Appendix A.4: Introduction to FEHT

A.4.1 Introduction

FEHT (pronounced 'feet') is an acronym for Finite Element Heat Transfer. The basic function provided by FEHT is the numerical solution using finite element of 2-D steady state and transient heat transfer problems. FEHT is intuitive to use and therefore can be learned very quickly by students and researchers.

A version of FEHT that allows up to 1000 nodes is provided on www.cambridge.org/nellisandklein. A commercial or professional version of FEHT can be obtained from:

F-Chart Software
http://fchart.com
email: info@fChart.com

A.4.2 Using FEHT

A.4.2.1 Introduction

FEHT provides three essential functions: Problem Definition, Calculations, and Output. The Problem Definition commands provide a drawing environment in which the mouse is used to draw the outlines of the materials with straight lines; a variety of drawing aids are provided to facilitate this process. Triangular elements of arbitrary size needed in the finite-element analysis are specified by the user by clicking the mouse button on the endpoints of the lines. The program monitors the discretization process to ensure that lines do not cross. Once a coarse triangular mesh is prepared manually, the program can automatically reduce the mesh size. The Problem Definition is completed by specifying the boundary and (for transient problems) the initial conditions. These specifications are made by double-clicking the line, node, or material with a mouse click (causing it to flash) and then selecting the desired specification from a pull-down menu.

Calculations are initiated from a pull-down menu. The program first checks to see that all materials are properly discretized and that the properties, boundaries, and initial conditions are specified. Any error detected during this process is marked and described in a separate window at the top of the screen. For transient problems, the computational method (Euler or Crank-Nicolson) and the start time, stop time and time step are selected from a dialog box; if no errors are detected, then the calculations are initiated.

A variety of output capabilities are provided. For steady-state problems, the potentials (temperature, voltages, magnetic potential, streamlines, or pressures) within the material may be shown at the nodal positions or in one of several types of contour plots. The potential (e.g., temperature) at the cursor position is displayed when the mouse button is depressed. The potential gradients (e.g., temperature gradient, current density, electrical or magnetic flux density) within the materials can be displayed using arrows that point in the direction of the gradient where the shaft length is proportional to the magnitude of the gradient. The flow of heat, charge, current, or magnetic flux across any element line may be determined by clicking the mouse button while the cursor is on the line. For transient heat transfer problems, the temperatures of selected nodes may be displayed in a temperature versus time plot. Heat flow can be plotted as a function of time. The contours and/or temperature gradients for each time step may be shown in sequence providing a 'movie' depicting the changes with time.

Section A.4.2.2 illustrates the solution of a simple heat transfer problem using FEHT from start to finish. Reference information is available from within the program by pressing the F1 key which brings up the online Help program. Section 2.7.2 (Extended Section E6, available on www.cambridge.org/nellisandklein) provides a summary of the finite-element method that is used by FEHT.

A.4.2.2 Getting Started

FEHT will run under the Microsoft Windows Operating systems. The installation program will create a menu item in the Programs section of the Start menu. Detailed help is available at any point in FEHT. Pressing the F1 key will bring up a Help window relating to the foremost window. Clicking the Contents button will present the Help index. Clicking on an underlined word (shown in green on color monitors) will provide help relating to that subject.

There are several example problems with completed problem definitions stored in the Examples subdirectory on the disk which can be accessed using the **Open** command in the **File** menu. These example problems illustrate the necessary triangular discretization and allow a review of the output commands after the **Calculate** command in the **Run** menu has been issued.

Commands are distributed among nine pull-down menus; a brief summary of their functions follows:

- The **File** menu provides commands for loading and saving work files, printing, and copying information to and from the clipboard.
- The **Subject** menu allows the problem discipline to be selected from the areas of heat transfer, electrical currents, electrostatics, scalar magnetostatics, bio-heat transfer, potential flow and porous media flow .
- The **Setup** menu commands allow specification of the unit system, size, scale, coordinate system (Cartesian or cylindrical), and problem type (steady-state or transient).
- The **Draw** menu contains the commands to outline a material, manually or automatically construct element lines, delete or group selected items, reposition nodes, and add text.
- The **Display** menu contains a variety of commands which affect the screen display including a Zoom command to enlarge a selected part of the screen.
- The **Specify** menu allows material properties, internal generation, boundary conditions and initial conditions to be specified.
- The **Run** menu contains commands to check a problem definition and to initiate or continue the calculations.
- The **View** menu provides the means to making one of the nine FEHT windows active.
- The **Examples** menu provides convenient access to a number of example problems developed for use with this text. Examples can also be opened with the **Open** command in the **File** menu.
- The **Help** menu provides access to the on-line help.
- To select a command, place the cursor on the desired menu title, press the mouse button, and while holding the button down, slide the cursor to the command you wish to execute;

