Finite Element Analysis

Hilary Term 2022

Finite Elements for Coupled Thermo-elastic Problems

MEng Engineering Science, University of Oxford

Janina Branke

Contents

1 Introduction

Finite Element Analysis is a powerful modelling tool for numerically solving various engineering problems. In this report, a MATLAB programme for FEM will be described, verified and applied to a cylindrical flange and pipe. The programme has been developed to analyse the effects of various physical conditions on the temperature, displacement and stress distributions within a component.

2 Description of Algorithms

The general method for using finite elements analysis is illustrated in Figure 1, where A and B represent the stiffness matrix and force vector respectively. For each element, the coordinates corresponding to its nodes are used to calculate the Jacobian and the gradient of the local shape function. These are used to find the local stiffness matrix, A^e_{ij} and local nodal force vector, B^e_i, which are used to assemble the global matrices A and B [1]. To account for different materials within the element, we must identify which elements are part of the tube or the flange from the mesh, so that the correct material properties can be used in calculations. The elements corresponding to each material are different element sets within the mesh.

2.1 Steady-state heat conduction problem

The objective is to create an FEM programme capable of finding the steady-state temperature distribution. The temperature at each node can be calculated from $AU = B$, solving for U, where A and B have been found by using the heat equation in a finite element programme. Dirichlet conditions may be used to apply temperatures at the nodes of boundaries.

2.2 Thermoelastic problem

Finding the displacement for the thermoelastic problem requires the temperature at each node, which is obtained above. We are simulating the interior of a very thick component in a single plane and are therefore use coefficients for plane strain in our calculations for stress at the nodes. The FEM system is constructed similarly to temperature, therefore also follows the steps shown in Figure 1. However, note that there are two displacements associated with each node: x and y, therefore our local stiffness matrix is of the size 6x6 as there are 3 nodes for each element. This means our global A matrix will be a $2n \times 2n$ matrix (where n is the number of nodes in the mesh) and our global B matrix will have dimensions $2n \times 1$. Neumann boundary conditions can be used to apply pressures at boundaries and Dirichlet boundary conditions can be applied to fix the node displacements at a certain value.

2.3 Stress

The stress function uses averaging over neighbouring elements to calculate stress. It has 5 inputs: the mesh, material properties, the reference temperature and the temperature and displacement outputs from the respective functions. Hooke's law is used to calculate stress from the strain, calculated from the thermoelastic problem. Iterating through each element of the Jacobian, the gradient of the local shape function and relevant coefficients are found and used with Hooke's law to find the stress at each node. The stress of the element is calculated to be the sum of the stresses at each node. The stress at each node is now found by summing the stresses from all adjacent elements and dividing by the number of adjacent elements.

2.4 Neumann boundary condition

The Neumann boundary is also shown in Figure 1. It changes the values of the B matrix by looping through each boundary. For each node on the boundary, find the x and y coordinates and use these to calculate the Jacobian and the normal vector to the edge. The traction vector can be found by multiplying the normal vector with the boundary condition applied (in the displacement case, this is the pressure). A local nodal force vector B_i^e is found using the traction and Jacobian, which is added to the global force vector B.

2.5 Dirichlet boundary condition

The Dirichlet boundary conditions specify the values the solutions should take at the nodes along a boundary. It is applied once matrices A and B are fully constructed and changes both matrices.

