



# **F E H T**

**A Finite Element Analysis Program**

**for the**

**Microsoft Windows  
Operating System**

**F-Chart Software**

**<http://www.fchart.com/>**

**email: [info@fchart.com](mailto:info@fchart.com)**

Box 44042, Madison, WI 53744

Copyright 1995-2016 by S.A. Klein

All rights reserved.

The authors make no guarantee that the program is free from errors or that the results produced with it will be free of errors, and they assume no responsibility or liability for the accuracy of the program or for the results which may come from its use.

FEHT is compiled in DELPHI XE10

V8.001

# *Table of Contents*

<b>Overview</b>	1
<b>Chapter 1: Getting Started</b>	5
Installing FEHT on your computer	5
Starting FEHT	5
Background Information	6
A Practice Work Session	8
<b>Chapter 2: Reference Information</b>	17
FEHT Windows	17
Selecting Objects	18
Entering Variable Properties and Boundary Specifications	19
Working with Menus	20
The <b>File</b> Menu	21
The <b>Subject</b> Menu	23
The <b>Setup</b> Menu	27
The <b>Draw</b> Menu	30
The <b>Display</b> Menu	34
The <b>Specify</b> Menu	37
The <b>Run</b> Menu	43
The <b>View</b> Menu and FEHT Windows	45
The <b>Help</b> Menu	55
The <b>Examples</b> Menu	56
<b>Chapter 3: The Finite-Element Method</b>	59
Guidelines for Choosing a Time Step	67
References	68
<b>Appendix A: Hints for Using FEHT</b>	69
<b>Appendix B: Program Information</b>	71

FEHT is an acronym for Finite Element Heat Transfer. FEHT was originally designed to facilitate the numerical solution of steady-state and transient two-dimensional conduction heat transfer problems. However, the fundamental equations describing conduction heat transfer, bio-heat transfer, potential flow, steady electric currents, electrostatics, and scalar magnetostatics are similar. The current version (Version 8) of FEHT has been designed to solve problems in all of these disciplines. It is designed for use on Windows operating systems XP, 7, 8, and 10.

Most practical conduction heat transfer problems can not be solved analytically. As a result, numerical solutions provide the only feasible way in which these problems can be solved. There are two common approaches: finite-difference and finite-element methods. In both approaches, the governing partial differential conduction equation subject to specified boundary (and for transient problems, initial) conditions is transformed into a system of ordinary differential equations (for transient problems) or algebraic equations (for steady-state problems) that are solved to yield an approximate solution for the temperature distribution. In the finite difference method, spatial discretization of the problem using a set of nodal points followed by application of energy balances and rate equations for each of the discrete segments directly results in a system of equations which are solved to obtain the temperature at each nodal point.

In the finite-element method, the partial differential equation is transformed into an integral form. Numerical approximation of the integral results in the system of algebraic or ordinary differential equations. In both the finite-difference and finite-element methods, the accuracy of the solution is improved as the number of nodes used to discretize the region is increased. The finite-element method has been chosen over the finite-difference method primarily because its use of triangular elements greatly simplifies the discrete approximation of non-rectangular geometries.

FEHT provides three essential functions: Problem Definition, Calculations, and Output. The Problem Definition commands provide a drawing environment in which the user draws the outlines of the materials with a series of straight lines that enclose an area. After it is defined, the properties of the material represented by the outline and the boundary and initial (for transient problems) conditions can be defined. These specifications are made by tagging the line, node, or material with a mouse click (causing it to flash) and then selecting the desired specification from a pull-down menu. The Auto Mesh command can then be applied to create a rough triangular mesh. The Reduce Mesh command can be used to reduce the mesh size.

Calculations are initiated from a pull-down menu. The program first checks to see that all materials are properly discretized and the properties, boundary, and initial conditions are specified. Any error detected during the checking 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 start, stop and time step are selected from a dialog box; if no errors are detected, 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 at the cursor position is displayed when the mouse button is depressed. The potential gradients (temperature gradient, current density, electrical or magnetic flux density) within the materials can be displayed by arrows pointing in the direction of the gradient with the shaft length proportional to the gradient magnitude. 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.

The motivation behind the development of FEHT was for instruction. Although undergraduate engineers are exposed to numerical solution methods, they are typically not in a position to solve the more interesting practical problems after completing the course due to a lack of experience. The problems encountered by students in an undergraduate course are typically one-dimensional or two-dimensional with very simple geometry that are used to illustrate the basic concepts. As a result, students do not have the opportunity to learn how to choose appropriate nodal networks as needed for complex geometries and/or boundary conditions, to apply variable nodal spacing, or to ensure smooth transitions at the interface between different materials. These subjects do not receive more attention because such analyses currently involve a prohibitive amount of programming effort and student time.

FEHT offers the advantages of a simple set of intuitive commands with which a novice can quickly learn to use for solving complex two-dimensional problems. FEHT is ideally suited for instruction in electrical engineering fields and mechanical engineering heat transfer and bio-engineering courses and for the practicing engineer faced with the need for solving practical problems.

The remainder of this manual is organized into three chapters and two appendices. Chapter 1, intended for the new user, illustrates the solution of a simple heat transfer problem from start to finish. Chapter 2 provides specific information on the various functions and controls for each of the FEHT windows and menu commands. Much of this reference information is available

from within the program by pressing the F1 key which brings up the online Help program. Chapter 3 provides a brief summary of the finite-element method with references to more complete descriptions.



---

# C H A P T E R 1

---

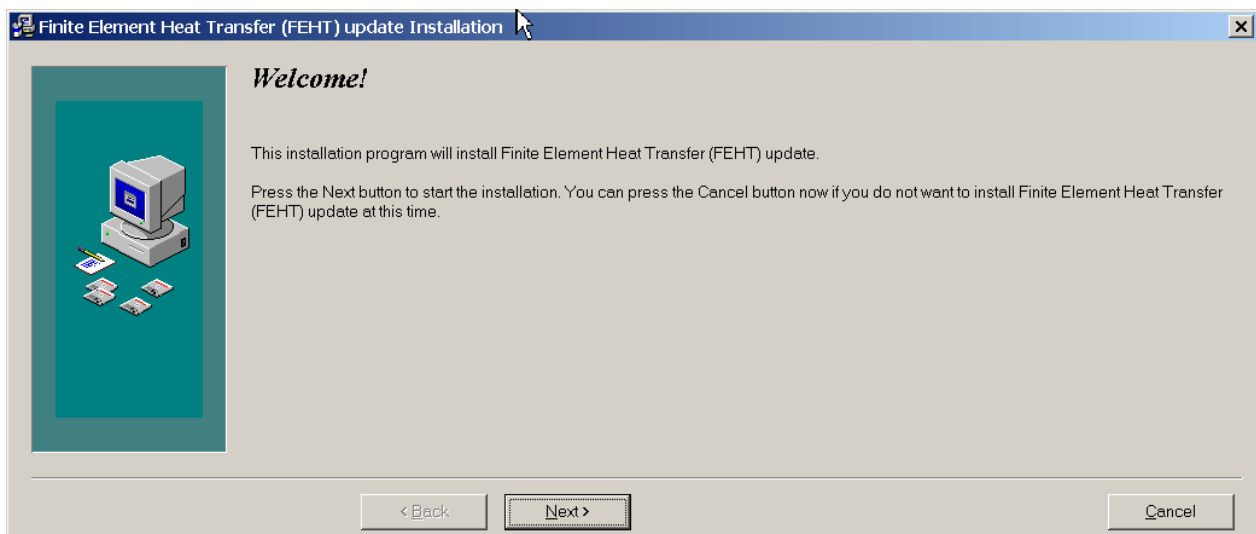
## Getting Started

---

This chapter, intended for the new user, will guide the solution of a simple heat transfer problem from start to finish. After trying the problem outlined in this chapter, you may wish to review the reference information in Chapter 2.

### *Installing FEHT on your Computer*

FEHT is a 32-bit application that will run under all modern Microsoft Windows 32 and 64-bit operating systems including Windows XP, 7, 8, and 10. An installation program named SETUP\_FEHT.exe is provided that must be run from the Windows operating system. Double-click on the program file to start the installation process.



The installation program will provide a series of prompts which will lead you through the complete installation of the FEHT program and its associated files.

### *Starting FEHT*

The installation program will create a menu item in the Programs section of the Start menu. The FEHT program is represented by the icon shown below. Both the program and FEHT Files having a .FET filename extension will be shown with this icon. Double-clicking on this icon will start the program.

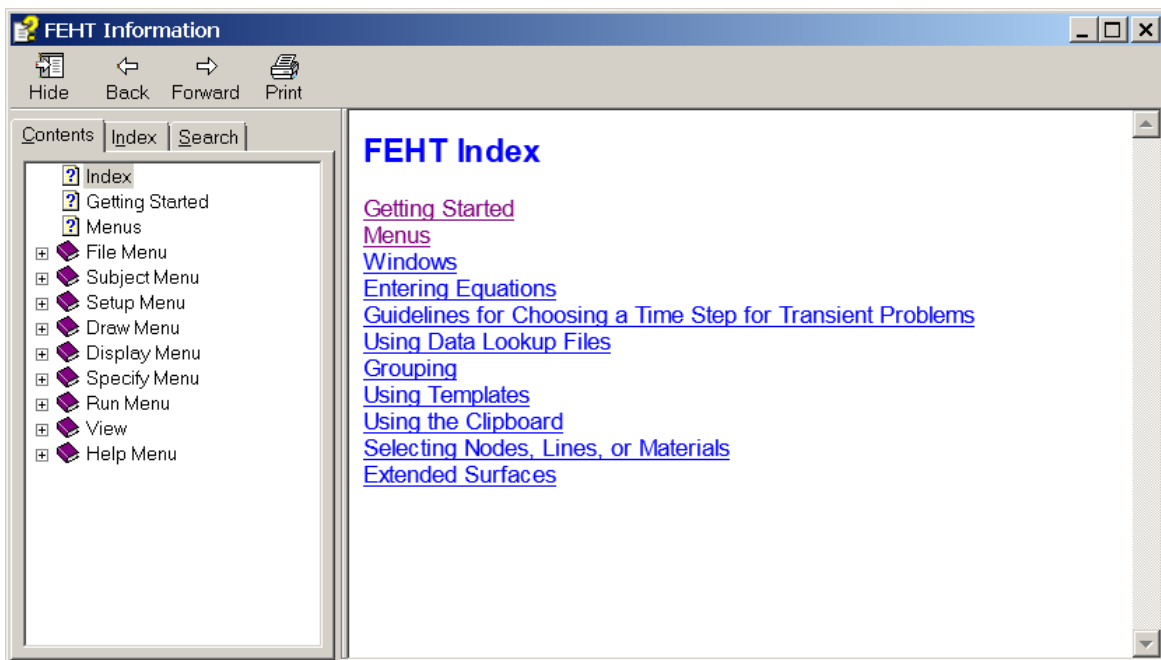




## Background Information

FEHT begins by displaying a dialog window showing the version number of the program and the owner registration number. This information will be needed if you request technical support. Click the OK button to dismiss the dialog window.

