2.7: Finite Element Solutions using FEHT

2.7.1 Introduction to FEHT

Sections 2.5 and 2.6 present the finite difference method for solving 2-D steady-state conduction problems. The finite difference method is very powerful, but it is not convenient for systems with irregular geometry. Many of the advanced heat transfer simulation tools that are available do not use finite difference techniques but rather the finite element method because it can be more easily applied to complex shapes. The finite element technique divides the solution domain into simply shaped regions (or elements) that are typically triangular or quadrilateral for two-dimensional (2-D) problems. Rather than relying directly on energy balances, as in the finite difference method, the finite element method assumes a computationally simple functional form (e.g., a plane or a polynomial) to describe the temperature distribution within each element; the coefficients of the solution are selected so that it most closely satisfies the underlying governing differential equation. The total solution links (or assembles) the individual solutions over each element in a manner that ensures continuity at the boundaries. The finite element technique applied to conduction heat transfer problems is discussed in Myers (1987) and Moeveni (2003) and presented in more detail in Section 2.7.2.

In this section, the program FEHT (Finite Element Heat Transfer) is presented. FEHT uses the calculation technique that is presented by Myers (1987) in order to solve 2-D conduction problems; additional features have been added to automate the specification of the problem and the calculations. It is not necessary to understand the finite element technique in order to use FEHT. A version of FEHT that is limited to 1000 nodes can be downloaded from www.cambridge.org/nellisandklein. In order to become familiar with FEHT, it is suggested that the reader stop and go through the tutorial provided in Appendix A.4, which can be downloaded from www.cambridge.org/nellisandklein.

There are many commercially available finite element software packages that are more powerful than FEHT. Widely used programs such ANSYS, COSMOS, and COMSOL can handle 3-D problems and provide the user with more control of the specification of the geometry (usually allowing the user to import geometry from solid modeling programs). These programs will automatically generate and refine the mesh, and they provide methods for examining and manipulating the solution. Some The example in this section illustrates the proper use of FEHT (coupled with EES) to solve heat transfer problems. The steps needed to solve steady-state heat transfer problems using any finite element software package are similar and include specifying (1) the geometry, (2) the material properties, (3) the boundary conditions, and (4) the mesh. The solution should always be verified for grid convergence, compared to your physical intuition, and if possible checked against an analytical solution.

EXAMPLE 2.7-1: Measurement of High Heat Flux Heat Transfer Coefficient

Figure 1 illustrates a device that is used to measure the heat transfer coefficient for a spray cooling system that is designed to remove high heat flux (on the order of 30 W/cm^2) with a small temperature difference. Fluid (often a dielectric fluid in systems that are designed for electronics cooling) is sprayed from one or more nozzles onto the heated surface where it forms a thin film and therefore provides a high heat transfer coefficient (on the order of $\overline{h}$ = 1 W/cm^2-K). The measurement device is a th = 1.0 cm thick plate of copper with conductivity k_{Cu} = 300 W/m-K. In order to obtain an average heat flux of 30 W/cm^2 over the top surface, a uniform heat flux of $\dot{q}''$ =5 W/cm^2 is applied to the bottom surface by electrical heaters. The top surface is cooled by the spray cooling system with fluid at T_∞ = 45ºC and an unknown average heat transfer coefficient ($\overline{h}$) that must be measured.

Three thermistors are embedded in the neck of the device leading to the top surface. The thermistors are placed along the center line and spaced 0.5 cm apart. The temperatures measured by the three thermistors are plotted as a function of position x (the distance along the axis of the device). The temperature gradient associated with these measurements is used to determine the heat flux delivered to the spray-cooled surface ($\dot{q}''_m$) and extrapolated to find the surface temperature of the sprayed surface ($T_{s,m}$). The measured heat transfer coefficient is determined based on these measurements according to:

