Modelling an elastic membrane with finite elements

Thomas Mitchell (71506388)

Introduction

The purpose of this report is to study the deformation of a rectangular elastic membrane of length L and width W (10 mm), under an applied load f (100 N/m^3). The problem could be described by Poisson's equation, Equation 1.

$$-\nabla \cdot (\tau \nabla u) = f \tag{1}$$

The dependent variable u was the displacement and the diffusion coefficient τ was the stiffness (with $\tau_0 = 1\ Pa$), Equation 2.

$$\tau(x) = \left(1 + 0.9 \sin \frac{\sqrt{2}\pi x}{W}\right)\tau_0 \tag{2}$$

The homogeneous Dirichlet boundary condition was applied to the membrane perimeter thereby containing the displacement at the boundary to zero (u = 0 m).

The finite element method was used to solve the governing partial differential equation (PDE). The problem was assumed to be 1-D and was analysed in MATLAB using linear and quadratic elements. These codes were verified against a COMSOL analysis. The simplifying assumption was subsequently evaluated by comparing the 1-D COMSOL model with a 2-D model for varying aspect ratios ε, Equation 3.

$$\varepsilon = L/W \tag{3}$$

Methods

The governing equation was rewritten for the 1-D case, Equation 4.

$$-\frac{d}{dx}\left(\tau \frac{du}{dx}\right) = f \tag{4}$$

To solve the problem, Equation 4 was rewritten using the product rule, Equation 5.

$$-\tau \frac{d^2u}{dx^2} - \frac{d\tau}{dx}\frac{du}{dx} = f \tag{5}$$

Using the method of weighted residuals, the governing DE was rewritten, Equation 6.

$$R(x) = \tau \frac{d^2\tilde{u}}{dx^2} + \frac{d\tau}{dx}\frac{d\tilde{u}}{dx} + f \tag{6}$$

Interpolation functions were introduced as the weighting functions in accordance with the Galerkin method, Equation 7.

$$\int_{x_1}^{x_2} \left(\tau \frac{d^2\tilde{u}}{dx^2} + \frac{d\tau}{dx}\frac{d\tilde{u}}{dx} + f\right) N_i\, dx = 0 \tag{7}$$

Integration by parts was used to lower the order of the $\frac{d^2\tilde{u}}{dx^2}$ term. The term was a product of three functions: $\frac{d^2\tilde{u}}{dx^2}$, $\tau(x)$, and $N_i(x)$. Equation 8 was used to find the integral.

$$\int_a^b uvdw = [uvw]_a^b - \int_a^b uwdv - \int_a^b vwdu \tag{8}$$

The integral was found using $u = \tau(x)$, $v = N_i(x)$, and $dw = \frac{d^2\tilde{u}}{dx^2}$, Equation 9.

$$\int_{x_1}^{x_2} \tau \frac{d^2\tilde{u}}{dx^2} N_i\, dx = \left[\tau \frac{d\tilde{u}}{dx} N_i\right]_{x_1}^{x_2} - \int_{x_1}^{x_2} \tau \frac{d\tilde{u}}{dx}\frac{dN_i}{dx} dx - \int_{x_1}^{x_2} N_i \frac{d\tilde{u}}{dx}\frac{d\tau}{dx} dx \tag{9}$$

Equation 9 was substituted into Equation 7 and simplified. The limits for the $\left[\tau \frac{d\tilde{u}}{dx} N_i\right]_{x_1}^{x_2}$ term were evaluated for $i = 1$ and $i = 2$. This gave the weak form equations, Equations 10a and 10b.

$$\tau \frac{d\tilde{u}}{dx}\bigg|_{x1} = \int_{x_1}^{x_2} \tau \frac{d\tilde{u}}{dx}\frac{dN_1}{d_x} dx - \int_{x_1}^{x_2} fN_1\, dx \tag{10a}$$

$$\tau \frac{d\tilde{u}}{dx}\bigg|_{x2} = -\int_{x_1}^{x_2} \tau \frac{d\tilde{u}}{dx}\frac{dN_2}{dx} dx + \int_{x_1}^{x_2} fN_2\, dx \tag{10b}$$

The stiffness matrix was assembled by adding all the 2x2 element stiffness matrices into a global 5x5 stiffness matrix. The global forcing vector was subsequently assembled by adding up all the element equations to form a 5x1 column vector. The internal boundary conditions were then reinserted into the global stiffness and forcing matrices to produce the final solution.

