# B1 Class - Overview

This problem set is provided for students and intended to check and consolidate theoretical knowledge of finite element methods by solving problems of structural mechanics in Matlab using the FEM Package. It is expected that generic computational programming experience would be sufficient to understand how solutions work in the original m-files. At the same time, detailed instructions contained in task statements help to ensure that Matlab language and implementation technicalities do not take too much time and attention. Nevertheless, students unfamiliar with Matlab may benefit from reading and watching Matlab tutorials (see http://mathworks.com) that cover the following topics:

- vectors, matrices and multidimensional scalar arrays, creation, indexing (numerical and logical), reshaping;
- arithmetic, matrix operations and solving linear algebraic systems;
- FOR- and WHILE-loops;
- Matlab program structure, function files, script files, workspaces;
- debugging, setting breakpoints, stepping through the code.

Solving problems with the FEM Package also involves other language constructs, however these problem sheets are designed to avoid them. These constructs include

- cell arrays;
- anonymous functions;
- structures.

The FEM Package code provided with the problem set is open and can be examined. The procedures included in the package are commented and optimised for clarity (rather than performance). That is, a student who wishes to explore the stiffness matrix assembly procedure `formStiffnessMatrix.m` will find there a loop that iterates over elements and calls local assembly subroutine, which contains the inner loop over Gauss points etc. Such investigation contributes to understanding of finite element methods and is encouraged, although it is not explicitly stated as part of the tasks.

# Problem sheet

`problem_inc_1.m` aims at solving a plane stress equilibrium problem of stretching of a bar. The desired boundary conditions are the fixed displacements at the right and left edges and traction free top and bottom sides (see Figure 1). As provided, though, the code does not stretch the bar.

Figure 1: Boundary conditions for `problem_inc_1.m`

1. Identify the part of the code that defines boundary conditions.
2. Add the Dirichlet boundary conditions allowing for such stretch.
3. Run the file to make sure that the undeformed state is obtained as a solution.
4. Try to also pull upward the bar in a combined stretch/bending load.