University of Oxford

Department of Engineering Science

# B1 Project

# Finite Elements for Coupled Thermo-elastic Problems

Hilary Term 2022

Candidate Number: XXXXXXX

# Contents

# 1 Introduction

The Finite Element Method is a numerical method used to solve partial differential equations that arise in many engineering and physics related problems. I am developing a MATLAB program to apply it to a thermo-elastic problem, with linear triangular elements.

Here, I consider half of a lead pipe carrying a hot liquid under high pressure, supported by a steel flange (Figure 1 of project brief [1]). My program starts by solving the steady-state heat conduction problem, considering the Dirichlet boundary conditions and distinguishing between both lead and steel components, then goes on to solve the thermo-elasticity problem in plain strain for the temperature distribution approximated by the thermal problem, now introducing Neumann boundary conditions too.

# 2 The Finite Element Programme

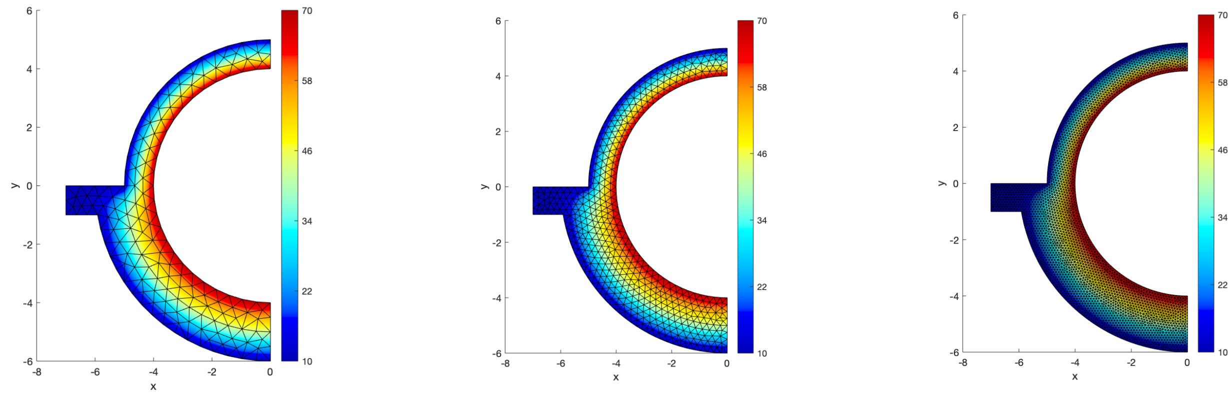
## 2.1 Structuring the Finite Element program with MATLAB

The overall structure of my Finite Element Program is shown below.

```
theMesh = readMesh('Meshes/tube_coarse.txt');  % PRE-PROCESSING, To read mesh from file
[C,D] = buildFESystemThermal(theMesh, vals);
Temp = C\D;                                    % Solving the Thermal problem
[A,B] = buildFESystemElastic(theMesh, vals, Temp);
Displacement = A\B;                            % Solving the Thermo-elastic problem
plotMesh(theMesh, Displacement, true, []);     % POST-PROCESSING, to plot results
```

## 2.2 Building the Finite Element System

The two functions, *buildFESystemThermal* and *buildFESystemElastic* are used to construct the thermal and thermo-elastic Finite Element systems respectively. Exploring the thermo-elastic problem, the function 'buildFESystemElastic' takes as inputs the finite element mesh, the temperatures field (as approximated by the thermal problem) and both the Neumann and Dirichlet boundary conditions to output the global stiffness matrix **A** and the corresponding global nodal vector **B**. The nodal displacements, to be stored in **U** are then found by solving the equation **AU = B**.




(a) Coarse mesh solution (b) Normal mesh solution (c) Fine mesh solution

Figure 2: Comparison of the solutions to the steady-state heat conduction problem obtained from (a) the coarse mesh, (b) the normal mesh and (c) the fine mesh

Applying the finite element method on each of the three meshes yields noticeably very similar solutions, as expected and shown above. Despite this, it is understood that the greater the number of nodes and elements dividing the overall domain, the more the finite element solution matches the exact solution; this the solution to the laplace equation $\nabla^2$U = 0 which, in 2D is solved by means of change and separation of variables, considering the same boundary conditions and utilising orthogonality relations to verify the accuracy of this finite element solution. As appealing as greater computational accuracy sounds with increased number of nodes, there is in fact a trade-off between this and greater computational time in which the program runs for, as well as computational complexity and expense which is something that could be considered when solving similar problems with similar methods of code.

