NTNU – NORWEGIAN UNIVERSITY OF SCIENCE AND TECHNOLOGY
FACULTY OF ENGINEERING
DEPARTMENT OF STRUCTURAL ENGINEERING

TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 1

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 hole 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).

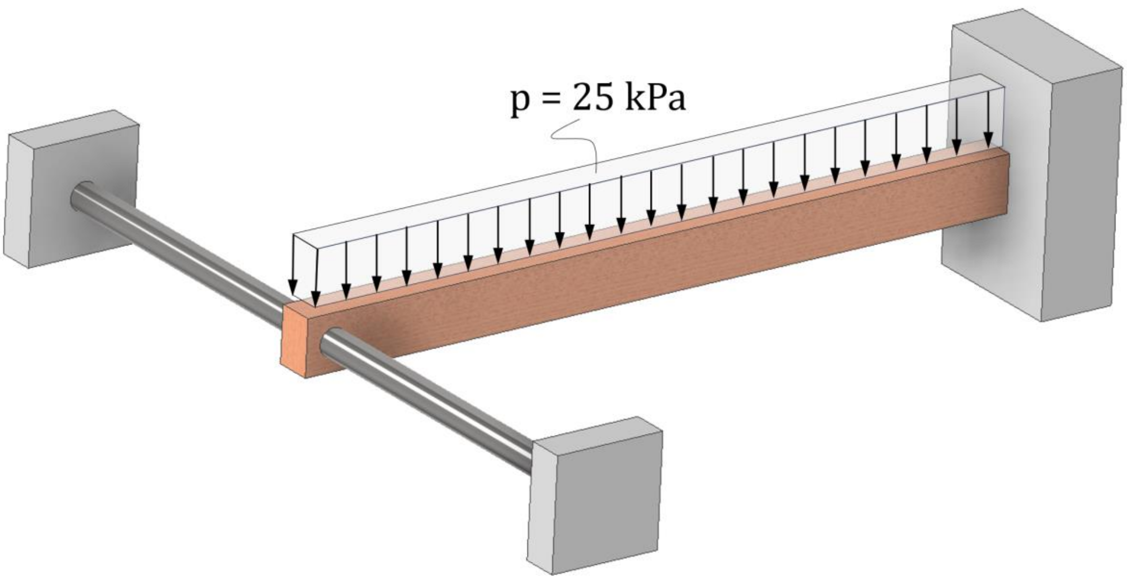


Figure 1 - A wooden beam supported by concrete at one end and a steel rod at the other end.

We will start by modeling the cantilever beam in Figure 2. 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.

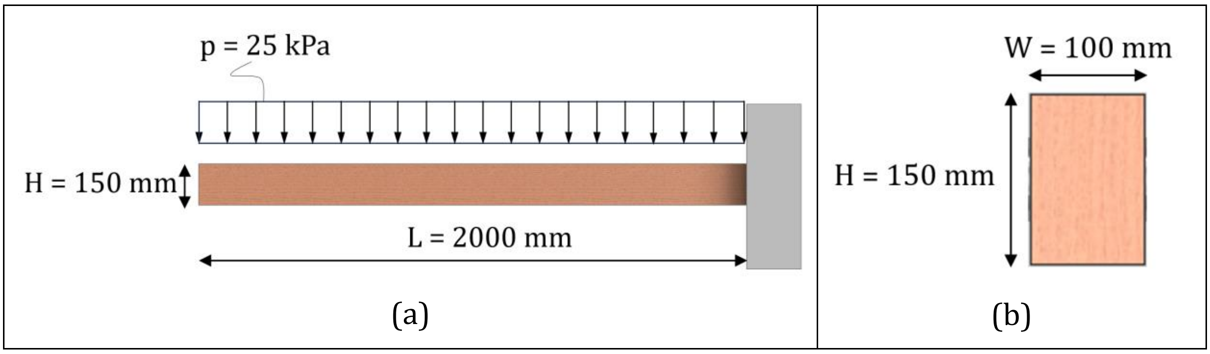


Figure 2 - A cantilever beam representing a simplified version of the structural system.

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) Q4 plane stress elements (CPS4 in Abaqus). Report your model in Abaqus by generating a figure of the model.

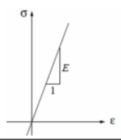
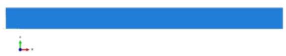
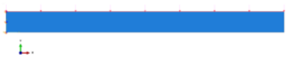
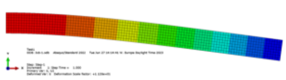
Part	Property	Assembly
Create the part geometry 	Define materials Define and assign sections to parts and regions 	Position the part for initial configuration 
Step	**Interaction**	**Load**
Define analysis steps and output requests	Not applicable for this example	Apply loads and BCs to regions or named 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.

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.

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?

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?

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?

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 4. 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.

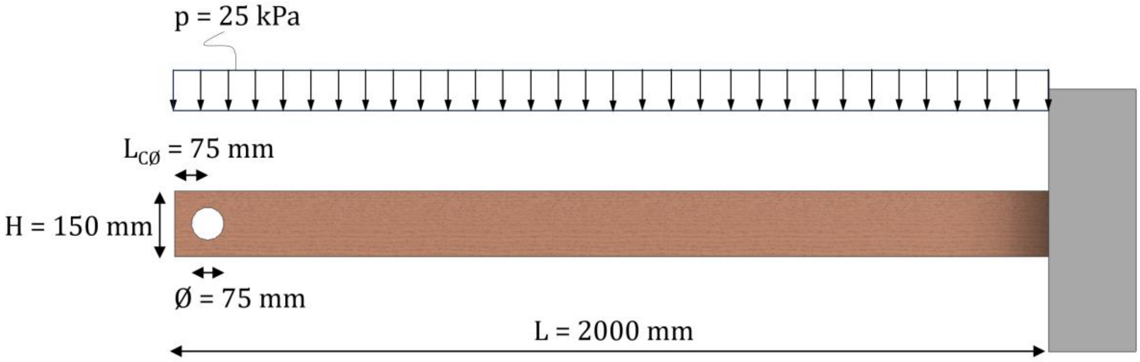


Figure 4 - Side view of the beam including the hole at the end of the beam.

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. Report your model in Abaqus by generating a figure of the model.

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

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.

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

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

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?