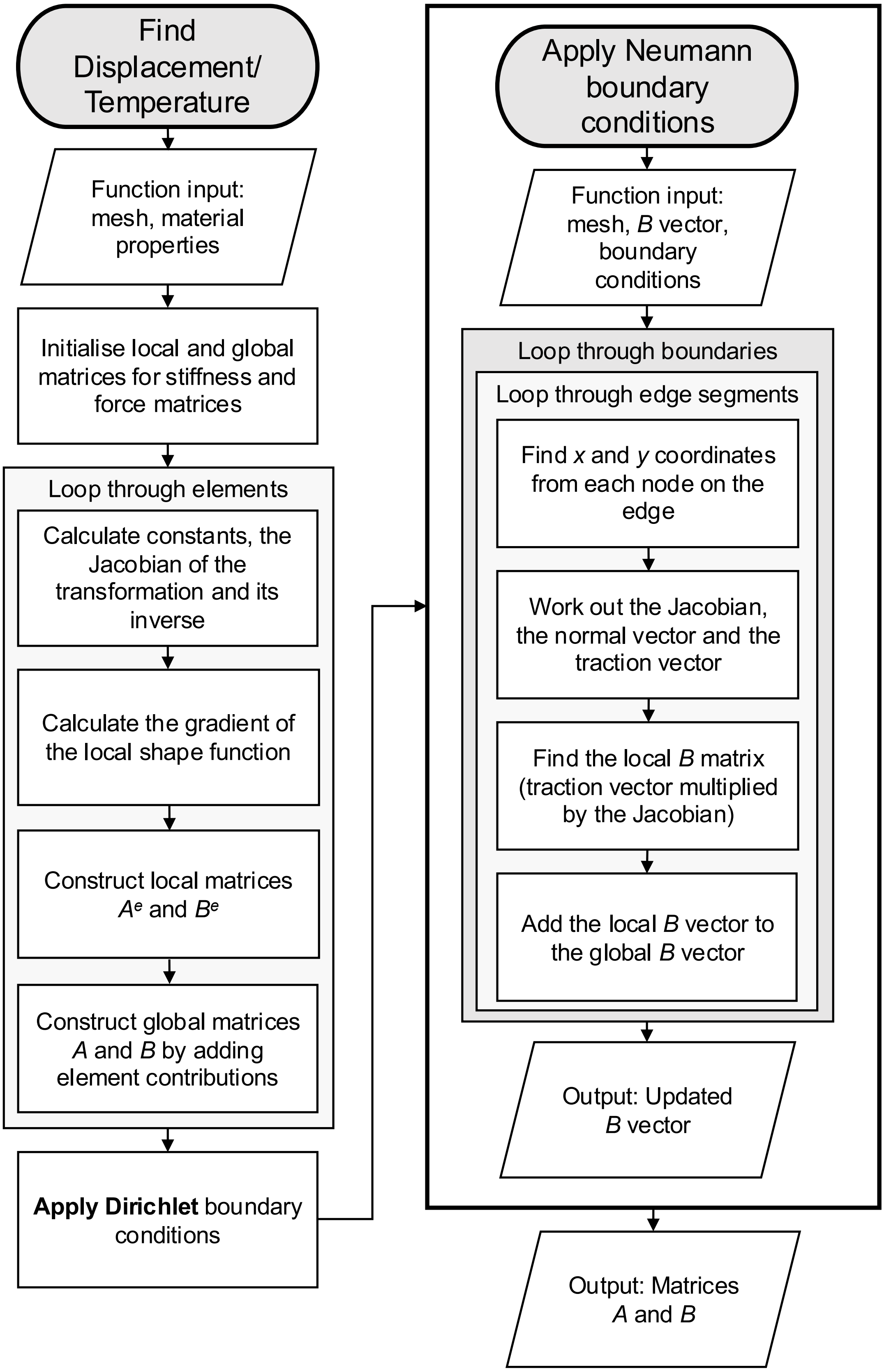

Figure 2.1: Flow chart for developing a finite element programme

3 Verifying the Code with Analytical Predictions

The FEM programme has been verified by comparing its results with analytical predictions. A simple, rectangular, 6 node, 4 element mesh was created, as illustrated in the following examples. In the first 5 tests, Dirichlet Boundary were used to set zero x displacement on the left side and zero y displacement on the bottom side. The ambient temperature in each of the tests is set to 20°. The block has dimensions 2x1 metres and was also used for debugging the programme, by comparing its results with an FEM system calculated by hand.