$$\overline{h}_m = \frac{\dot{q}''_m}{\left(T_{s,m} - T_f\right)} \tag{1}$$

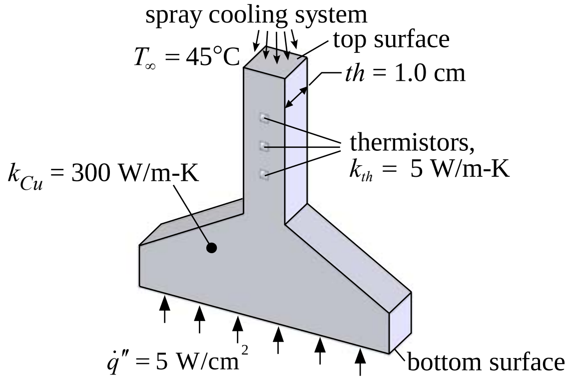

Figure 1: Measurement device for spray cooling heat transfer coefficient.

All surfaces of the plate, except for the top and bottom, are well-insulated and can be considered to be adiabatic. The problem is symmetric about the center-line of the device and therefore a half-symmetry, 2-D model can be used to simulate the measurement system, as shown in Figure 2. (Note that the key points that define the geometry are indicated in Figure 2, with units indicated in m.) A uniform heat transfer coefficient of $\overline{h}$ = 1 W/cm^2-K is applied to the top surface in the model. The thermistors are relatively large and can be modeled as having square cross-section, 2.0 mm on each side. The conductivity of the thermistors is k_{th} = 5 W/m-K. Because the thermistor conductivity is much lower than that of the surrounding copper, the presence of the thermistors distorts the local temperature distribution and induces some measurement error.

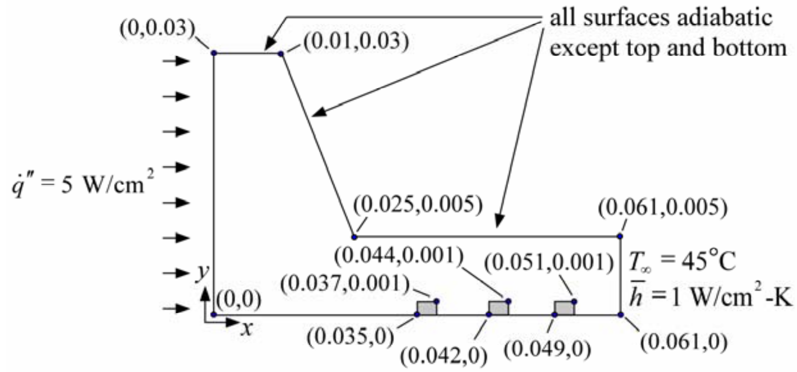

Figure 2: Two-dimensional representation of the measurement device (positions of key points are indicated in units of m)

a.) Prepare a 2-D simulation of the measurement device using FEHT.

Due to the insulation, the plate does not experience heat transfer from the front or back surfaces and therefore the problem is not an extended surface problem, but rather a simple 2-D conduction problem that can be solved by selecting Heat Transfer under the Subject menu. The geometry is most easily specified by setting up a grid using the Scale and Size option under the Setup menu. An appropriate scale is that each 1 cm square of the screen represents a 0.005 m square of the device; the origin is set within the screen.

The device is drawn, approximately, using the Outline option from the Draw menu. The nodes are subsequently positioned exactly by double-clicking on each node and entering their exact coordinates, as indicated in Figure 2. The thermistors are drawn using the same technique. The result is shown in Figure 3.

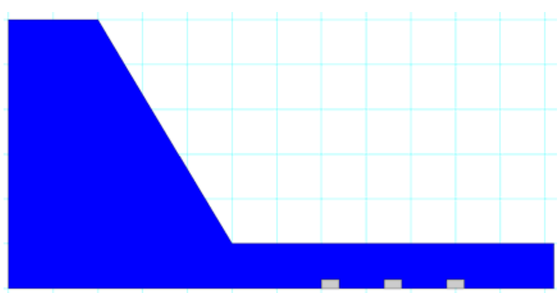

Figure 3: Problem geometry.

