Examination

MHA021 Finite Element Method

Date and time:	2025-01-16, 08.30-12.30
Instructors:	Jim Brouzoulis (phone 2253) and Knut Andreas Meyer (phone 1495). An instructor will visit the exam around 09:30 and 11:30.
Solutions:	Example solutions will be posted within a few days after the exam on the course homepage.
Grading:	Will be posted on Canvas on Jan 31 the latest.
Review:	Friday Jan 31, and Monday Feb 3, 12-13 in meeting room Newton, third floor, Mechanical Engineering building.
Permissible aids:	Chalmers type approved pocket calculator. **Note**: A formula sheet is available as a pdf-file alongside with this exam thesis.

Exam instructions

All exam problems require a hand-in on paper. For some of the problems, it may be convenient to also use Python including CALFEM. However, note that a complete solution requires deriving and stating all equations hand-written (or computer formatted, not plain text) separately from the code. If you use Python and CALFEM as part of your solutions, you must make sure to also hand in any Python code you have written yourself. You do this by saving your files under `C:\__Exam__\Assignments\` in the appropriate sub-directories created for each problem. When doing so, it is strongly recommended that you write your anonymous exam code in any Python files that you want to hand in.

Note that to get Python up and running on the exam computer additional steps are required. Please see the instructions in the folder `Python setup`.

You can utilize these files by copying appropriate files into the sub-directories for the problem where they are needed. Should you need to refer to the CALFEM manual, you can find this also (excluding the examples section) under the folder `Calfem documentation`.

Finally, remember to save any open Python files before you log-out from the computer when you are finished with the exam.

Problem 1

Consider a thin-walled circular shaft as shown in Figure 1 with total length $L = 2.0$ m. The shaft has a thickness $h = 2\,\text{mm}$ and (average) radius $r = 30\,\text{mm}$. It is made of steel with shear modulus $G = 70\,\text{GPa}$. The right end of the shaft is clamped and the left end is loaded by a torque with magnitude $T_0 = 300\,\text{Nm}$ as shown in Figure 1.

The strong form for this problem is given as: find φ such that

$$\begin{cases} \dfrac{\mathrm{d}}{\mathrm{d}x}\left(G\,I_\mathrm{p}\,\dfrac{\mathrm{d}\varphi(x)}{\mathrm{d}x}\right) = 0 \\ T(0) = -T_0 \\ \varphi(L) = 0 \end{cases}$$

where I_p is the polar moment of area, which for a thin-walled circular cross-section is $I_\mathrm{p} = 2\pi\, r^3\, h$ and $\varphi(x)$ is the rotation angle of the cross-section at coordinate x. In addition, the sectional torque is given through the constitutive relationship $T(x) = G\,I_\mathrm{p}\,\dfrac{\mathrm{d}\varphi(x)}{\mathrm{d}x}$.

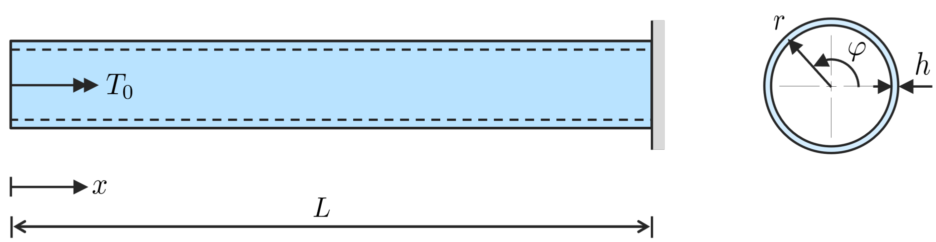

Figure 1: Clamped shaft subjected to a torque T_0 at the free end.

Tasks:

a) From the strong form, derive the weak form. **(1p)**

b) From the weak form, derive the global finite element formulation $\mathbf{K}\,\mathbf{a} = \mathbf{f}$. Then for the case of a linear approximation of the rotation, $\varphi(x)$, derive the element stiffness matrix $\mathbf{K}^e$ for an element with nodes x_i and x_{i+1}. **(1p)**

c) Consider the shaft discretized into two equally long linear elements. Determine the explicit system of equations $\mathbf{K}\,\mathbf{a} = \mathbf{f}$ and determine the rotation angle at $x = 0$. **(2p)**

d) Consider a modified problem where the fixed support is replaced with a rotational spring with stiffness $s = 3 \times 10^4\,\text{Nm}$. This gives a new boundary condition at L (of a mixed/Robin type): $T(L) = -s\,\varphi(L)$. For this modified problem, determine the new system of equations (you do not have to determine the rotation as in task c. **(2p)**

Problem 2: Heat equation

In this problem, we shall solve the heat equation,

$$\boldsymbol{\nabla}^{\mathrm{T}}\boldsymbol{q} = h \text{ in } \Omega_{\mathrm{F}} \tag{2.1}$$

on the simple rectangular domain, Ω_{F}, below.

