TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 4

Case Study 4 will revisit the wooden beam from Case Study 1, but now we will model the beam using 3D solid elements. A workshop on how to model the different aspects addressed in this case study is uploaded to Blackboard (see "*Workshop4.pdf*" in the folder "Case studies"). In Task 1, we will start by modeling a simple cantilever beam before moving to a more complicated geometry and boundary conditions by introducing the two holes in the beam in Task 2.

Learning outcome:

- Modelling of 3D solid problem
- Convergence studies
- Evaluate energy output (strain energy and hourglass energy)
- Visualization and post-processing of results in Abaqus/CAE
- Introduction to the Abaqus Python scripting interface

Problem description

Figure 1 shows a very similar wooden beam to that used in Case Study 1, i.e., a wooden beam 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. The new aspect of this case study is an additional hole in the center of the beam (as shown in Figure 1), which makes it necessary to use 3D solid elements in Abaqus.
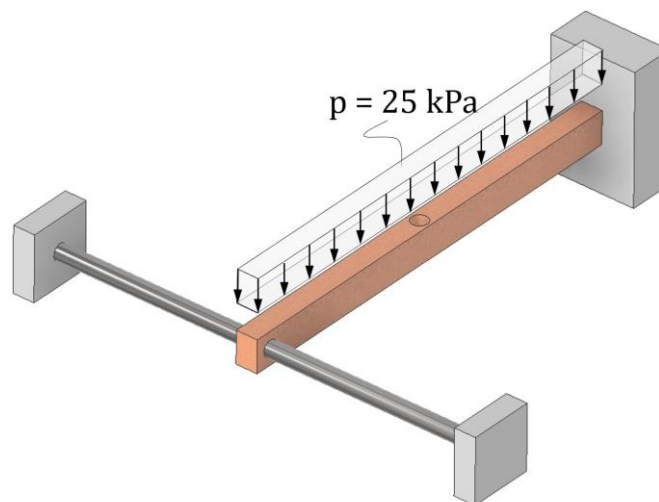

Figure 1 - A wooden beam supported by concrete at one end and a steel rod at the other end.

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.
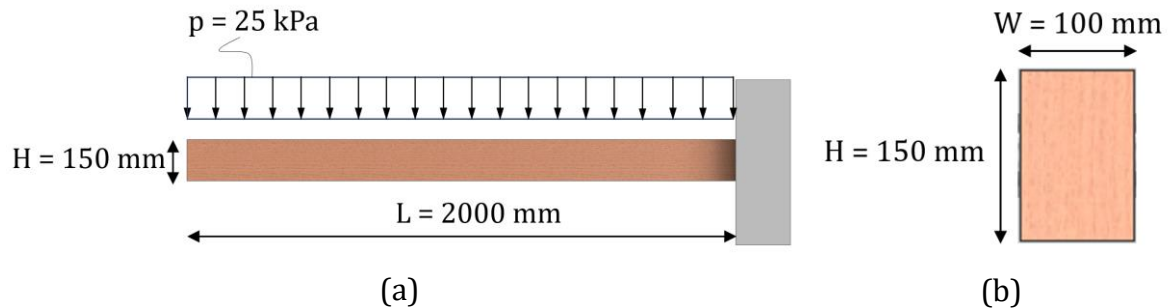

(a)   (b)

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

Load: $p = -0.025$ N/mm^2 (downwards)

Material data: $E = 10\,000$ N/mm^2, $v = 0.30$, $\rho = 500$ kg/m^3, $\sigma_y = 20$ N/mm^2

Task 1

a) The cantilever beam should be modeled using 8-node brick elements with reduced integration (C3D8R in Abaqus) and 50 mm characteristic size. Report your model in Abaqus by generating a figure of the model.

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 displacement 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., evaluate results on an element-by-element basis. Discuss the results.

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

c) Evaluate the maximum bending stress and the maximum displacement of the cantilever 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?

2

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) Evaluate the internal strain energy (ALLIE in the OBD history output) and the artificial strain energy (ALLAE in the ODB history output). How can we use these energies to evaluate the performance of the FEA?

f) We will now change the element type to a C3D8 element. This is an element with selective reduced integration and should provide more accurate results for bending-dominated problems. Model the beam using element sizes of 50.0 mm and 25.0 mm. How do the FEA predictions compare to the predictions using 2D plane stress elements (CPS8 in Abaqus) in Case Study 1?