The material properties for the copper can be specified by clicking within the device (but not within the thermistors) and selecting Material Properties under Specify. Select Copper from the list of materials and specify the conductivity so that it matches the problem statement (as shown in Figure 4; note that the density and specific heat do not matter as this is not a transient problem).

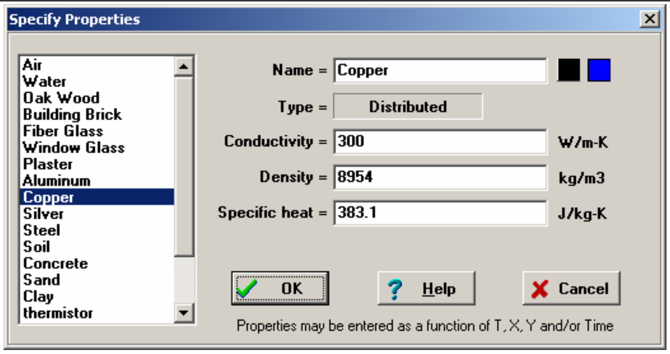

Figure 4: Specification of material properties for copper.

The thermistor conductivity must be specified; click on each of the 3 thermistors while holding the shift key and then select Material Properties from the Specify menu. There is no thermistor material to select from the list of materials; therefore, select "not specified" from the list and name the material "thermistor". Set the conductivity of the material to 5 W/m-K.

The boundary conditions must be set. The centerline and outer surfaces are adiabatic; these boundaries can be selected by holding down the shift key and clicking on each of the boundaries in turn. Select boundary conditions from the Specify menu and choose a zero heat flux. Boundaries can also be selected by holding the left mouse button down and "dragging" a selection box around the desired boundaries. The heat flux boundary condition at the lower surface is a specified heat flux (note that the heat flux must be converted into SI units). The convection boundary at the top surface must also be specified.

Remove the grid and the patterns by selecting Hide Pattern and Hide Grid from the View menu. Draw a mesh in the copper and thermistor using the Element Lines selection from the Draw menu. A very crude is suggested as it can be refined easily. Note that FEHT requires every node to be connected with triangular elements. Select Check from the Run menu to ensure that the mesh is complete and all boundary conditions have been specified. Then select Calculate from the Run menu. The mesh should be refined (select Reduce Mesh from the Draw menu) while some characteristic of the solution is monitored in order to ensure that the mesh is sufficient. It is possible to selectively refine the mesh within a specified material (e.g., the thermistors) by selecting that material and then selecting Reduce Mesh. The locations of the thermistors is the most critical region for this problem so it is advisable to refine the mesh in this region; a refined mesh is illustrated in Figure 5.

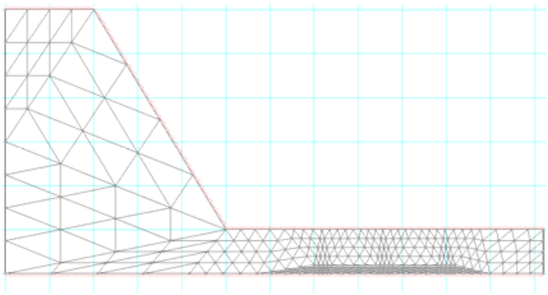

Figure 5: A refined mesh.

The problem is solved by selecting Calculate from the Run menu and the temperature distribution can be examined by selecting Temperature Contours from the View menu (Figure 6).

