Examination

MHA021 Finite Element Method

Date and time:	2025-08-25, 14.00-18:00
Instructors:	Jim Brouzoulis (phone 2253) and Knut Andreas Meyer (phone 1495). An instructor will visit the exam around 14:30 and 16:30.
Solutions:	Example solutions will be posted within a few days after the exam on the course homepage.
Grading:	Will be posted on Canvas on September 15, the latest.
Review:	Tuesday September 16, 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 distributed axial load, with linearly increasing intensity, given as $b(x) = b_0 \frac{x}{L}$ [N/m]. The strong form for this problem is given as: find the axial displacement $u(x)$ such that

$$\begin{cases} -\dfrac{\mathrm{d}}{\mathrm{d}x}\left[EA\dfrac{\mathrm{d}u}{\mathrm{d}x}\right] = b \quad 0 < x < L \\ u(0) = 0 \\ N(L) = EA\dfrac{\mathrm{d}u}{\mathrm{d}x}(L) = 0 \end{cases}$$

where E is Young's modulus, A is the cross-sectional area.

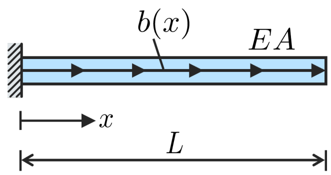

Figure 1: Clamped bar studied in Problem 1.

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) Consider the bar discretized into three equally long linear elements. Determine the explicit system of equations $\mathbf{K}\,\mathbf{a} = \mathbf{f}$. Let $E = 200\,\mathrm{GPa}$ and $A = 2 \times 10^{-5}\,\mathrm{m}^2$, $b_0 = 10\,\mathrm{kN/m}$ and $L = 1\,\mathrm{m}$. Approximate the load as piecewise constant over each element. **(1.5p)**

d) The exact solution to the strong form is $u(x) = \frac{b_0\,x(3L^2 - x^2)}{6EAL}$ and the corresponding normal force is $N(x) = \frac{b_0(L^2 - x^2)}{2L}$.

Determine the error in axial displacement at $x = L$ and error in normal force at $x = 0$ for the discretization above. **(1.0p)**

e) In subtask c the load was assumed piecewise constant, now consider the actual variation of $b(x)$ and determine the (consistent) element load vector $\mathbf{f}_1^e$, and solve the problem again with this load vector **(1.5p)**

Problem 2: Heat equation

In this problem, we shall solve the heat equation,

$$\boldsymbol{\nabla}^{\mathrm{T}}\boldsymbol{q} = 0 \text{ in } \Omega \tag{2.1}$$

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

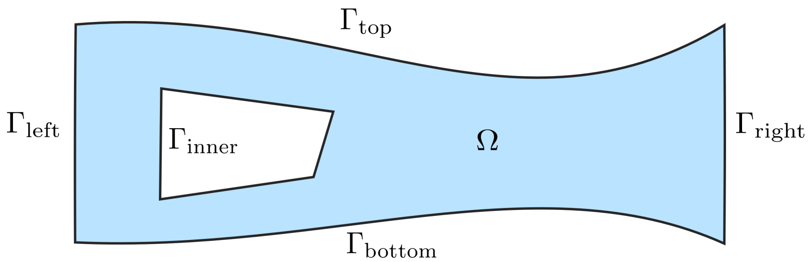

Figure 2: The plate with a hole to be analyzed in this problem.

On the left and right boundaries, Γ_{left} and Γ_{right}, the temperatures, T_{left} and T_{right}, are $0\,^{\circ}\mathrm{C}$. The inner boundary, Γ_{inner}, is heated by $2\,\mathrm{W/m^2}$ (going into the body). The material is isotropic with heat conductivity, $k = 1\,\mathrm{W/(m^{\circ}C)}$, such that the heat flux vector can be written $\boldsymbol{q} = -k\,\boldsymbol{\nabla}T$.