then release the mouse button. Menus (or commands within a menu) which are not presently accessible are dimmed, as, for example, **Specify** in the menu bar shown above. Dimmed items can not be selected.

Online Help for any command in the pull-down menus can be obtained by first pressing and holding the mouse button down with the cursor positioned on the menu item and then pressing the F1 key.

An example problem is a steady-state heat transfer analysis of a furnace wall shown in Figure A.4-1. The air within the furnace is maintained at 560°C. The outside surface of the furnace is exposed to air at 30°C. The furnace wall is made of brick. The problem is to determine the temperature distribution in the brick wall and the total rate of heat transfer through the wall.

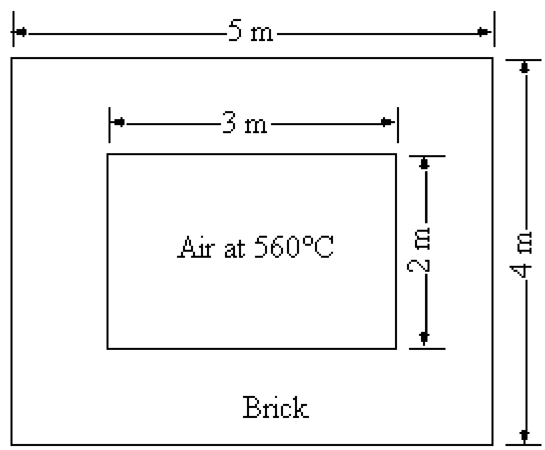

Figure A.4-1: **Furnace wall.**

By default, FEHT is configured for steady-state heat transfer problems in Cartesian coordinates. These characteristics apply to this practice problem and do not need to be changed. It is usually best to set the unit system, scale, and grid spacing at the start of a problem although they can be changed at any time. Pull down the Setup menu and select the **Scale and Size** command which will bring up a dialog window (Figure A.4-2) in which the scale attributes can be entered.

The small circles in the Scale and Size dialog window shown below are called radio buttons. Radio buttons control the unit system for heat transfer problems. To change the unit system, move the mouse to position the cursor on the appropriate button and click the mouse button; as usual, we will select SI units. A reasonable scale for this problem is to have 1 cm on the screen represent 0.25 m. Any item within a box can be changed. The unit for length is, by default, centimeters but it can be set to mm, cm, m, or km by clicking in the units box to the right of the scale value. (In the English system, the length unit can be inches, feet, yards, or even miles.) Click in the scale units box until it displays m for meters. Then enter 0.25 in the scale edit box. Note that double-clicking within any edit box causes the characters to be highlighted (shown in inverse). Typing any character will replace the highlighted field with the entered character. X0 and Y0 designate the location of the origin of the coordinate system on the screen *in screen coordinates*. The default values, X0=0.0, Y0=0.0 correspond to the origin being placed at the lower left of the screen. Gridlines make the drawing easier to prepare. Grid spacing is specified in the same coordinates as for the drawing. Set the grid spacing as shown below. Click the OK button or press the Enter key to set these scale attributes.