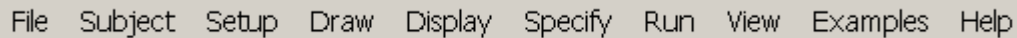
Detailed help is available at any point in FEHT. Pressing the F1 key will bring up a Help window relating to the foremost window. Clicking the Contents button will present the Help index shown below. Clicking on an underlined word will provide help relating to that subject.



There are several example problems with completed problem definitions stored in the Examples subdirectory on the disk that 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. Detailed descriptions of the commands appear in the next chapter.





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 create a mesh, reduce the mesh, delete or group selected items, reposition nodes, and add text.

The **Display** menu contains a variety of commands that 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 function of each window are described in Chapter 2.

The **Examples** menu provides convenient access to a number of example problems. Examples can also be opened with the **Open** command in the **File** menu.

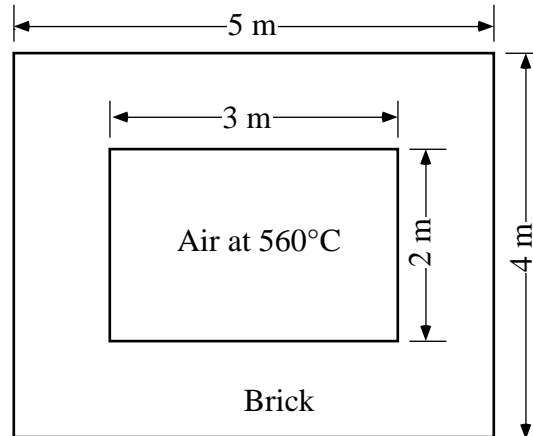
The **Help** menu provides access to the on-line help as well as to the F-Chart Software web site.

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.

**A Practice Work Session**

The example problem is a steady-state heat transfer analysis of a furnace wall shown in the figure below. The air within the furnace is maintained at  $560^{\circ}\text{C}$ . The outside surface of the furnace is exposed to air at  $30^{\circ}\text{C}$ . 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.

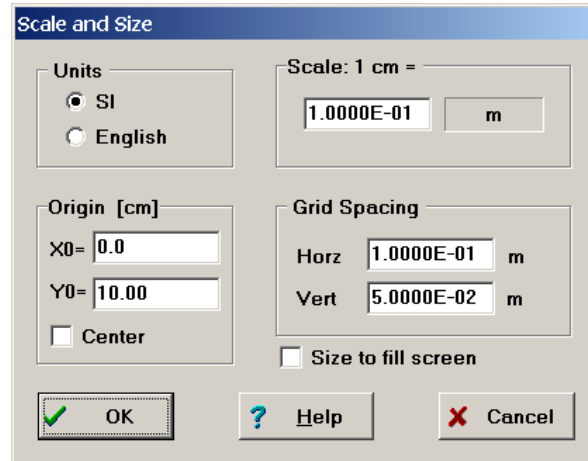


By default, FEHT is configured for steady-state heat transfer problems in Cartesian coordinates. These characteristics apply to this practice problem and do not need to be changed.

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 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. A reasonable scale for this problem is to have 1 cm on the screen represent 0.1 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 miles.) Click in the scale units box until it displays m for meters. Then enter 0.1 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 and origin to suit the size of your monitor and the dimensions of your problem. Click the OK button or press the Enter key to set these scale attributes.



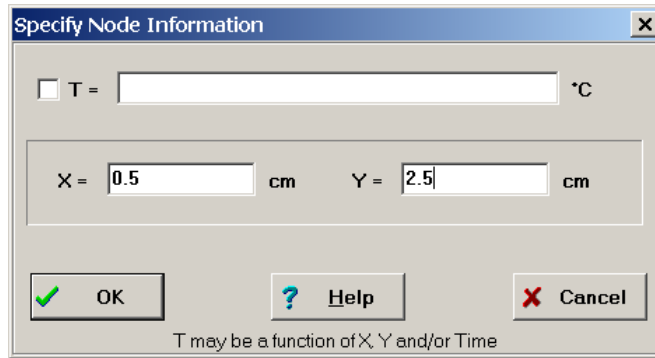
The first step is to sketch material outlines. It is easier to prepare a scale drawing with a coordinate grid. Select **Show Grid** from the **Display** menu. Select **Outline** from the **Draw** menu. Since this problem is symmetrical, we only need to analyze one quarter of the furnace. Note that the 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<sup>1</sup>. 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.

At this point, you may wish to exactly locate the node positions since it unlikely that you were able 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

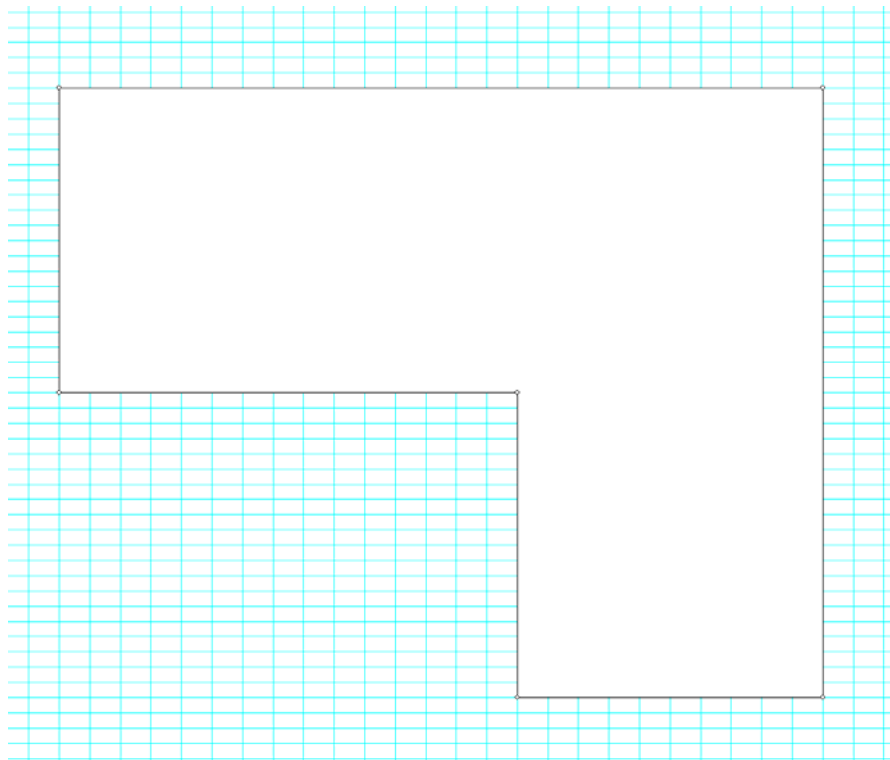
---

<sup>1</sup> Note: It is not necessary to position the mouse cursor at the exact desired position. Just place the cursor close to the desired position. The exact position can be later entered with the Boundary Conditions command in the Specify menu.

coordinates, not the node temperature. Enter the coordinates X=0.50 and Y=2.50 as shown below. FEHT will not let the node move to a position which causes existing lines to cross, so there is some restriction on the coordinates that you enter. Note the FEHT displays the coordinates in exponential notation. The E means ‘times ten to the power of’. Repeat this process for all of the other nodes so that they are exactly positioned at the proper locations. After entering the coordinates of all of the nodes, save the file as Example 1.fet using the **Save** command in the **File** menu.

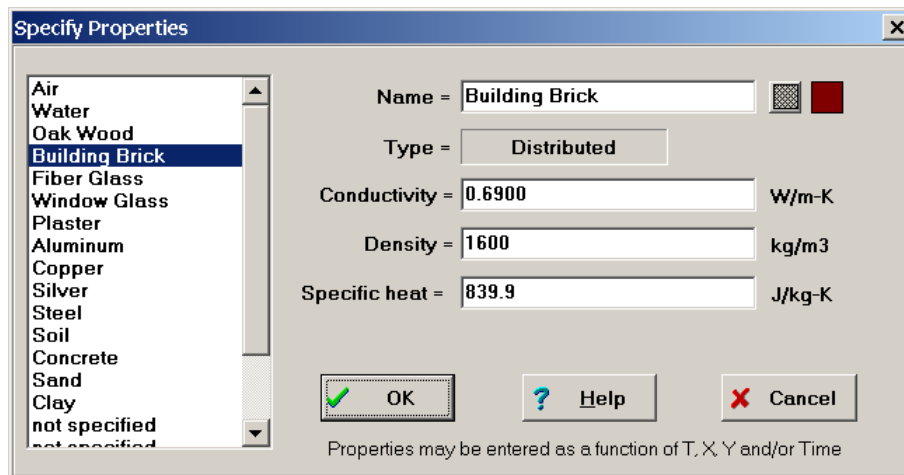


The screen should now look like this.