Task 2

We will now increase the complexity of the model and introduce the two holes in the beam. The dimensions and position of the holes are given in Figure 3. 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.
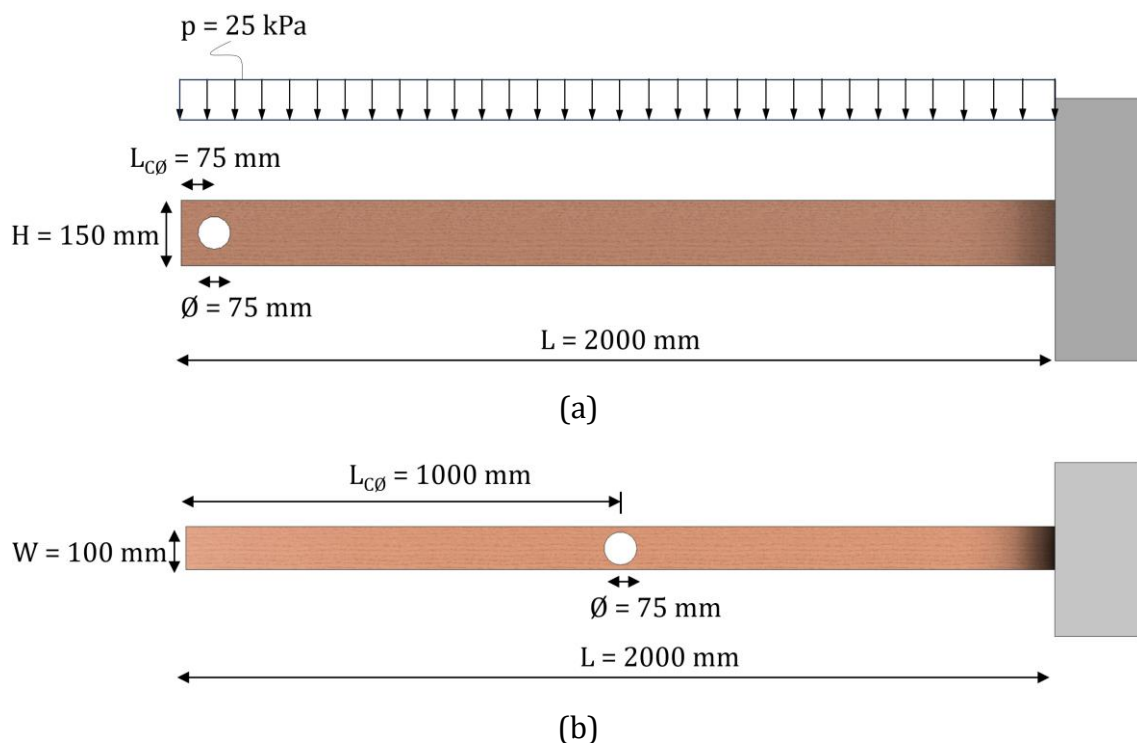

Figure 3 – (a) Side view of the beam including the hole at the end of the beam and (b) top view of the beam illustrating the hole at the center of the beam.

a) Include the holes in the geometry of the part and apply appropriate boundary conditions. The beam should be modeled with a global element size of 12.5 mm. Use element type C3D8R. Report your model in Abaqus by generating a figure of the model.

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 beam on top of the undeformed shape.

2) Contours plot of the vertical displacement 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.

c) Evaluate the internal strain energy (ALLIE in the OBD history output) and the artificial strain energy (ALLAE in the ODB history output). What is your evaluation of the performance of the FEA with an element size of 12.5 mm and element type C3D8R?

d) We will now change the element type to a C3D8 element. Re-run the model with the new element type. Evaluate the maximum von Mises stress, the maximum vertical displacement, and the ratio ALLAE/ALLIE. Compare the results for the two element types.

e) 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 principal stress components. Why should we check both the principal stress components and the von Mises stress?

f) Change the element type to a C3D8 element for the refined mesh. Re-run the model with the new element type, and evaluate the maximum von Mises stress, the maximum vertical displacement, and the ratio ALLAE/ALLIE. Compare the results for the two element types.

g) Keep the size of the global elements as in e) with element type C3D8R and increase the pressure load by a factor of 4 (p = 0.1 MPa). Comment on your observations. How does the von Mises stress compare to the elastic limit of the material ($\sigma_y = 20$ N/mm^2)? Do you trust the structural integrity of the model for the new (increased) pressure loading?

h) Voluntary: We will now briefly introduce the Python scripting interface in Abaqus. Python is the standard programming language for Abaqus scripting and is used in several ways. We will pay special attention to the various modules required for modeling the cantilever beam.

This is done by inspection of the journal (.jnl) file. This file contains the Python configuration commands used to build your model in the Abaqus/CAE GUI and is automatically generated by Abaqus in your Working directory.

Try saving your .jnl file as a .py file. You should probably clean up some of the commands because they are redundant, i.e., Abaqus saves your entire "clicking"-journey from the

Abaqus/CAE GUI. See "Workshop4.pdf" for more information and an explanation of how the Python commands in the journal file relate to the modules in Abaqus/CAE.

Reopen Abaqus. Choose **File → Run script →** Choose your .py file containing the Python configuration commands to be run to build your model in Abaqus. Execute the script by choosing the .py file and click **OK**. Does it work? If so, then you may start to parameterize your model in the .py file. Relevant candidates for parameterization are the physical dimensions of your model, the mesh size, and the loading.