Student Name:
Student ID:

Nonlinear Finite Element Method (Spring 2024)

Mid-term Project

Due Date: 5:00pm, April 17th, 2024

---

1. **Problem 1 (60%)** Implement the Finite Element method in Matlab for stress analysis of 2D linear elasticity problems by completing the predefined Matlab functions.

   (a) Functions in *main.m*, *plot_results.m*, *quadplot.m* and *constitutive.m* are completed. *main.m* is the main program of the Matlab code, which shows the steps for a Finite Element analysis process. Read the code starting from *main.m* to understand the meaning of our data structures and follow the calling order of the subroutines.

   (b) Implement the functions marked with "TODO" comments following the instructions in the code:

B_matrix.m :	formation of the B matrix
Boundary_coditions.m :	set up the displacement boundary conditions
Enforce_BC.m :	enforce the essential boundary conditions to degrees of freedoms
F_vector.m :	formation of the external force vector
g_center.m :	calculate the barycenter of each element
generate_mesh.m :	generate the mesh, i.e., node coordinates and element connectivity table
K_matrix.m :	develop the K stiffness matrix

Note that the important data structures defined in the Matlab code are

$$x\_a = \begin{pmatrix} x^{(1)} & y^{(1)} \\ x^{(2)} & y^{(2)} \\ \vdots & \vdots \\ x^{(N)} & y^{(N)} \end{pmatrix}, \quad elem = \begin{pmatrix} I^{(1)} & J^{(1)} & \dots & K^{(1)} \\ I^{(2)} & J^{(2)} & \dots & K^{(2)} \\ \vdots & \vdots & \vdots & \vdots \\ I^{(M)} & J^{(M)} & \dots & K^{(M)} \end{pmatrix}, \quad xg = \begin{pmatrix} x^{(1)} & y^{(1)} \\ x^{(2)} & y^{(2)} \\ \vdots & \vdots \\ x^{(M)} & y^{(M)} \end{pmatrix}.$$

where $x^{(i)}$ and $y^{(i)}$ in x_a are the coordinates of node $i \in \{1, 2, \dots, N\}$, $\{I^{(q)}, J^{(q)}, \dots, K^{(q)}\}$ are the indices of the nodes of element $q \in \{1, 2, \dots, M\}$, i.e. *elem* is the connectivity table, and $x^{(q)}$ and $y^{(q)}$ in xg are the coordinates of the barycenter of element q. The **B** matrix is defined as a Matlab cell structure, i.e.,

$$\mathbf{B}(q) = \begin{pmatrix} \frac{\partial N_I^{(q)}}{\partial x} & 0 & \frac{\partial N_J^{(q)}}{\partial x} & 0 & \dots & \frac{\partial N_K^{(q)}}{\partial x} & 0 \\ 0 & \frac{\partial N_I^{(q)}}{\partial y} & 0 & \frac{\partial N_J^{(q)}}{\partial y} & \dots & 0 & \frac{\partial N_K^{(q)}}{\partial y} \\ \frac{\partial N_I^{(q)}}{\partial y} & \frac{\partial N_I^{(q)}}{\partial x} & \frac{\partial N_J^{(q)}}{\partial y} & \frac{\partial N_J^{(q)}}{\partial x} & \dots & \frac{\partial N_K^{(q)}}{\partial y} & \frac{\partial N_K^{(q)}}{\partial x} \end{pmatrix} \quad \text{for } q \in 1, 2, \dots, M$$

2. **Problem 2 (40%) Problem 2.** Consider a static linear elasticity problem on the trapezoidal panel domain as shown in Figure 1. The vertical left edge is fixed. The bottom and the right vertical edges are traction free, i.e. $\bar{\mathbf{t}} = \mathbf{0}$. Traction $\bar{\mathbf{t}} = (0 \quad -2 \times 10^4)^T$ N/m is applied on the top horizontal

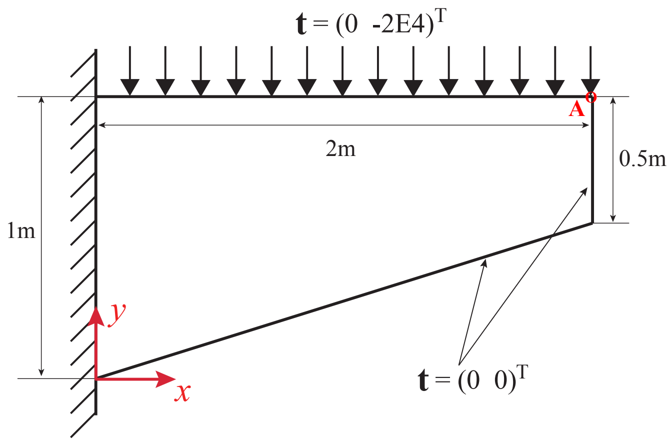

Figure 1:

edge. Material properties include Young's modulus $E = 3 \times 10^7$ Pa and Poisson's ratio $\nu = 0.3$. Plane stress conditions are considered, i.e., the elastic moduli matrix is given by

$$\mathbf{C} = \frac{E}{1-\nu^2} \begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{pmatrix}.$$

(a) Discretize the domain using 2, 100 and 2000 triangular elements, respectively(Hint: generate the meshes following the pattern shown in Figure 2(a) and implement your algorithm in the Matlab function named generate_mesh.m). Find the stress distribution of the domain using the Matlab code developed in Problem 1.

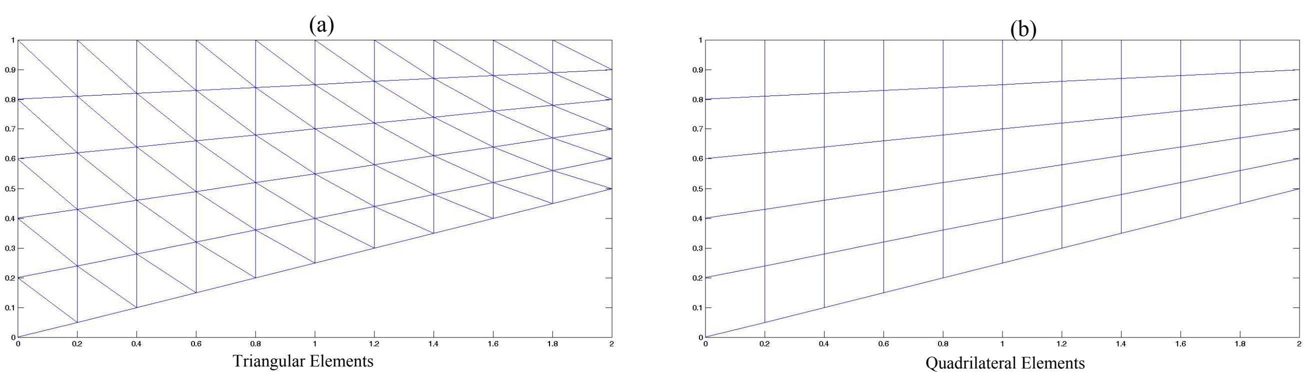

Figure 2:

(b) Discretize the domain using 1, 50 and 1000 quadrilateral elements, respectively (Hint: generate the mesh following the pattern shown in Figure 2(b) and implement your algorithm in the Matlab function named generate_mesh.m). Find the stress distribution of the domain using the Matlab code developed in Problem 1.

(c) Compare the displacement of point A (Figure 1.) in y-direction, i.e. $u_y(x_A, y_A)$, obtained from (a) and (b) for the three meshes. Calculate the convergence rate of your solution for the triangular meshes and quadrilateral meshes, respectively.