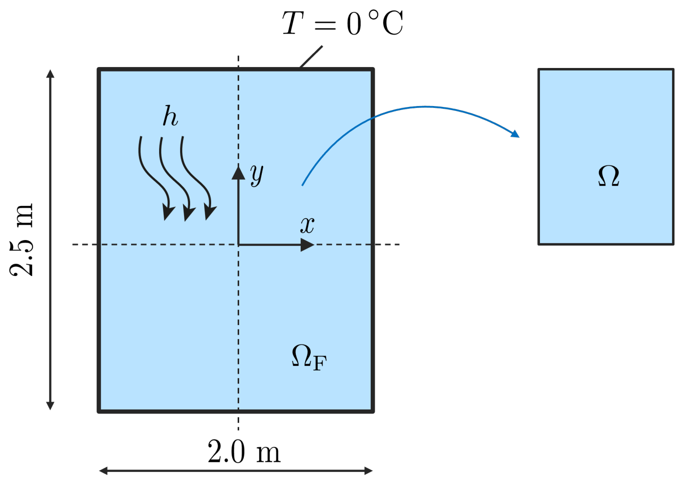

Figure 2: Full domain of Problem 2 with loads and boundary conditions. The right image shows the reduced domain Ω which should be analyzed.

The temperature, $T = 0\,^\circ\mathrm{C}$, is known around the entire boundary. The material is isotropic with heat conductivity, $k = 1\,\mathrm{W/(m^\circ C)}$, such that the heat flux vector can be written $\boldsymbol{q} = -k\,\boldsymbol{\nabla}T$. The thickness can be set to $1\,\mathrm{m}$ and a constant volumetric heat source, $h = 1\,\mathrm{W/m^3}$ is acting on the entire domain. Due to symmetry, we can (and will) reduce the computational costs by only analyzing the upper right quarter ($x \geq 0$ and $y \geq 0$). This domain will be denoted Ω.

To solve the problem, you are given a mesh `mesh_data.mat` of Ω with linear triangle elements consisting of the following parts,

- `Coord`: The coordinates of each node `[num_nodes,2]`
- `Dofs`: The degree of freedom for each node `[num_nodes,1]`
- `Edof`: The element degrees of freedom matrix `[num_elements,4]`
- `Ex` and `Ey`: The element x and y coordinates, `[num_elements,3]`
- The degrees of freedom for the nodes located at

- The bottom of Ω: `bottom_dofs`
- The right side of Ω: `right_dofs`
- The top of Ω :`top_dofs`
- The left side of Ω: `left_dofs`

In addition, you also get the solution, `solution.mat`, stored as `solution_vector`. This can be used to validate your solution as well as use for the postprocessing tasks if the FE solution is not found.

Hint: You can use the code provided in the file `Problem_2.py` to load these data structures.

Tasks:

a) Draw the domain Ω and state the complete strong form for the problem on Ω including the boundary conditions. **(0.5p)**

b) Derive the weak form of the heat equation. **(0.5p)**

c) Derive the global FE form of the heat equation for the problem at hand. **(0.5p)**

d) Use CALFEM to solve the FE problem and note that you can use the CALFEM function `flw2te` to calculate the element contributions. **(2.5p)**

e) Calculate the temperature at the point, $\boldsymbol{x} = [0.650\,\mathrm{m}, 0.375\,\mathrm{m}]^{\mathrm{T}}$, located in element number 18. **(1.0p)**

f) Explain how the code in task b should be modified (what must be added and what can be removed) if the Dirichlet boundary conditions are replaced by Robin (convection) boundary conditions. How would the maximum temperature change (assuming the surrounding temperature is the same temperature as previously prescribed by the Dirichlet conditions)? **(1.0p)**

Problem 3

Assume a mechanical FE simulation has been performed of a component with thickness $t = 0.015\,\mathrm{m}$. The material is assumed linear elastic with Young's modulus $E = 210\,\mathrm{GPa}$ and Poisson's ratio $\nu = 0.3$. The simulation used a discretization of linear isoparametric triangular elements as can be seen in Figure 3. In this problem, you will only consider a single element (among those