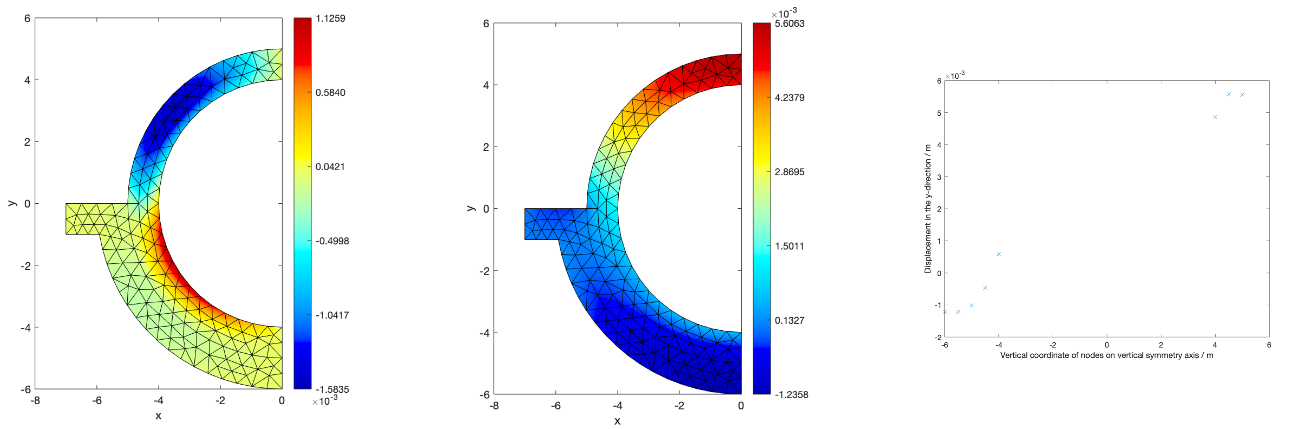
Something else to consider would be the elements used to divide the mesh. Here, linear triangular elements are used however for this same problem (as well as the thermo-elastic), an equal number of second-order triangular elements with six nodes and six shape functions each (Figs. 2.10 and 2.12 of [2]) could be used. Similar to what is described above, this will further reduce the error of the finite element solution compared to that of the analytical solution as the shape functions, quadratic in nature (with changing gradients) are free to more accurately contribute to both the stiffness matrices **C** and **A**, and the nodal vectors **D** and **B**. This however does double the number of nodes, as well as the dimensions of both the stiffness matrix and nodal vector, further increasing computational complexity and time.

The figures in Figure 2 show the temperature at the internal surface of the pipe to be at 70°C, as well as the temperatures on the external surface of the pipe and on the flange to be 10°C, as expected; this showing the Dirichlet boundary conditions to be accounted for exactly as they should.

# 4 The thermo-elastic problem in plane strain

Task 2 involves modifying the finite element program to now solve the overall thermo-elastic problem in plane strain. The thermo-elastic problem, unlike the thermal problem, is a two degree-of-freedom problem and each node has a component of its displacement in the x-direction, and another in the y-direction.

Figure 3 shows the finite element solutions to the displacement problem, presented on two contour plots of the nodal displacement values in both the x- and the y-directions and supported by a profile of the y-displacement along the vertical symmetry axis.