3.1 Testing the temperature distribution

To test the code responsible for finding temperature, Dirichlet boundary conditions were used to set a temperature of 10° and 100° to the left and right sides respectively. Figure 3.1 shows a uniform temperature, as expected. The colour in the centre of the block is 55°, which is midway between the two boundary values.

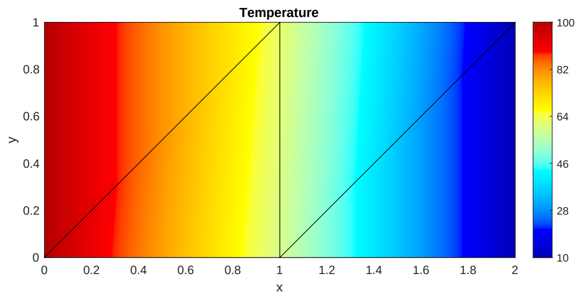

Figure 3.1: Block with temperature gradient

3.2 Applying a displacement to one side

Figure 3.2 shows a block with a displacement of 5m applied to the right side. The block's nodes are at ambient temperature. The displacement induces a stress within the element as follows:

$$\sigma_{xx} = \frac{E}{(1+v)(1-2v)}((1-v)\varepsilon_{xx} + v\varepsilon_{yy})$$
$$= \frac{15\times10^9}{(1+0.3)(1-2(0.3))}((1-0.3)\times\frac{5}{2}) = 5.048\times10^{10} \quad (1)$$

$$\sigma_{yy} = \frac{E}{(1+v)(1-2v)}((1-v)\varepsilon_{yy} + v\varepsilon_{xx})$$
$$= \frac{15\times10^9}{(1+0.3)(1-2(0.3))}((0.3)\times\frac{5}{2}) = 2.163\times10^{10} \quad (2)$$

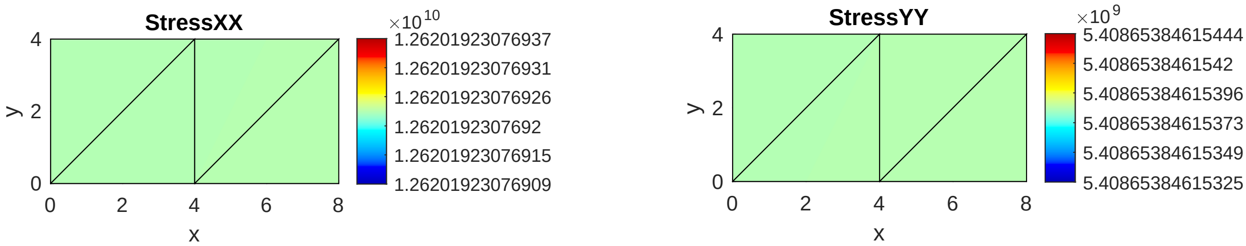

Figure 3.2: Temperature and displacement distributions throughout flange and pipe

3.3 Temperature difference of the block and its surroundings

A temperature of 70° was applied to all nodes of the mesh. Figure 3.3 shows the x and y displacements of the block due to thermal expansion because of the block's temperature difference to the ambient temperature (20°).

$$\varepsilon = (1+v)\alpha\Delta T \quad (3)$$

$$\varepsilon = (1+0.3)\times 28\times10^{-6}\times 50 = 1.82\times10^{-3} \quad (4)$$

$$u_x = \varepsilon\times l = 1.82\times10^{-3}\times 2 = 3.64\times10^{-3}\text{ m} \quad (5)$$

$$u_y = \varepsilon\times h = 1.82\times10^{-3}\times 1 = 1.82\times10^{-3}\text{ m} \quad (6)$$

Both the analytical and numerical calculations yield an x displacement of 3.64×10^{-3} m and a y displacement of 1.82×10^{-3} m. Since their solutions align, the programme correctly uses temperature to calculate displacement.