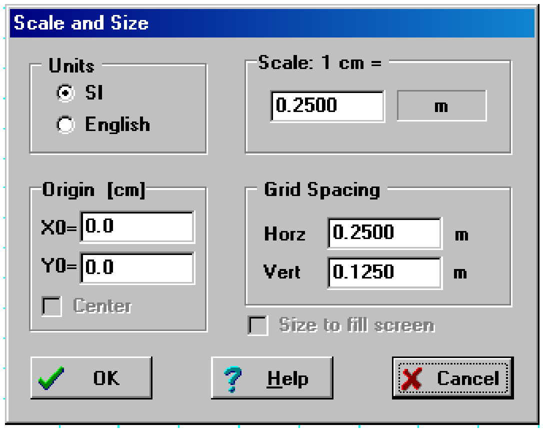

Figure A.4-2: Scale and Size dialog window

The first step is to sketch material outlines. It is easier to prepare a scale drawing with a coordinate grid. Select **Show Grid** from the **Display** menu (if it is not already selected). Select **Outline** from the **Draw** menu. Since this problem is symmetrical, we only need to analyze one quarter of the furnace. Note that the *x* and *y* coordinates of the cursor position for the selected unit system and scale are shown in the small window at the upper left, just below the menu bar. Move the mouse to locate the cursor at position x=0.50 m, y=2.50 m (Note that because the mouse position is represented in discrete pixels, it may not be possible to have the cursor positioned exactly at the desired position. In this case, just place the cursor close to the desired position. It is possible to enter the exact node coordinates with the Boundary Conditions command in the Specify menu.)

Click the mouse to fix a node at the corner. The first node is shown as a small closed circle. Now, position the cursor at *x*=3.00 m, *y*=2.50 m and click the mouse. Hold the shift key down to provide a drawing aid for horizontal, vertical, or 45° lines. Click on the remaining corners at *x*=3.00 m, *y*=0.50 m; *x*=2.00 m, *y*=0.50 m; *x*=2.00 m, *y*=1.50 m; and *x*=0.50 m, *y*=1.50 m. Now click on the first corner. The outline must begin and end on the same node without crossing any existing lines. At this point, the outline will flash, indicating that the outlining process is completed and the material within the flashing boundary is selected. The outline number and name are shown in the center information window below the menu bar. The area enclosed by the outline is shown in the right information window. The screen should appear as in Figure A.4-3.

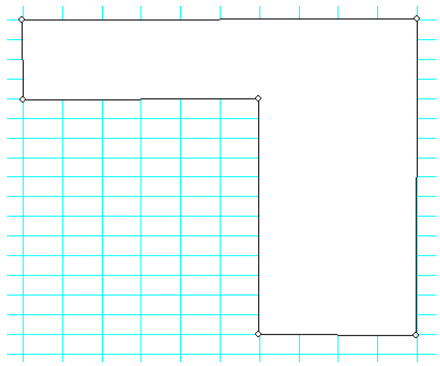

Figure A.4-3: **Quarter symmetry model of furnace.**

A material must be selected (flashing) in order to specify its properties. A material can be selected by clicking the mouse anywhere within it outline; it is automatically selected just after it has been drawn. Select **Material Properties** from the **Specify** menu. A property dialog box will appear with default property names listed on the left. Choose the material to be brick by clicking on Building Brick in the list on the left. You can choose the pattern which will be used to identify the material by holding the mouse button down while the cursor is in the pattern box. A pop-up palette will appear with the possible patterns. The color of the pattern can also be selected in the same manner by clicking in the Color box. The thermal properties of brick are displayed. These properties can be changed to other values. They may also be entered as a function of temperature and position. Leave the brick properties at their default values. The properties dialog box should now appear as shown in Figure A.4-4.

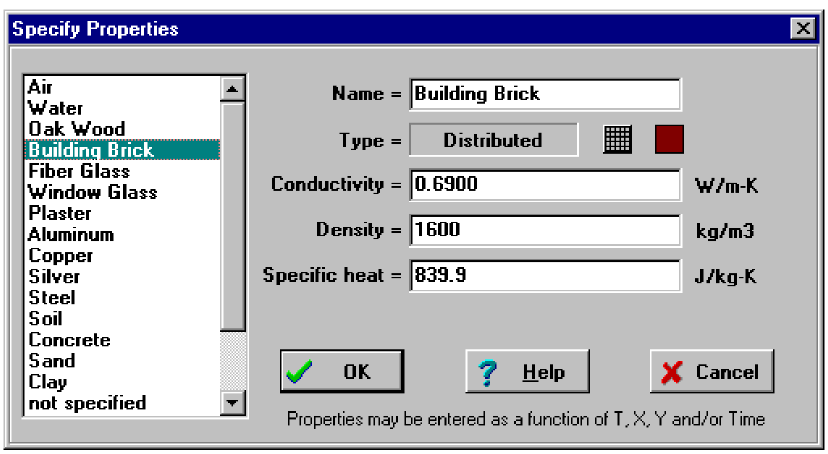

Figure A.4-4: **Specify Properties dialog window.**

Click the OK button and the screen will be redrawn with the pattern you have chosen identifying the material.

