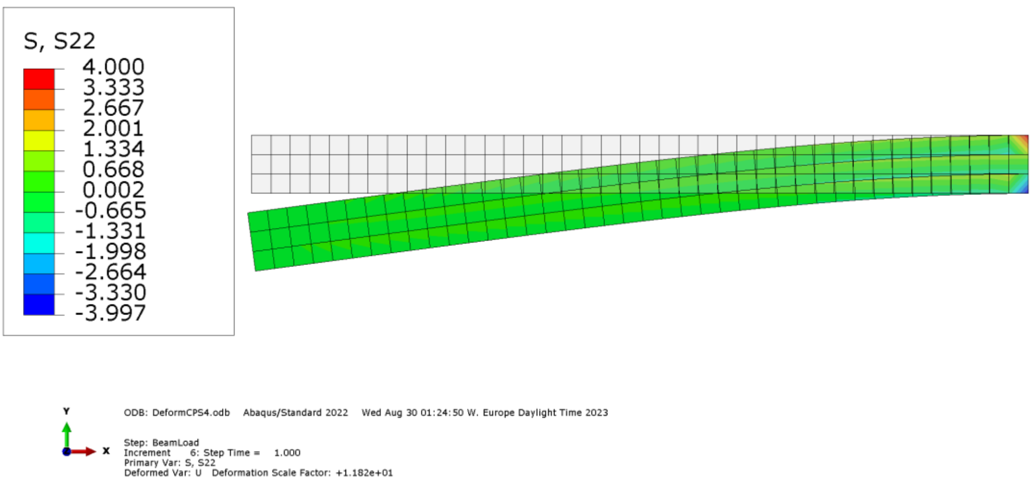


Figure 11 - Contours plot of the transverse normal stress (S22) on the deformed shape.

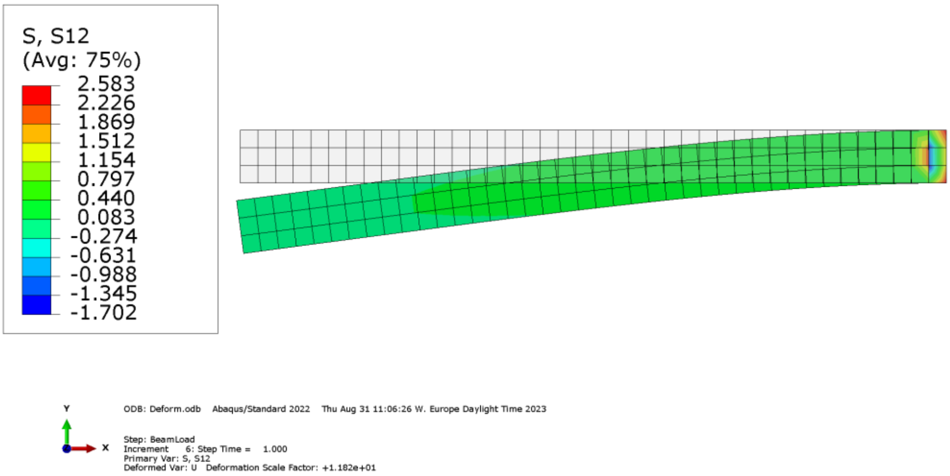


Figure 12 - Contours plot of the shear stress (S12) on the deformed shape.

The explanation for the deviations is mainly related to the relatively coarse element size (50 mm) and the element type (CPS4). The element solution with CPS4 is in general too stiff. A more refined mesh, elements with reduced integration (element type CPS4R in Abaqus), higher order elements (e.g., element type CPS8 in Abaqus), or beam elements will result in a softer structural response and thus larger deflections. Note that in the particular case of beam elements, this may not represent better predictions for the stresses (only deflections).

The stresses tend to localize in the corners at the fixed boundary (top and bottom at the right edge of the beam) because we have prevented the contraction of the structure. Smaller elements give an increase in the maximum stress. This is not necessarily wrong, but a consequence of how we represent the boundary condition.

d) Re-run the model using a characteristic element size of 25.0 mm and 12.5 mm. Compare and discuss the results against those obtained in b). What happens with the computational time and the memory requirements?

Element size [mm]	Deflection (U2) [mm]	Maximum von Mises stress [MPa]	Maximum normal stress (Integration point) [MPa]	CPU time [s]	Memory (Minimum memory required) [MB]
50	16.92	12.67	13.33(11.19)	0.6	17
25	17.61	13.68	14.57(13.07)	0.1	17
12.5	17.80	15.03	15.89(14.62)	0.9	18

We observe that the deflection has converged towards the elementary beam theory, while the stress levels are slightly overestimated at the finest element size (12.5 mm).

Note that the maximum value of the stress in the contour plot/field does not necessarily coincide with the stress value in the integration points. The contours plot also includes the extremities of the geometry, while the location of the integration points is determined by the element type.

