Examination

MHA021 Finite Element Method
VSM167 Finite Element Method – basics

Date and time:	2023-08-21, 14.00-18.00
Instructors:	Martin Fagerström (phone 1300) and Jim Brouzoulis (phone 2253). An instructor will visit the exam around 15:00 and 17:00.
Solutions:	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 8 September the latest.
Review:	For a review of the exam corrections, please make an appointment with your examiner.
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 MATLAB including CALFEM. If you use MATLAB and CALFEM as part of your solutions, you must make sure to also hand in any MATLAB 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 MATLAB files that you want to hand in. Finally, **it is also absolutely necessary that you write the name of your computer on the cover page for the exam!**

Note that most CALFEM files (but not all) are provided for your convenience. In addition, please note that the CALFEM function `extract.m` also exists in the CALFEM directory as `extract_dofs.m` (to avoid a conflict with a built-in MATLAB function). These CALFEM finite element files can be found under the directory `C:\__Exam__\Assignments`. 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 `C:\__Exam__\Assignments`.

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

Problem 1

Consider the bar in Figure 1 which is supported by a spring at the right end. The axial displacement $u(x)$ of the bar is given as the solution to the boundary value problem

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

where E is Young's modulus, A is the cross-sectional area, $b(x)$ is an axially distributed force [N/m], and k is the spring stiffness [N/m].

Tasks:

a) **Derive the weak form of the problem and be mindful of the boundary conditions**. **(2.0p)**

b) **Derive the (global) FE formulation from the weak form in sub-problem a)**. **(2.0p)**

c) Consider the bar discretized into three equally long linear elements. **Determine the explicit system of equations $\mathbf{K}\,\mathbf{a} = \mathbf{f}$ and determine the axial displacement at** $x = L$. Let E and A be constants, $k = \frac{2EA}{L}$ and $b = \frac{P}{L}$ (constant). **(2.0p)**

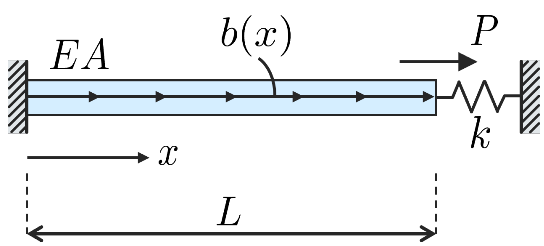

Figure 1: Spring supported bar considered in Problem 1.

Problem 2

Consider a square membrane with side length $L = 4\,\mathrm{m}$ and loaded with a distributed load $q = 150\ \mathrm{N/m^2}$ in the z-direction as shown in Figure 2; the membrane is pre-tensioned with a force $S = 1000\ \mathrm{N}$.

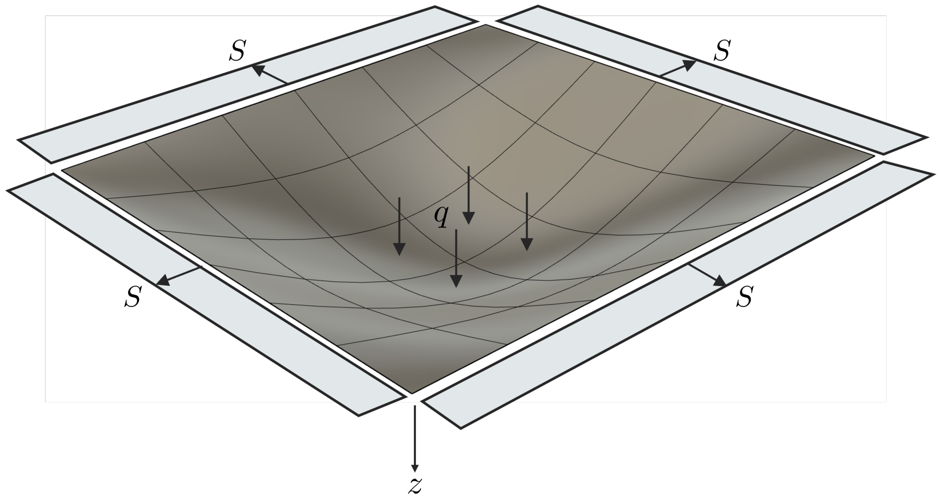

Figure 2: Square membrane considered in Problem 2. Note that the distributed load q acts over the entire surface.

