ME345 Modeling and Simulation, Spring 2012
Case Study 2
Assigned: Friday March 23, 2012
Due: Friday April 20, 2012

<u>Note 1</u>: You are STRONGLY encouraged to complete the assignment prior to this due date!

<u>Note 2:</u> Each of the Case Studies will be done in groups of 4 students. Each student is expected to contribute equally to the group work. No two students can be in the same group for more than one Case Study (as discussed in class). Students will be asked to complete an anonymous Group Member evaluation after submitting the Case Study assignment. Each group member should know exactly how the other members approached and solved the problem.

PROBLEM 1. (40% of grade)

In a manufacturing process, a transparent film is being bonded to a substrate as shown in Figure 1 below. To cure the bond at a temperature T_0, a radiant source is used to provide a heat flux q_0'' (W/m^2), all of which is absorbed at the bonded surface. The back of the substrate is maintained at a constant temperature T_1 while the free surface of the film is exposed to air at T_∞ and a convection coefficient h. Assume the following conditions: T_∞ = 20 °C, h = 50 W/m^2 K, and T_1 = 30 °C. Find the heat flux q_0'' (W/m^2) that is required to maintain a bonded surface temperature of T_0 = 60 °C. (Suggestion: to some extent you can set up the work on the Parts below separately; be efficient in the use of your group's time and resources.)

Part A. We can model this as a one-dimensional heat transfer problem. Using just 1D convection and conduction finite elements, draw the equivalent circuit by hand. Using the finite element analysis approach by hand demonstrated in lecture, derive the global stiffness matrix with the known temperatures and external applied heat fluxes shown in matrix form.

Hint 1: the problem has three elements in this case: convection at the top surface, conduction through the film, and conduction through the substrate. Be careful when considering units.

Hint 2: if you re-number the nodes from top to bottom, node 1 has a temperature of T_∞ and an unknown heat flux, T_2 has an unknown temperature and zero external applied heat flux (all heat flux is assumed to be applied at node 3), T_3 has a desired temperature of 60 °C and the necessary external applied heat flux (this is an unknown that you are trying to calculate), and T_4 is the known temperature labeled T_1 above with unknown external applied heat flux; hence you should have 4 equations and 4 unknowns.

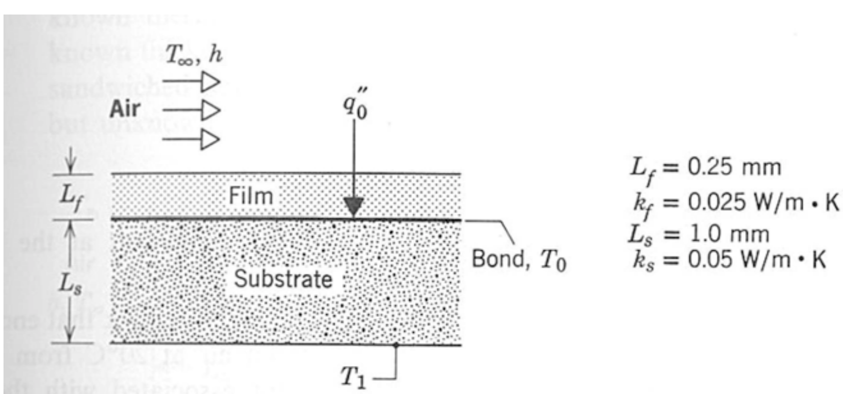


Figure 1. Schematic of manufacturing process for Problem 1.

Part B. Using your favorite math tool, solve the system of four equations and four unknowns to determine the necessary heat flux q_0'' (W/m^2) to maintain a temperature of 60 °C at the bonded surface.

Part C. The objective is now to compare your analytical solution with a three-dimensional CAE analysis. Using the heat flux q_0'' (W/m^2) found in Part B for all cases, conduct a DESIGN SCENARIO where the lateral dimensions of the square film (in mm) are 5, 10, 15, 20, 25, and 30mm on a side. *Assume that the same convection conditions at the top of the film are also present on all sides of the film.* In your report describe:

C1. In a little detail, the specifics of how you set up the DESIGN SCENARIO in SolidWorks Simulation. (I am specifically looking for you to describe how you automated the process, while you can check all cases by hand if you wish, it is not enough to run individual analyzes for each analysis; I am specifically asking for the Design Scenario tool to be used.)

C2. In your report, discuss the sensibility of your results in terms of how changing the lateral dimensions of the film and substrate alter the temperatures that you found.