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:

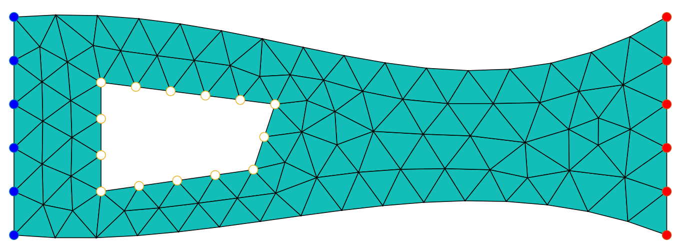

Figure 3: Markers show the nodes included in the boundary sets.

- `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]`

- Information about the inner boundary
  - The edge degree of freedoms: `inner_edgedofs`: `[num_inner_edges,2]`
  - The edge x-coordinates: `inner_ex`: `[num_inner_edges,2]`
  - The edge y-coordinates: `inner_ey`: `[num_inner_edges,2]`

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

Tasks:

a) Based on the description above, state the boundary conditions on equation form for every part of the boundary **(0.5p)**

b) Derive the weak form of the heat equation, using the problem-specific boundary conditions **(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 calculate the norm of the temperature degree of freedoms. **(3.0p)**
Hint 1: You may use the CALFEM function `flw2te` to calculate the element contribution.
Hint 2: If you are not able to implement the non-zero Neumann boundary conditions, you may set $T_{\text{right}} = 10\,°\text{C}$ instead of $0\,°\text{C}$ to keep working with this problem. Please state clearly if you use this option.

e) Calculate the heat flux in each element and report the heat flux vector in element 10, 20, and 30. **(1.5p)**
Hint 1: You may use the CALFEM function `flw2ts` to calculate the heat flux.

Problem 3

Assume a plane stress 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 constitutive matrix is then given as

$$\boldsymbol{D} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

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.

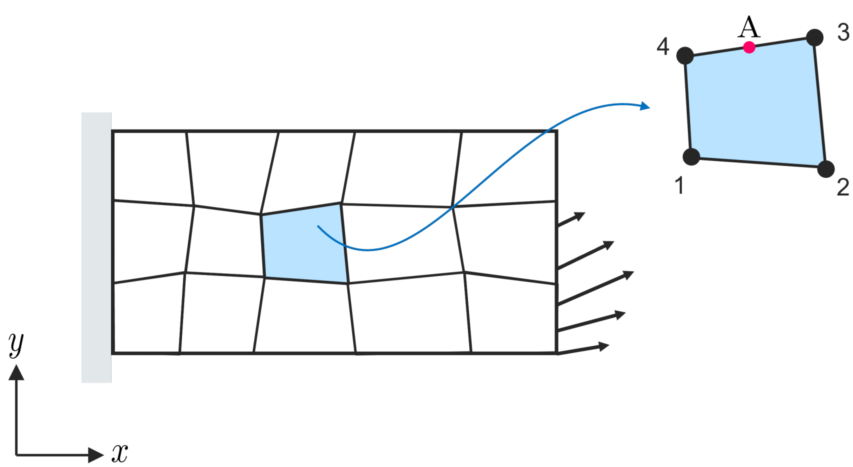

Figure 4: Illustration of component with discretization and the considered element in Problem 3.

Tasks:

a) The point A is exactly in the middle between nodes 3 and 4. What are the local, $\boldsymbol{\xi}_{\mathrm{A}}$ and global, $\boldsymbol{x}_{\mathrm{A}}$, coordinates of point A? **(1p)**

b) Given an FE-solution with the following nodal displacements of the element (given as 10^{-5} 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, $\boldsymbol{u}_{\mathrm{A}}$, strain, $\boldsymbol{\epsilon}_{\mathrm{A}}$, and stress, $\boldsymbol{\sigma}_{\mathrm{A}}$ at point A **(3p)**

c) Use numerical integration with 2x2 Gauss quadrature points to calculate the element area, $A^{\mathrm{e}} = \int_\Omega \mathrm{d}\Omega$. **(2p)**