MHA021 Finite element method
Re-exam 2022-08-22, 14:00-18:00

Examiner: Jim Brouzoulis.

Instructor: Martin Fagerström (phone 070-224 8731) will visit the exam around 15:00 and 17:00.

Solution: Example solutions will be posted within a few days after the exam on the course homepage.

Grading: The grades will be reported to the registration office on 9 September 2022 the latest.

Review: It will be possible to review the grading at the Division of Dynamics. Please contact Jim Brouzoulis to schedule an appointment for reviewing the exam corrections.

Permissible aids: Chalmers type approved pocket calculator and MATLAB with the CALFEM toolbox. On the computer, you can find the CALFEM manual (excluding the examples section) and the CALFEM finite element method functions. **Note**: A formula sheet is also appended to this exam paper.

Exam instructions

Please note that the solutions to some of the problems require or benefit from being solved by use of MATLAB and CALFEM. Be extra careful to read the instructions for these problems.

The CALFEM finite element files are provided under the directory `C:\__Exam__`. Should you need to refer to the CALFEM manual, you can find this also (excluding the examples section) under `C:\__Exam__`. Please note that the CALFEM function `extract.m` also exists in the CALFEM directory as `extract_dofs.m` (to avoid possible conflicts with the built-in MATLAB-function `extract.m`).

Please also note that we will collect all files saved under the directory `C:\__Exam__\MHA021` and subdirectories. This in order to be able to review these files during the exam corrections. Therefore, it is very important that you save all your files under `C:\__Exam__\MHA021` in the appropriate subdirectories created for each problem. **It is also absolutely necessary that you write the name of your computer on the cover page for the exam! Please also write your anonymous exam code inside each MATLAB file.**

Finally, remember to close MATLAB and log-out from the computer when you are finished with the exam.

Problem 1

Consider the water filtering system depicted in Figure 1. From the left, ground water is flowing into the sand filter with varying cross section $A(x) = A_0\,(1 + 0.2x/L)$ and of length L. The total amount of water passing the inlet at the left boundary is $Q_L = q_L A(x = 0)$, and the water pressure at the filter outlet is p_R.

Since the cross-section $A(x)$ of the filter varies "slowly" along its length, the amount of water flowing through the filter, denoted $q(x)$, can be descried well by Darcy's law as:

$$q(x) = -k(x)\frac{\mathrm{d}p(x)}{\mathrm{d}x}$$

where $k(x) = k_0\,(1 + 0.2x/L)$ is the so-called permeability and $\frac{\mathrm{d}p(x)}{\mathrm{d}x}$ is the gradient of the so-called "pore pressure" (i.e. the pressure of the water flowing through the porous sand at position x). Furthermore, since water can only enter and leave the sand filter at $x = 0$ and $x = L$, respectively, the 1D balance equation for the water flow in the sand filter is given by:

$$\frac{\mathrm{d}}{\mathrm{d}x}\left(A(x)q(x)\right) = 0.$$

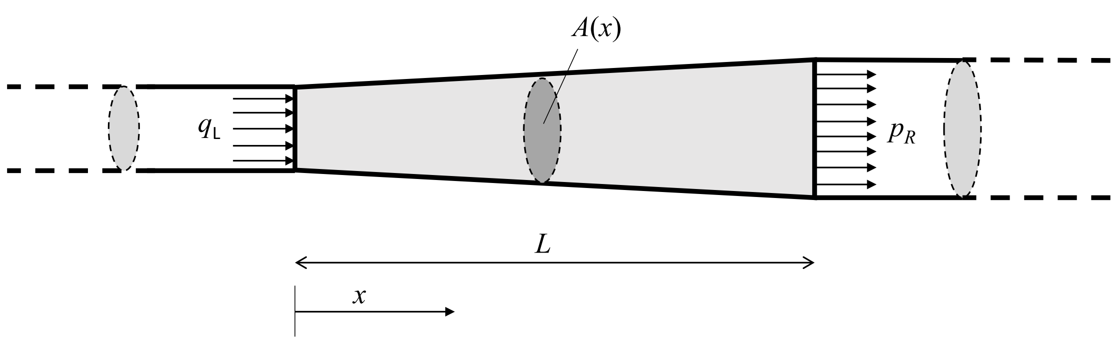

Figure 1: Sketch of the 1D elastic bar with varying cross-section and distributed load.