Click the mouse button anywhere within the outline. This will cause it to flash, indicating that it 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. A material must be selected (flashing) in order to specify its properties. A material can be selected by clicking the mouse anywhere within its 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 as described in Chapter 2. Leave the brick properties at their default values. The properties dialog box should now appear like this.



Click the OK button and the screen will be redrawn with the pattern you have chosen identifying the material. You may wish to hide the pattern so that you can better see the nodes in the outline. You can do this by selecting **Hide Patterns** from the **Display** menu.

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. 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.

The dialog box titled "Specify Boundary Conditions" has a close button (X) in the top right corner. It contains four rows of options, each with a checkbox, a text input field, and a unit label:

- Temperature [ ] °C
- Heat Flux [0] W/m<sup>2</sup>
- Conv. Coef [ ] W/m<sup>2</sup>-K
- Fluid Temp. [ ] °C

At the bottom, there are three buttons: "OK" (with a green checkmark), "Help" (with a question mark), and "Cancel" (with a red X). Below the buttons is the text: "Boundary information may be a function of Time and/or T".

The two boundary lines are now shown with bold lines to indicate that the boundary conditions have been specified.

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. Enter a convection coefficient of 5 W/m<sup>2</sup>-K and a fluid temperature of 30°C. The dialog window will appear as shown. Click OK.

The dialog box titled "Specify Boundary Conditions" has a close button (X) in the top right corner. It contains four rows of options, each with a checkbox, a text input field, and a unit label:

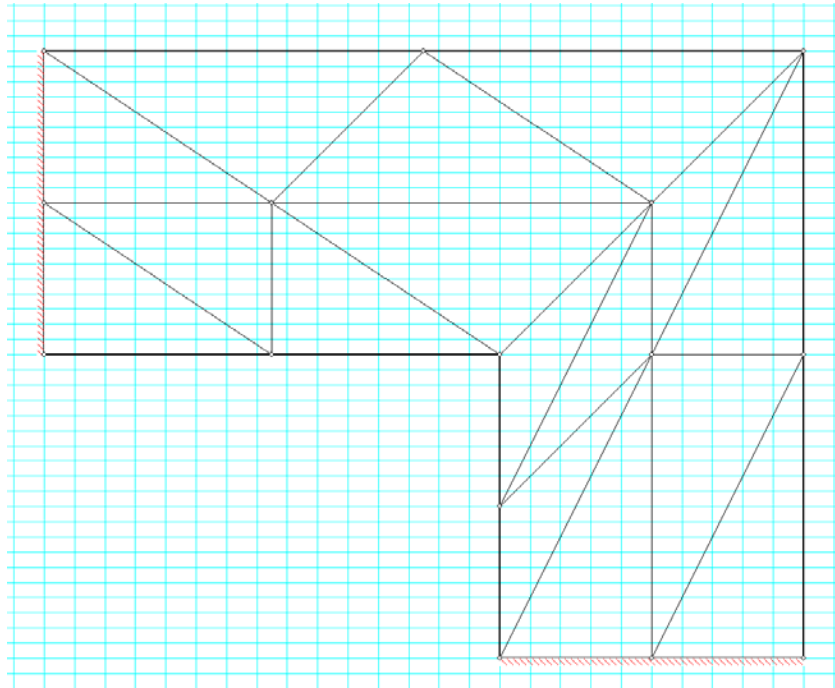
- Temperature [ ] °C
- Heat Flux [ ] W/m<sup>2</sup>
- Conv. Coef [5] W/m<sup>2</sup>-K
- Fluid Temp. [30] °C

At the bottom, there are three buttons: "OK" (with a green checkmark), "Help" (with a question mark), and "Cancel" (with a red X). Below the buttons is the text: "Boundary information may be a function of Time and/or T".

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<sup>2</sup>-K and a fluid temperature of 560°C. Click the OK button. Save the file.

To complete the problem definition, it is necessary to discretize the brick material into triangular elements. This task can be done manually (using the **Element Lines** command in the **Draw** menu) or automatically using the **Auto Mesh** command in the **Draw** menu. Select the **Auto Mesh** command. EES will break the outline into triangular elements as shown.





The **Auto Mesh** command first breaks the outline into a crude mesh and then reduces the mesh once. If you wish to see the crude mesh, select the **Undo** command. Use the **Reduce Mesh** command to return to the display above.

Although there it is usually unnecessary, you can adjust the mesh that has been created using the **Reposition Nodes** command in the **Draw** menu. You can add additional elements manually to the existing mesh (or create the mesh from the start) by selecting the **Element Lines** command from the **Draw** menu. Click on an existing node or line to form one end of the element line. Then click on a node or line that will become the other end of the element line. The following rules apply to manual element line construction.

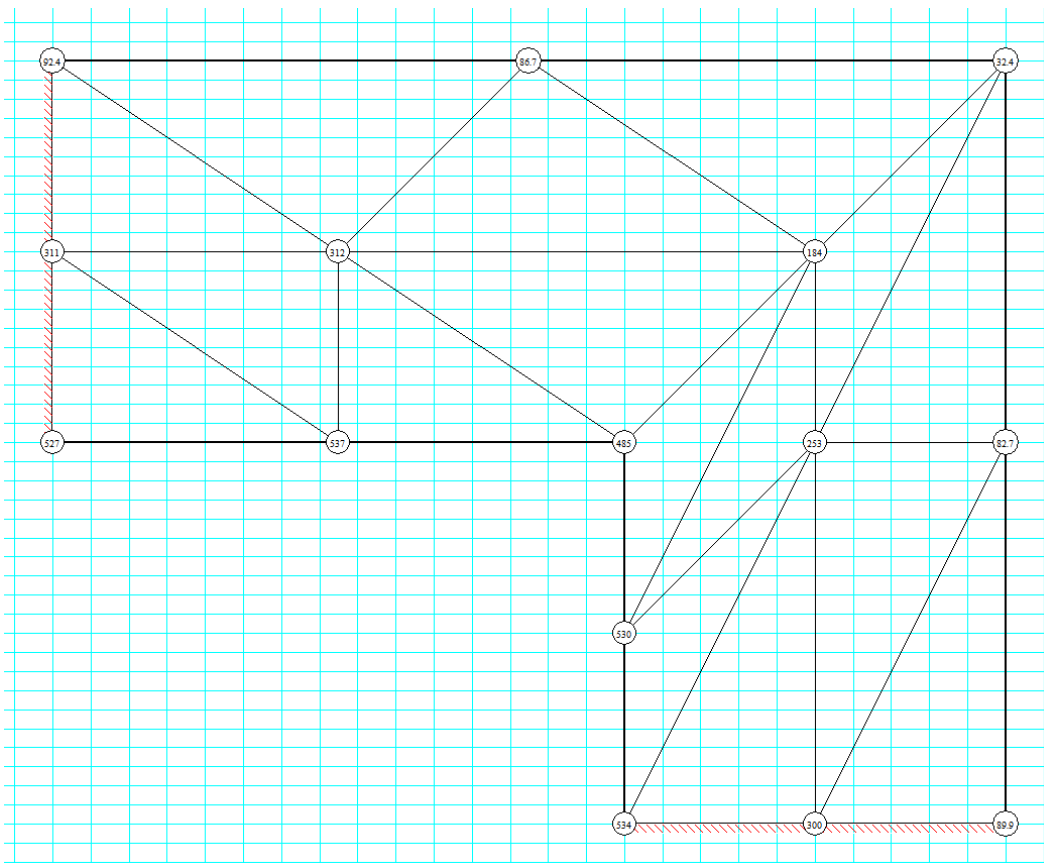
- i. The first end of the line must be on an existing line or node. A new node will form at this point if one is not already there.
- ii. Element lines can not cross existing lines.
- iii. Clicking in the area surrounding the drawing or pressing the Esc key cancels the **Element Lines** command in the **Draw** menu.
- iv. It is not necessary to construct a coarse mesh since the **Reduce Mesh** command in the **Draw** menu can refine a mesh once it has been constructed.

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

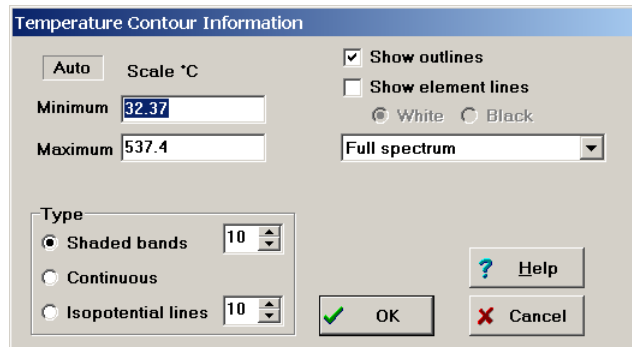
specified. Any errors detected will be listed in the information window at the upper right of the screen, just below the menu bar. This example problem is assumed to be steady-state. Had this been a transient problem (by selecting **Transient** from the **Setup** menu), a dialog box would have appeared 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. When the calculations have been completed, the dialog box will display the elapsed time and other information.