For all practical purposes, the difference in CPU time is neglectable and irrelevant from an engineering point of view. Actually, as a rule of thumb, one should not pay too much attention to CPU time before it exceeds a certain threshold (~10-20 s). Below this threshold, it is not the FEA itself that takes up the main part of the CPU time, but other operations on the computer may affect the time report in the Message file. The same goes for the memory requirements for these small models. The memory requirements are found in the Data file.

e) We will now change the element type to a CPS8 element. This is a more advanced element with additional degrees of freedom and should provide more accurate results for bending-dominated problems. Model the beam using 3 x 40 (50.0 mm) and 6 x 80 (25.0 mm). How do the FEA predictions compare to the elementary beam theory? Discuss plausible explanations for the observed deviations. What will be the consequence of increasing the height of the beam? Do you expect smaller or larger deviations from the elementary (Euler-Bernoulli) beam theory?

Element type	Element size [mm]	Deflection (U2) [mm]	Maximum von Mises stress [MPa]	Maximum Normal stress [MPa]	CPU time [s]	Memory (Minimum memory required) [MB]
CPS4	50	16.92	12.67	13.33(11.19)	0.6	17
	25	17.61	13.68	14.57(13.07)	0.1	17
CPS8	50	17.85	13.34	14.46(12.88)	0.5	17
	25	17.86	14.98	15.81(14.38)	0.7	17
CPS4I*	50	17.83	13.42	13.17(11.27)	0.6	17
	25	17.85	14.21	14.03(12.83)	0.6	17

* *CPS4I is not asked for in the task but is included for educational purposes because these types of elements will be treated later in the lectures. See the Lecture plan on Blackboard. The incompatible element will be further explained in the lectures. In general, a linear element experiences shear locking when subjected to bending. Thus, linear elements are not recommended for bending problems, especially if you use a coarse mesh. Quadratic elements are an alternative but are more computationally expensive. The incompatible element is therefore often preferred in such problems*

if the elements are not too distorted since the accuracy declines rapidly with increasing distortion for these kinds of elements.

It is clear from the results that the displacement has more or less converged for the coarsest mesh for the higher-order (CPS8) and incompatible (CPS4I) elements. Thus, CPS8 and CPS4I elements result in a softer structural response and larger deformations. This is also reflected in the stress levels, which have slightly increased for the CPS8 and CPS4I elements.

The consequence of increasing the height of the beam (while keeping the length constant), will be that shear deformations play a larger role in the response of the beam. Hence, the assumption of neglecting shear deformations will be challenged, and the elementary beam theory is not ideal for comparison. The deviations between the FEA predictions and the elementary beam theory will therefore increase with increasing height of the beam. That is, the displacements will be larger in the FEA compared to the elementary beam theory. The user should consider using the Timoshenko beam theory for comparison instead if the height of the beam is increasing. Timoshenko beam theory includes shear strains and will be a better comparison for the FEA predictions.

We will now increase the complexity of the model and introduce the hole in the beam. The dimensions and position of the hole are given in Figure 13. The beam is still embedded in the concrete wall at one end and supported by the steel rod at the hole at the opposite end of the beam.

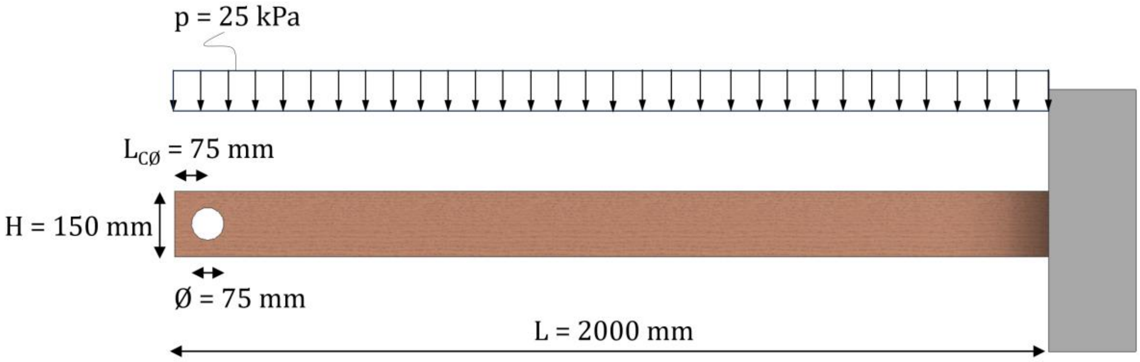


Figure 13 - Side view of the beam including the hole at the end of the beam.

f) Include the hole in the geometry of the part and apply appropriate boundary conditions. Follow the detailed instructions given in *Workshop1.pdf*. The beam should be modeled with a global element size of 12.5 mm. Use element type CPS4.

Solution:

Report your finite element model in Abaqus by generating a figure of the model:

Use **File → Print → Destination** = **File** to make a *.png, *.ps, *.eps, *.tif, or another format to plot the model and include this in your report.