(a) Displacement in x-direction

(b) Displacement in y-direction

(c) Displacement profile along vertical symmetry axis

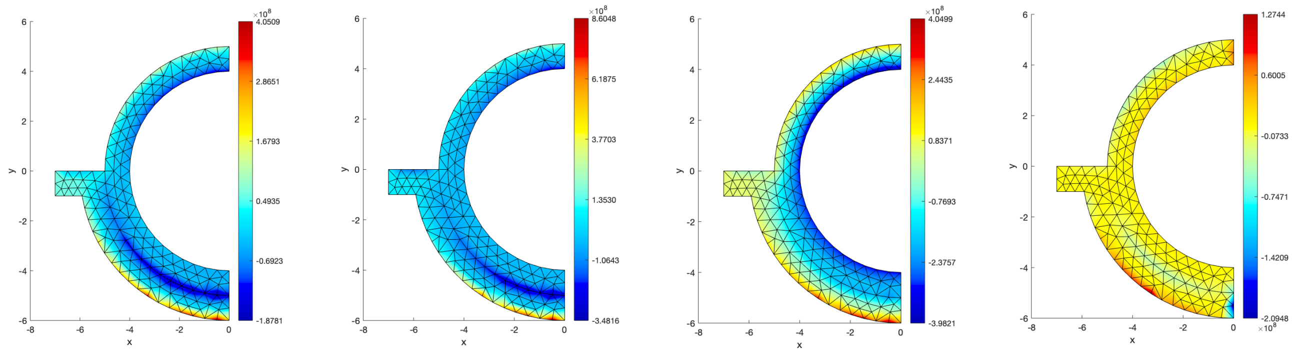
Figure 3: Finite element solution to the elastic part of the thermo-elastic problem

The function *buildFESystemElastic* is implemented here to solve for the displacement field; it allows for the extraction of the nodal displacement values in both the x- and the y-directions, in which corresponding contour plots are formed. It accounts now not only for the displacement-related Dirichlet boundary conditions but also the additional Newmann boundary conditions too, both described on page 2 of the project brief [1]. This time, it is noted that the bottom horizontal and vertical sides of the flange have zero displacement (accounted for in Dirichlet conditions) and the internal pressure of the pipe is at 10 bar, while the external is at atmospheric pressure (accounted for in Neumann). Although not explicitly stated, care must also be taken here to also constrain the x-displacement at the surfaces where symmetry is considered, to account for the right-hand half of the problem (this, again via the Dirichlet conditions). Similar to the thermal problem, the different materials are accounted for in the form of their different material properties, detailed on page 2 of the project brief [1].

Both plots in Figure 3 show the displacements at the vertical side of the flange, and its bottom horizontal

## 6.2 Calculating stresses

By calling the function *FindStress*, contour plots of the stresses are produced; these can be seen below.




(a) Direct stress $\sigma_{xx}$ (b) Direct stress $\sigma_{yy}$ (c) Direct stress $\sigma_{zz}$ (d) Shear stress $\sigma_{xy}$

Figure 7: Calculated (a) direct stress $\sigma_{xx}$, (b) direct stress $\sigma_{yy}$, (c) direct stress $\sigma_{zz}$ and (d) shear stress $\sigma_{xy}$ at the nodes from finite element solution

The four contour plots for the stresses associated with each node, shown in Fig. 7, are such that to represent this plain strain thermo-elasticity problem where $\epsilon_{zz}$ = $\gamma yz$ = $\gamma_{xy}$ = 0. This problem constraint highlights the consideration of either a long pipe or a pipe constrained at both ends, as opposed to a plain stress situation which evidently is not the case here as the out of page (axial) stress, $\sigma_{zz} \neq$ 0 throughout.

A constant $\sigma_{zz}$ throughout the pipe is usually expected with a plain strain elasticity problem due to it's dependence only on the inlet and outlet pressures and the inner and outer radii. Noticeably not constant here, it draws a link to the solution to the thermal problem, and highlights the influence of temperature on $\sigma_{zz}$ by means of the additional thermal term -E$\alpha\theta$/(1-2$\mu$) in calculating the stresses (except for $\sigma_{xy}$) which has particularly influence on the inner radius of the pipe due to the higher boundary temperature.

It can be seen that the region of greatest compressive direct stress is that of the outer radius of the pipe in contact with the flange. The displacements here, particularly in the y-direction are non-zero and represent the pipe expanding out into the steel flange which as expected raises the stress here as the steel flange resists deformation. Although expected to be high, the stress fields are admittedly of higher magnitudes than expected with them being in the order of thousands of bar, much larger than the inlet pressure of 10 bar. It is also surprising that the shear stress on the symmetry boundaries are not zero, as expected again to account for the right hand side of the problem.

# References


[1] L Brassart. *Finite element for coupled thermo-elastic problems*. Michaelmas Term, 2021.

[2] L Brassart. B1 finite elements. 2021.