Data to be used for the current problem:

$A_0 = 0.2$ dm^2,
$L = 3$ dm,
$Q_L = 0.2$ dm^3/s,
$p_R = 100$ kPa,
$k_0 = 6 \cdot 10^{-6}$ m^2/(Pa·s) - Note the unit!!

Tasks on the next page!

Tasks:

(a) **Derive and state the strong form of the problem in terms of the primary unknown pressure field $p(x)$.** **(1.0p)**

(b) **Derive the corresponding weak form for the same problem.** **(1.0p)**

(c) **Derive the FE-form for the same problem using Galerkin's method**. Specify the contents (in general terms) of any matrices or vectors you introduce. **(1.0p)**

(d) Discretise the sand filter region into three linear elements with equal length. Then **compute the resulting element stiffness matrix K^e for the middle element**. Note that the numerical values are expected, so you may benefit from using MATLAB for some of the computations. **(1.5p)**

(e) **Compute all known components of the global load vector of the resulting FE-problem, and also define which component(s) that are unknown until the problem has been solved.** Again, actual numerical values are expected for all components that can be computed prior to solving the problem. **(1.5p)**

Problem 2

A sealant with rectangular cross section Ω containing an elliptical hole is placed between two insulating walls to minimise the heat outflow from a heat chamber, cf. Figure 2. The temperature of the air in the heat chamber is T_{hot} and the ambient temperature of the air on the outside is T_{cold}. Furthermore, inside the sealant flows cooling water with the temperature T_w.

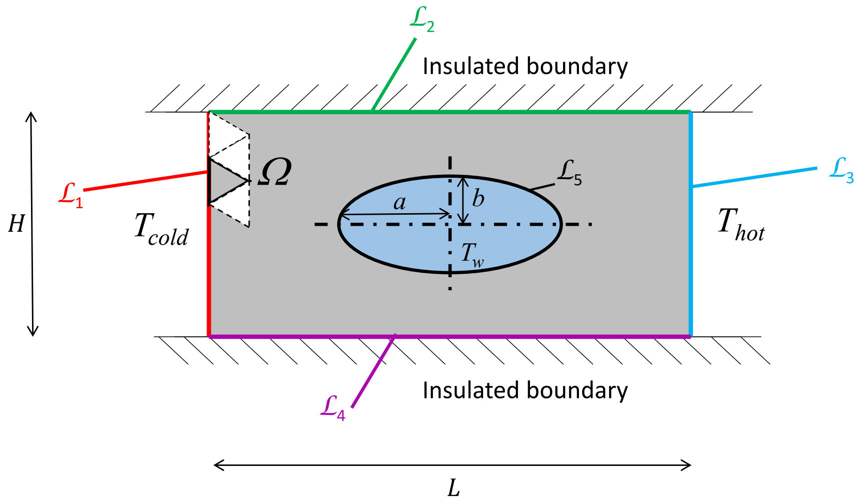

Figure 2: Sketch of the 2D heat flow problem considered in Problem 2. Please note that a few elements of the mesh considered in subtask c) and shown in Figure 3 are also indicated in the figure.

The sealant material is assumed to be isotropic (w.r.t heat flow) and obey Fourier's law $\boldsymbol{q} = -k\boldsymbol{\nabla}T$. Furthermore, the material in the sealant is such that the heat conductivity is k and that the heat transfer coefficient between the sealant and air is α_{air}. Between the sealant and water it is α_w.