Part D. FOR ONE OF THE LATERAL DIMENSIONS IN PART C above, demonstrate the use of quarter-symmetry in your model. Compare the results of your quarter-symmetric model with the results your full model. (Hint: you may need to adjust the value of the applied heat flux when using quarter symmetry boundary conditions?)

PROBLEM 2. (30% of grade)

A rectangular alloy steel plate with two holes is subject to a 20,000N tensile load as shown in Figure 2. You are asked to run a Design Optimization in SolidWorks Simulation to determine if the plate can be made lighter (i.e. the holes made larger). As shown in Figure 2 below, in the initial size of the plate is 50mm in height, 120 mm in length, and 5mm in thickness. (NOTE: In SolidWorks Simulation Xpress there are limitations with respect to the optimization; specifically the number of parameters that can be varied simultaneously, so do not use Xpress.)

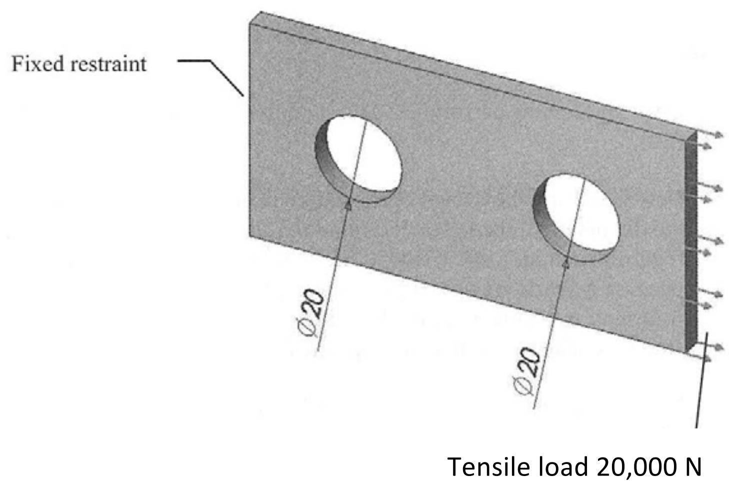


Figure 2. Initial geometry for Design Optimization study. Assuming the origin is (0,0), the center of the left hole is (30, 25); the right hole is mirrored about the centerline of the plate.

Part A. For the given geometry and conditions in Figure 2, determine the maximum stress in the plate. *Quantitatively* discuss the sensitivity of your results to the size of your mesh with examples. Discuss your result.

Part B. Now perform a Design Optimization on the plate using SolidWorks Simulation, maintaining the outer dimensions of the plate but changing the diameters of the holes in the plate, with the goal of minimizing the weight of the structure. The diameters of the holes can be different. The maximum possible diameters of the holes are assumed to be 40 mm. Minimize the weight of the plate while ensuring that the von Mises stresses are less than the yield stress. (Suggestion: you are strongly encouraged to find sources describing how to conduct Design Optimizations using SolidWorks Simulation.) Be sure to discuss:

- in some detail, the procedure of conducting the Design Optimization in SolidWorks Simulation
- the values of the weight pre- and post-optimization
- the impact of a Factor of Safety on your analysis (illustrate using a realistic example)

Part C. It has been suggested that an alternative manner to reduce the weight of the plate is to change the material. You have been asked to consider two materials: a high performance aluminum and a high performance polymer (plastic). For each of these materials, conduct an analysis similar to the one above to determine the whether these materials will work and, if so, determine an optimal size of the holes for each material. (Note: Do you really want to limit your choices to a small number of materials in a given database?)

Problem 3 (30% of grade)

Consider the stress concentration curves for a flat plate with a hole subjected to axial tension shown in Figure 3. These curves were developed to give designers a way to estimate the stress concentration factor K, which is defined as:

$$K = \frac{highest\ value\ of\ actual\ stress\ on\ notch, fillet, hole, etc}{NOMINAL\ stress\ as\ given\ by\ ELEMENTARY\ equation\ for\ MINIMUM\ cross\sec tion}$$

You are asked by your supervisor to compile a report on the following:

A. Describe *in words* the utility of having access to values for the stress concentration factor. Is *K* a function of material properties?

B. Conduct a finite element analysis to verify the stress concentration factors provided in the table. (To do this, run a few analyses for different geometries and compare the numbers to the curve shown in Figure 3.)