Next we will set the boundary conditions. The vertical line at x=0.50 m and the horizontal line at y=0.50 m are lines of symmetry and therefore there is zero heat flow across these lines. Move the cursor to a point near the center of one of these lines and click the button. The line should now be flashing. Move the cursor to the center of the other line and click. Both lines should now be flashing. Once a boundary is selected (flashing), the **Boundary Conditions** menu item in the **Specify** menu becomes accessible. Select this menu item to bring up the Boundary

Conditions dialog window (Figure A.4-5). Enter 0 (for adiabatic conditions) in the Heat Flux box and click the OK button. A check mark will automatically appear in before the word Heat Flux to confirm that you are setting a heat flux boundary condition.

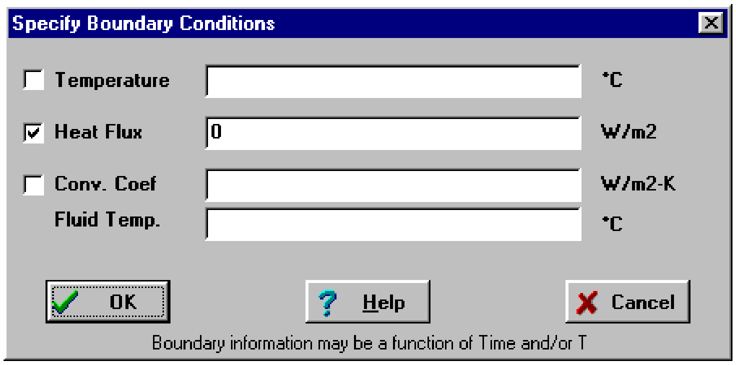

Figure A.4-5: **Specify Boundary Conditions dialog window.**

The two boundary lines are now shown with bold lines and cross-hatching to indicate that the boundary conditions have been specified and that the boundary condition is adiabatic.

The inside and outside walls of the brick are convective boundaries. Click the mouse at a point near the center of each outside line causing them to flash. Again, select **Boundary Conditions** from the **Specify** menu and enter a convection coefficient of 5 W/m^2-K and a fluid temperature of 30°C. The convection boundary information for the inside furnace walls is entered in the same manner. Select both boundaries and again issue the **Boundary Conditions** command. Enter the convection coefficient of 10 W/m^2-K and a fluid temperature of 560°C.

At this point, you may first wish to exactly locate the node positions if you were unable to do so while drawing with the mouse. Click on the node at the upper left of the drawing and then select the **Boundary Conditions** command in the **Specify** menu. (Note that, as a short cut, you can accomplish the same result by double-clicking the mouse on the node.) The Specify Node Temperature dialog window will appear with edit boxes for the node temperature and for the X and Y coordinates of the node. In this case, we wish to alter the coordinates, not the node temperature. Enter the coordinates x=0.50 and y=2.50 (Figure A.4-6). FEHT will not let the node move to a position which causes existing lines to cross. Note the FEHT displays the coordinates in exponential notation. The E means 'time ten to the power of'. You may wish to repeat this process for all of the other nodes so that they are exactly positioned at the proper locations.

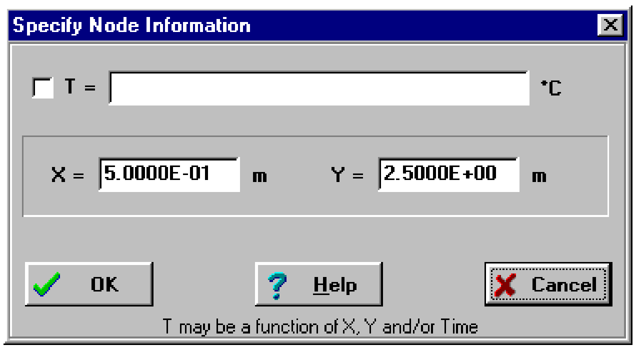

Figure A.4-6: **Specify Node Information dialog window.**

To complete the problem definition, it is necessary to discretize the brick material into triangular elements. The triangular elements are formed by placing element lines within the material. The positions of these lines and the number of triangular elements are determined by the user. One purpose of FEHT is, in fact, to make it easy to explore the effect of the discretization.