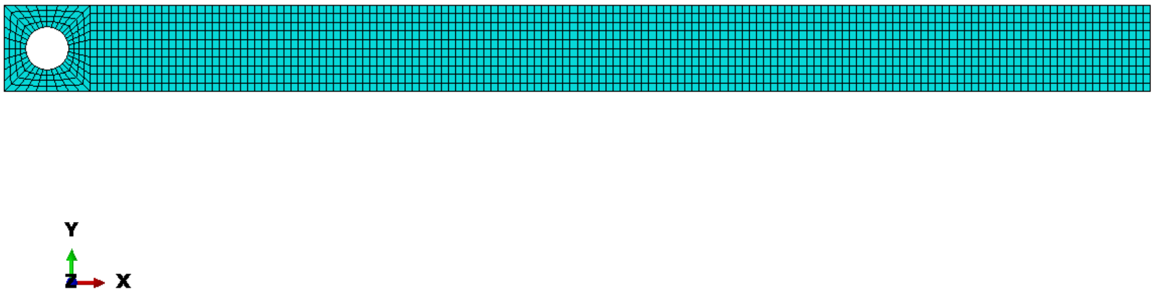


Figure 14 - Finite element model of the beam with a hole represented by 2D plane stress elements.

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

1) The deformed shape of the beam on top of the undeformed shape

2) Contours plot of the vertical deformation on the deformed shape

3) Contours plot of the von Mises stress and the maximum principal stress (Max. Principal in Abaqus) on the deformed shape

Solution:

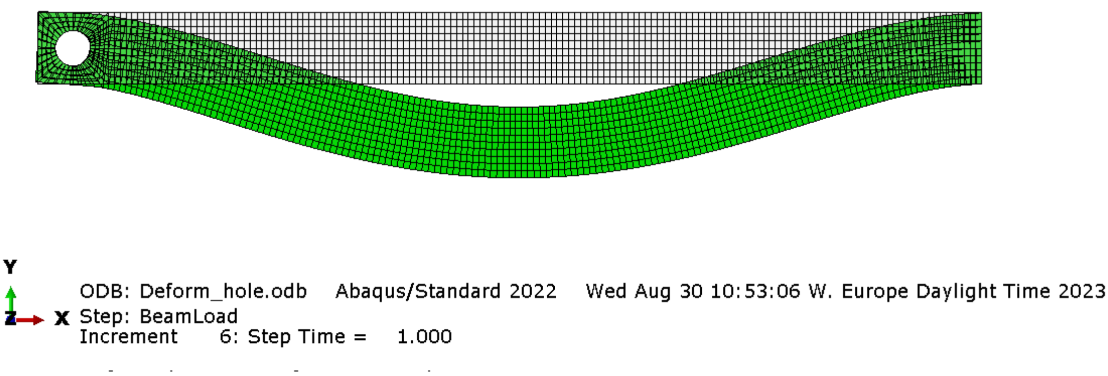


Figure 15 - The deformed shape of the beam on top of the undeformed shape.

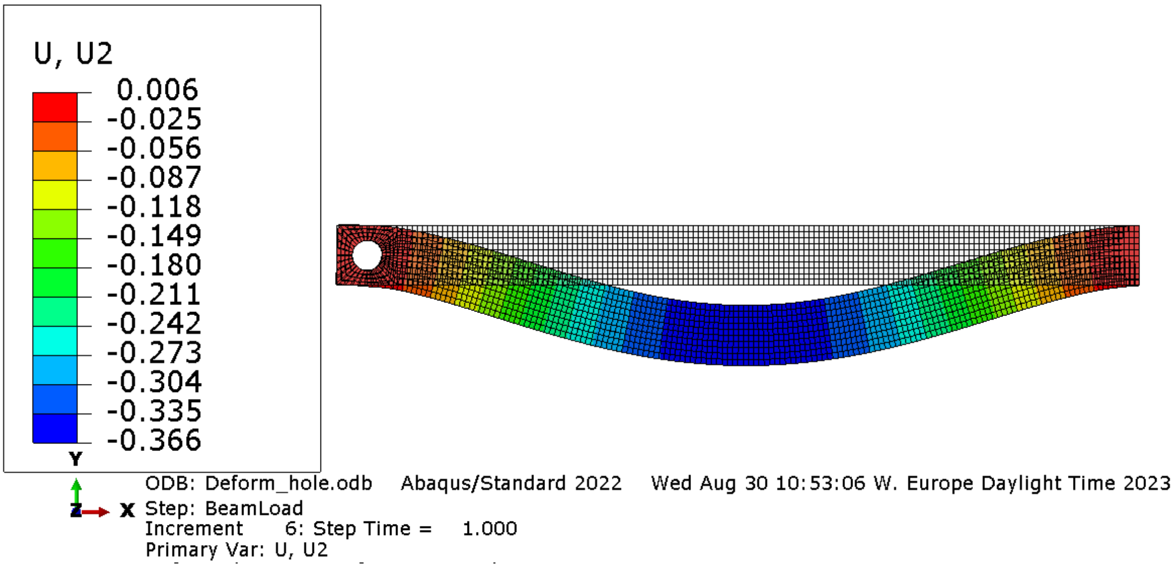


Figure 16 - Contours plot of the vertical deformation on the deformed shape.

