Examination

MHA021 Finite Element Method

Date and time:	2025-04-16, 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 April 29 the latest.
Review:	Tuesday April 29 , 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.

Each Python file that you create must contain your anonymous code as a comment on the first line, e.g.

Code snippet 1 Replace with **your** anonymous code

```
# MHA021-1234-XYZ
```

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 the bar in Figure 1 which is loaded by a force P at its left end and a distributed axial load $b(x)$ [N/m]. The strong form for this problem is given as: find the axial displacement $u(x)$ such that

$$\begin{cases} -\dfrac{\mathrm{d}}{\mathrm{d}x}\left[E\,A\dfrac{\mathrm{d}u}{\mathrm{d}x}\right] = b \quad 0 < x < L \\ u(L) = 0 \\ \dfrac{\mathrm{d}u}{\mathrm{d}x}(0) + \dfrac{P}{E\,A} = 0 \end{cases}$$

where E is Young's modulus, A is the cross-sectional area and $b(x) = b_0$ (constant).

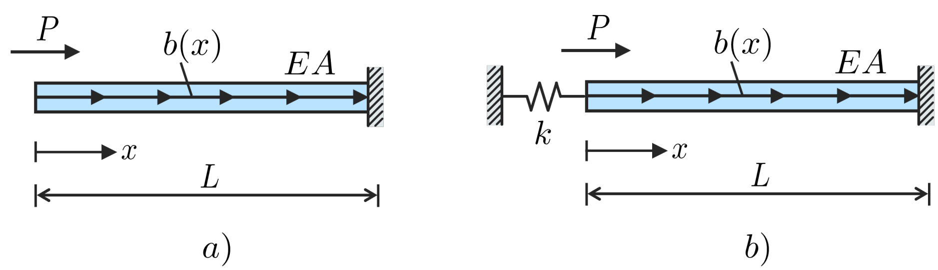

Figure 1: Clamped bar studied in Problem 1. Subfigure a is used for subtask a-d, and subfigure b is used for task e.

Tasks:

a) From the strong form, derive the weak form. **(1.0p)**

b) From the weak form, derive the global finite element formulation $\mathbf{K}\,\mathbf{a} = \mathbf{f}$. **(1.0p)**

c) For the case of a linear approximation of the displacement, $u(x)$, derive the element stiffness matrix $\mathbf{K}^e$ and element load vector $\mathbf{f}_l^e$ (associated with the distributed axial force b) for an element with nodes x_i and x_{i+1}. **(1.0p)**

d) Consider the bar discretized into two equally long linear elements. Determine the explicit system of equations $\mathbf{K}\,\mathbf{a} = \mathbf{f}$ and determine the axial displacement at $x = 0$. Let $E = 200\,\text{GPa}$ and $A = 2 \times 10^{-4}\,\text{m}^2$, $P = 1\,\text{kN}$, $L = 1\,\text{m}$ and $b_0 = -2P/L$. **(1.5p)**

e) If a translational spring with stiffness $k = \frac{3EA}{L}$ is added to the free end ($x = 0$) the corresponding boundary condition is changed to $\dfrac{\mathrm{d}u}{\mathrm{d}x}(0) + \dfrac{P}{E\,A} = \dfrac{k}{E\,A}u(0)$. Determine the new system of equations (as in task d) and determine the axial displacement at $x = 0$. **(1.5p)**

Problem 2: Heat equation

In this problem, we shall solve the heat equation,

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

i for the inhomogeneous plate with a hole, Ω, shown below.

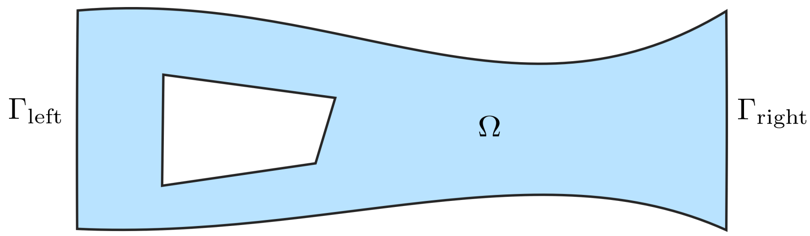

Figure 2: The plate with a hole to be analyzed in this problem.

The temperatures, $T_1 = 20\,°\mathrm{C}$, is prescribed on Γ_{left}, and $T_2 = 0\,°\mathrm{C}$ on Γ_{right}. The remaining boundaries are insulated. The material is isotropic with heat conductivity, $k = 1\,\mathrm{W/(m°C)}$, such that the heat flux vector can be written $\boldsymbol{q} = -k\,\boldsymbol{\nabla}T$. The thickness can be set to $0.01\,\mathrm{m}$, and the internal heat supply, $h = 0.5\,\mathrm{W/m^3}$

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

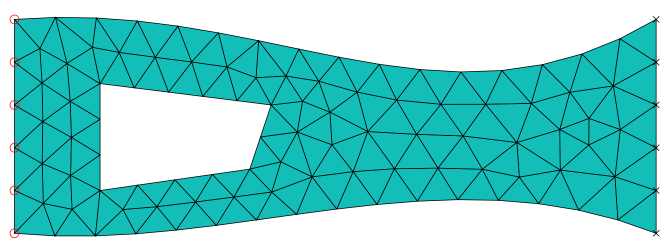

Figure 3: Markers show the nodes included in the boundary sets.