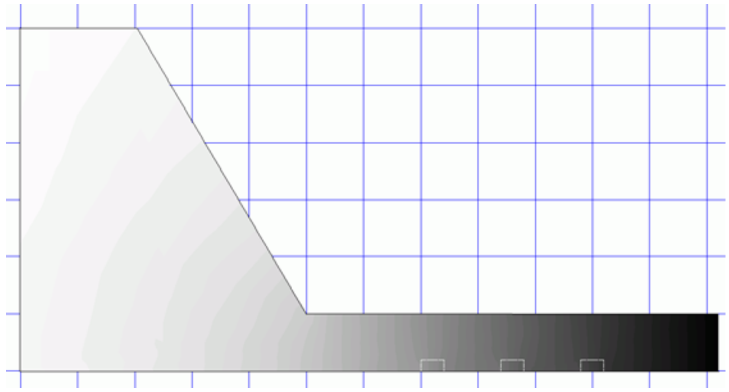

Figure 6: Contour plot of the temperature distribution.

In order to manipulate the temperature distribution more easily, it can be exported to EES. Select Tabular Output from the View menu in order to display a table that contains all of the nodal locations and the associated temperature. Click on Select All and then select "Save as" to bring up a dialog. Save the table as an EES Lookup Table in a location of your choice. Start EES and select Open Lookup Table from the Tables menu. Navigate to the previously saved file and select it in order to open the tabular data in EES. A lookup table in EES will be generated (Figure 7).

Lookup Table

EXAMPLE 2p7dash1

	1 X [m]	2 Y [m]	3 T [°C]	4 Node Balance [W/m]
Row 1	0	0	122.1	93.75
Row 2	0	0.03	123.7	93.75
Row 3	0.01	0.03	122.6	-7.105E-15
Row 4	0.025	0.005	114.2	-1.688E-14
Row 5	0.061	0.005	75	-187.5
Row 6	0.061	0	75	-187.5
Row 7	0.035	0	104	6.439E-15
Row 8	0.035	0.001	103.6	-2.472E-14

Figure 7: Lookup table containing the solution

The temperature at an arbitrary position can be obtained based on the lookup table using EES' Interpolate2D function. The Interpolate2D function returns an interpolated value from tabular data; the function requires six arguments:

Z = INTERPOLATE2D('Table Name',X,Y,Z,X=value1,Y=value2)

where 'Table Name' is the name of an existing lookup table (or a lookup table that is stored on the disk). The arguments X and Y must correspond to columns in that table and are the independent variables while Z, the dependent variable, must also correspond to a column in the table. The last two arguments set the values of the two independent variables. Note that the column names and units have been imported with the data and therefore all variables should have their units set and the problem should be checked for unit consistency.

To evaluate the temperature at $x = y = 0$:

```
"EXAMPLE 2.7-1: Measurement of a High Heat Flux Heat Transfer Coefficient"
$UnitSystem SI MASS RAD PA K J
$Tabstops 0.2 0.4 0.6 3.5 in

x=0 [m]                                         "x-position"
y=0 [m]                                         "y-position"
T=Interpolate2D('EXAMPLE 2p7dash1',X,Y,T,X=x,Y=y)  "2-D interpolation from table of nodal data"
```

which provides the temperature of 122.1°C. By default, the Interpolate2D function uses a bi-quadratic interpolation with the 16 nearest data points; an optional 7th argument, N, can be used to vary the interpolation algorithm. If N is positive then a multi-quadric radial basis function is used and if it is negative then a bi-quadratic method is used. The absolute value of the argument is the maximum number of data points to use in the interpolation. You can adjust the value of this parameter and see if it makes any difference in the interpolated value.

Using the Interpolate2D command, it is possible to plot the temperature as a function of x at $y = 0$ (i.e., along the centerline of the device); a parametric table is created that includes the variables x and T. The x column of the table is altered so that it runs from 0 to 0.061 m. A plot is generated based on this table and the result is shown in Figure 8.