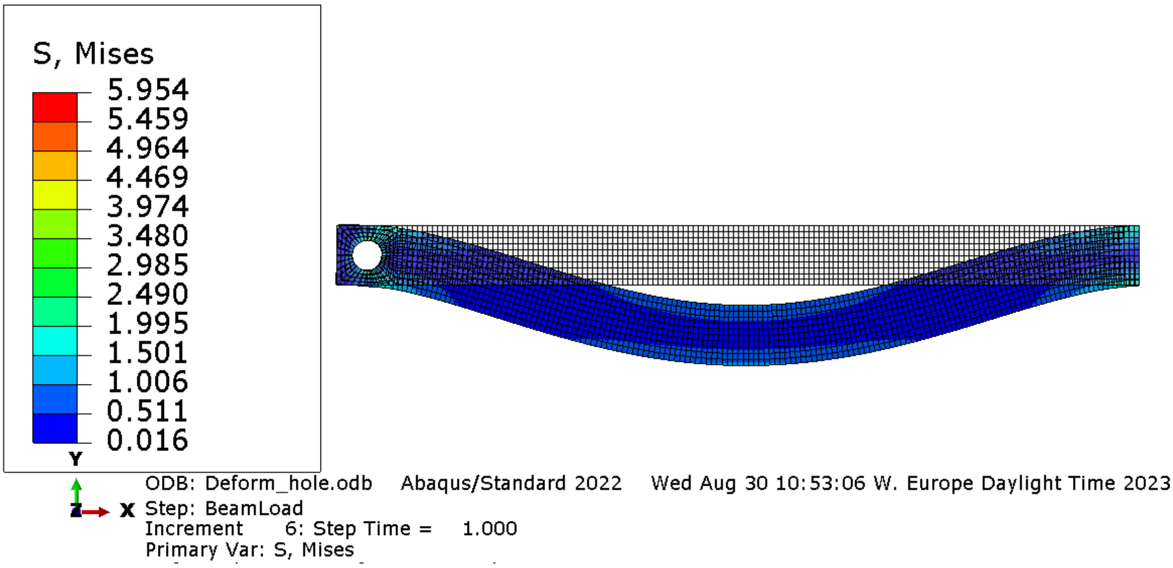


Figure 17 - Contours plot of the von Mises stress on the deformed shape.

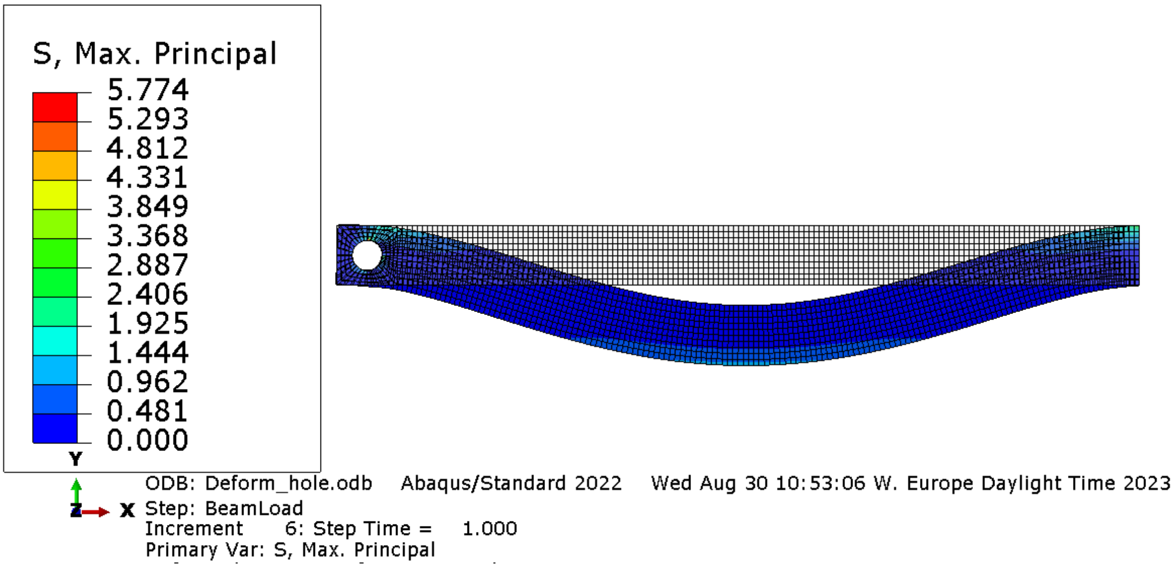


Figure 18 - Contours plot of the maximum principal stress (Max. Principal in Abaqus) on the deformed shape.