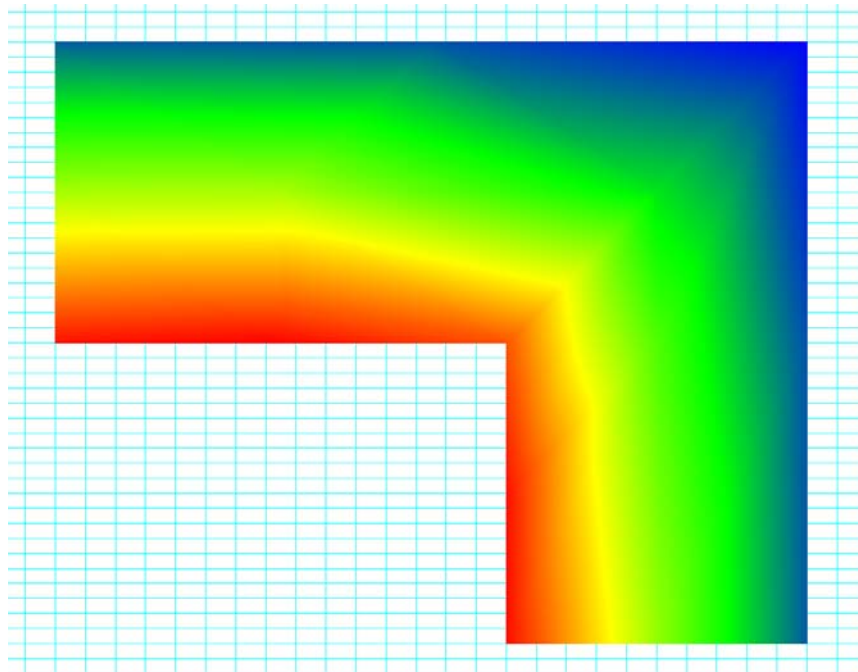
Click the Continue button. A number of the output display windows in the **View** menu will now be accessible. The temperatures within the wall can be displayed at each node by selecting **Temperatures** from the **View** menu. Press and hold the left mouse down and move it within the outline. The temperature at the cursor location will be displayed in the status bar at the upper left of the screen, replacing the cursor coordinates.



Temperature can also be displayed as a contour plot by selecting **Temperature Contours** which will bring up the following dialog window.



Three types of contour plots are available, a banded plot showing gradations of hot to cold in 5 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. The scale is shown in the status bar at the top of the screen.

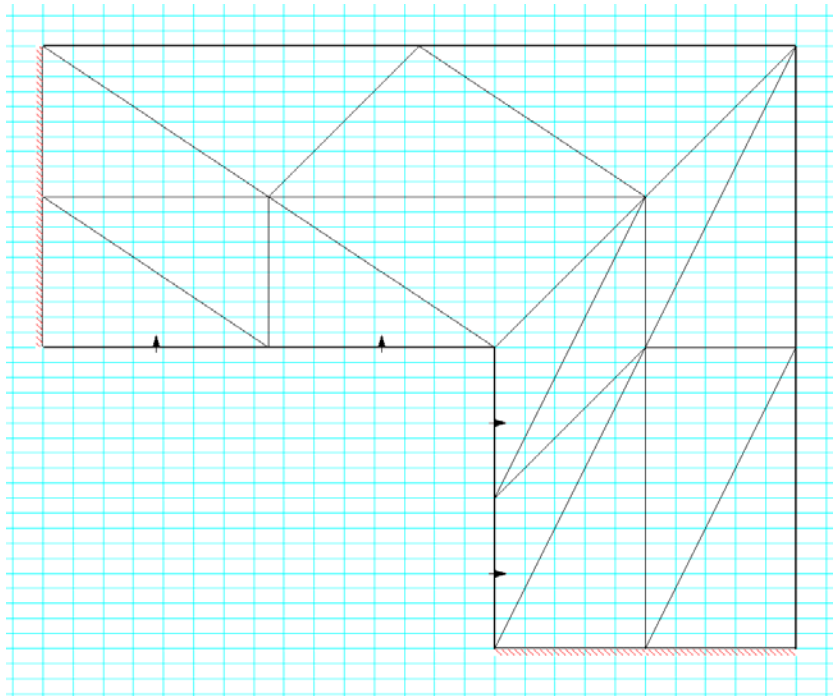


In either the temperature or contour plot output, the temperature at the cursor position will be displayed at the upper left of the screen below the menu bar when the mouse button is depressed.

One objective of this problem was to determine the total heat flow through the brick wall. Select **Heat Flows** from the **View** menu. (It is also possible to determine the heat flow using

the **Nodal Balances** command, as described in Chapter 2.) The screen will be redrawn with the nodes hidden. Click on any line segment of the inside wall. Hold the Shift key down to add the selected boundaries the previous selection. An arrow will appear indicating the direction of heat flow for the selected boundaries. 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 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 976.28 W. The total heat flow through all sides of the wall is then 3904.8 W.

X=1.65 Y=4.38 m	Steady-State	Q[14,6]=207.1 W/m Sum(Q)=976.2 W/m
-----------------	--------------	------------------------------------



Select **Input** from the **View** menu to return to the drawing window. At this point, you may wish to explore. Try using smaller triangular elements. FEHT will automatically reduce the mesh size for you if you select the **Reduce Mesh** command in the **Draw** menu. Will the smaller mesh significantly change the heat transfer rate? How refined does the mesh need to be? Compare the heat flows on the outside walls of the furnace with that on the inside. Should they agree? Do they?

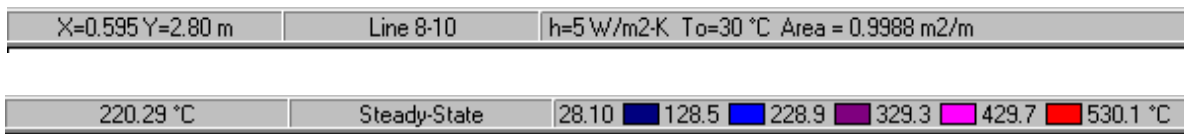
---

## *Reference Information*

---

### *FEHT Windows*

The FEHT program interacts with the user through three small information windows located just below the menu bar. The specific functions and capabilities of each window are described later in the View Menu section of this chapter.



Examples of the information display in the information windows are shown above. The leftmost window ordinarily shows the coordinates at the cursor position in the scale selected with the **Scale and Size** command in the **Options** menu. However, when the **Potentials**, **Contours**, or **Gradients** window is foremost, this window will display the potential (e.g., temperature) or gradient at the cursor position while the mouse button is depressed.

The center window has two functions. When **Input** is selected from the **View** menu), it identifies the selected object, such as a selected line, node, or frame. When any of the output display options are selected from the View menu, this box will display steady-state or, for transient problems, the time for which the Potentials, Contours, or Gradients, Heat Flows, or Nodal Balances output is displayed. Clicking the mouse within this window will cause the time to increment to the next timestep and the output window to be refreshed.

The larger information window on the right displays information appropriate to the current selection or situation. For example, if a line is selected, as shown above, the boundary conditions for this line are displayed. Error messages appear in this window as do the legends for contour and gradient plots.

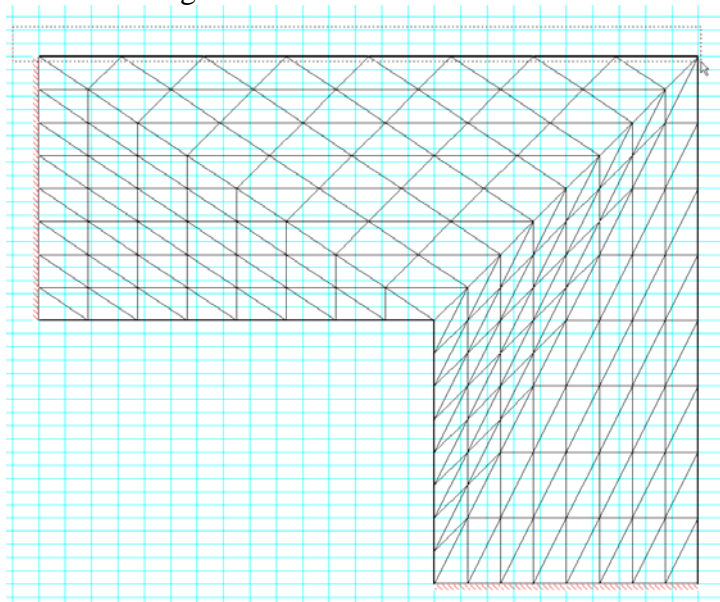
### Selecting Objects

Problem definitions consist of material outlines, element lines and nodes. These components must be “selected” in order to specify boundary conditions, properties, etc. A node is selected by placing the cursor on the node and clicking the mouse button. A line is selected by clicking the mouse button near the center of the line. To select a material, click the mouse button on a point within the material outline, but not near an existing node or line. Selected components will be indicated by flashing.

FEHT first checks to see if the mouse click is on a node, then on a line, and then on a frame. You can hide the nodes and/or lines using the **Show/Hide Nodes** and **Show/Hide Lines** commands in the **Display** menu to make the selection process easier. Alternatively, use the **Zoom** command in the **Display** menu to simplify selection of closely spaced components.

Multiple selections of the same type can be made. For example, clicking two nodes in sequence will select both nodes. To unselect an item which has been selected and is currently flashing, click on it a second time.

Lines that form the border of a material can also be selected with a selection rectangle in the Input and Heat Flows windows. To select a series of connected border lines, press and hold the left mouse button down at a point outside of any material. While holding the mouse button down, slide the mouse to a new location. A dotted line rectangle will appear as shown below. All border lines within the rectangle will be selected when the mouse button is released.



Nodes that form the border of a material can also be selected with a selection rectangle in the Nodal Balance window. To add to an existing selection, hold the Shift key down while creating the selection rectangle.

Lines are ordinarily selected in order to specify boundary conditions. As a short-cut, you can double-click the mouse on a line to immediately bring up the Boundary Conditions dialog box. Similarly, double-clicking within a material outline will immediately bring up the Material Properties dialog box.

Selection of multiple nodes, lines, or groups is simplified by the **Group** command in the **Draw** menu. When the objects are grouped, selecting any one of the objects within a group causes all of the object in the group to be selected. For example, you may wish to group all boundaries which have the same boundary conditions. To override the group selection, press and hold the Option key while selecting the object.