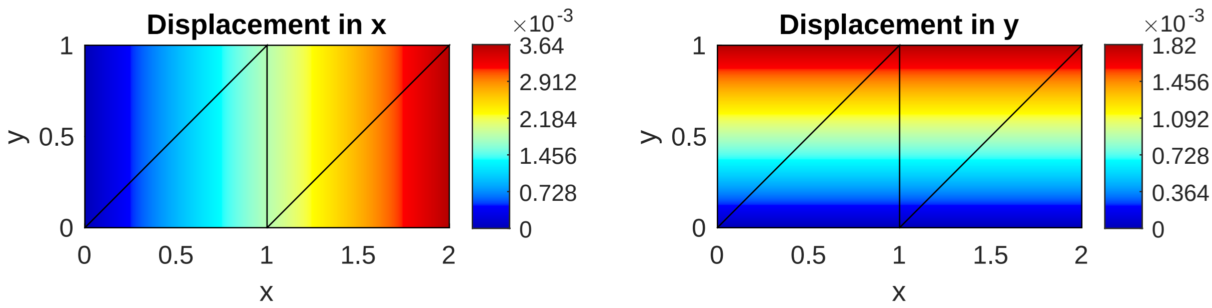

Figure 3.3: Applying a temperature difference

3.4 Applying a pressure

The block in Figure 3.4 has been set to 20° (ambient temperature) and a pressure of 1 bar has been applied to the right-hand side. The analytical displacements are consistent with the programme's output.

$$\varepsilon_{xx} = \frac{1-v^2}{E}(\sigma_{xx} - \frac{v}{1-v}\sigma_{yy}) = \frac{1-0.3^2}{15\times 10^9}(1\times 10^5) = 6.067\times 10^{-6} \tag{7}$$

$$u_x = 6.067\times 10^{-6}\times 2 = 1.213\times 10^{-5}\text{ m} \tag{8}$$

$$\varepsilon_{yy} = \frac{1-v^2}{E}(\sigma_{yy} - \frac{v}{1-v}\sigma_{xx}) = \frac{1-0.3^2}{15\times 10^9}\left(-\frac{0.3}{1-0.3}(1\times 10^5)\right) = -2.6\times 10^{-6} \tag{9}$$

$$u_y = -2.6\times 10^{-6}\times 1 = -2.6\times 10^{-6}\text{ m} \tag{10}$$

3.5 Applying a temperature and a pressure

Applying the conditions from both 3.3 and 3.4 (temperature difference of 50° of the block to its surroundings and a pressure of 1 bar) causes the displacements to be a linear sum of the displacements from the previous cases, as demonstrated in Figure 3.5, as well as analytically. Displacements vary uniformly across the element.