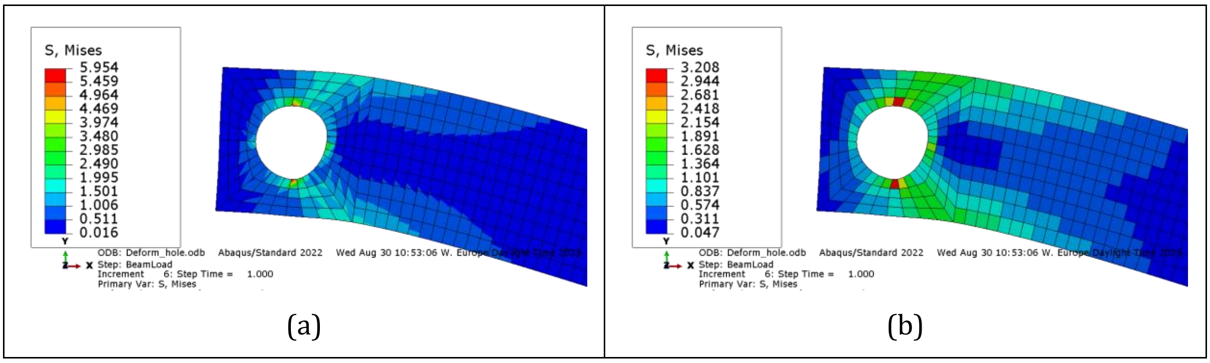


Figure 19 - Close-up view of the contours plot of the von Mises stress in the vicinity of the hole. (a) Averaged von Mises stress and (b) Quilt contour type of the von Mises stress (element-by-element representation).

h) Refine your mesh by reducing the global element size by a factor of 2 and re-run the analysis. Comment on your observations. Also, check the other stress components.

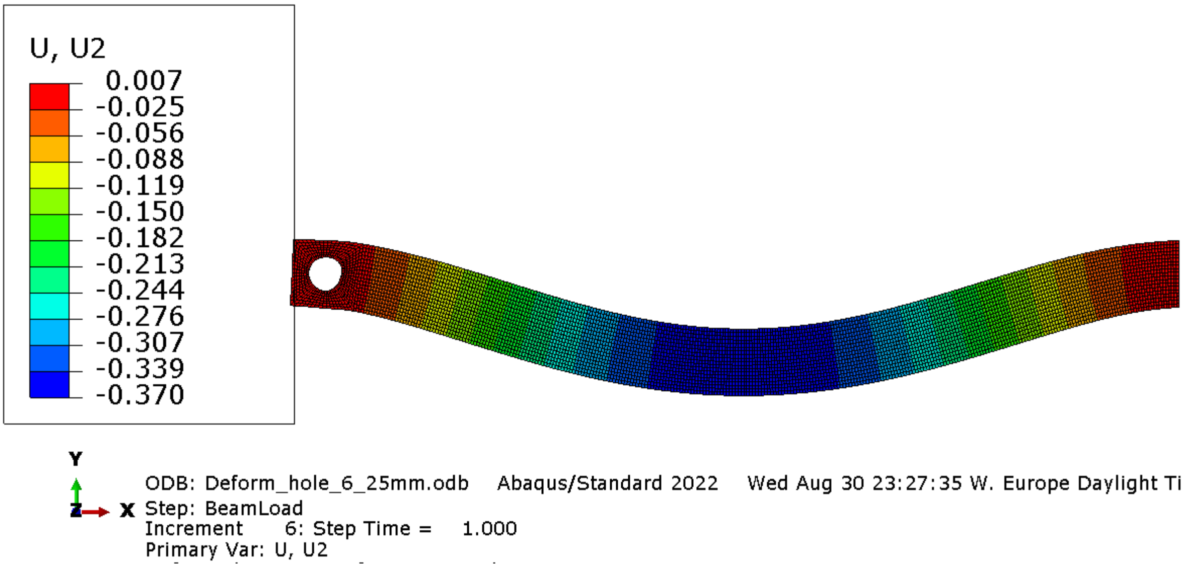


Figure 20 - Contours plot of the vertical deformation on the deformed shape.

