AME40541/60541: Finite Element Methods
Homework 1: Due Monday, February 8, 2021

Problem 1: (10 points) Complete the tutorial in `comsol_heat2d.pdf` on the course website. Submit a plot of the temperature distribution and the temperature at $(x, y) = (0.3\text{m}, 0.1\text{m})$.

Problem 2: (30 points) From S. Govindjee, UC Berkeley. In this problem you will examine the behavior of linear vs quadratic elements in trying to estimate the stress around a hole in a plate.

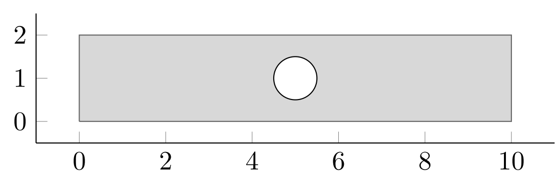
1) Start COMSOL: 2D analysis, structural mechanics, stationary analysis.

2) Under *Global Definition* and *Parameters*

   - Right-click *Parameters* and define a new parameter named *mesh_size* (under *Name*). We will be varying the *Expression* value; set it to 0.05 for now.

3) Create the geometry as follows:

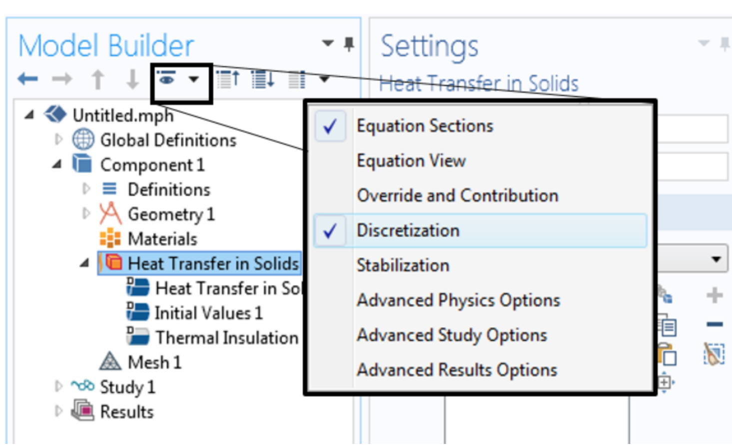
   - Rectangle: $0 \leqslant x \leqslant 10\text{m}$ and $0 \leqslant y \leqslant 2\text{m}$
   - Circle: center $(5, 1)\text{m}$ and $r = 0.5\text{m}$
   - Take the difference of the rectangle and the circle (using *Booleans and Partitions*, *Difference* under the *Geometry* tab) to create the domain shown in the figure. If you have trouble selecting the appropriate object when creating the difference, use the middle button to cycle through objects while hovering the mouse over them.



   - Add a point at $(x, y) = (0, 1)$ for later use

   Our objective is to compute the maximum von Mises stress $\max \sigma_{mises}$ throughout the domain using different mesh densities and element types.

4) Within *Solid Mechanics* set the *2D Approximation* to *Plane Stress* and the *Thickness* to 1 mm. Also set the *Discretization* to *Linear*. If you do not see *Discretization* in the *Solid Mechanics* settings, you can add it through the *Model Builder* window.



5) Set the elastic properties as: Young's modulus = 70 GPa and Poisson's ratio = 0.33.

6) Add a *Prescribed Displacement* and a *Prescribed Load*. On the left edge, fix the displacement in the x-direction to be zero. At the point $(0, 1)$, fix the displacement in both directions. On the right edge, set the load type to *Total Force* with a value of 1 kN in the x-direction.

7) Define a *User-controlled mesh* with *Maximum element size* set to be *mesh_size* (your parameter). This forces the mesh to have elements roughly the size of the parameter value.

8) Right-click *Study* and add a *Parametric Sweep*. Left-click *Parametric Sweep* and add *mesh_size* as a parameter name using the plus sign and in the *Parameter values* field enter:
1 0.8 0.6 0.4 0.2 0.1 0.09 0.08 0.07 0.06 0.05.

9) If you now solve (*Compute* under *Study*), COMSOL will solve the problem using the sequence of meshes you defined.

10) The maximum von Mises stress happens at the top and bottom of the hole. To extract the needed information, use the *Derived Values* option with a *Point Evaluation*. To get the correct variable in *Point Evaluation* change *Expression* to *solid.mises* by typing it in or using the menus. Make sure the *Data set* is *Study1/Parametric Solutions 1*. If you now *Evaluate All* (*Results* tab), it will create a table of values for your for each mesh density.

11) Repeat this study with quadratic elements. Go back to *Solid Mechanics* and change the *Discretization* to *Quadratic*. Also go to the *Mesh* tab and change the size to 2∗*mesh_size*. This will keep the number of nodes close to that of the linear case (more fair comparison). Go to *Study* and *Compute* a new solution. Re-evaluate your *Point Evaluation*. This will add a new column to your results table.

12) Export the table with both columns (linear and quadratic elements) and create a plot of mesh size vs. maximum von Mises stress (use a semi-logx plot). Don't forget that your quadratic elements have elements roughly twice the size of your linear elements (even though it won't be reflected in your table). Comment on the meaning of what you observe. Also export a plot of the contours of the von Mises stress over the plate.