No heat is assumed to flow out of the plane shown in Figure 2. Thus, the problem can be considered as a 2D heat flow problem. For such a problem, the weak form of the heat balance equation is generally defined as:

$$\int_A \left(\boldsymbol{\nabla}^T v\right) \boldsymbol{D} t \boldsymbol{\nabla} T \,\mathrm{d}A \;\; = \;\; \int_A Q t \,\mathrm{d}A - \int_{\mathcal{L}} v q_n t \,\mathrm{d}\mathcal{L}$$

where A is the cross-section domain, v is an arbitrary weight function, $\boldsymbol{D} = -k\mathbf{1}$ is the constitutive heat conductivity matrix, $\mathbf{1}$ is the 2 x 2 identity matrix, $\boldsymbol{\nabla} = [\partial/\partial x \quad \partial/\partial y]^T$ is the gradient operator, t is the thickness (here constant), Q is any external heat supply and q_n is the boundary heat outflux (positive if heat leaves the body).

Tasks on the next page!

Tasks:

(a) **Considering symmetry, identify the smallest part of the cross section that can be analysed by FE and state the complete weak form for this particular problem.** For full points you need to include a sketch of the simulation domain where you clearly indicate the different boundary parts related to your weak form. **(1.5p)**

(b) Introduce a suitable linear finite element approximation, use Galerkin's method and **derive the corresponding FE-form of the current problem.** Make sure to clearly indicate the contents of any matrices that you introduce. **(2.0p)**

(c) Consider the shaded element in Figure 3 (also indicated in Figure 2). For this element, implement a MATLAB function that computes the two convective element contributions (one load vector contribution and one stiffness[1] matrix contribution) given the following input: **(2.5p)**

- nodal x and y coordinates
- thickness t (here $t = 1$)
- heat transfer coefficient α
- ambient (outside) temperature

Also report the values of these contributions given the following input: $L = 0.1$ m, $H = 0.3$ m, $\alpha = 10$ W/(m^2 °C), $T_{cold} = 20$ °C.

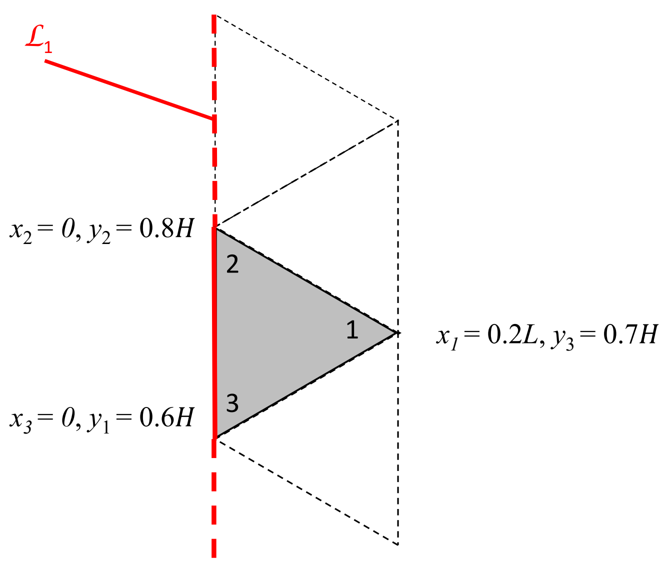

Figure 3: The shaded element with one edge along $\mathcal{L}_1$ which is to be considered in Problem 2c.

Problem 3

The weak form of a 2D plane stress elasticity problem can be written as

$$\int_A t\left[\tilde{\nabla}\boldsymbol{v}\right]^{\mathrm{T}}\boldsymbol{\sigma}(\boldsymbol{u})\,\mathrm{d}V = \int_A t\,\boldsymbol{v}^{\mathrm{T}}\,\boldsymbol{b}\,\mathrm{d}V + \int_{\Gamma_g} t\,\boldsymbol{v}^{\mathrm{T}}\,\boldsymbol{g}\,\mathrm{d}S + \int_{\Gamma_h} t\,\boldsymbol{v}^{\mathrm{T}}\,\boldsymbol{h}\,\mathrm{d}S$$

$$\boldsymbol{u} = \boldsymbol{g} \text{ on } \Gamma_g$$

where t is the thickness, $\boldsymbol{v}$ is an arbitrary weight function, $\boldsymbol{\sigma}$ is the stress vector, $\boldsymbol{b}$ is the body load vector, $\boldsymbol{g}$ are prescribed displacements on Γ_g, $\boldsymbol{h}$ are prescribed tractions on Γ_h.