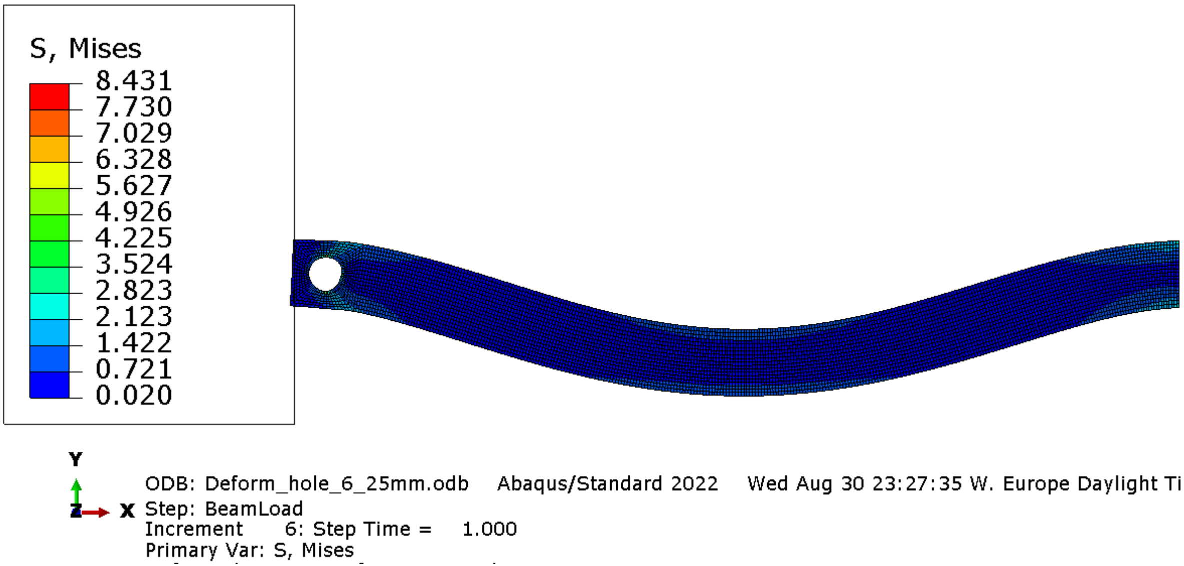


Figure 21 - Contours plot of the von Mises stress on the deformed shape.

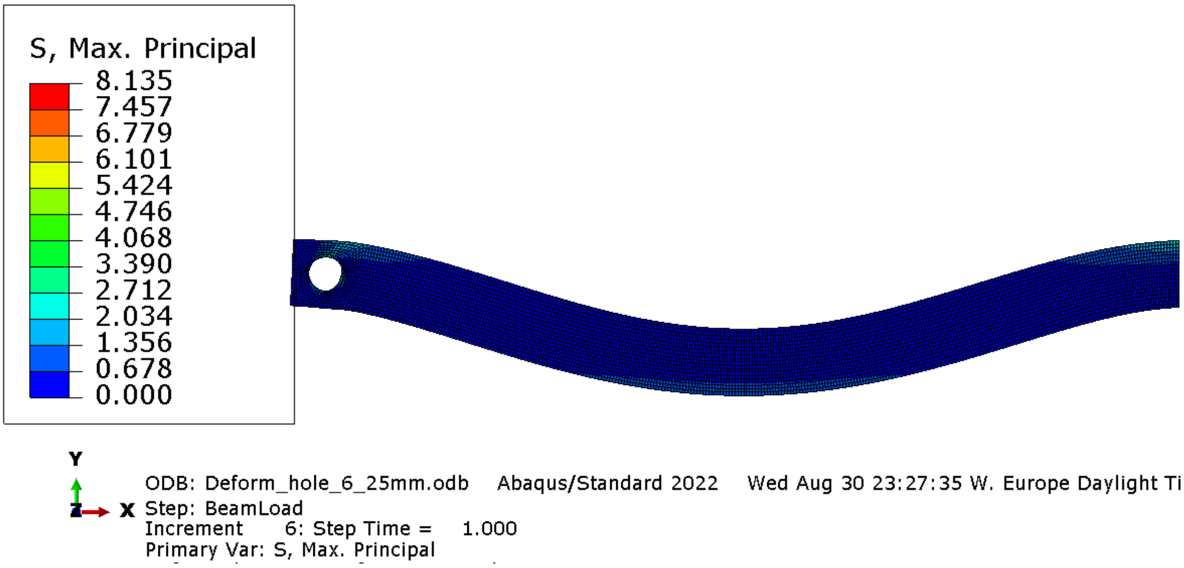


Figure 22 - Contours plot of the maximum principal stress (Max. Principal in Abaqus) on the deformed shape.

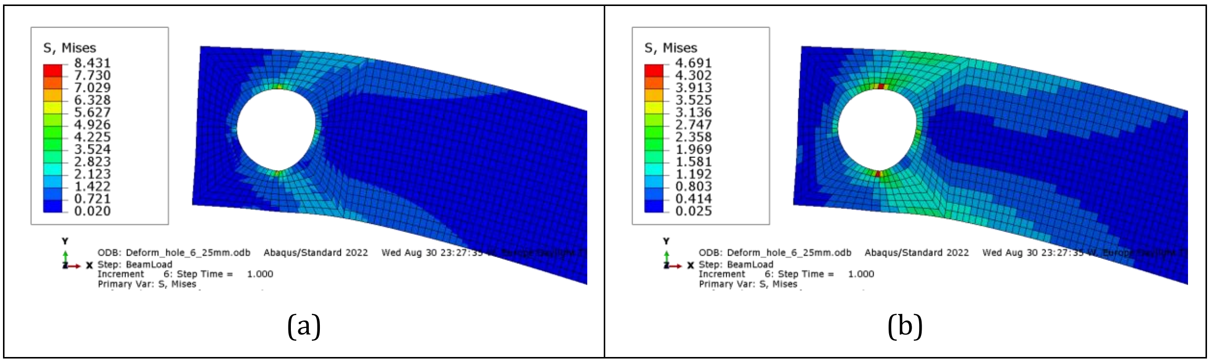


Figure 23 - Close-up view of the contours plot of the von Mises stress in the vicinity of the hole. (a) Averaged von Mises stress and (b) Quilt contour type of the von Mises stress (element-by-element representation).