### ***Entering Variable Properties and Boundary Specifications***

FEHT will accept numerical constants or equations for property information, boundary specifications and internal generation. For example, the thermal conductivity of a material may be entered as a function of temperature and/or position. Boundary specifications, such as the temperature or the convection coefficient along a line, may be a function of time as well. The numerical constants or equations are entered directly into the appropriate edit field of the dialog box. The edit field will automatically scroll to accommodate long equations.

The rules relating to the use of mathematical operators in equations conform closely to those used in FORTRAN, C, or Pascal. For example, a property value, *e.g.*, thermal conductivity can be entered as  $(3 + 0.04 * T) / X$ . FEHT will interpret this expression as the sum of 3 and the product of 0.04 and T all divided by X. The thermal conductivity is evaluated at each nodal point. The temperature and position at the point would be used to evaluate the property value. The \* indicating multiplication is required. Spaces are ignored. Upper and lower case letters are treated identically. The caret symbol (^) or \*\* can be used to indicate raising to a power.

In the above equation, T is the temperature. (V is used for voltage, M for magnetic potential, S for stream function value, and H for head). Spatial position is represented by X and Y in Cartesian coordinates or by R and Z in cylindrical coordinates. The units of the spatial coordinates must be the same as those used in the drawing, as shown in the upper right information window. Boundary conditions may be a function of time, represented by the symbol Time. Time will have units of seconds or hours, depending on the choice made in the Transient Calculation dialog box.

FEHT equations may also make use of the following built-in functions.

<b>abs(z)</b>	returns the absolute value of z
<b>cos(z)</b>	returns the cosine of z, where z is in radians
<b>exp(z)</b>	returns the exponential of z
<b>ln(z)</b>	returns the natural logarithm of z
<b>log(z)</b>	same as ln(z)
<b>max(a,b)</b>	returns either a or b, whichever is larger
<b>mod(a,b)</b>	returns the remainder of a/b
<b>min(a,b)</b>	returns either a or b, whichever is smaller
<b>sin(z)</b>	returns the sine of z, where z is in radians
<b>sqrt(z)</b>	returns the square root of z
<b>sqwv(z)</b>	square-wave function with amplitude ranging between +/-1; z is in radians, similar to the sin(z) function
<b>trunc(z)</b>	returns the value of z with the decimal part set to zero

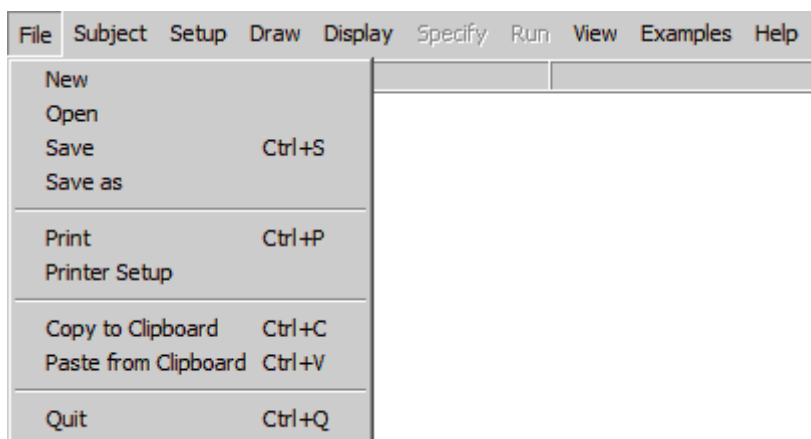


### *Working with Menus*

Commands are distributed among nine pull-down menus appearing at the top of the screen. (A tenth menu can be created under user control to display a grouped set of problems.) 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. Many of the menu items have command key equivalents. That is, they can be executed either by using the mouse or by pressing the keys indicated in the menu. Help for any menu item can be made to appear by pressing the F1 key with the mouse positioned on the menu command.

The remainder of this chapter provides detailed descriptions of each of the menu commands, starting with the **File** menu. Menu names and command names appearing in the menus will be shown in bold face. Much of the information presented here is available within the program by pressing the F1 key while holding the mouse button down on the menu item.

## File Menu



**New** initiates a new work session. All variables will be reset and the screen will be cleared. If an unsaved problem definition exists, you will be asked if you wish to save your work, as if the **Save As** command were issued.

**Open** will allow you to access and continue working on any file saved previously with the **Save** or **Save As** commands. After the confirmation for unsaved work, a dialog window will appear showing the names of all previously saved FEHT files in the current directory. FEHT files have a .fet filename name extension. If you wish, you can view the files in another directory or on another drive. The directory can be changed by clicking on the names within the Directories: list and the drive designation can be changed by clicking within the Drives list. To select a file, click on the file name in the file name list or enter the file name in the File Name edit box at the upper left. It is not necessary to supply the .fet filename.

**Save** will save your problem definition with the same file name (shown after Save in the menu) and on the same disk it was last saved. For a new work session, you will be prompted to supply a file name, just as if the **Save As** command were selected. All input information concerning the problem definition including property data is saved on the disk. The output is not saved but it can be restored by the **Calculate** command. The Save menu item is dimmed after the save operation until a change is made in the problem definition. The filename appears in the title bar of the Input window.

**Save As** provides the same function as the **Save** command except that it will first prompt you to supply a file name. This command allows you to save the problem definition with another name or on another disk drive than used previously. A dialog window will appear in which you must supply a file name. The filename will be shown as 'untitled'. Enter the file name of your choice in its place to save a copy of the file with this file name.

**Print** will print the contents of the window currently being displayed. You may first want to set printer options such as the scaling factor or Landscape/Portrait mode with the **Printer Setup** command.

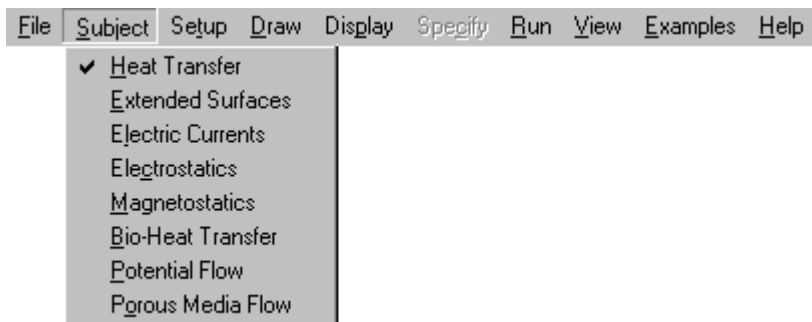
**Printer Setup** allows the printer, the paper orientation, the scaling factor and other printer options to be specified. These options will then be in effect when the **Print** command is issued and they will remain in effect until FEHT is terminated.

**Copy To Clipboard** copies what you see on the screen to a temporary memory storage area called the Clipboard thereby allowing the output of this program to be exported to other applications, such as a word processor or a drawing program. The contents of all windows except the Report window are copied into a bitmap format. The Report window contents are copied as tab-delimited which can be pasted directly into a spreadsheet program or a word processor.

**Paste From Clipboard** copies the contents of the Clipboard to the screen. The purpose of this command is to allow a drawing prepared by another application to be imported into FEHT as a drawing template. To use this feature, it is first necessary to prepare the drawing in a drawing program. The drawing may be either in bitmap or MetaPict format. The drawing must then be copied to the Clipboard (using the application's Copy command). The imported drawing can not be used directly in the problem definition. Rather, it provides a template that may be traced. It is particularly convenient to use a template when the drawing involves circles or other non-linear shapes. Once pasted onto the screen, the template can be moved and resized with the **Size/Move Template** command in the **Draw** menu. Use the **Hide Template** command in the **Display** menu to remove the template from view.

**Quit** provides a graceful way to exit the program. If there have been any changes to your file, you will be given the opportunity to save the file as if the **Save As** command were issued.

The remaining items in the File menu are recently accessed filenames. Selecting any of these filenames opens the file. This capability may be disabled for network installations.

**Subject Menu**

The **Subject** menu controls the type of problem which is to be solved. The default configuration is for heat transfer problems. The subject should be selected before any drawing is started since selecting a subject will initiate a new work session, just as if the **New** command were issued. In addition, default property data for the selected subject will be read in and the menus and dialog boxes will be reconfigured to conform with the selected subject. A check mark identifies the currently selected subject.

**Heat Transfer configures** FEHT for steady-state or transient heat transfer problems. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \dot{q} - \rho c_b (T - T_b) - \rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \dot{q} - \rho c \frac{\partial T}{\partial t}$$

where

- $T$  is temperature [ $^{\circ}\text{C}$  or  $\text{K}$ ,  $^{\circ}\text{F}$  or  $\text{R}$ ]
- $\dot{q}$  is the local internal generation rate per unit volume [ $\text{W}/\text{m}^3$  or  $\text{Btu}/\text{hr}\text{-ft}^3$ ]
- $\rho$  is the local density [ $\text{kg}/\text{m}^3$  or  $\text{lb}_\text{m}/\text{ft}^3$ ]
- $c$  is the local specific heat [ $\text{kJ}/\text{kg}\text{-K}$  or  $\text{Btu}/\text{lb}_\text{m}\text{-R}$ ]
- $t$  is time [sec or hours]
- $k$  is the thermal conductivity [ $\text{W}/\text{m}\text{-K}$  or  $\text{Btu}/\text{hr}\text{-ft}\text{-R}$ ]

