ME345 Modeling and Simulation, Spring 2017
Case Study 2
Assigned: Friday March 24, 2017
Due: Thursday April 20 (by 11:59pm on CANVAS)

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

Note 2: Each of the Case Studies will be done in groups of 5 students. Each student is expected to contribute equally to the group work. No two students can sign up to 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.

Note 3: One report per group should be submitted through CANVAS by the deadline indicated above. All team members are responsible for all aspects of the project work.

PROBLEM 1. (30% of grade)

As shown in Figure 1 below, hot water flows through pipes that are embedded in a concrete slab (a section of which is shown below). The temperature of the water inside the pipes is 60 °C, with a corresponding heat transfer coefficient of 220 W/m^2 K. Assume that the thermal conductivity of concrete is 1.0 W/m K.

Part A: Using the conditions shown in Figure 1, determine the temperature (maximum temperature, as well as the temperature distribution) at the top surface of the concrete slab.

Part B: Assuming that the heat transfer coefficient of the water remains constant at 220 W/m^2 K (i.e. that it is independent of the water temperature), at what water temperature does the top surface of the slab freeze?

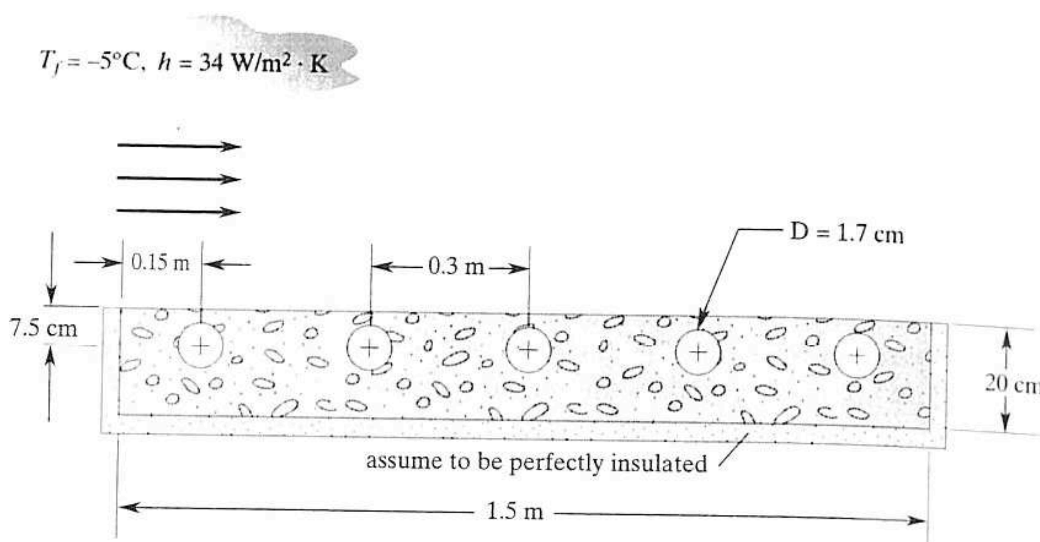


Figure 1. Cross section of problem for heat transfer analysis in Problem 1.

Part C: How does the length of the slab (perpendicular to the cross section shown in Figure 1) impact the results of the simulation. Assuming that the external convection conditions are the same on the front and back surfaces of the slab in the perpendicular direction, how long must the slab be such that heat transfer in the perpendicular direction can be neglected?

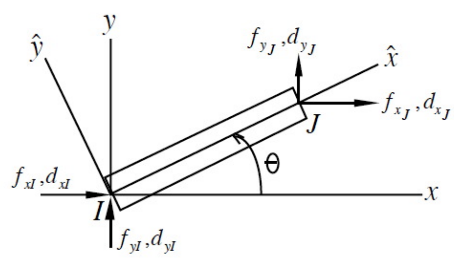
Part D: You are tasked with the goal of trying to slightly adjust the above setup in order to be able to decrease the water temperature necessary to prevent freezing at the surface (due to the considerable savings that may be able to be achieved by having to heat the water to a lower temperature). Specifically, you are asked to investigate how the depth of the pipes and their relative spacing (both in relation to the top wall as well as the relative spacing of the interior pipes with respect to each other). The pipes should remain symmetrical about the center of the slab (although you may consider only using 4 pipes if desired).