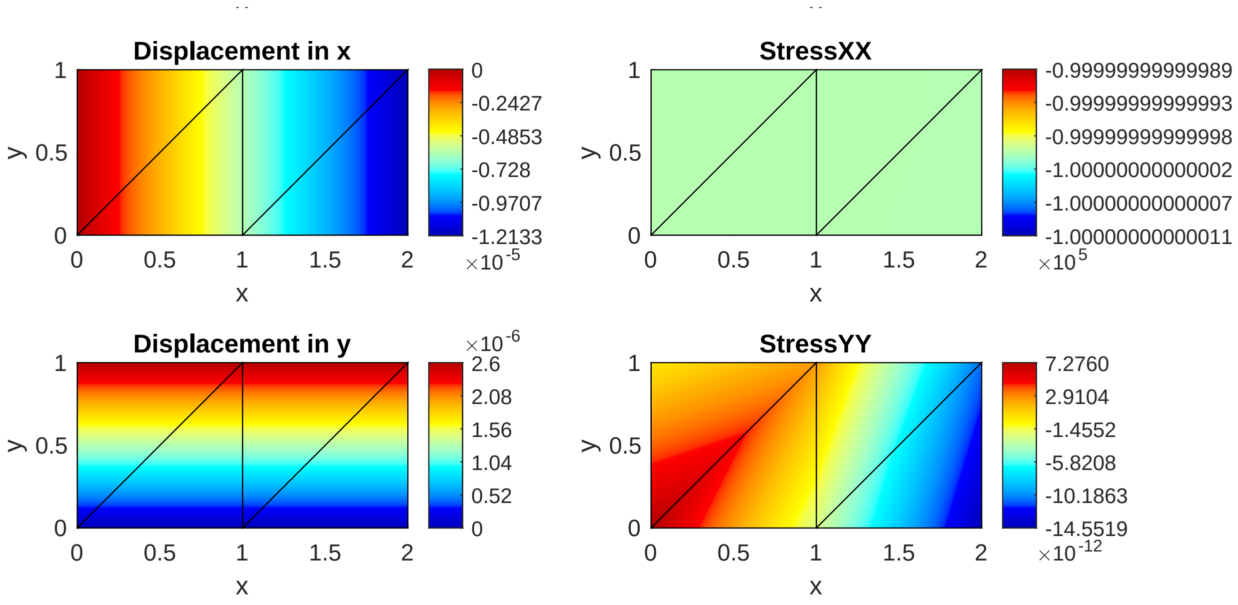

Figure 3.4: A pressure of 1 bar applied to the right edge

$$u_x = 3.64 \times 10^{-3} - 1.2134 \times 10^{-5} = 3.6279 \times 10^{-3} \text{ m} \tag{11}$$

$$u_y = 1.82 \times 10^{-3} + 2.6 \times 10^{-6} = 1.8226 \times 10^{-3} \text{ m} \tag{12}$$

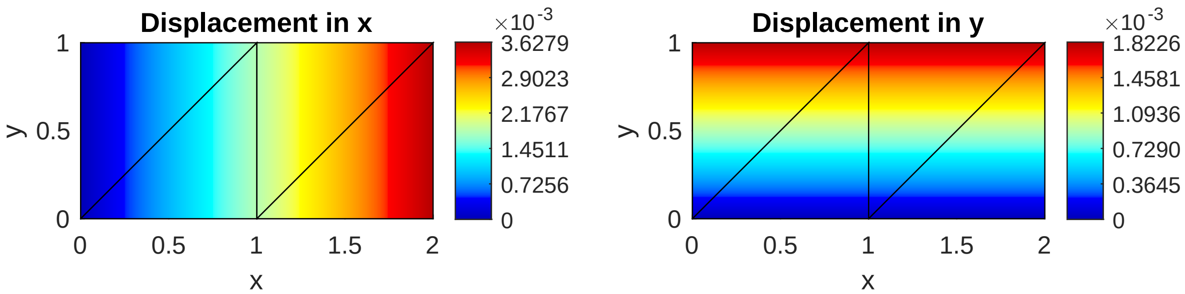

Figure 3.5: Combined displacements for temperature difference and pressure

3.6 Thermal stress

To test the thermal stress, the entire rectangle has been fixed in Figure 3.6: setting the x displacements of both vertical sides and the y displacements of both horizontal sides to zero. The temperature of the rectangle has been set to 70° and the stress in plane strain can be found as:

$$\sigma = -\frac{E\alpha\Delta T}{1-2v} = -\frac{15\times10^{9}\times28\times10^{-6}\times50}{1-2\times0.3} = -52.5\times10^{6}\ \text{Nm}^{-2} \tag{13}$$

