Department of Applied Mechanics and Biomedical Engineering, Indian Institute of Technology, Madras.

Fundamentals of Finite Element Analysis (AM 5450)

Assignment 8 – 2D Isoparametric Quad Elements for Elasticity Problems

Submission Date: 26October 2024

Instructions:

I. MATLAB/PYTHON scripts used to prepare plots/figures/results should be attached with your answer.
II. Scan all the pages of your hand-written answers, MATLAB/PYTHON scripts, output results, and figures into a PDF file
III. Submissions sent after the deadline will not be evaluated.

Question 1

Consider a cantilever beam subjected to a surface shear force of 500 kN/m and a tip load of 600 N as shown in Figure 1. Solve this 2-D plane-stress elasticity problem using **Isoparametric Quad elements**. The Young's modulus of elasticity and Poisson's ratio for the beam material are 80 GPa and 0.3, respectively. Compare the obtained FE (i) Displacements at each node, as well as the (ii) Strains and (iii) Stresses over the elements with the **(a)** solution from ABAQUS/ANSYS in the tabular form, **(b)** solution using CST elements obtained in the previous assignment. Plot the deformed shape and contour plots of these results. (Generate mesh data – node co-ordinates and node-element connectivity - in ABAQUS/ANSYS to use in your FE code.)

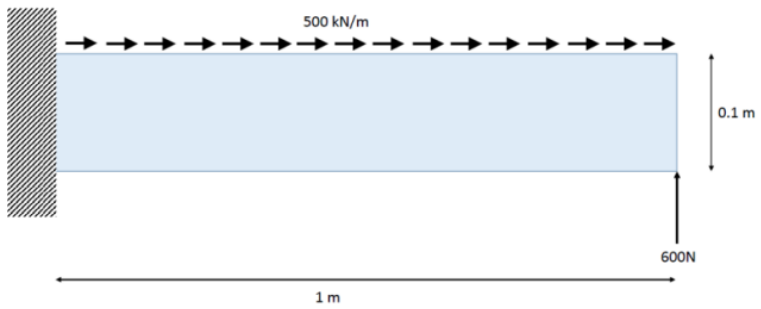

Figure 1

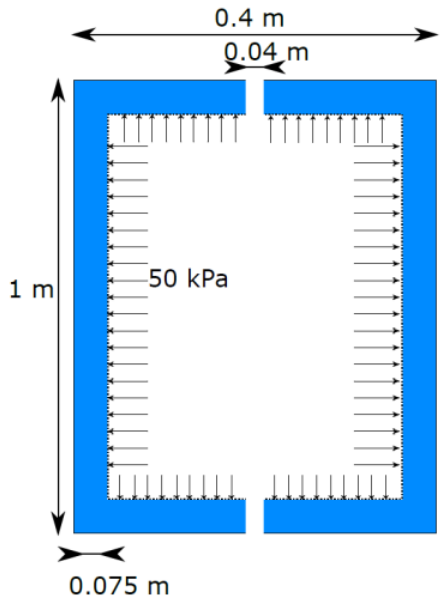

Figure 2

Question 2

Consider a cylindrical pressure vessel of height 1 m with closed ends as shown in Figure 2. The outer diameter is 0.4 m, and the diameter of the holes present in the end caps is 0.04 m. If the vessel is subjected to a uniform internal pressure of 50 kPa, determine the longitudinal and hoop stresses developed in the structure using **Isoparametric Quad elements**. Report the maximum values and location. Assume a wall thickness of 75 mm for the vessel. Compare the results with **(a)** the analytical solution for thin-walled pressure vessels, and **(b)** with the results obtained using the CST element in previous assignment. (Generate mesh data – node co-ordinates and node-element connectivity - in ABAQUS/ANSYS to use in your FE code.)