ME 114 Lab: Finite Difference

Mechanical and Aerospace Engineering | San Jose State University

A concrete dam used in applications of fluid mechanics can also be subject to large temperature gradients due to large heat loads and sinks. Large temperature gradients are generally not good from the standpoint of thermally induced stress and deflection. However, the boundary conditions are such that the temperature solution is difficult to obtain analytically. In this lab, you will explore the application of the finite difference technique to numerically solve for the resulting temperature distribution.

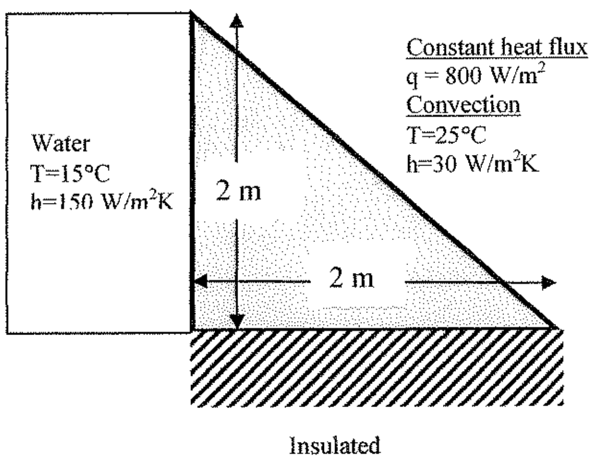

Figure 1. Cross section of concrete dam

a) Consider a long concrete dam (k=0.6 W/mK) with the triangular cross section shown. To the left of the dam, heat is convected to a large mass of water with a temperature of 15°C and a heat transfer coefficient of 150 W/m^{2}K. On the bottom boundary, the concrete is insulated. On the diagonal surface, there is a solar heat flux of $\dot{q}$=800 W/m^2, as well as convection to air at temperature 25°C and a heat transfer coefficient of 30 W/m^{2}K. We are interested in obtaining the 2D steady-state temperature distribution within the concrete dam. To start this assignment, assume that your stepsizes (i.e. Δx and Δy) are equal to 1 m.

b) Repeat your numerical simulation with a stepsize of 0.1m.

Steps:

1) Set up Excel to perform the iterations under "Tools", "Options", "Calculation". Set the calculation to manual, the maximum number of iterations to 1,000, and the maximum change to 0.001. The "maximum change" box tells Excel to consider your solution converged when the changes from one iteration to the next are less than 0.00001°C. Check the "iteration" box. Turn off "Recalculate before save." When you wish to calculate your results, hit F9. **Always save before hitting F9!** If you mistakenly enter an undefined number, it will spread throughout your spreadsheet once you hit F9, forcing you to start over. To save time, scroll down so you can't see the numbers on the screen before you start iterating. Otherwise, Excel will continuously update the numbers on the screen as it iterates 10,000 times, which makes the process take several times longer.

2) Outline your solution space on the spreadsheet by adding color to the cells where you'll be entering equations. **Enter a rough estimate of the final temperature in each cell (a temperature of 40°C is a good starting point).** Do not make these initial guesses equal to or less than T_∞. Save your spreadsheet before continuing.

3) Determine the equations that you will use in each cell. You must derive any formulas that we did not cover in the lecture yourselves. You may check the equations that you derive with the lab TA.

4) Enter your equations in the cells. Put constants (k, T_∞, Δx, h, $\dot{q}$, and any other repeated quantities) near the top of your spreadsheet to save time. Normally when you copy and paste an equation, the cell reference changes. If you want to copy a cell but continue to reference the constants, use the following format: B1 instead of B1. After inputting the equations for a small group of cells, save your results and hit F9. **If you get an undefined number anywhere, close your spreadsheet without saving and re-open the previous version.**

5) Continue this process until you have entered equations into all of the cells. You may have to hit F9 several times for the iteration process to be complete. You will know that it's complete if the word "Ready" in the lower left hand corner of the screen doesn't change when you hit F9.

Results and Discussion

1. Show the results of your numerical simulations in a table, and label them with the appropriate captions.
2. Where are the maximum and minimum temperatures located for both solutions, and what are their values? Are the locations of the max and min where you expected them to be?
3. What additional information does the more refined solution give you compared to the coarse model, if any? Comment on the pros and cons using smaller stepsizes.