Figures 15 and 19 show that the vertical displacement does not change much between the two discretizations. The stress fields show a more evident effect of the mesh refinement. That is, we observe a rather large increase in stress from the 12.5-mm mesh (Figures 17-19) to the 6.25-mm mesh (Figures 21-23). This is expected when modeling stress concentrations. It would be recommended to also evaluate the performance of other element types (e.g., CPS8 in Abaqus). The stress concentrations at the top and bottom of the hole may be related to the boundary conditions on the left edge of the hole, restricting the hole from horizontal translation. It would therefore be recommended to investigate the influence of boundary conditions (around the hole) further.

Note: A stress concentration is a location in a solid material where the stress is higher than the surrounding material. Stress concentrations can occur if there are some irregularities in the geometry (e.g., notches and holes) or in the material (e.g., aggregates in concrete). High local stresses can cause the material to fail. One should therefore design the geometry to avoid such concentrations in the stress field. If holes or notches cannot be avoided in the design, the oval shapes are preferred before sharp corners.

i) How would you evaluate the accuracy of your solution, e.g., in terms of deflections and stress levels?

Solution:

Following up on the previous answer **h)**, good practice would be to run convergence studies on the element size, element type, and boundary conditions. For instance as that conducted for the cantilever beam in **d)** and **e)**. Convergence would then be evaluated based on displacement and stresses. However, displacements are often preferred because they are solved directly in the FE solver. Stresses (and strains) are interpolated within the element and may be less accurate.

Global displacements may also be evaluated similarly as that in **c)**, by comparison to elementary beam theory. One could also vary the boundary conditions on the left perimeter of the hole to investigate the influence of the constraints. From an elementary beam theory point of view, this will be idealized as a roller support that is free to move horizontally. That is, the maximum vertical displacement should occur (0.5785 x 2000 mm =) 1157 mm from the fixed end of the beam and the following solution:

$$\sigma_{x,\max} = \frac{M}{I}\frac{H}{2} = 3.33 \text{ MPa}, \qquad v_{\max} = 5.416 \cdot 10^{-3}\frac{qL^4}{EI} = -0.77 \text{ mm}$$

j) Elaborate on the physical characteristics of plane stress states. Can you give examples of situations where these assumptions are valid?

Solution:

Plane stress states:

- Defined by $\sigma_z = \tau_{yz} = \tau_{xz} = 0$
- Appears in thin plane structures, where the thickness (in the z-direction) is much smaller than the dimensions in the x- and y-direction. (NB: We must not confuse plane structures with plates, where the loads and displacement are in the z-direction)
- Examples: Pressure vessels (norsk: trykktank), in the web (norsk: steg) of a beam, junction plates (norsk: knuteplater) and in wall panels (norsk: veggskive)

k) Voluntary: Increase the height H of the cantilever beam (e.g., by a factor of 5) and re-run the simulation. How do the FEA predictions compare to elementary beam theory? You may notice that the deviations in displacements between theory and the FEA become larger. Can you explain why? (Hint: Elementary beam theory only applies to slender beams where the shear deformation is small). How does the stress component σ_y vary across the plane structure? What is σ_y according to elementary beam theory?

Solution:

The solution given by the elementary (Euler-Bernoulli) beam theory is:

$$\sigma_{x,\max} = \frac{M}{I}\frac{H}{2} = 0.53 \text{ MPa}, \qquad v_{\max} = \frac{qL^4}{8EI} = -0.14 \text{ mm}$$

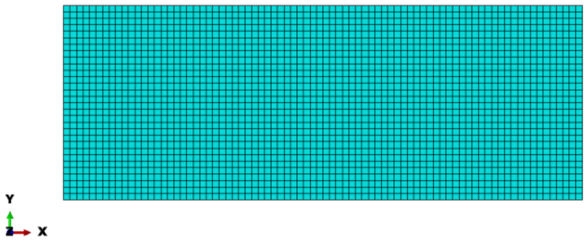


Figure 24 - Finite element model of the cantilever beam with new height (H = 750 mm) represented by 2D plane stress elements.

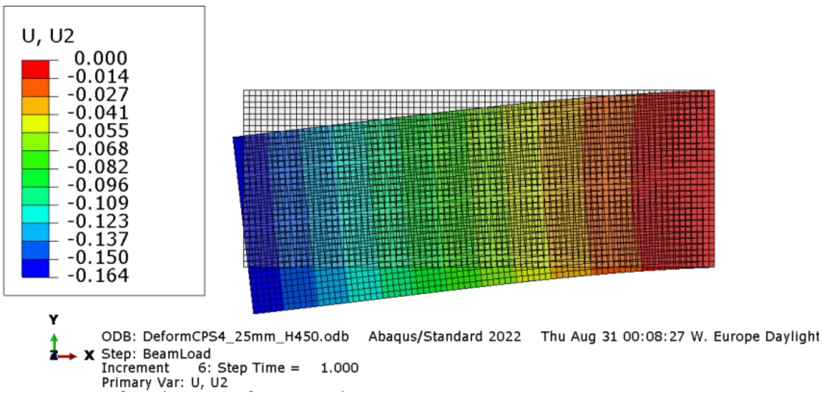


Figure 25 - Contours plot of the vertical deformation (U2 in Abaqus) on the deformed shape.