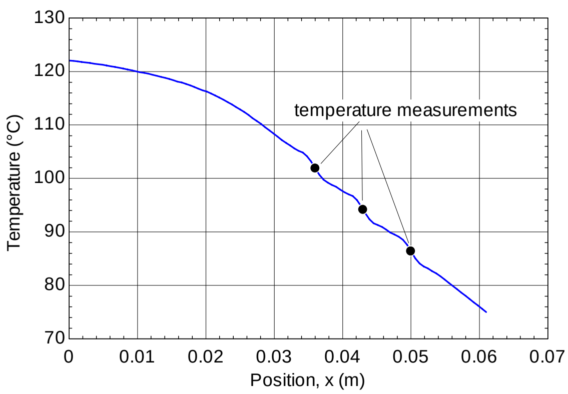

Figure 8: Temperature distribution along the center line of the device.

The thermistors are mounted at y = 3.6 cm, 4.3 cm, and 5.0 cm; the temperatures at these locations correspond to the measurements that are used to infer the heat flux and surface temperature which are, in turn, used to evaluate the heat transfer coefficient. These temperatures are obtained in another parametric table and overlaid onto Figure 8.

The temperature measurements are used to obtain a best-fit line; this is accomplished by selecting Curve Fit from the Plots menu to bring up the Curve Fit Plotted Data dialog window (Figure 9). Select the 2nd data set (that corresponds to the three temperature measurements) and fit a polynomial of order 1 to the data. EES will determine the best fit values of the coefficients a_0 and a_1 for the line:

$$T_m = a_0 + a_1\,x$$

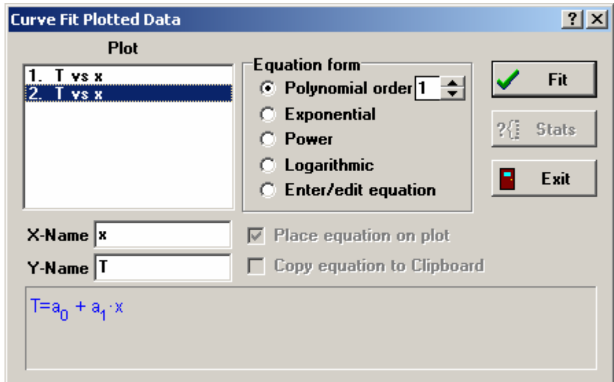

Figure 9: Curve Fit Plotted Data dialog window

The best fit values are a_0 = 141.82°C and a_1 = -1108.77°C/m. Select Copy Equation to Clipboard and paste the solution into the Equation Window. The surface temperature is obtained by extrapolating to the top surface of the device, x_s= 0.061 m.

$$T_{s,m} = a_0 + a_1\,x_s$$

```
a_0=141.820035 [C]                         "coefficients of curve fit"
a_1=- 1108.76793 [C/m]
x_s=0.061 [m]                              "position of the surface"
T_s_m=a_0+a_1*x_s                          "extrapolated surface temperature"
```

The measured heat flux is obtained according to:

$$\dot{q}''_m = -k_{Cu}\frac{dT_m}{dx} = -k_{Cu}\,a_1$$

```
k_Cu=300 [W/m-K]                           "copper conductivity"
q``_m=-k_Cu*a_1                            "inferred heat flux"
```

The measured heat transfer coefficient ($\overline{h}_m$) is obtained according to:

$$\overline{h}_m = \frac{\dot{q}''_m}{\left(T_{s,m} - T_\infty\right)}$$

```
T_infinity=45 [C]                          "fluid temperature"
h_bar_m=q``_m/(T_s_m-T_infinity)            "measured heat transfer coefficient"
```

Running the example indicates a measured heat transfer coefficient $\overline{h}_m$ =11400 W/m^2-K or 1.14 W/cm^2-K which is 14% in error relative to the value specified in the finite element analysis (1.0 W/cm^2-K). The error is related both to the temperature non-uniformity associated with the contraction (manifested as the initial, non-linearity in the temperature distribution near y = 0 in

Figure 8) as well as the temperature fluctuations associated with the presence of the thermistors (manifested as the ripples in the temperature distribution in Figure 8).