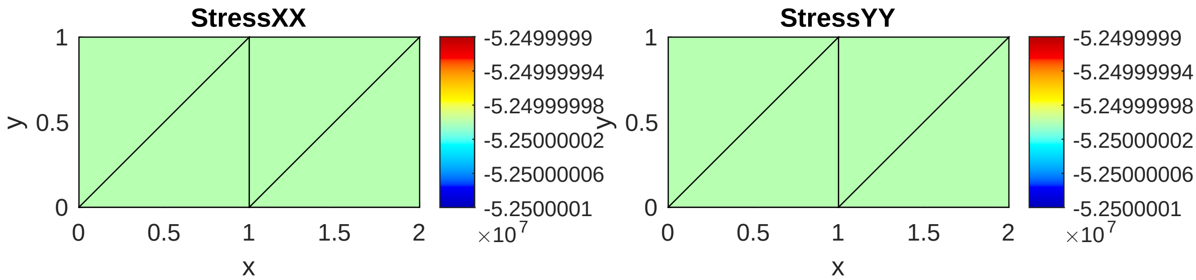

Figure 3.6: Thermal stresses

4 Flange and Pipe Problem

The tested MATLAB code was applied to a flange and pipe problem using given coarse, normal and fine meshes. The pipe and flange are subject to boundary conditions of 70° and 10 bar on the internal surface and 10° and atmospheric pressure on the external surface. The flange has also been fixed to the wall at two boundaries.

4.1 Steady-state heat conduction programme applied to flange and pipe

Figure 4.1 illustrates the smooth variation of temperature throughout the component from 70° on the inside to 10° on the outside. We have not accounted for convection, meaning these temperatures have been applied directly to the nodes using the Dirichlet boundary condition. Each element uses linear interpolation between nodes, however the slight shift in gradient is caused by the flange and pipe having different thermal conductivities.

4.2 Thermoelastic problem applied to the flange and pipe

For the thermoelastic problem, Dirichlet boundary conditions were applied in the x direction at the centre as there is no displacement due to symmetry, as well as in the x and y directions at the fixed edges. Figure 4.1 shows the x and y displacements, which mirror the boundary conditions. Figure 4.2 shows the variation of the y displacements along the symmetrical axis. Overall, the flange and pipe are displacing outward, especially in the vertical direction, therefore we need to ensure that it is not built adjacent to other components, as this could cause unexpected stresses.

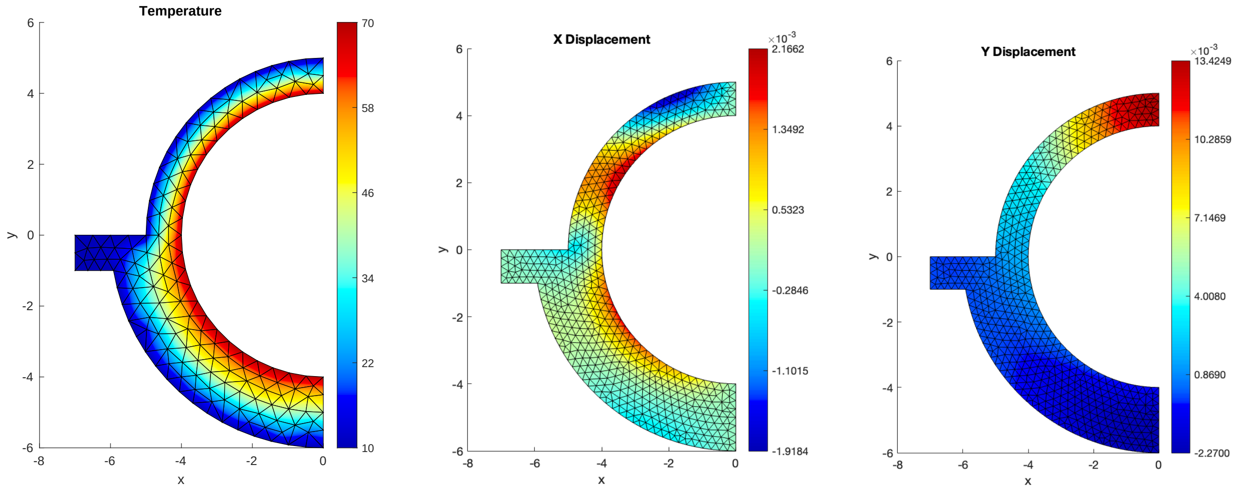

Figure 4.1: Temperature and displacement distributions throughout flange and pipe