- `Coord`: The coordinates [m] 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 [m], `[num_elements,3]`

- The degrees of freedom for the nodes located at
  - The right side of Ω, Γ_{right}: `right_dofs`, `[num_dofs_right,1]`
  - The left side of Ω, Γ_{left}: `left_dofs`, `[num_dofs_left,1]`

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

Tasks:

a) 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 including the boundary conditions **(0.5p)**

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

d) Explain (with equations) how the equation system can solved while accounting for non-zero Dirichlet boundary conditions via partitioning of the equation system. **(1.0p)**

e) Show that for the three-noded triangle element, the flux, $\boldsymbol{q}$, is constant inside each element **(1.0p)**

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

Problem 3

Assume a mechanical FE simulation has been performed of a component with thickness, $t = 0.025\,\mathrm{m}$. 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 bilinear isoparametric quadrilateral elements as can be seen in Figure 4. In this problem, you will only consider a single element (among those used in the FE simulation). The coordinates for the four nodes (on the element level) are

$$\boldsymbol{x}_1^{\mathrm{e}} = [0.014, 0.010]^{\mathrm{T}}\ \mathrm{m}, \quad \boldsymbol{x}_2^{\mathrm{e}} = [0.021, 0.009]^{\mathrm{T}}\ \mathrm{m}, \quad \boldsymbol{x}_3^{\mathrm{e}} = [0.018, 0.018]^{\mathrm{T}}\ \mathrm{m} \quad \boldsymbol{x}_4^{\mathrm{e}} = [0.012, 0.016]^{\mathrm{T}}\ \mathrm{m}$$

Note: In this task you may not use any CALFEM routines.

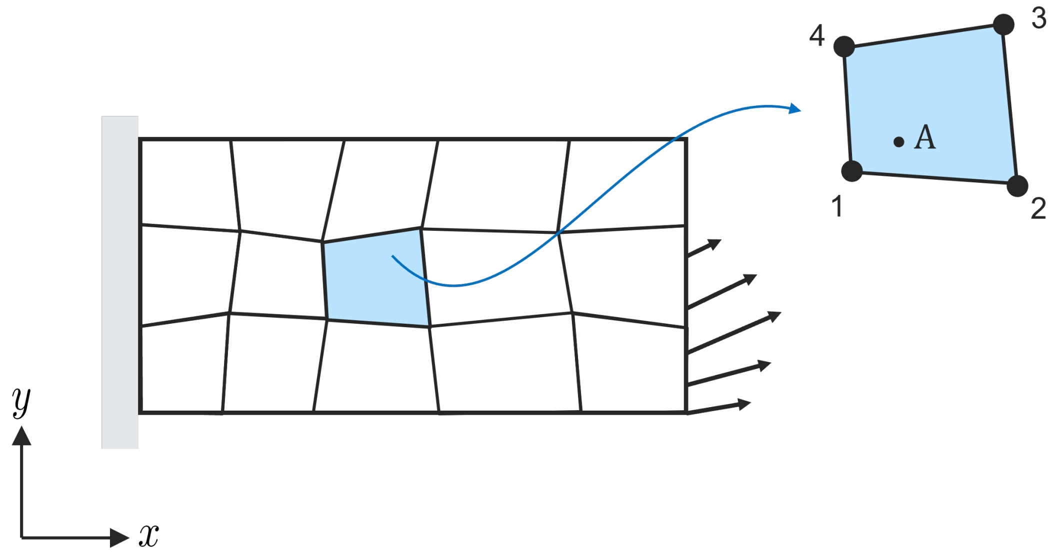

Figure 4: Illustration of component with discretization and the considered element in Problem 3.

Tasks:

a) Consider a point A within the element with local coordinates $[0.25, 0.15]^{\mathrm{T}}$ in the parent domain. (The point is shown in the right part of Figure 4.) 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 (given as $10^{-5}\,\mathrm{m}$),

$$\boldsymbol{u}_1^{\mathrm{e}} = [4.1, 1.0]^{\mathrm{T}}, \quad \boldsymbol{u}_2^{\mathrm{e}} = [3.5, 1.6]^{\mathrm{T}}, \quad \boldsymbol{u}_3^{\mathrm{e}} = [3.8, 1.7]^{\mathrm{T}}, \quad \boldsymbol{u}_4^{\mathrm{e}} = [3.7, 1.5]^{\mathrm{T}}$$

calculate the displacement vector, $\boldsymbol{u}_{\mathrm{A}}$, at point A **(1p)**

c) Using a single Gauss quadrature point, calculate the element stiffness matrix **(4p)**

$$\boldsymbol{K}^{\mathrm{e}} = \int\limits_{\Omega^{\mathrm{e}}} \left[\boldsymbol{B}^{\mathrm{e}}\right]^{\mathrm{T}} t \boldsymbol{D} \boldsymbol{B}^{\mathrm{e}} \mathrm{d}\Omega$$

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.