Due Date: April 12, 2016

COMPUTER PROJECT # 2
ME/AE 6212 Advanced Finite Element Analysis

Consider plates with circular, elliptical, and rectangular holes at the center (as shown in figures on next page), subjected to tensile stresses. The material to be used has a modulus of elasticity E = 200 GPa, Poisson's ratio ν = 0.3. Use ABAQUS to analyze the plane stress problem.

(a) Use a full plate model (for the three cases), with 3 node triangular elements. Take plate thickness 0.02 m. Plot the deformed shape and von-Mises stress distribution.

(b) Use a symmetric quarter plate model (for the three cases), with 3 node triangular elements. Plot the deformed shape and von-Mises stress distribution.

(c) Compare and tabulate the maximum displacements and the maximum von-Mises stresses for cases (a) and (b). Perform the convergence study using atleast three different element sizes (coarse to fine). For your verification, calculate the ratio of maximum stress obtained to nominal stress and compare it with theoretical stress concentration factor from the handbook (or literature). Include this in your table.
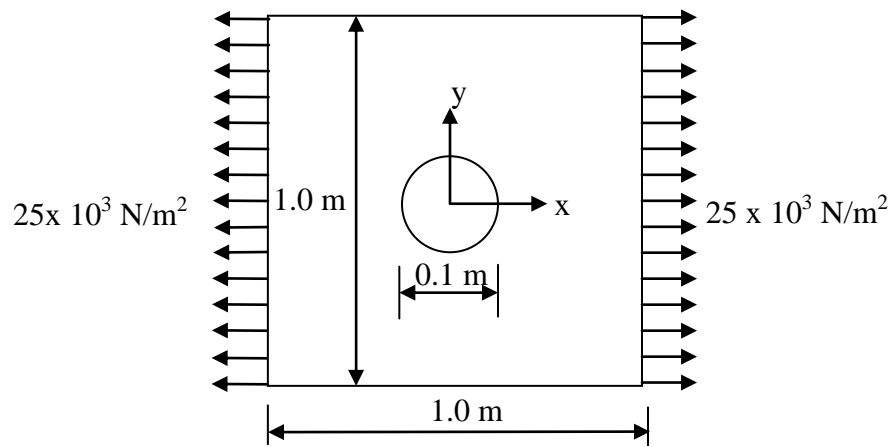
Note:

- In ABAQUS, the shell edge load is defined as force per unit length.

- If you are using a student version of ABAQUS, the number of elements is limited to 1000. Accordingly, use appropriate number of elements and perform the convergence study.
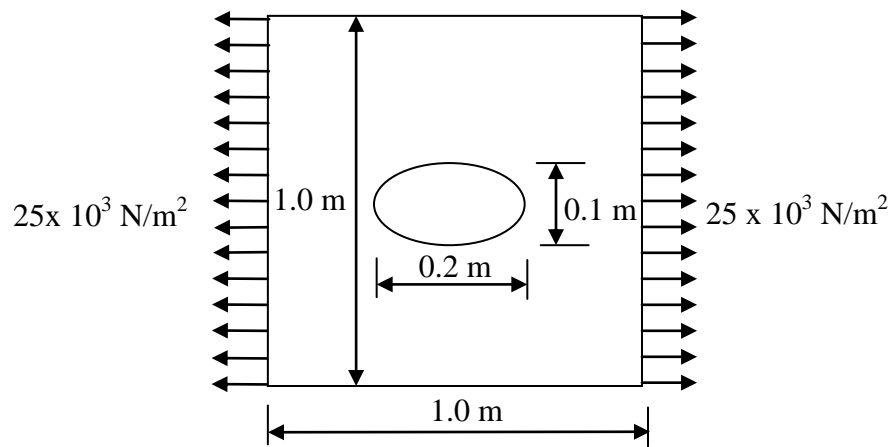
The report should include the following:
1. Cover page (Title, name, etc.)
2. Statement of the problem
3. Procedure/Related equations
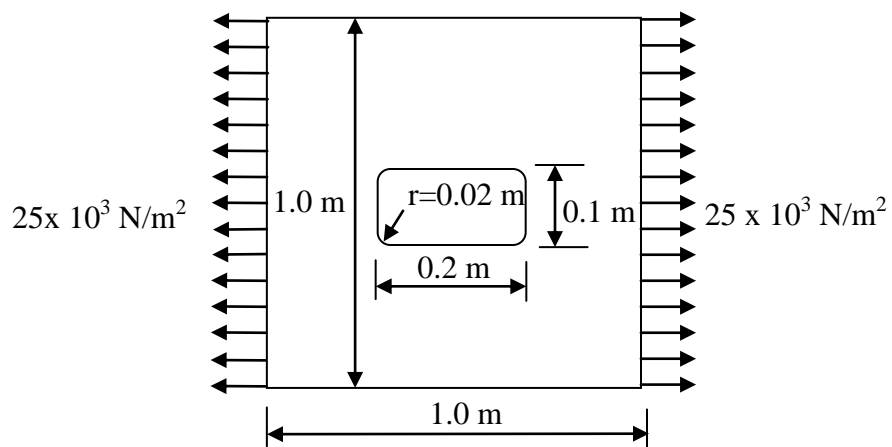4. Summary of results with units and discussion of results
5. Sample output
The report should not exceed 20 pages.

25×10^3 N/m²   1.0 m   y   x   0.1 m   1.0 m 25×10^3 N/m²

(a) Plate with a circular hole

25×10^3 N/m²   1.0 m   0.1 m   0.2 m   1.0 m 25×10^3 N/m²

(b) Plate with an elliptical hole

25×10^3 N/m²   1.0 m   r=0.02 m   0.1 m   0.2 m   1.0 m 25×10^3 N/m²

(c) Plate with a rectangular hole