CE 205: Finite Element Method: Project Assignment

Instructor: Dr. Narayan Sundaram*

March 25, 2024

This is a MATLAB programming project assignment. You must submit your assignment as a single zip file containing (1) A pdf with all derivations, results, plots, and discussions (2) All MATLAB .m files for your project in a single folder inside the zip file. Show all work clearly and name files appropriately. Plots must be labeled properly. This assignment has FOUR pages.

I. Problem description

1. Write a plane stress MATLAB FE code to analyze the problem of stress concentration due to a central circular hole in a remotely loaded square aluminum plate of given dimensions. The half-width / half-height of the plate $H = W = 254$ mm, and the radius of the hole $R = 25.4$ mm, giving an $R : W$ ratio of 1:10. For aluminum, $E = 70$ GPa and $\nu = 0.33$. The out-of-plane plate thickness t^* is 5 mm. The plate is subjected to a stress σ_{yy}^0 of 10 MPa along its top face.

   You can use quarter-symmetry to model the plate (see Fig. 1), with appropriate boundary conditions along the symmetry lines. Due to the distant location of the points of load application, the nodal forces need only be statically equivalent to the resultant of the remote stresses – eliminating the need to calculate consistent nodal loads.

2. Carry out a convergence study using meshes of isoparametric Q4 elements to estimate the value of the stress concentration factor K_c. This is defined as

$$K_c = \frac{\sigma_{\theta\theta}^{max}}{\sigma_{yy}^0} \frac{W - R}{W}$$

   The second factor in this equation accounts for the reduced load carrying section in a finite plate with a hole. Here, $\sigma_{\theta\theta}^{max}$ is the maximum hoop stress at the edge of the

*Department of Civil Engineering, Indian Institute of Science

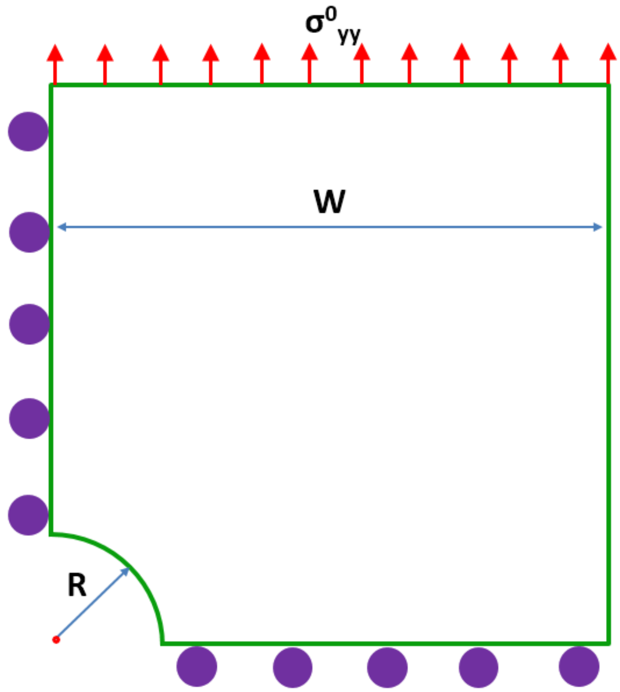

Figure 1: One quarter of a square plate with a circular hole

hole. The value of K_c should converge as you refine the mesh. Record the convergence by recording K_c against the number of elements along the edge of the hole N_h, the total number of elements N_{el}, and degrees of freedom.

Hint: This problem requires a high density of Q4 elements near the edge of the hole, while elements further away can be much larger as our region of interest lies close to the hole. Your mesh should, therefore, be 'graded' as shown in Fig. 2. This exercise is infeasible without element size transitions, you would require too many elements!

3. Repeat the convergence study using meshes of isoparametric T6 quadratic triangular elements. You should again use a graded mesh. Again, record K_c against N_h, N_{el} and the number of degrees of freedom.

4. Make a plot of the hoop stress $\sigma_{\theta\theta}$ vs θ for both your converged meshes. You will need to do a coordinate transformation of your FE stresses.

5. Plot the original and deformed hole shapes for both converged meshes. You can use a displacement scaling factor of 10-50 to show the deformed shape better.

