CE 205: Finite Element Method: Homework IV

Instructor: Dr. Narayan Sundaram*

March 11, 2024

This is a guided programming assignment, taking you through the three essential steps of any FE analysis: Pre-processing, solution, and post-processing, while making you write code for each part. You must submit all source code. Show all work clearly. Plots must be labeled legibly and completely. This homework has THREE pages.

1. Obtain a symbolic expression (using MATLAB symbolics, Mathematica, or manually) for the stiffness matrix of a plane-stress Q4 element given the input parameters a, b, E, ν, and t^{el}. Start with the $[B]$ matrix of a Q4 element and then use $\int\int [B]^T[E][B]dxdy$ as shown in the demo in class. As a test, write down the matrix $[k]/(E\,t^{el})$ for an element with $\nu = 0.25$.

2. Write a MATLAB function 'Q4stiffness' that returns the (numerical) stiffness matrix of a rectangular Q4 element given the values of the input parameters : Elastic moduli E, ν, the element half-dimensions a, b, and the element thickness t^{el}. You can copy and use the symbolic expressions for the stiffness matrix entries derived in part (1) above to generate this function. The return value should be a symmetric 8 x 8 matrix of numbers for the given input parameters.

3. Write a MATLAB function 'makemesh' that generates a valid finite element mesh of Q4 elements over a beam of given length L and height h. This function should also accept, as input, the number of elements along the span n_h and the number of elements through the height, n_v. The outputs of this meshing function should be a matrix of elements of size $n_{elems} \times 4$, containing the node numbers in each element, as well as corresponding matrices of nodal coordinates.

4. Check that your FE mesh generator in part (3) produces a valid, topologically and geometrically correct mesh. One way to do this is to write MATLAB code to produce a plot of the mesh with all the nodes and elements labeled. For instance, a mesh of

*Department of Civil Engineering, Indian Institute of Science

4×2 elements for an $L = 1.0\ m$, $h = 0.1\ m$ beam has 8 Q4 elements and 15 nodes. It has a matrix of element connectivities given by:

$$\begin{bmatrix} 1 & 2 & 7 & 6 \\ 2 & 3 & 8 & 7 \\ 3 & 4 & 9 & 8 \\ 4 & 5 & 10 & 9 \\ 6 & 7 & 12 & 11 \\ 7 & 8 & 13 & 12 \\ 8 & 9 & 14 & 13 \\ 9 & 10 & 15 & 14 \end{bmatrix}$$

The corresponding plot of the mesh is shown in Fig. 1. Notice the node numbers (blue) and element numbers (black). The green A identifies the first edge in every element.

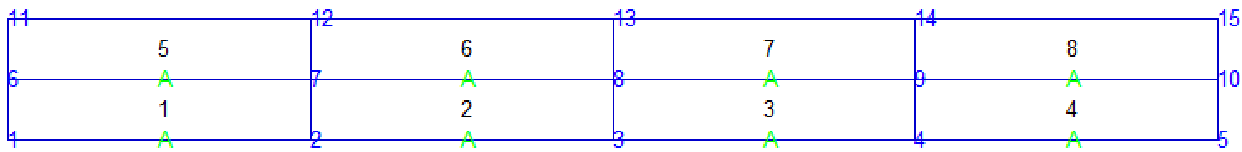

Figure 1: Sample 4×2 mesh of Q4 elements

Sometimes, one can produce a so-called shrink plot where the interior element sides are slightly shrunk to show the element shapes, as shown in Fig. 2.

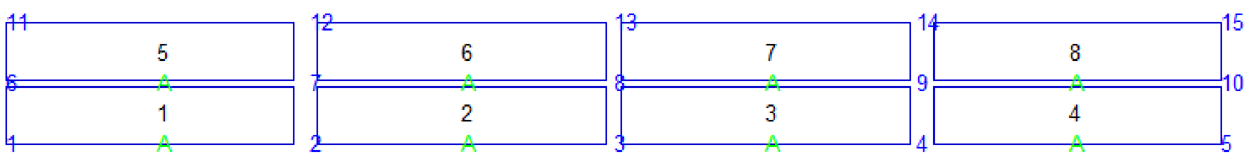

Figure 2: Shrink plot of the same mesh

5. Now consider an $L = 1.0$ m long rectangular cross-section aluminum beam, of height $h = 0.1$ m (slenderness 10) and out-of-plane thickness 0.05 m. The left-end of the beam is fixed (cantilevered), and the right end is tip-loaded. For aluminum, $E = 70$ GPa and $\nu = 0.33$. The tip load is applied downward, $P = -2000$ N.

   Use the functions developed above to write a MATLAB program to analyze this tip-loaded cantilever problem using Q4 elements. The user should be able to specify the number of elements over the span n_h and through the thickness n_v.

   Proceed systematically through assembly, boundary condition and load application specification, and solution of the reduced system. All nodes along the left-end of the cantilever should be fixed. The tip-load should be divided in half and applied to the top-rightmost and bottom-rightmost nodes in your mesh. For example, in Fig. 1, this would be loads of $P/2$ each applied to nodes 5 and 15.

6. Calculate a closed-form expression for the tip deflection, v^{th}, using linear elasticity. What is its value for the present problem?
   Hint: The elasticity solution coincides with the strength-of-materials solution in this case.

7. Conduct a mesh convergence study using your FE program. Use the vertical displacement of the bottom-rightmost node (e.g. this would be node 5 in Fig. 1) v^{tip} to check for convergence, while comparing against the analytical expression v^{th} above. Do this study for the following sets of mesh parameters:

n_h	n_v
10	$1, 2, 4$
20	$1, 2, 4, 8$
40	$1, 2, 4, 8, 16$
60	$1, 2, 4, 8, 16$
80	$1, 2, 4, 8, 16$

   This is a total of 22 FE analyses using your code. Make a table of v^{tip}/v^{th} for these values of n_h and n_v. Comment on the convergence rate and on this parametric study. What does this tell you about optimal element use for this problem?

8. Write a MATLAB code to calculate the element stresses for this problem. For the 10×1 and 80×16 analyses, plot the FE σ_{xx} as a function of x at $y = -h/2$, the lower face of the beam. Superimpose the analytical σ_{xx} stress. What do you observe?
