Name____________________________________

ME 786/886
Fall 2022
Prof. I. Tsukrov

Homework #9 (Lab 5)

STRESS CONCENTRATION AROUND CIRCULAR HOLE:
FINITE ELEMENT ANALYSIS
(Due 12/01/22)

Background. It is well known that holes produce concentration of stresses. For example, the stress distribution in a continuous rectangular plate subjected to uniaxial tension *p* is uniform (Fig. 1a). However, in the presence of defects, the stresses around defects are much higher. If one considers a circular hole having a diameter much smaller than the size of the plate (circular hole in infinite plate), the stress near the hole is $\sigma_{yy} = 3p$ while far from the hole it is still $\sigma_{yy} = p$ (Fig. 1b).

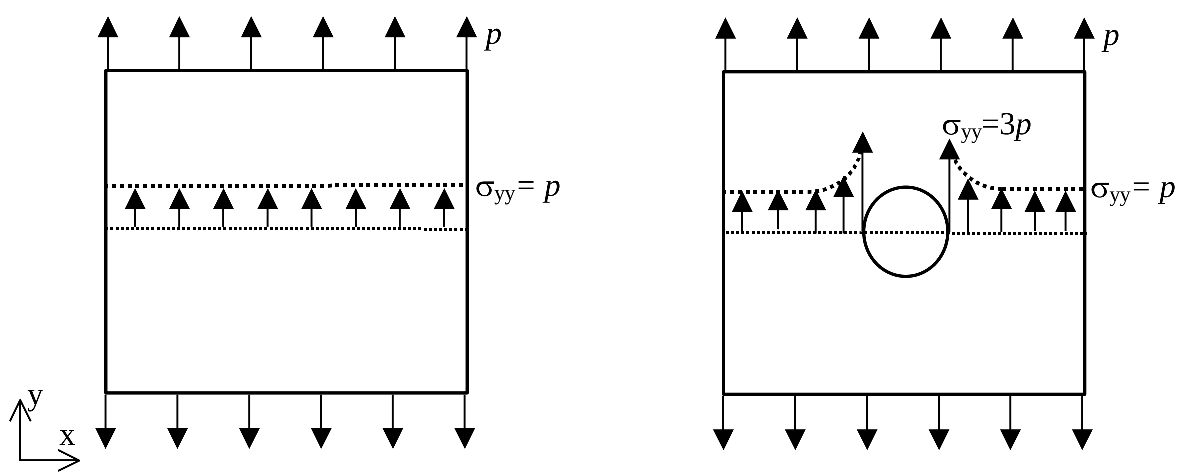

Figure 1a Figure 1b

Assignment. Perform plane stress finite element analysis of a square plate of thickness t = 6 *mm* with a circular hole of radius R = 12 *cm* subjected to tensile stretching of p = 100 *MPa* (Fig. 2a). Assume the plate to be made of steel with E = 220 *GPa* and ν = 0.2. Investigate how the relative size of the hole influences the stress concentrations by considering square plates with the side length (1) $a = 40\,cm$, (2) $a = 60\,cm$ and (3) $a = 80\,cm$.

Note that due to symmetry, only 1/4th of the model may be simulated with appropriate boundary conditions (Fig. 2b). For all parts of your analysis, work with a quarter of the plate. To generate FEA meshes, use information provided in the appendices. Using MARC perform the analysis outlined below.