6. Calculate and make contour / shaded-contour plots of the von Mises stress σ_{vm}:

$$\sigma_{vm} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_{xx}-\sigma_{yy})^2+(\sigma_{yy}-\sigma_{zz})^2+(\sigma_{zz}-\sigma_{xx})^2+6\left(\sigma_{xy}^2+\sigma_{yz}^2+\sigma_{xz}^2\right)}$$

for both your converged meshes. Explain your method of stress extrapolation, if any.

7. Comment on which element type is more suitable for this application. Report the minimum number of elements that is sufficient to get a converged K_c for each element type.

II. Implementation

To implement the code you can proceed as follows. You must report the results for items 2, 3, and 5 below in your project report pdf. Of course all source code must be submitted as well.

1. Write a MATLAB function 'isoparamQ4stiffness' that returns the 8×8 stiffness matrix of a plane stress, isoparametric quadrilateral Q4 element given the values of the input parameters : Elastic moduli E, ν, the nodal coordinates $(x_1, y_1) \cdots (x_4, y_4)$, and the element thickness t^{el}. The function should allow a flag to choose between full and reduced integration.

2. Test that your Q4 isoparametric element is working by using it (with full integration) in place of the rectangular Q4 elements in HW #4 and any one mesh in HW #4, Q7. The displacements should be identical to within numerical precision in both cases. Check that the values of the Jacobian J for a typical rectangular element are as expected at all integration points.

3. Write a MATLAB function 'isoparamT6stiffness' that returns the 12×12 stiffness matrix of a plane stress, isoparametric, quadratic triangular element given the relevant input parameters. Recall that the physical element has 6 nodal coordinates $(x_1, y_1) \cdots (x_6, y_6)$. Use the $r - s$ element formulation with full numerical integration using three interior sampling points. You can test the element's correctness by (1) Checking the Jacobian for a straight-sided element with side nodes at the mid-points of edges and (2) Using it to solve the cantilever problem

4. Write the core solver functions that perform assembly of the global stiffness matrix, apply symmetry and any other boundary conditions to get the reduced stiffness matrix, formulate the load vector, and solve for the unknown nodal d.o.f.s.

5. You can use any third-party mesh generator of your choice for mesh generation (e.g. Gmsh) or write your own. However, you must report which mesh generator you used, and include detailed pictures of the meshes you use with a zoom on areas near the hole. The mesh generator that you use must have the ability to generate graded meshes, as explained earlier (see Fig. 2).

6. Write the postprocessors: These are functions that calculate stresses at the element centroids, update the coordinates to get the deformed shape of the plate, etc.

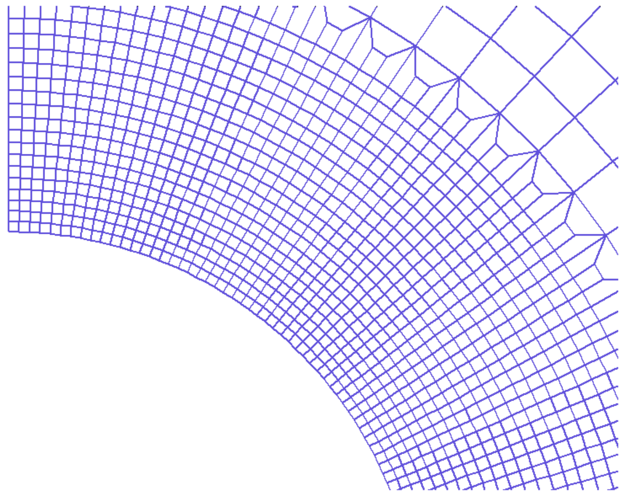

Figure 2: An example of a Q4 mesh with grading and element size transition. This shows a portion of the mesh near the hole.

7. Write helper functions, e.g. to perform coordinate transformations, calculate Jacobians, do visualizations etc.

As you code, watch out for the following common mistakes:

- Lack of testing to see if the element stiffness matrix function is correct.
- Meshes where the elements are not all consistently oriented, i.e. not checking that the Jacobians are all positive
- Q4 meshes that have too many d.o.f.s to be solved with dense matrix linear algebra. This should not be a problem in this project if the meshes are correctly optimized.
- Inconsistent mid-node node numbering in T6 elements.
- Incorrect application of boundary conditions (e.g. omitting some nodes) or loads.