It is easier to construct element lines when the brick pattern is hidden; the pattern can be removed by selecting **Hide Patterns** from the **Display** menu. The grid lines are no longer needed; select **Hide Grid** from the **Display** menu. Select **Element Lines** from the **Draw** menu. Move the cursor to the upper left node and click the button. Now, click at point at x=1 m, y=2.00 m. (Note that is it not important to exactly locate the nodes. Anywhere near this point is fine.) A node will be created at this point and an element line will be drawn between the two nodes. Continue this process of constructing element lines until the screen appears as shown in Figure A.4-7.

You have considerable freedom in selecting your mesh and your mesh does not need match Figure A.4-7; however, the following rules apply to manual element line construction:

- The first end of the line must be on an existing line or node. A new node will form at this point if one is not already there.
- Element lines can not cross existing lines.

Clicking in the area surrounding the drawing or pressing the Esc key cancels the **Element Lines** command in the **Draw** menu. It is only necessary to construct a coarse mesh since the **Reduce Mesh** command in the **Draw** menu can refine a mesh once it has been constructed.

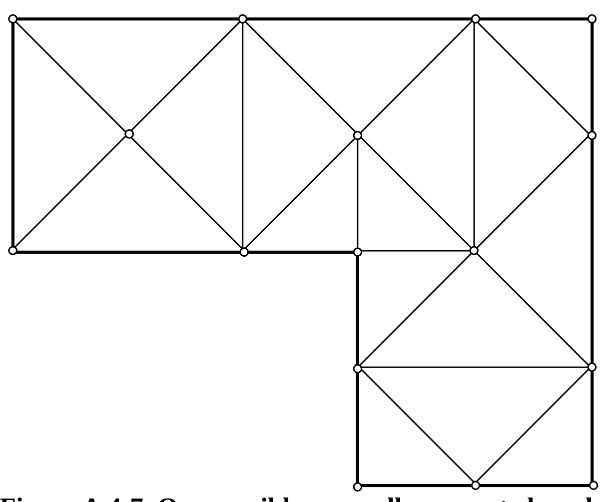

Figure A.4-7: **One possible manually generated mesh.**

The problem definition is complete. Select the **Calculate** command from the **Run** menu to initiate the calculations. FEHT will first check the problem definition to ensure that the distributed materials are properly discretized and all properties and boundary conditions are specified. Any errors detected will be listed in the information window at the upper right of the screen, just below the menu bar. This example problem is assumed to be steady-state. Had this been a transient problem (by selecting **Transient** from the **Setup** menu), a dialog box would have appeared in which the start, stop and step times would be entered. If no errors are found, a dialog window will appear indicating that the calculations are in progress (Figure A.4-8). When the calculations have been completed, the dialog box will display the elapsed time and other information.

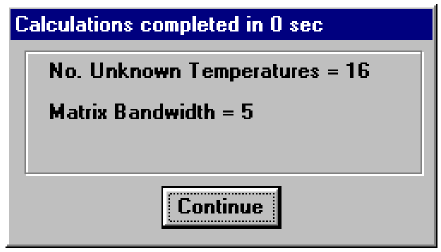

Figure A.4-8: **Calculations completed message.**

Click the Continue button. A number of the output display windows in the **View** menu will now be accessible. The temperatures within the wall can be displayed at each node by selecting **Temperatures** from the **View** menu (Figure A.4-9).

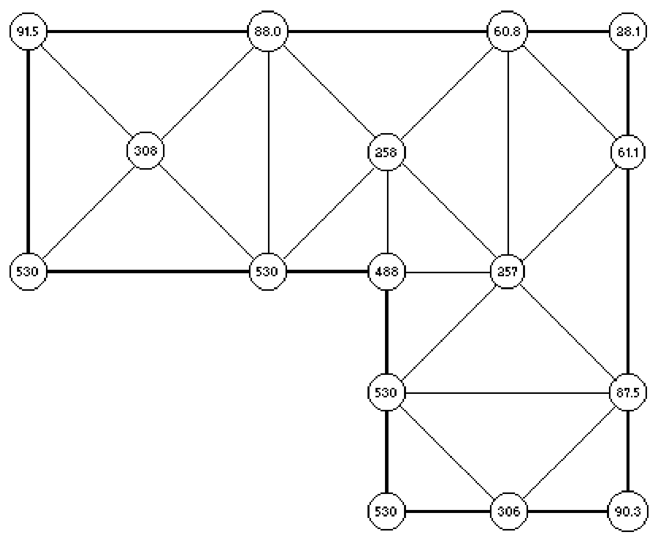

Figure A.4-9: **Temperatures at each node.**

Temperature can also be displayed as a contour plot by selecting **Temperature Contours** which will bring up the dialog window shown in Figure A.4-10.