C. Be sure to comment on the accuracy of your solution by considering the following:

   a. Verify that you can accurately utilize half- and quarter-symmetry in the analysis

   b. Demonstrate convergence of your solution as the number of elements in the model increases

   c. Compare your results by repeating some of the analysis using a different finite element package. For example, if you used ANSYS as the primary tool, compare your results with SolidWorks Simulation, or vice versa. Be sure to comment on the quantitative similarities and differences you find.

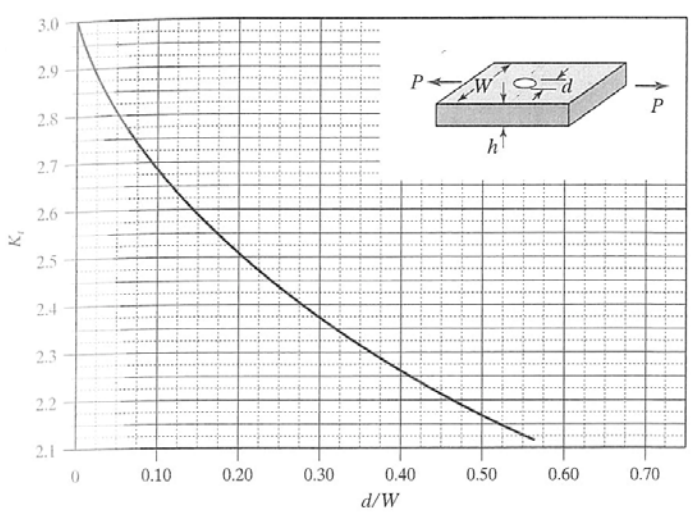


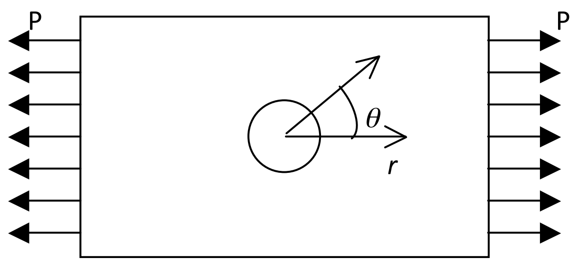
Figure 3. Stress concentration factor for 'plate with a hole' in tension (top) (from Design of Machine Components, 8th Edition, Spotts, Shoup, and Hornberger, 2004)

D. When d/W is very close to zero (i.e. an 'infinite' plate), the stresses in cylindrical (two-dimensional polar) coordinates can be written as

$$\sigma_{rr} = \frac{P}{2}\left(1-\frac{a^2}{r^2}\right)+\frac{P}{2}\left(1-\frac{4a^2}{r^2}+\frac{3a^4}{r^4}\right)\cos 2\theta$$

$$\sigma_{\theta\theta} = \frac{P}{2}\left(1+\frac{a^2}{r^2}\right)-\frac{P}{2}\left(1+\frac{3a^4}{r^4}\right)\cos 2\theta$$

$$\sigma_{r\theta} = -\frac{P}{2}\left(1+\frac{2a^2}{r^2}-\frac{3a^4}{r^4}\right)\sin 2\theta$$



where *a* is the radius of the hole, *r* is the radial coordinate with *r=0* located at center of the circle, and θ is the angular coordinate as drawn below. This assumes that stresses in the *z* direction (the thickness direction of the plate) are zero, a condition known as plane stress.

Evaluate the stresses at the location where $r = a$ and θ=0 and θ=90 degrees. Comment on the value for the hoop stress ($\sigma_{\theta\theta}$) in relation to the stress concentration factor found via FEM for the case where d/W approaches zero.

Report:

You will need to submit an engineering report of your analysis to the company president (i.e. the professor). The report should be **well-organized, clear, and concise**, and at minimum address the points listed above The report should also provide a justification of why the results that you have obtained are sensible. **NOTE: just submitting the software output without your analysis and discussion is NOT acceptable. EACH PROBLEM SHOULD BE WRITTEN AS A SEPARATE 'CHAPTER', BUT SUBMITTED AS A SINGLE REPORT.**

While it is natural to discuss your work and progress with your colleagues (i.e. classmates), **individual group analyzes and reports are required [this cannot be emphasized enough – work that fails to meet this requirement will not be given credit for the assignment]**. Your supervisor (i.e. the TA) is also available to answer thoughtful questions as you work on the project, but it would be unprofessional to overly rely on your supervisor to complete your project.