UNIVERSITY OF CALIFORNIA BERKELEY
Department of Civil Engineering
Spring 2016

Structural Engineering,
Mechanics and Materials
Professor: S. Govindjee

Lab 3

In this lab you will examine the behavior of linear versus quadratic elements in trying to estimate the stress concentration factor around a hole in a plate. In this lab you will learn how to define parameters, compute the response of the system for a range of those parameters, and how to conveniently extract information about the computation. From an intellectual point of view, you will learn the power of quadratic approximation versus linear approximation; you will also learn that commercial programs do not always live up to their billing. In this particular case, you will see that COMSOL has trouble converging the answer correctly, though it does get it correct to a few digits.

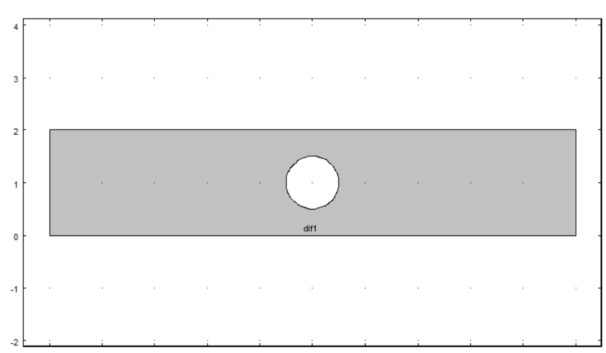
1. Start COMSOL and select a 2D analysis with the Structural Mechanics:Solid Mechanics module. Use stationary analysis.

2. Under *Global Definition* add *Parameters*.

    - Left-click *Parameters* and define a new parameter named *mesh_size*; just type it the box under *Name*. In the *Expression* box enter a number like 0.05. The exact value is mostly irrelevant. When you perform your convergence study, you set the actual values.

3. Create a geometry as follows:

    - Rectangle, with $0 \leq x \leq 10$ m and $0 \leq y \leq 2$ m
    - Circle, with center at $(x, y) = (5, 1)$ m, and $r = 0.5$ m
    - Add a point at $(x, y) = (0, 1)$ for later use.



Our objective will be to compute the stress concentration factor around the hole under axial loading using different mesh densities and element types. In this context, the stress concentration factor is

$$K = \frac{\max \sigma_{\text{mises}}}{\sigma_{\text{applied}}} \tag{1}$$

4. Within *Solid Mechanics* set the *2D Approximation* to *Plane Stress* and the *Thickness* to 1 mm. Also set the *Discretization* to *Linear*; later you will come back to this tab and set this to *Quadratic*. With the solid mechanics chosen, one can directly set the discretization with out adding an explicit sub-option like you did in the last lab – though that method also works here.

5. Set the elastic properties so that you have a Young's modulus of 70 GPa and a Poisson's ratio of 0.33.

6. Add a *Prescribed Displacement* and a *Prescribed Load*. On the left edge, fix the displacements 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 *Size* set to be *mesh_size* (your parameter). This forces the mesh to have elements of roughly the size of the parameter value (though they are smaller) around the hole.

8. Right-click *Study* and add a *Parametric Sweep*. Left-click *Parametric Sweep* (if not auto selected), 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, the program will solve in sequence the given problem using the mesh sizes that you have specified.

10. The maximum von Mises stress happens at the top and bottom of the hole. To conveniently 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 to set the point correctly in the *Selection* window. Also set the *Data set* to *Solution 2* (this is the parametric sweep).

    If you now *Evaluate* (right-click *Point Evaluation* or use the equal sign at the top of the *Point Evaluation Settings* frame), it will create a table of values for you for each mesh density. It is a good idea to rename the table that it creates to something meaningful.

11. Now 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 and will in effect make for a fairer comparison.

Go to *Study* and *Compute* the answer anew. Now re-evaluate your *Point Evaluation*. This will add a new column to your results table (keep track of the meaning of the columns or evaluate to a new table to keep your data organized).

12. Export your tables for the linear case and the quadratic case. Compute the stress concentration factors for each mesh density and element type. Make some nice plots (try a semi-logx plot). Put it altogether into a report that shows the results and explains the meaning of what you observe. Add some plots of the results, such as the contours of the von Mises stresses over the plate.

13. Finished early?

    (a) Decrease the mesh parameter so that you are convinced that you have a converged result to 3 digits.

    (b) Add a third curve to your study which uses cubic elements (use *3*mesh_size* for the max element size).

    (c) Determine if changing the Young's modulus changes the stress concentration factor, note $E > 0$ is the thermodynamically valid range.

    (d) Determine if changing the Poisson's ratio changes the stress concentration factor; note $-1 \leq \nu < 1/5$ is the thermodynamically valid range.