Stress Concentration in a Plate with 2D Finite Element Triangles

A plate in plane-stress has a $0.01 meter$ diameter hole in it. In this homework, you will be using 2D finite elements to calculate the maximum $|\sigma_x|$ stress in the plate (where x is 'to-the-right' and y is 'up'). The plate is $0.05 meters$ wide and $0.05 meters$ tall, as shown in the below picture. The plate has a thickness of $0.001 meters$ (into the paper). The picture also shows the simple triangle-mesh that you will be using. The plate is loaded with a uniform horizontal pressure/traction (to-the-right) on its right edge with a total load of $1000N$. The boundary condition is $u = 0$ on the left edge of the plate. The plate is made of steel with $E = 210 \times 10^9 N/m^2$ and Poisson's ratio $\nu = 0.3$.

You have been provided with the list of 5314 triangle elements and the associated list of 2773 nodes in the below ASCII text files:

elems.out
nodes.out

You have also been provided a list of the left-edge nodes in the below ASCII file:

leftlist.out

and a list of the right-edge nodes in the below ASCII file:

rightlist.out

Use the 3node triangle-element (Constant-Strain-Triangle [CST]) to find the maximum $|\sigma_x|$ stress in the plate. Compare it to the theoretical value from Figure A-15-1 below, which gives a $|\sigma_x|_{\max}$ of $6.25 \times 10^7 N/m^2$. (You will find the FEM agrees within 0.6% even with these simple CST's.) For the nodal-loads, it is adequate to simply apply the total $1000N$ equally to each of the nodes on the right edge (the nodes in file rightlist.out), i.e., 1000/51 Newtons to each node.

The associated reduced-stiffness matrix solves in ***<u>6seconds or 2.35seconds</u>*** in MATLAB on my desk PC, e.g., depending on what command you use:

```
>> tic; disp_reduced = inv(Kreduced) * nodal_load_reduced
Elapsed time is 6.082989 seconds.
>> tic; disp_reduced = K_reduced \ nodal_load_reduced; toc
Elapsed time is 2.356513 seconds.
```

even using plain dense computations and no sparsity-exploitation. Make sure your computer will store about a 5494x5494 matrix of floating-point values before you begin. A 5500x5500 matrix in double precision is only 242MB; my desk PC has 8GB of RAM (8e9 / 242e6 ≈ 33); so, unless you have a very, very tiny computer, you should not have any memory problems on this hw, even using dense-matrix methods.

You are not allowed to use a canned FEM software for this homework. You must write the for-loops and turn in at least a hardcopy of the files [MATLAB, C, Fortran, whatever] that you used to solve this homework problem.

HINT: You will need to constrain a single node to have zero y-displacement; otherwise, your reduced stiffness matrix will be singular and your Ax=b solver may give complaints.

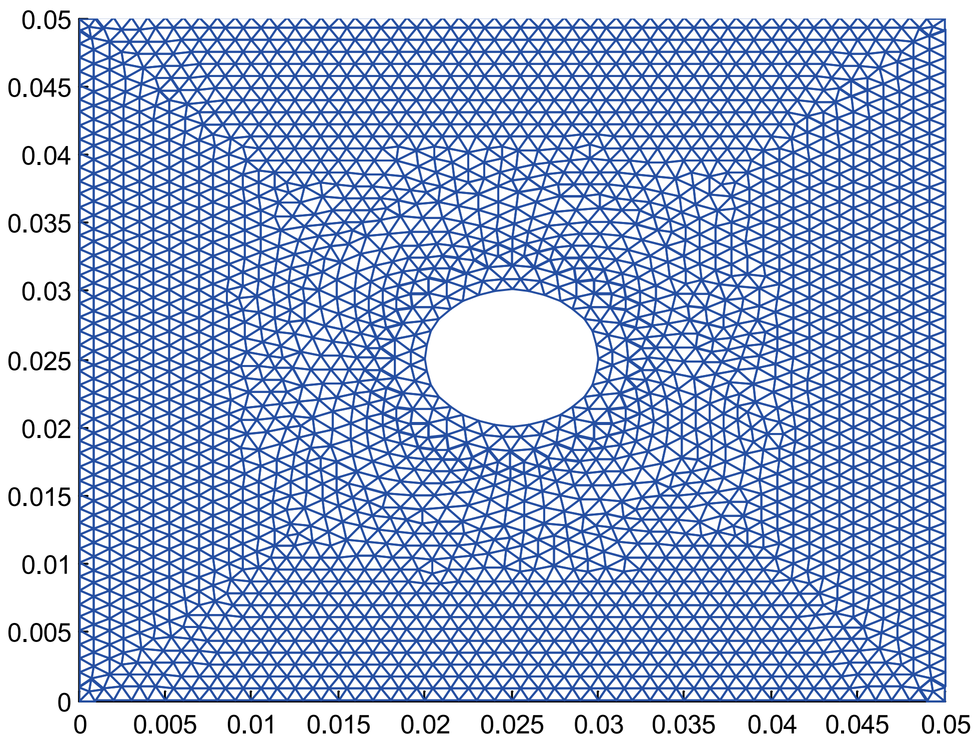
0.05
0.045
0.04
0.035
0.03
0.025
0.02
0.015
0.01
0.005
0
0
0.005
0.01
0.015
0.02
0.025
0.03
0.035
0.04
0.045
0.05

TABLE A-15

Charts of Theoretical Stress-Concentration Factors K_t^*

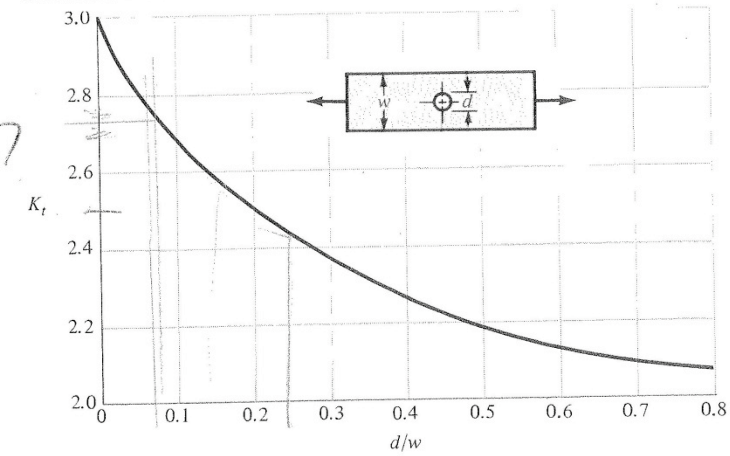

FIGURE A-15-1

Bar in tension or simple compression with a transverse hole. $\sigma_0 = F/A$, where $A = (w - d)t$ and t is the thickness.