Let us now consider a thermoelastic material behaviour on the form

$$\boldsymbol{\sigma} = \boldsymbol{D}\left(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{\mathrm{th}}\right),$$

where $\boldsymbol{\varepsilon}^{\mathrm{th}}$ are the thermal strains and $\boldsymbol{\varepsilon}$ are the total strains given by $\boldsymbol{\varepsilon} = \tilde{\nabla}\boldsymbol{u}$. For an isotropic material the thermal strains are computed as

$$\boldsymbol{\varepsilon}^{\mathrm{th}} = \begin{pmatrix} \varepsilon_x^{\mathrm{th}} \\ \varepsilon_y^{\mathrm{th}} \\ \gamma_{xy}^{\mathrm{th}} \end{pmatrix} = \overline{\alpha}\,\Delta T \begin{pmatrix} 1 \\ 1 \\ 0 \end{pmatrix},$$

where ΔT is a given temperature change from the reference temperature $\Delta T = T(x, y) - T_0$. Finally, the consitutive matrix $\boldsymbol{D}$ is given as

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

Tasks on the next page!

Tasks:

(a) From the weak form above, **show that accounting for a change in temperature, of a structure, leads to an additional load vector in the local FE form according to:** **(2.0p)**

$$\boldsymbol{f}^e_{\mathrm{th}} = \int_{A^e} t\,\overline{\alpha}\,\Delta T\,\boldsymbol{B}^{\mathrm{T}}\,\boldsymbol{D}\begin{pmatrix}1\\1\\0\end{pmatrix}\mathrm{d}A$$

.

(b) A previous FE simulation has resulted in a temperature distribution $T(x, y)$ as sketched in the figure below. We are now interested in performing an elastic analysis with the temperature field as input. Consider the highligted triangular element which has the following nodal coordinates.

$$N_{16} = (150, 140)\ \mathrm{mm} \qquad N_{43} = (200, 180)\ \mathrm{mm} \qquad N_{78} = (190, 110)\ \mathrm{mm}$$

The element approximation is linear and from the previous simulation the nodal temperatures have been determined as:

$$T_{16} = 45\,^{\circ}\mathrm{C}, \qquad T_{43} = 64\,^{\circ}\mathrm{C}, \qquad T_{78} = 27\,^{\circ}\mathrm{C}.$$

Determine the thermal load vector $\boldsymbol{f}^e_{\mathrm{th}}$ for this particular element using a three point Gauss integration. This integration scheme is found in the formula sheet. **(4.0p)**

Data to be used for the current problem:

The reference temperature is $T_0 = 15\,^{\circ}\mathrm{C}$,
$E = 210$ GPa,
$\nu = 0.3$,
$t = 10$ mm,
$\overline{\alpha} = 12 \cdot 10^{-6}\,1/^{\circ}\mathrm{C}$,

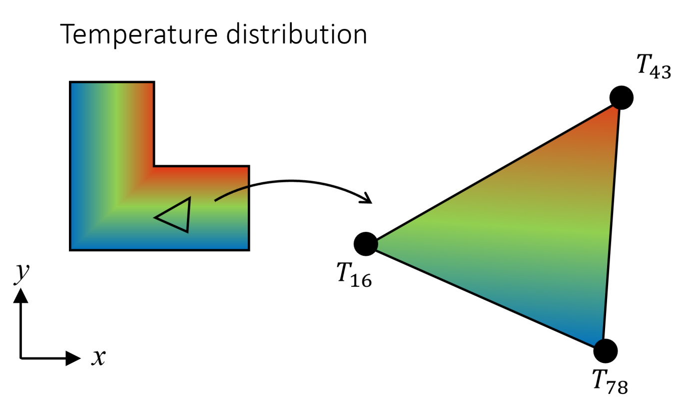

Figure 4: Illustration of the temperature field from a FE simulation and the element studied in Problem 3.