PROBLEM 2. (40% of grade)

Note: In class we have discussed several 1D finite elements. In this problem you will be asked to apply this knowledge to tackle a 2D truss problem. Further description of the 2D truss element and an example problem are provided in the class notes here (although not explicitly covered in class): http://personal.stevens.edu/~ffisher/me345/Intro_FEM_notes1.pdf

The element stiffness matrix for a 2D truss is global coordinates is given as

$$\begin{Bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{Bmatrix} = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{Bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{Bmatrix}$$



where $c = \cos\theta$ and $s = \sin\theta$, and A, E, and L are the cross-sectional area, modulus, and length of the element.

For the truss element shown in Figure 2, assume that the elements are made of aluminum (E = 70 GPa) with a cross-sectional area of 0.002 m^2. Assume a yield strength of 55 MPa for all elements.

Part A: Using the definition of the 2D truss element presented above, determine the deflections of points B, C, and D in the truss by implementing the finite element approach 'by hand'. (Please use a calculator to perform any matrix calculations.) Be sure to report the global stiffness and how it was assembled.

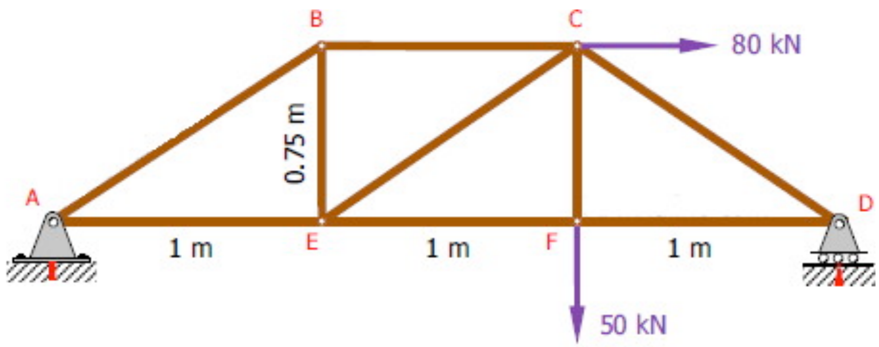


Figure 2. Truss problem for analysis in Problem 2.

Part B: To confirm your analysis, use a CAE package of your choice to determine the nodal displacements of points B, C, E, and F. Be sure to compare your results to the values obtained from Part A. Be sure to also report the stresses found for each element.

Part C: For three elements (of your choosing), use either the Method of Joints or the Method of Sections to determine the internal forces in the element. Compare this result to forces found in Part B. (Note it will be necessary to use the relationship between forces and stresses in this comparison.)

Report:

Your group 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.

PROBLEM 3. (30% of grade)

A rectangular alloy steel plate with two holes is subject to a 25,000N tensile load as shown in Figure 3. 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 3 below, in the initial size of the plate is 50mm in height, 120 mm in length, and 5mm in thickness.

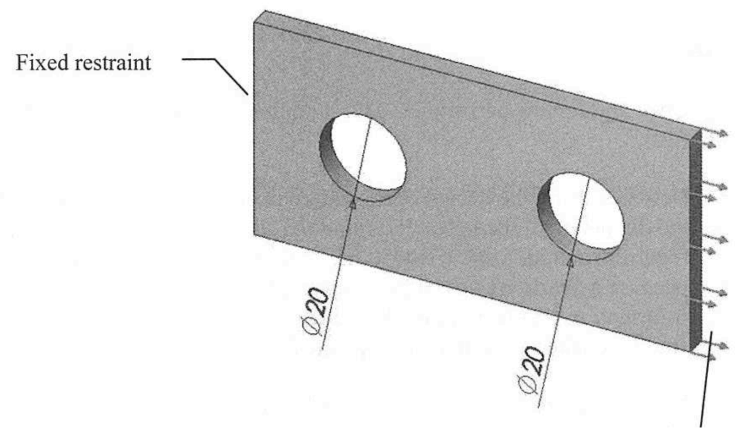


Figure 3. Initial geometry for Design Optimization study. Assuming the origin is (0,0), the center of the left hole is (30, 25); the right hole is mirrored about the centerline of the plate.

Part A. Select and justify an appropriate Factor of Safety to be used in the analysis. You may assume that the loads and environmental conditions in which the part will be operated are generally understood, and that the part will have infrequent (if any) inspections.

Part B. For the given geometry and conditions Figure 3, determine the maximum stress in the plate. *Quantitatively* discuss the sensitivity of your results to the size of your mesh with examples (specifically, conduct a simulation with different mesh densities and describe how this impacts your results). Discuss your results.

Part C. Now perform a design optimization on the plate, 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. Be sure to discuss:

- in some detail, how you conducted the design optimization
- 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 D. 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 software database?)