Finite Element Analysis - CE50130

TERM PROJECT

Assigned: 04/04/24 Submission due: 25/04/24

The objective of this study is to perform a thermostructural analysis on a rectangular plate, ABCD, as depicted in Figure 1, measuring 7.6 cm × 5 cm. The plate has fixed boundary conditions on the top (AB) and bottom (CD) edges, while heat transfers from 100°C to 0°C from the left (AC) to the right (BD) edge.

The governing equations consist of the heat transfer equation and the equilibrium equation. The heat transfer equation is given by: $\frac{\partial}{\partial x}\left(k_x \frac{\partial \mathrm{T}}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial \mathrm{T}}{\partial y}\right) = f(x, y)$, where k_x and k_y are the thermal conductivities in the x and y directions respectively, and $f(x, y)$ represents the heat source. The equilibrium equation is: $\nabla \cdot \boldsymbol{\sigma} + b = 0$, where $\boldsymbol{\sigma}$ is the stress tensor and b represents the external body force.

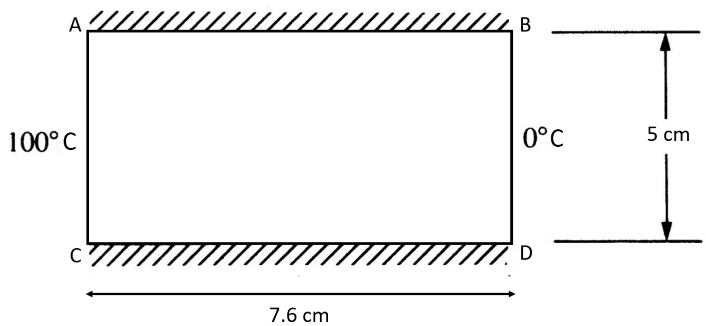

Figure 1

a. Following the weak form approach, derive the element stiffness matrix and force vector. (Refer to Introduction to FEM by J.N.Reddy for deriving weak forms of the governing equations.)

b. Using these derived weak forms, develop a MATLAB program for Finite Element Method (FEM) analysis. Use 4-noded isoparametric elements for plane stress analysis of the plate. Use reasonably fine mesh for analysis. (Use the scalar field element for the temperature and the vector field element for the displacements.) Note that:

   i. The MATLAB program should read an input file containing geometry, boundary conditions, and mesh details.

   ii. Gaussian integration is to be used to determine the stiffness matrix numerically.

   iii. Use the staggered approach for solving temperature first by solving the heat transfer equation and then the equilibrium equation to obtain displacements in the code.

   iv. The primary objectives include determining the temperature distribution within the plate and analyzing the resulting stress distribution due to the heat transfer.

   v. Material properties are provided as follows: Young's modulus, E $= 2 \times 10^5$ MPa, Poisson's ratio, $\nu = 0.30$, coefficient of thermal expansion, $\alpha = 7.2 \times 10^{-6}/^{\circ}C$ and thermal conductivity, $k_x = k_y$ = 45 W/m-K. Take the element thickness, t = 1mm.

c. Prepare a detailed report including the input file for nodes and connectivity details, well-commented FEM code. Plot the temperature and stress distribution.