**Extended Surfaces** configures FEHT for steady-state or transient, two-dimensional problems involving extended surfaces. The term ‘extended surface’ refers to a thin solid of thickness  $\delta$  in the  $z$ -direction that experiences two-dimensional  $(x,y)$  conduction within its boundaries and convection from its surfaces to an adjoining fluid. A common extended surface is the *fin*, which is used in applications to enhance heat transfer from a base surface to an adjoining fluid. The governing equation in Cartesian coordinates is

$$\delta \left[ \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) \right] + q_a'' + \delta \dot{q} = 2h(T - T_\infty) + \delta \rho c \frac{\partial T}{\partial t}$$

where

$\delta$	thickness of extended surfaces [m or ft]
$T$	temperature [ $^{\circ}\text{C}$ or $\text{K}$ , $^{\circ}\text{F}$ or $\text{R}$ ]
$q_a''$	applied surface heat flux [ $\text{W}/\text{m}^2$ or $\text{Btu}/\text{ft}^2$ ]
$\dot{q}$	rate of internal generation rate per unit volume [ $\text{W}/\text{m}^3$ or $\text{Btu}/\text{hr}\cdot\text{ft}^3$ ]
$h$	convection coefficient between surfaces and the adjoining fluid [ $\text{W}/\text{m}^2\cdot\text{K}$ or $\text{Btu}/\text{hr}\cdot\text{ft}^2\cdot\text{R}$ ]
$T_{\infty}$	temperature of adjoining fluid [ $^{\circ}\text{C}$ or $\text{K}$ , $^{\circ}\text{F}$ or $\text{R}$ ]
$\rho$	density [ $\text{kg}/\text{m}^3$ or $\text{lb}/\text{ft}^3$ ]
$c$	specific heat [ $\text{kJ}/\text{kg}\cdot\text{K}$ or $\text{Btu}/\text{lb}\cdot\text{R}$ ]
$t$	time [s or h]

Note that, with this option,  $x$  and  $y$  are the dimensions of the fin. The fin thickness,  $\delta$ , is the dimension into the screen. The fin is assumed to be thin such that there are no gradients in the direction into the screen. Convection is assumed to occur on both sides of the fin and that is the reason for the factor of 2 appearing before the convection coefficient. If convection only occurs on one side of the fin, the convection coefficient should be reduced by a factor of 2. The fin heat transfer coefficient, fluid temperature, and surface heat flux are specified with the **Surface Properties** menu item. The fin thickness is specified in the **Material Properties** dialog.

**Electric Currents** configures FEHT to solve two-dimensional, steady-state or transient problems in electrical conduction. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( \sigma \frac{\partial V}{\partial x} \right) + \frac{\partial}{\partial y} \left( \sigma \frac{\partial V}{\partial y} \right) = C \frac{\partial V}{\partial t}$$

where

$V$	is electric potential [V]
$\sigma$	is the local conductivity [S/m]
$C$	is the capacitance [F/m]
$t$	is time [sec or hours]

**Electrostatics** configures FEHT to solve two-dimensional steady-state problems in electrostatics. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( \epsilon \frac{\partial V}{\partial x} \right) + \frac{\partial}{\partial y} \left( \epsilon \frac{\partial V}{\partial y} \right) + \rho = 0$$

where

$V$	is electric potential [V]
-----	---------------------------

- $\epsilon$  is the electric permittivity [F/m]  
 $\rho$  is the volume charge density [C/m<sup>3</sup>]

**Magnetostatics** configures FEHT to solve scalar two-dimensional steady-state problems in magnetostatics. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( \mu \frac{\partial V}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial V}{\partial y} \right) = 0$$

where

- $V$  is magnetic scalar potential [V]  
 $\mu$  is the magnetic permeability [H/m]

**Bio-Heat Transfer** configures FEHT to solve steady-state or transient, two-dimensional heat transfer problems in biological tissue. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \dot{q} - wc_b(T - T_b) - \rho c \frac{\partial T}{\partial t}$$

where

- $T$  is the tissue temperature [°C or K, °F or R]  
 $T_b$  is the blood temperature [°C or K, °F or R]  
 $\dot{q}$  is the local metabolic heat generation rate per unit volume [W/m<sup>3</sup> or Btu/hr-ft<sup>3</sup>]  
 $\rho$  is the local tissue density [kg/m<sup>3</sup> or lbm/ft<sup>3</sup>]  
 $c$  is the local tissue specific heat [kJ/kg-K or Btu/lbm-R]  
 $c_b$  is the blood specific heat [J/kg-K or Btu/lbm-R]  
 $t$  is time [sec or hours]  
 $k$  is the tissue thermal conductivity [W/m-K or Btu/hr-ft-R]  
 $w$  is the local perfusion [kg/m<sup>3</sup>-sec or lbm/ft<sup>3</sup>-hr]

The tissue properties - thermal conductivity, density, and specific heat - and perfusion rate are specified with the **Material Properties** dialog. The blood properties - temperature and specific heat - are specified with the **Blood Properties** dialog. The metabolic heat generation rate is specified in the **Generation** menu item.

**Potential** configures FEHT to solve two-dimensional, steady-state, potential flow problems. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( \frac{\partial \psi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\partial \psi}{\partial y} \right) = 0$$

where

$\psi$  is the stream function [m<sup>2</sup>/sec]. The fluid velocities,  $u$  and  $v$ , in the  $x$  and  $y$  directions respectively, are related to  $\psi$  by

$$u = \frac{\partial \psi}{\partial y} \quad v = -\frac{\partial \psi}{\partial x}$$

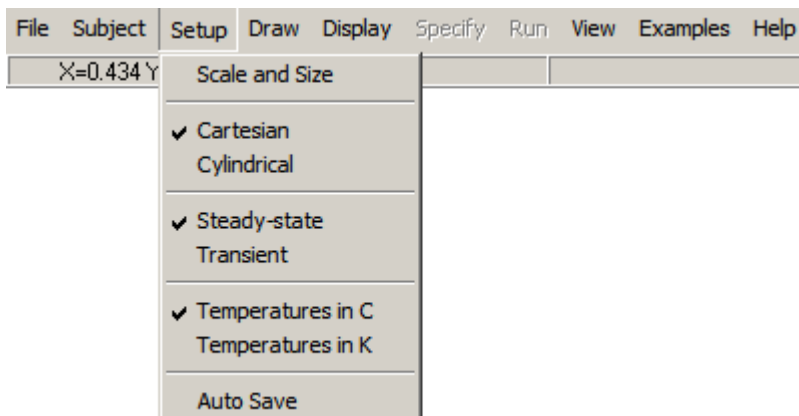
**Porous Media Flow** configures FEHT to solve two-dimensional steady-state or transient problems in flow through porous media, such as occur in underground water flows. The governing equation in Cartesian coordinates is

$$\frac{\partial}{\partial x} \left( k \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial H}{\partial y} \right) - S \frac{\partial H}{\partial t} = 0$$

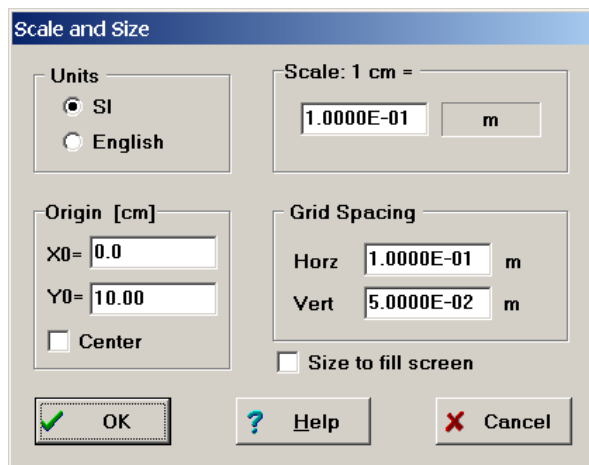
where

- H is hydraulic head [m]
- k is the local hydraulic conductivity [m/sec]
- S is the specific storage [1/m]

Setup Menu



**Scale and Size** will display the dialog window shown below in which the unit system (for heat transfer problems, English or SI), the position on the screen of the origin of the coordinate system, the length scale and units, and the grid spacing can be specified. In addition, capability is provided to center the drawing or resize the drawing to fit the video screen.



The unit system controls in the upper left are applicable only for heat transfer problems. All other problem types use SI units.

The origin of the coordinate system is positioned by specifying values in the Origin box at the lower left. In the Cartesian coordinate system, these are designated as X0 and Y0. In cylindrical coordinates, R0 and Z0 are used. Setting X0 (or R0) to 0 and Y0 (or Z0) to 0 will place the origin of the coordinate system at the bottom left corner of the screen. The origin values are specified in centimeters (or inches in English) in actual screen coordinates. The Center checkbox selected the coordinate system origin so as to center the drawing on the screen, overriding the values supplied for the origin.

The drawing scale and units are specified in the Scale box at the upper right. The value indicated in the this box is the length which is represented by 1 cm (or 1 inch in English)



units on the screen. Click on the box displaying the units to cycle it through a number of units choices. Each click will rotate the unit to the next possible unit type and automatically convert the scale and grid spacing value. The scale value can be changed by clicking the mouse button in the scale rectangle followed by editing. Note that the drawing size itself does not change but rather just the scale. The 'Size to fill screen' checkbox can be used to change the drawing size as explained below.

A grid will be placed on the screen if the **Show Grid** command in the **Display** menu is selected. The horizontal and vertical spacing of the lines forming the grid are controlled by the values in the Grid Spacing box at the lower right. The spacing value is entered in the unit system and scale of the drawing. The horizontal and vertical position of the cursor corresponding to the selected scale, unit system, and origin are shown in the information window at the upper left just below the menu bar.

The Scale and Size commands should normally be issued before a drawing is started, but it can be applied to an existing drawing. Changing the unit system will result in automatic conversion of all defined quantities, but will not affect the display. Changing the origin of an existing drawing will shift its location on the screen. Changing the scale will not change the displayed size on the screen, but it will affect the actual dimensions of the drawing.

The Size to fill screen checkbox provides a means of changing the displayed size of a drawing. If this option is selected, the existing drawing will be rescaled so that it fills the Input Window in the unzoomed mode. The scale and grid spacing will be adjusted accordingly. This option is most useful when a drawing was initiated on a computer having a smaller or larger video display than currently available.

The 'Size to fill screen option' can be used to set the drawing to any desired size. To shrink the drawing, make the Input Window small by first clicking on the size control at the upper right of the screen, and then resize the window by dragging an edge. Next, select the Scale and Size command and click the 'Size to fill screen' checkbox. The drawing will be resized to fit the smaller window. Finally, click the size control again to restore the Input window to its original full-screen size.