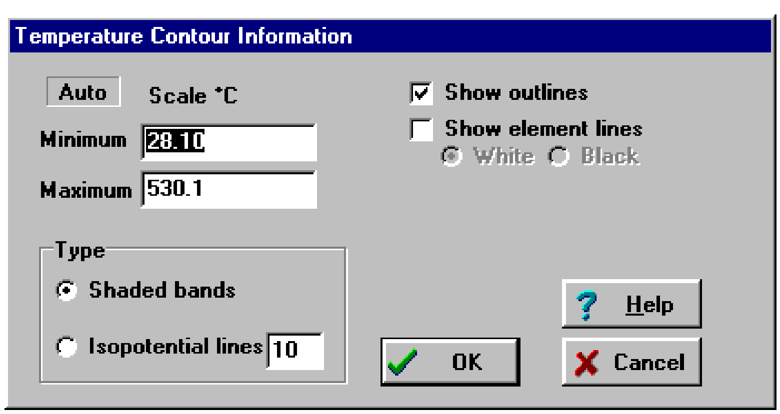

Figure A.4-10: **Temperature Contour Information dialog window.**

Three types of contour plots are available, a banded plot showing gradations of hot to cold in a specified number of sections, a continuous color plot showing temperatures as colors, and a contour plot of lines of constant temperatures. The minimum and maximum values in the contour plot can be entered manually or FEHT will automatically find the limits if you click in the User/Auto box at the upper left. Click the OK button or press the enter key to show the contour plot (Figure A.4-11).

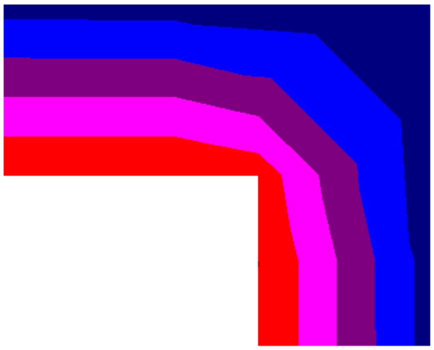

Figure A.4-11: **Contour plot**

In either the temperature or contour plot output, the temperature at the cursor position will be displayed at the upper left of the screen below the menu bar when the mouse button is depressed.

It is possible to determine the total heat flow through the brick wall. Select **Heat Flows** from the **View** menu. The screen will be redrawn with the nodes hidden. Click on any line segment of the inside wall and an arrow will appear indicating the direction of heat flow. The magnitude of the heat flow is shown in the information window at the top of the screen below the menu bar. Clicking on adjacent lines forming the inside boundary in a clockwise or counterclockwise manner will allow the heat flows to be summed. Alternatively you can select multiple lines by pressing the mouse outside of the material and dragging it while holding the mouse down to create a selection rectangle. All lines that lie completely within the selection rectangle will be selected. After selecting all of the lines, you may wish to group them using the **Group** command in the **Draw** menu. Clicking on any one of the lines in a group selects all lines in that group. For one quarter of the problem, the heat flow through the furnace wall is 961.8 W. The total heat flow through all sides of the wall is then 3847.2 W.

Select **Input** from the **View** menu to return to the drawing window. At this point, you may wish to explore. Compare the heat flows on the outside walls of the furnace with that on the inside; they should agree. Try using smaller triangular elements. FEHT will automatically reduce the mesh size for you if you select the **Reduce Mesh Size** command in the **Draw** menu. As the mesh is refined the solution should approach some limit indicating that the numerical model has converged.

This problem was a steady 2-D problem without convection occurring from the face of the material. If **Extended Surfaces** is selected from the **Subject** menu then it becomes possible to specify convection from the faces; the problem remains 2-D provided that the material is thin enough (as determined using a Biot number).