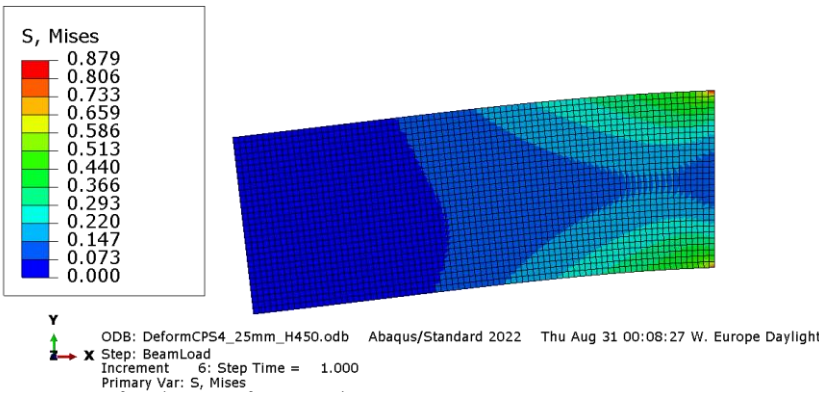


Figure 26 - Contours plot of the von Mises stress on the deformed shape.

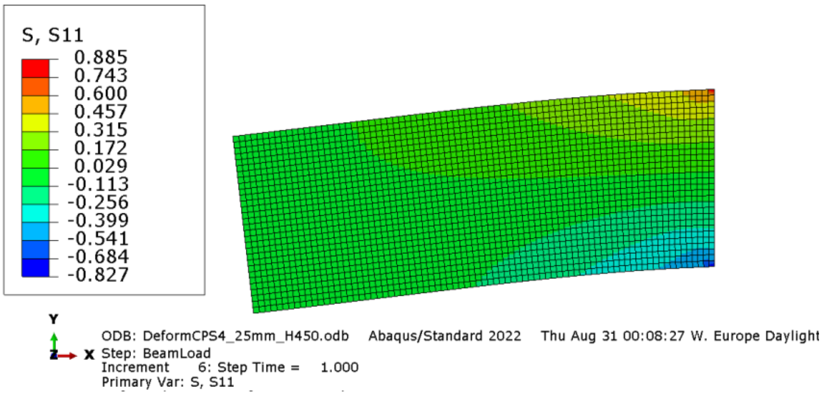


Figure 27 - Contours plot of the longitudinal normal stress (S11) on the deformed shape.

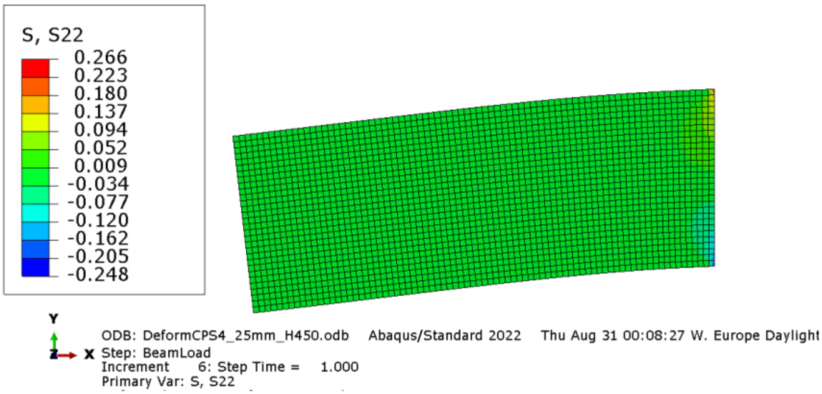


Figure 28 - Contours plot of the transverse normal stress (S22) on the deformed shape.

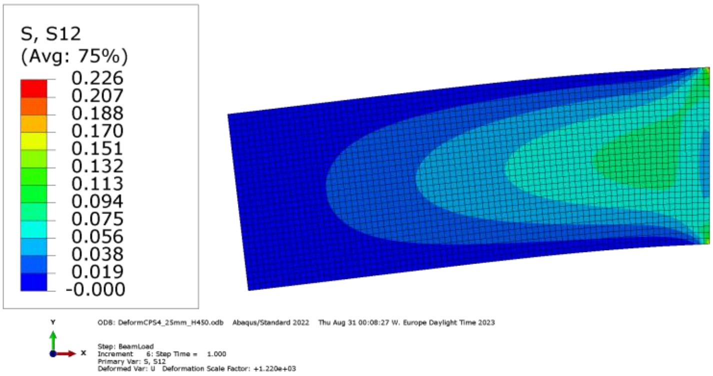


Figure 29 - Contours plot of the shear stress (S12) on the deformed shape.

See also answer in **e)**. The deviations between the FEA predictions and the elementary beam theory are as expected. Increasing the height of the beam means that the displacements will be larger in the FEA compared to the elementary beam theory. The user should consider using the Timoshenko beam theory for comparison instead if the height of the beam is increasing. Timoshenko beam theory includes shear strains and will be a better comparison for the FEA predictions.