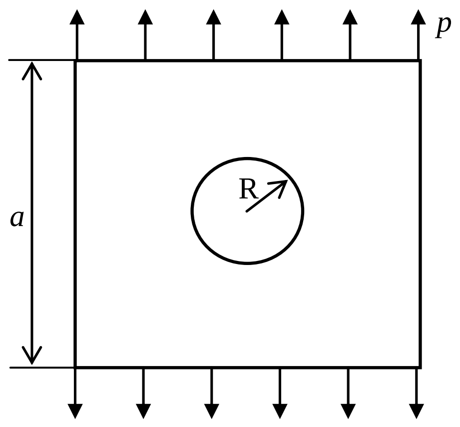

Figure 2a

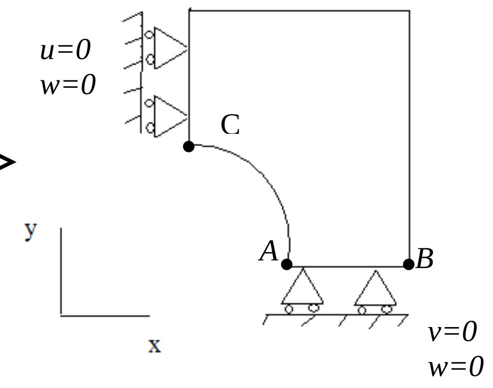

Figure 2b

1. First, consider case (1): square plate with side $a = 40\,cm$ having a circular hole in the center. Start out by creating a mesh of 4-node quadrilateral elements using Automesh as described in **Appendix 2D**. Apply the appropriate boundary conditions, and run the problem assuming plane stress. (Choose *plate.mud* as a file name when you save your model.) Record the values of vertical stress σ_{yy} at points *A* and *B* and horizontal stress σ_{xx} at point C shown in Fig. 2b.

2. Refine the mesh using different Automesh divisions and observe whether the calculated values of stress at points *A*, *B*, and *C* change.

3. Use the results of your simulations (nodal values of stress) to plot the distribution of stress σ_{yy} along the line *AB* for both original and refined mesh. (Plot them on one graph. You can use Excel, Matlab or any other software to plot your graph.) Do you observe any difference? Comment.

4. Repeat steps 1-3 for cases (2) $a = 60\,cm$ and (3) $a = 80\,cm$.

5. For the results obtained using the original mesh of case (1), report the value of stress σ_{yy} at the Gauss quadrature point closest to point *A* (you can get this value from *plate_job1.out* file). Now, graphically display your results in MARC (choose *Results* -> *Open* -> *plate_job1.t16*. Select *Numerics* plot for σ_{22}). How does the value at the quadrature point compare with the stress shown at point *A*?

6. Use **Appendix 3D** to generate a 3D mesh based on your original 2D mesh from step (1). Perform 3D analysis (*job* $\rightarrow$ *analysis dimension* $\rightarrow$ *3D; element types* $\rightarrow$ *solid* $\rightarrow$ *7*). Present picture with distribution of σ_{yy} stress, compare its max value with the 2D plane stress prediction.

Your report should include:
1. Problem statement.
2. Description of all meshes used (mesh plots with boundary conditions are required).
3. Graps and tables.
4. Discussion of results (2-4 paragraphs). Your discussion should address the questions asked in the above outline.

APPENDIX 2D

a. To use MARC/MENTAT AUTOMESH

1. In *"Geometry&Mesh"* add points A, B, and C as well as top left and right corner points. Also create a center point for the hole.
2. Select *Curves* → *Arc Cen/Pnt/Pnt* then *Curves* → *Add* → Select the center point then points A and C to create the curve.
3. Select *Curves* → *Line* → then connect the rest of the points to create the plate geometry.
4. Select *Automesh* → *Planar* and set divisions under *Quadrilaterals (Overlay)* to 12, 12 then click *Quad Mesh!* Then select all of your curves and right click in the graphics window to obtain the mesh. Note that you can enter different division values for a more or less refined mesh.

b. To have Gauss quadrature point data in the *plate_job1.out* file:

Jobs → *Structural* → *Job Results* → *Output File* → change *"No Element Results" and "No Node Results" to "Full Element Results" and "Full Node Results"*

c. Use *Edge Load* with units of force per area. Also note the direction it is applied.

APPENDIX 3D

a. To generate a 3D mesh by extrusion of the 2D model in *z*-direction. Assuming that you use meters as you length units, the following sequence of commands has to be performed

Geometry&Mesh → *Expand* → *Translation* → *0, 0, 0.002*, *Repetitions* → *3*

and select *Expand Elements* <all of your elements>

Geometry&Mesh → *Sweep* → *All*

b. Change geometric properties to *3D solid* for all elements. Check *Boundary Conditions*, and reassign constraints and loads if needed.

c. To assign a distributed pressure to a 3D surface, choose *B.C.* → *Face Load* → *<value of pressure with the appropriate sign>*, and choose the desired element faces.

d. Refine the mesh in the thickness direction:

Geometry&Mesh → *Subdivide* → *Divisions: 1, 1, 2* <select all of your elements>

Geometry&Mesh → *Sweep* → *All*