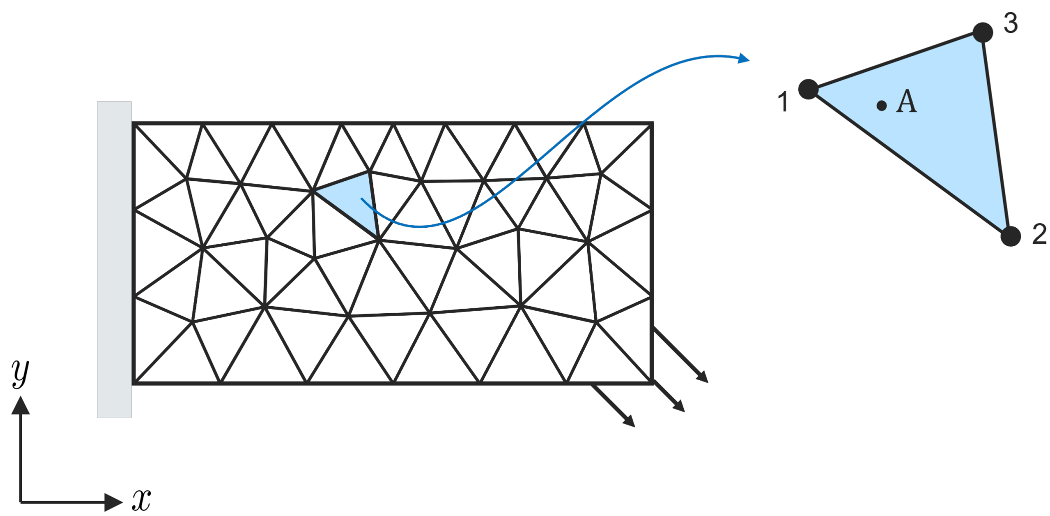

Figure 3: Illustration of component with discretization and the considered element in Problem 3.

used in the FE simulation). The coordinates for the three nodes (on the element level) are

$$\boldsymbol{x}_1^{\mathrm{e}} = [0.010\,\mathrm{m}, 0.014\,\mathrm{m}]^{\mathrm{T}}, \qquad \boldsymbol{x}_2^{\mathrm{e}} = [0.021\,\mathrm{m}, 0.009\,\mathrm{m}]^{\mathrm{T}}, \qquad \boldsymbol{x}_3^{\mathrm{e}} = [0.015\,\mathrm{m}, 0.018\,\mathrm{m}]^{\mathrm{T}}$$

Tasks:

a) Consider a point A within the element with local coordinates $[0.2, 0.1]^{\mathrm{T}}$ in the parent domain. (The point is shown in the right part of Figure 3.) For this point, calculate the corresponding global coordinates $\boldsymbol{x}_{\mathrm{A}} = [x_{\mathrm{A}}, y_{\mathrm{A}}]^{\mathrm{T}}$. **(1p)**

b) Given an FE-solution with the following nodal displacements of the element

$$\boldsymbol{u}_1^{\mathrm{e}} = [4.1, 1.0]^{\mathrm{T}} \times 10^{-5}\,\mathrm{m}, \qquad \boldsymbol{u}_2^{\mathrm{e}} = [3.5, 1.6]^{\mathrm{T}} \times 10^{-5}\,\mathrm{m}, \qquad \boldsymbol{u}_3^{\mathrm{e}} = [3.8, 1.7]^{\mathrm{T}} \times 10^{-5}\,\mathrm{m}$$

calculate for point A: the displacement vector, $\boldsymbol{u}_{\mathrm{A}}$, strains, $\boldsymbol{\epsilon}_{\mathrm{A}}$, and stresses, $\boldsymbol{\sigma}_{\mathrm{A}}$. Assume that the component is in a state of plane stress where the constitutive matrix is given as

$$\boldsymbol{D} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}.$$ **(3p)**

c) Using numerical integration with a single integration point, compute the element volume load vector $\boldsymbol{f}_l^e = \int_{A^e} [\boldsymbol{N}^e]^{\mathrm{T}} \boldsymbol{b}\, t \, \mathrm{d}A$ for the considered element. The body force vector is a function of the global coordinates and is given as $\boldsymbol{b}(x, y) = [2x, 3xy]^{\mathrm{T}} \times 10^6 \, \mathrm{N/m^3}$. **(2p)**

Remark: using only one integration point will not integrate the integrand exactly (for this particular problem) but this is disregarded for the sake of simplifying the problem.