Code verification

The 1-D elastic string was solved using two linear elements, and then with two quadratic elements using MATLAB (elastic_membrane_study.m). The same problem was then solved in COMSOL (1D_membrane.mph). Figure 1 shows a plot of the linear and quadratic solutions from the MATLAB and the COMSOL analyses.

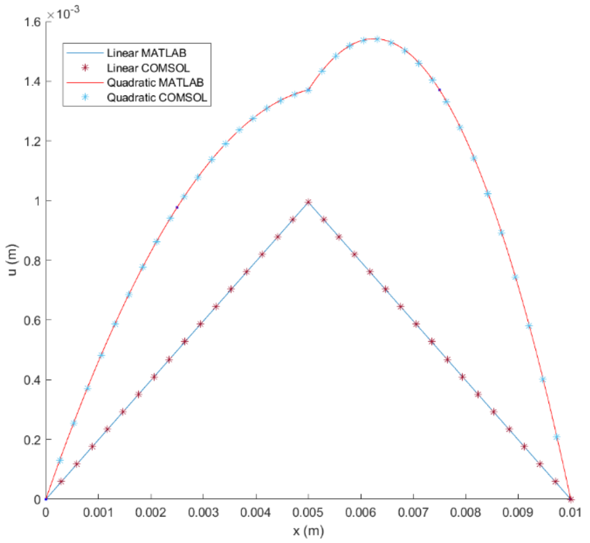

Figure 1: Plot of the linear and quadratic solutions from the MATLAB and the COMSOL analyses.

Figure 1 shows that the results from the MATLAB and COMSOL analyses are the same, thereby verifying the integrity of the analytical derivation.

Mesh convergence

A mesh convergence study was performed with the MATLAB code for both linear and quadratic elements (mesh_convergence_study.m). Starting with two elements, the mesh was iteratively refined to seven elements. Table 1 shows the displacement at the centre (x = W/2) and the relative error (compared to the finest mesh) of the linear case. Table 2 shows the quadratic case.

Table 1: Displacement at the centre and relative error for different numbers of linear elements.

Number of elements	Displacement (mm)	Relative error
7	1.40	0.00%
6	1.39	1.18%
5	1.32	5.71%
4	1.28	8.77%
3	1.11	21.0%
2	0.995	29.1%

Table 2: Displacement at the centre and relative error for different numbers of quadratic elements.

Number of elements	Displacement (mm)	Relative error
7	1.50	0.00%
6	1.50	0.17%
5	1.49	0.51%
4	1.48	1.31%
3	1.45	3.17%
2	1.37	8.52%

The relative errors of the quadratic study were smaller than those from the linear study, which indicated that the quadratic elements converged more quickly than the linear study. As such, the quadratic elements performed better overall and will be used for the next task.

Aspect ratio study

An aspect ratio study was performed in COMSOL on a 2-D model using quadratic elements (2D_membrane.mph). A range of aspect ratio (ε) values were iterated through to determine when the relative error of the displacement was less than 1% for the 1-D and 2-D models. It was found that this occurred when $\varepsilon = 2.6$ or when $L = 26\,mm$. Beyond this length it can be assumed that the 1-D case approximates the 2-D case throughout the cross section of the membrane. Figure 2 shows the convergence of the central displacement value as the aspect ratio tends to infinity.

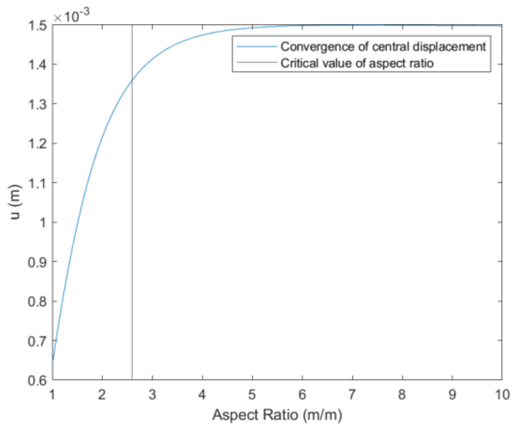

Figure 2: Convergence of central displacement.

Conclusion

In summary, the finite element method can be used to solve PDEs. Quadratic elements approximate solutions to higher fidelity than linear elements. For $L \geq 26\,mm$, 1-D approximations can be used.