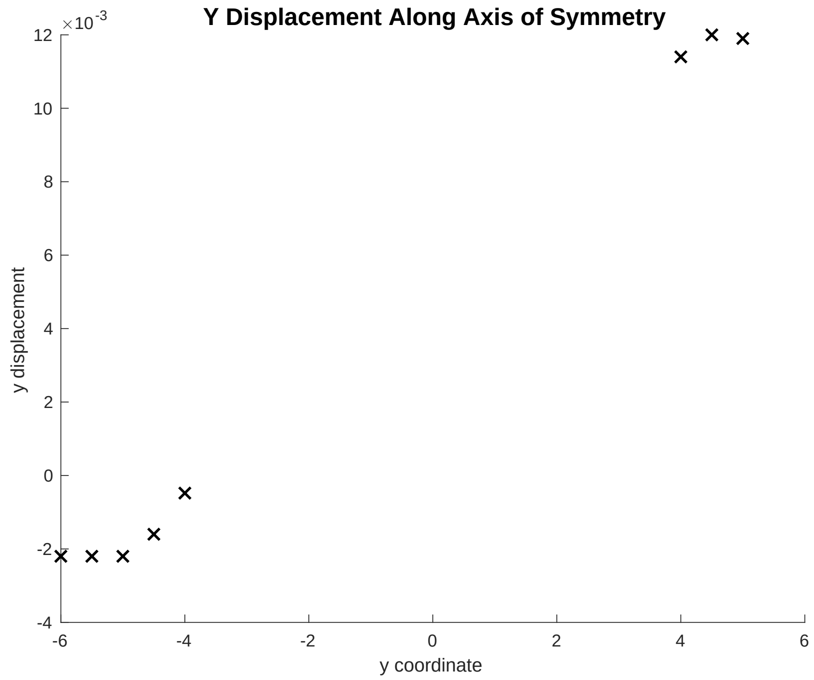

Figure 4.2: y displacement along the axis of symmetry

4.3 Stresses throughout the flange and pipe

Figure 4.3 shows the stress distributions throughout the flange. We can see that the maximum stress is the stress at the bottom of the flange in the x direction (79.7 MPa). This value is larger than the ultimate failure stress of lead (12 MPa) [2]. To prevent failure, we could support the flange and pipe system at the bottom of the flange. Since our model assumes perfect insulation at the surfaces, neglecting heat transfer with the surroundings, it likely overestimates the temperature and displacement gradients compared to real-world conditions. Therefore, these stress predictions account for a worst case scenario.