The deflection $w(x, y)$ (translation in z-direction) of the membrane satisfies the boundary value problem

$$\begin{cases} -\operatorname{div}\left(\nabla w(x,y)\right) &= \dfrac{q}{S} \quad \text{in } \Omega \\ w(x,y) &= 0 \quad \text{on } \Gamma_{\mathrm{g}} \end{cases}$$

with notations as in Figures 2 and 3.

Tasks:

a) **Derive the weak form of the problem**. Make sure not to exclude any terms due to specific boundary conditions at this stage **(2.0p)**

b) Based on a), **derive the (global) finite element form of the problem** with test functions according to the Galerkin method. **Also show what the element $\boldsymbol{B}^{\mathrm{e}}$-matrix looks like** for a linear triangular element with 3 basis functions N_1^e, N_2^e and N_3^e. **(2.0p)**

c) Using the symmetry of the problem, consider 1/8th of the domain (Ω') as indicated in the right of Figure 3. Assume that the domain Ω' is discretized with a single 3-noded triangular

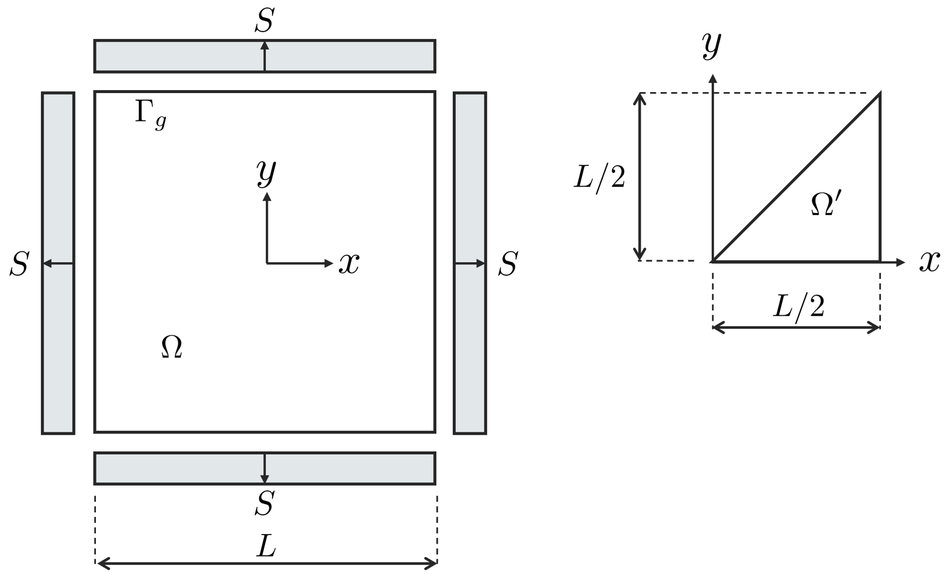

Figure 3: Shows the original domain of the problem for Problem 2 (left), and the reduced domain when considering symmetry (right).

element. With node numbering starting in the lower left corner and then moving counter-clockwise, the basis functions become $N_1 = \frac{1}{2}(2-x)$, $N_2 = \frac{1}{2}(x-y)$, and $N_3 = \frac{y}{2}$. **Determine the system of equations $\mathbf{K}\,\mathbf{a} = \mathbf{f}$ and determine the center-point deflection, i.e.** $w(0,0)$. **(2.0p)**

Problem 3

A cantilever beam with an applied linearly varying traction along the top surface is to be analysed under the assumption of plane stress, cf. left part of Figure 4. The length of the beam is $b = 4\,\mathrm{m}$, the height is $a = 0.2\,\mathrm{m}$, the (out-of-plane) thickness is $t = 0.1\,\mathrm{m}$ and the maximum traction magnitude at $x = 0$ is $h_0 = 10\,\mathrm{MPa}$.