**Cartesian** instructs FEHT to consider the problem to be two-dimensional in an X-Y coordinate system. The Cartesian option assumes that the object has infinite extent in the direction normal to the screen. Heat flows (or analogous quantities for electric fields) are calculated on a per unit (foot or meter) depth. The position of the origin (*i.e.*, the position on the screen where  $x=0$  and  $y=0$ ) can be changed with the Set Units and Scale command. The axes become visible if the **Show Axes** command in the **Display** menu is selected.

**Cylindrical** instructs FEHT to consider the problem to be axi-symmetric. Only the portion of

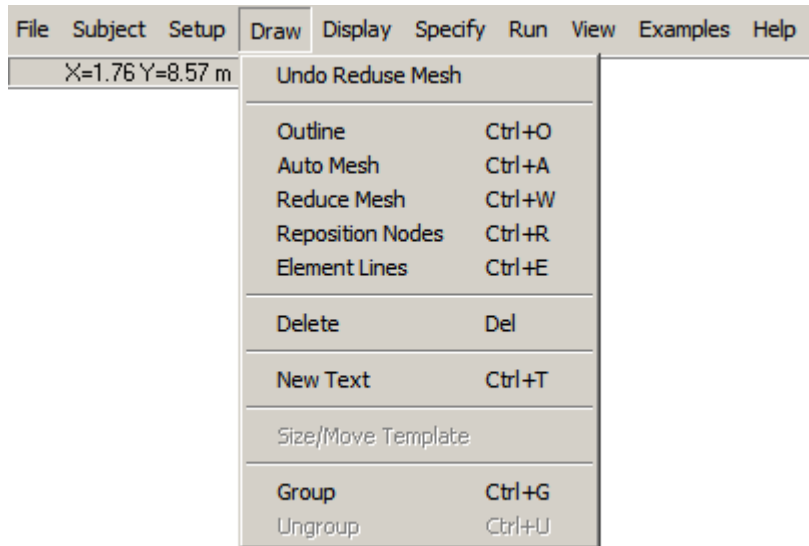
the object to the right of the vertical centerline can be drawn. The vertical centerline is initially placed about one-half inch from the left edge of the screen. The position of the centerline can be changed with the **Scale and Size** command. The centerline can be made invisible by selecting **Hide Axes** command in the **Display** menu.

**Steady-State** instructs FEHT to consider the problem to be steady-state. This command, which appears only for heat transfer, electric currents and porous flow problems, will disable the **Initial Conditions** menu item (**Specify** menu). The steady-state solution will be calculated after the **Calculate** command (**Run** menu) is issued. The **Potentials vs Time** and **Flow vs Time** windows (**View** menu) are of course not applicable and will be disabled. This command can be entered at any time. A check mark will appear in front of Steady-State in the menu to indicate that it has been selected.

**Transient** instructs FEHT to consider the current problem to be transient (*i.e.* time dependent). This command is available only to heat transfer, electric currents, and porous flow problems. The **Initial Conditions** command (**Specify** menu) becomes accessible for transient problems. The start time, stop time, and time step are entered when the **Calculate** command (**Run** menu) is issued. The Transient command can be entered at any time. A check mark will appear in front of Transient in the menu to indicate that it has been selected. By default, a problem is assumed to be steady-state.

**Temperatures in C** and **Temperatures in K** specify the units in which temperatures will be displayed for heat transfer problems. (In the English system, these commands would apply for F and R, respectively.) The temperature scale in use is indicated by a check mark in front of the menu item. Changing the scale will convert all defined temperatures.

**Auto Save** saves a backup copy of the file whenever the problem definition is changed. This option has a check preceding the menu name when it is selected. The **Backup** command in the **File** menu can be used to open the saved backup file.

**Draw Menu**

**Undo** reverses the effects of the last change made to the problem definition. The Undo menu name will change to Undo Outline, Undo Lines, Undo Auto Mesh, etc. after using any of these drawing commands. Selecting the **Undo** command will restore the problem to the condition it was in before that last command was issued. An internal copy of the problem information is made in memory when any of the drawing commands are issued in order to support the Undo command. Up to 16 copies are retained in memory.

**Outline** is used to define the outline of a material. Each material is considered to have a single set of property values (or equations) and a single internal generation value (or equation). Boundary conditions must be specified for the lines forming the outline of a material. The material outline definition begins by positioning the cursor at a point on the boundary of the material and clicking the mouse button. A node (designated by a small circle) will appear at the corner. (The first node will be displayed as a solid circle, while the remaining nodes are displayed as open circles.) From then on, a line between the node and the cursor will be indicated. The cursor is then moved to the next boundary point at which a node is desired. The node is fixed by clicking the mouse button. This process is continued until the mouse is again clicked on the first node (solid circle), creating the outline.

In constructing the outline of a material which borders an existing material, it is necessary to trace over the common border, clicking the mouse button on each node in common. Material outlines may share a common border, but may not overlap. To aid in precisely locating nodes, the horizontal or vertical position of the cursor can be locked in place by holding down the Shift or Ctrl key, respectively. You can escape from the Outline command at any point by pressing the Esc key.

**Auto Mesh** creates a relatively crude triangular mesh for the selected outline. If no outline is selected, the mesh will be created for all outlines in the problem description. It is recommended that boundary conditions be set before using **Auto Mesh**. Note that the outline must not have any existing element lines in order for this command to work. After creating a crude mesh, **Auto Mesh** calls **Reduce Mesh** to reduce the size of the triangular elements. Use the **Undo** command if you wish to reverse this action.

**Reduce Mesh** can be applied only after all distributed materials are discretized into triangular elements. This is usually done with the **Auto Mesh** command. Mesh reduction can be applied to one or more selected material outline(s) or to all outlines.

One of two mesh reduction algorithms are used, depending on whether any outlines are selected. When the Reduce Mesh command is given when no outlines are selected, the mesh size of all outlines is reduced by forming new elements constructed with lines connecting the center points of existing lines. Each existing element then results in four new elements. The geometry of the original discretization is preserved. When the Reduce Mesh command is given while one or more outlines are selected (*i.e.*, flashing), the mesh reduction occurs only for the selected outlines by adding a node at the centroid of each existing element in the outline and connecting this new node to the vertices of the element. Then, if the element lines are not on the border of an unselected outline, they are bisected and nodes at each bisection point are connected to the centroid node. The result is that each existing element is reduced into 3 to 6 new elements.

It is advantageous to set the boundary conditions (or group lines with the **Group** command in this menu before applying **Reduce Mesh**, since this command may produce many more boundary lines. Alternatively, set the boundary conditions before reducing the mesh.

**Reposition Nodes** allows the horizontal and/or vertical position of existing nodes to be changed. After selecting this command, place the cursor on the node which is to be moved and move it to its new location while holding down the mouse button. (The horizontal or vertical position of the node can be locked in place by holding down the Shift or Ctrl key, respectively.) **Reposition Nodes** remains in effect until the mouse is clicked on something other than an existing node.

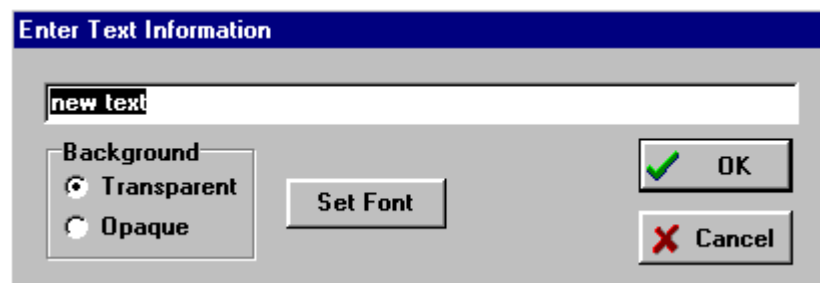
**Element Lines** allows manual control over forming a triangular mesh. This command is normally not needed but it can be used to improve a mesh that was automatically created with the **Auto Mesh** command. After selecting the **Element Lines** command, the cursor must be placed on an existing node or line. Clicking the mouse button fixes one end of the new element line, creating a node if needed. The cursor is then moved to the other end of the line which is fixed by clicking the mouse. This process is repeated as needed. To aid in precisely locating nodes, the horizontal or vertical position of the cursor can be locked in

place by holding down the Shift or Ctrl key, respectively. Element lines can not be constructed in a lumped material. Press the Esc key to escape from the Element Lines command. Note that it is not necessary to finely discretize the material as this can be done automatically using the **Reduce Mesh** command.

**Delete** can be used to delete one or more element lines, nodes, or material outlines. To delete one or more element lines, select the lines to be deleted by clicking the mouse button on each line center, causing it to flash. Then select Delete. Note that the borders of materials are not considered to be element lines and cannot be deleted.

To delete one or more nodes, click the mouse button on the nodes causing them to flash and then issue the Delete command. A node may be deleted only if it is connected to two or fewer lines. It may be necessary to delete element lines before deleting the node. Nodes on the border of a material may be deleted, but care should be exercised so that the resulting line does not cross any existing lines. Clicking the mouse button on any point within a material (but not on a node or line) will select that material. Delete will delete all nodes and element lines within the material, as well as the material outline.

**New Text** allows text to be entered on the selected window. Note that the text will appear only on the window which is foremost with this command is issued. The following dialog window will appear.



Enter the text which is to appear on the window in the edit box. The text background is controlled by the radio buttons. Transparent displays only the letter, whereas Opaque clears a rectangle for the text. Clicking the Set Font button displays a dialog window in which the font, size, and color of the text can be specified. Click OK or press the Enter key when the text specifications are completed. The text can be repositioned by pressing and holding the mouse button down within the text rectangle while sliding the text to its new location. Double-clicking on a text item will bring up the above dialog window in which further changes can be made.