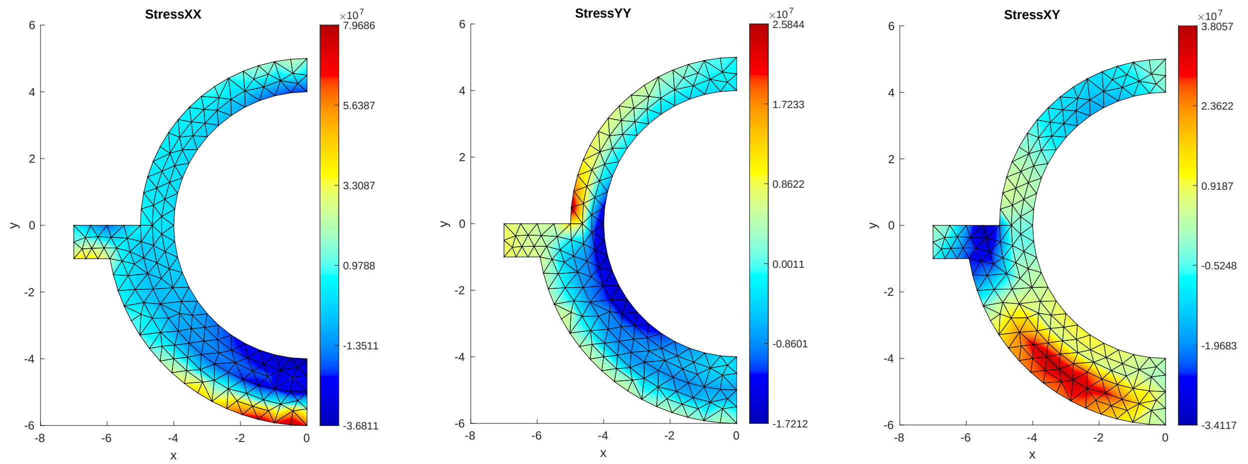

Figure 4.3: Stress distribution throughout flange and pipe

5 Possible Improvements

5.1 Different mesh sizes

The times taken to compute the temperature, displacements and stresses is shown in Table 1. Different mesh sizes could be combined to minimise the computational time, while maintaining accuracy since the numerical solution approaches the analytical solution with more elements. This could be done by starting with a coarse mesh to find elements of high rates of changes and then applying smaller meshes to only those areas.

Table 1: Time taken to run programme given different meshes

Coarse Mesh	Normal Mesh	Fine Mesh
0.460788 sec	2.303736 sec	138.595270 sec

5.2 Additional support

To keep the stresses within the flange and pipe to a minimum, an additional support at the bottom of the flange has been applied by using Dirichlet conditions to set the two most bottom nodes to zero x and y displacement. However, as shown in Figure 5.1, the stress in the x direction has only reduced by a very marginal amount (from 7.9686×10^7 Nm^{-2} to 7.7090×10^7 Nm^{-2}). Therefore, other methods such as using a different material or altering the pipe's dimensions may be better.

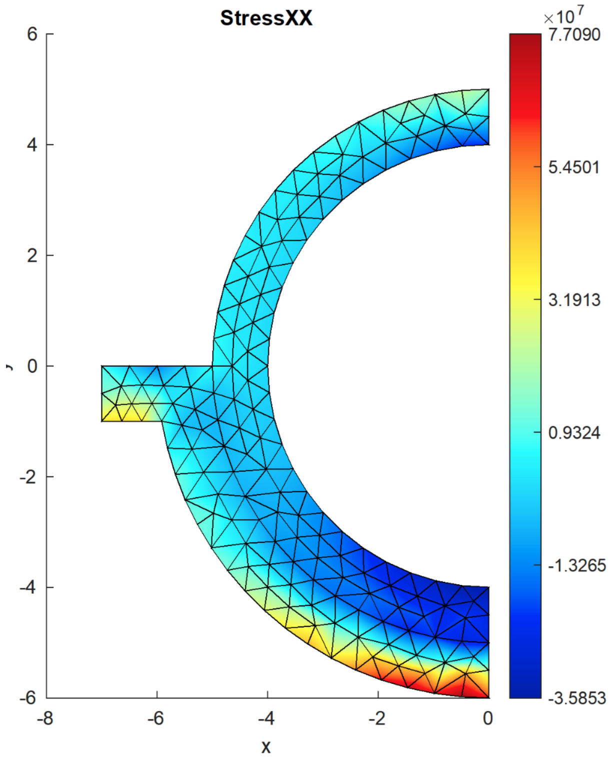

Figure 5.1: Stress in x direction when flange is supported at the bottom

6 Conclusion

The Finite Element Model introduced in this report has successfully captured the system's responses to temperature and pressure variations, as demonstrated by our tests and analytical solutions. In practice, this FEM code could be used to predict the location of likely failure in the pipe by finding the area of highest stress. This is useful, as it allows engineers to add reinforcement to those areas as well as carry out predictive maintenance.

References

[1] T.J.R. Hughes, *The finite element method. Linear static and dynamic finite element analysis*, Dover Publications, 2000

[2] A. M. Howatson, P. G. Lund and J. D Todd, *Engineering tables and data*, 2009