The domain is discretised with 4-node isoparametric bilinear elements *of equal size* according to the right part of Figure 4 (note that the mesh is much coarser than appropriate to simplify the analysis).

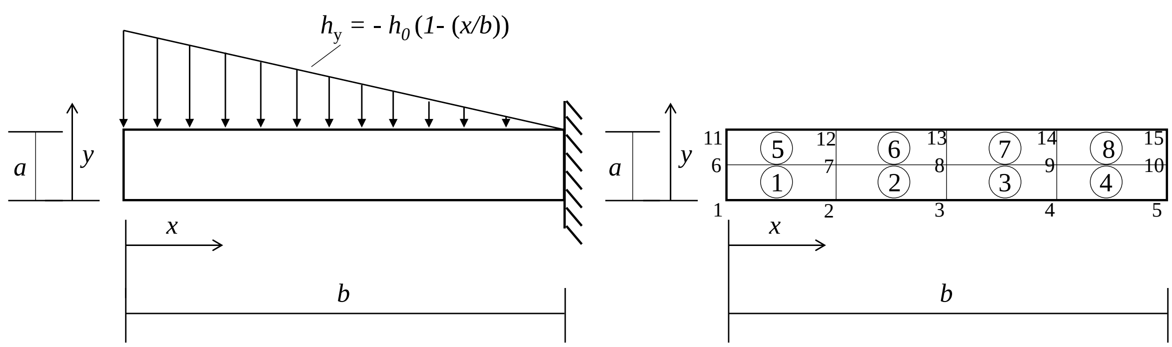

Figure 4: (left): Cantilever beam analysed in Problem 3. (right): The discretisation (mesh) into 4-node bilinear isoparametric elements. Encircled numbers denote element numbers (e.g. ① denotes element number 1), whereas regular numbers denote node numbers.

The 2D elasticity equation on weak form is written as:

$$\int_A \left(\tilde{\nabla}\mathbf{v}\right)^{\mathrm{T}} \mathbf{D}\tilde{\nabla}\mathbf{u}\, t\, \mathrm{d}A = \int_A \mathbf{v}^{\mathrm{T}}\mathbf{b}\, t\, \mathrm{d}A + \int_{\mathcal{L}_g} \mathbf{v}^{\mathrm{T}}\mathbf{t}\, t\, \mathrm{d}\mathcal{L} + \int_{\mathcal{L}_h} \mathbf{v}^{\mathrm{T}}\mathbf{h}\, t\, \mathrm{d}\mathcal{L}$$

for any domain with prescribed displacements $\mathbf{u} = \mathbf{g}$ along $\mathcal{L}_g$ and prescribed tractions $\mathbf{t} = \mathbf{h}$ along $\mathcal{L}_h$.

Tasks:

a) **Derive the FE form of the given problem and provide a sketch of the domain and explain any regions (domains, surfaces, edges etc.) you introduce**. Make sure to specify (in general terms) the contents of any vectors or matrices you introduce. **(2.0p)**

b) **Determine the element boundary load vector $\mathbf{f}_b^e$ for element ⑤ in the given mesh. Also explain how this element load contribution can be assembled in the global load vector f** with proper consideration of a global degree of freedom numbering of your choice. Please note that you may give a symbolic answer, i.e. in terms of h_0, a and so on, and you need not to simplify the answer to a single term for each load component. **(2.0p)**

Hint: If you want, you can (of course) use numerical integration. Then, a convenient change of variable (from a global coordinate x to a local variable ξ) may be $x = \frac{x_i+x_j}{2} + \xi\frac{x_j-x_i}{2}$ where x_i and x_j are the x-coordinates of the two element nodes along the boundary ($x_j > x_i$).

c) For the lower left element (element ①), **compute the Jacobian matrix and its determinant associated with the isoparametric mapping in the midpoint of the element. Also explain how you can verify the correctness of the value of the determinant given the geometry of the element. (2.0p)**