Modeling with PDE, MA 461
Assignment 10: Heat Flow Problem
Due date: 04/05/2021

In this assignment we use COMSOL Multiphysics to investigate 2-D heat flow in a heated metal plate. **Email the instructor the mph file and upload your Report (see the end of the assignment) to Canvas.**

The Problem

The uniform metal plate with thermal diffusivity $1m^2/s$ is represented by a rectangle with opposite corners at $(-0.5, -0.8)$ and $(0.5, 0.8)$. It is assumed that the plate has a hollow crack or cavity represented by the rectangle with opposite corners at $(-0.05, -0.4)$ and $(0.05, 0.4)$. The left side of the plate is heated to $100°C$, while on the right side the heat is flowing out at a constant rate of $10W/m^2$; all other boundaries (the top, bottom, and interior sides of the cavity) are assumed insulated. The entire plate is initially at $0°C$.

The PDE for the temperature $u(x, y, t)$ at time t seconds at a point in the plate with coordinates (x, y) takes the form

$$\frac{\partial u}{\partial t} = \Delta u.$$

The region is bounded on the outside by the boundary of the large rectangle, and on the inside by the boundary of the inner rectangle. The boundary conditions are given by the Dirichlet condition

$$u = 100$$

on the left side, and the Neumann condition

$$\frac{\partial u}{\partial n} = -10$$

on the right side, and all other boundaries are assumed to take the Neumann condition

$$\frac{\partial u}{\partial n} = 0.$$

COMSOL Multiphysics

Specify 2D domain and choose *Mathematics→Classical PDEs→Heat Equation.* (Follow the procedure of A Wave Animation in the lecture notes.)

Specifying the Domain

Domains are constructed by the addition and subtraction of primitive domains such as rectangles, polygons, and ellipses. In the Model Builder window right click *Geometry 1* and choose *Rectangle* to draw the cavity first. Specify *Width* as 0.1 and *Height* as 0.8. Then set the base corner with x-coordinate -0.05 and y-coordinate -0.4. Click *Build Selected.* Repeat the procedure to build the larger rectangle of 1×1.6. Next right click *Geometry 1* and choose *Booleans and Partitions* $\rightarrow$ *Difference*. Activate *Objects to add* and choose the larger rectangle by clicking on it in the *Graphics* window. You should see r2 showing up as chosen. Similarly, choose the smaller one (r1) as the object to subtract. Click *Build Selected*. You should be able to see the intended domain in the *Graphics* window.

Specifying the PDE and Initial Condition

Click *Heat Equation 1* under *Heat Equation* in the Model Builder window. In the Heat Equation window click *Equation*. Set $d_a = 1$, $c = 1$, and $f = 0$. In the *Initial Values* window the initial condition has been set as 0 by default.

Specifying the Boundary Conditions

The boundary conditions have been set as zero flux (homogenous Neumann boundary condition) by default. You need to reset the right side as -10. For this you add a node to the model builder tree by right-clicking *Heat Equation* and choose *Flux/Source*. In the *Flux/Source* window choose boundary 8 (by clicking the right side of the larger rectangle) and set g as -10. (Keep q as 0.) Similarly, set Dirichlet boundary condition to the left side. Also check *Zero Flux 1* to make sure other boundary conditions are set correctly.

Specifying the Mesh

The COMSOL Multiphysics routines use the finite element method to approximate the PDE solution. This means that the region is divided into a number of triangular subregions, and the solution is then assumed to be continuous and piecewise planar on each of the subregions. This method is particularly useful when the Domain of the PDE has curved boundaries, as is common in many practical problems. COMSOL Multiphysics has automatic routines for performing the initial triangularization of the domain, and any subsequent refinements that might be needed for greater accuracy.

To initialize the mesh click *Mesh 1* in the Model Builder window. In the Mesh window the Element size is set as *Normal* by default. Change it to *Fine*. Then click the green button *Build All* to build the mesh.

Solving and Plotting

To set the time span of solution click *Step 1: Time Dependent* in the Model Builder window. In the Time Dependent window set *Times* as *range (0,0.15,3)*. This is to start the computation at $t = 0$, finish at $t = 3$, with time step 0.15 to save results from the computation. You can solve the equation by pressing the "=" button in the Home tab. After solving the problem view an animation.

The Report

Write a short report describing the movement of heat in the plate.

1. Besides the animation you may see an arrow plot that describes the temperature gradient: In *2D Plot Group 1* tab choose *Arrow Surface*. To see a clear plot right-click

Arrow Surface 1 in the model builder tree and select *Plot In New Window*. Similarly, you can view each of the surface plots under *2D Plot Group 1* in the model builder tree in a seperate window.

2. Can you see a slowdown of the increasing in temperature from the animation and draw some conclusion? From the arrow plot can you make some conclusion on the direction of the heat flow (i.e. heat flux) in terms of the temperature gradient?

3. Finally, determine the temperature at the middle of the back of the cavity (i.e. the point $(-0.05, 0.0)$) at time $t = 3$: In the Model Builder window right-click *Data Sets* and choose *Cut Point 2D*. In the *Cut Point 2D* window enter -0.05 for x and 0.0 for y. You may click *Plot* in that window to verify the point that you chose. Right-click *Derived Values* in the Model Builder tree and choose *Point Evaluation*. In *Point Evaluation* window select *Cut Point 2D 1* for *Data set* and *Last* for *Time selection*. Click *Evaluate* in that window. What do you find under *Table 1* tab beneath the Graphics window?

Hand in a surface plot, a surface plot with height and an arrow plot for the solution at $t = 3$ with your report. Also email me the mph file at ynzeng@uab.edu.