**Size/Move Template** becomes accessible only after a template has been copied with the **Paste from Clipboard** command (**File** menu). The purpose of this command is to reposition and/or resize a template drawing so that it may be 'traced' during the problem definition.

After selecting Size/Move Template, the template drawing will be displayed in front of all everything else in the Input Window along with a boundary rectangle and a size box. To reposition the template, press and hold the mouse button down anywhere within the rectangle (except within the size box) and drag the template to a new location while holding the mouse button down. The template will be redrawn after the button is released. To change the size of the template, press and hold the mouse button down in the size box at the lower right corner of the boundary rectangle. Drag the mouse to the new lower right corner. The template will be scaled and redrawn after the button is released.

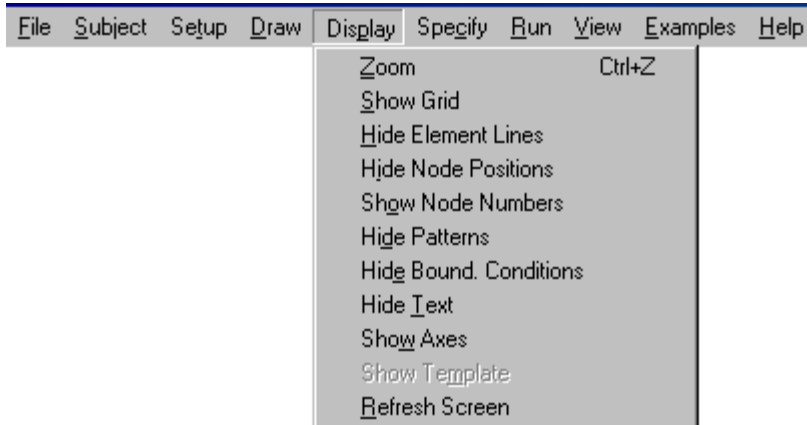
Use the **Hide Template** command (**Display** menu) to remove the template from the screen when it is no longer needed.

**Group** becomes available when two or more lines, nodes, or outlines have been selected. The Group command causes the selected items to be grouped into a single entity. After these items are grouped, selecting any one of the items causes all in the group to be selected. (Depressing the Ctrl key while selecting the item disables the group selection.) The grouping is in effect only for the window in which it was defined. Use the **Ungroup** command in this menu to undo the group specification.

As an example, all boundary lines having identical boundary conditions can be grouped. In this way, the boundary conditions for all of these lines can be easily changed just by selecting one of the lines. Similarly, all nodes on a boundary can be grouped in the **Nodal Balances** window so that the flow across that boundary can be easily determined.

If an item appears in more than one group, the most recent group will be chosen when that item is selected. Grouped boundary lines are preserved when the **Reduce Mesh** command (**Draw** menu) is applied. However, groups are destroyed when the **Delete** command is issued.

**Ungroup** becomes available when a previously established group of lines, nodes, or outlines has been selected. The group is established with the **Group** command. Ungroup destroys the group structure relating these items.

*Display Menu*

**Zoom (Normal Size)** allows a selected part of the screen to be enlarged. After the Zoom command is given, a small rectangle will appear with its upper left corner at the position of the cursor. Move the cursor to the upper left corner of the screen area which is to be enlarged and click the mouse button. Next move the cursor to the lower right corner. The rectangle will grow as needed indicating the viewing area of the enlarged screen. Click the mouse button to complete the Zoom command. The enlarged screen will be drawn with scroll bars at the bottom and right of the screen. The scroll bars or the arrow keys can be used to locate the viewing area anywhere within the drawing. All drawing and output alternatives operate with either the normal size or enlarged screen. You may find it convenient to enlarge the drawing while creating or selecting closely packed nodes or element lines. When the zoom command is effect, the Zoom menu item is changed to **Normal Size**.

**Show (Hide) Grid** superimposes your drawing on a rectangular grid. The distance between the grid lines can be set with the **Scale and Size** command (**Setup** menu). When the grid is made visible, the menu toggles to **Hide Grid**.

**Hide (Show) Element Lines** controls the visibility of the element lines which divide a material into triangular elements. You may wish to hide element lines to produce an uncluttered display of your drawing or to make it easier to select a material containing many closely packed element lines. Hidden element lines cannot be selected. The **Hide Element Lines** command toggles the menu item to **Show Element Lines**.

**Hide (Show) Node Positions** removes (or restores) the node position markers from the screen. The position of nodes is represented either by a small circle (if **Hide Node Numbers** is in effect) or by a larger circle containing the node number. You may wish to hide node positions to produce an uncluttered display of your drawing or to make it easier to select materials or element lines. Nodes can not be selected if they are hidden. The node position

markers are restored with the **Show Node Positions** command which replaces **Hide Node Positions** in the menu.

**Show (Hide) Node Numbers** controls the type of marker used to indicate the node positions. Each node is identified with a node number. Lumped materials are identified with a number. It is necessary to know the node numbers for some output operations. They may be displayed with the **Show Node Numbers** command. Circles at each node position will be drawn around the node number when this command is in effect. Lumped materials are treated in a manner similar to nodes. A number for the lumped material will be displayed at the position within the material at which the mouse was last clicked. Nodes associated only with lumped materials cannot be selected and are shown in inverse. The node numbers can be hidden with the **Hide Node Numbers** command which replaces **Show Node Numbers** in the menu. Note that the node number of the last selected node is also displayed in the information window at the upper left of the screen whether or not the **Show Node Numbers** command is in effect. Node numbers may be changed by FEHT during delete operations and when a material is changed between lumped and distributed. The node numbers showing on the screen are not used in the calculations. In order to minimize the bandwidth, FEHT reorders the nodes and thereby reduce computational effort.

**Hide (Show) Patterns** removes (or restores) the identifying pattern for each material type from the screen. The patterns are selected with the **Material Properties** command. In some cases, it may be easier to position nodes and element lines if the pattern is hidden.

**Hide (Show) Bound. Conditions** changes the screen display so that boundaries and nodes for which boundary conditions have been specified are (or are not) represented in bold (*i.e.*, double thickness). Shown in this way, it is easy to see where boundary conditions have and have not been specified. Hide Bound. Conditions causes these lines and nodes to be shown normally.

**Hide (Show) Text** temporarily removes (or restores) the text entered with the **Text** command.

**Show (Hide) Axes** controls the visibility of the horizontal and vertical axes ( $X=0$ ,  $Y=0$ ) in Cartesian coordinates and the center line ( $R=0$ ) in cylindrical coordinates. If the centerline or axes do not appear after the command is selected, it may be that the axes are currently at the bottom left of the screen or off the screen. The origin of the coordinates system can be changed with the **Scale and Size** command (**Setup** menu).

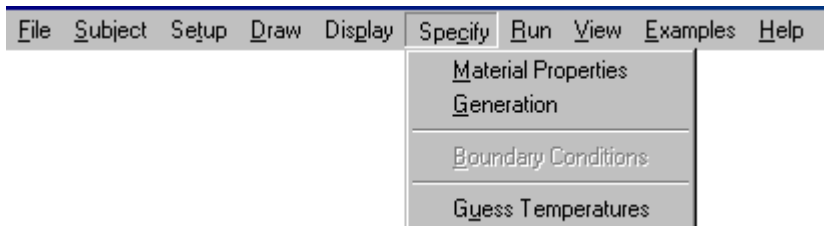
**Show (Hide) Template** controls the visibility of the template drawing which can be traced over to simplify a problem definition. This command is accessible only after a template



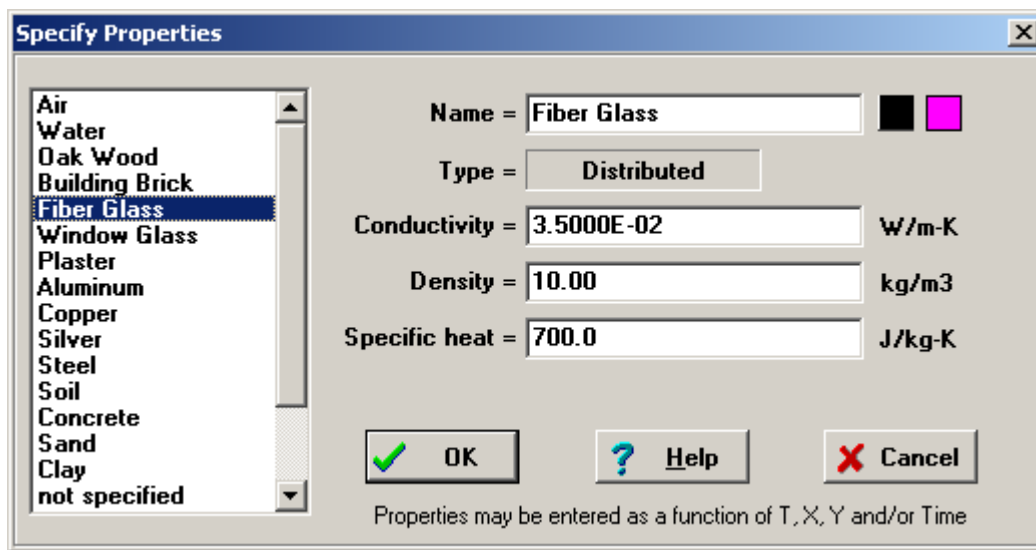
has been copied from the Clipboard with the **Paste from Clipboard** command (**File** menu). The template is displayed behind existing materials unless the **Size/Move Template** option (**Draw** menu) is selected.

**Refresh Screen** simply causes the screen to be redisplayed. It should not be needed under normal circumstances.

**Specify Menu**



**Material Properties** allows the thermal properties for up to twenty different materials to be specified for use in the current problem. To specify properties, you must first select one or more materials by clicking the mouse button on a point within the outline (but not on an internal line or node). The outline of selected materials will flash if the material is distributed. A lumped material will acknowledge selection by flashing the circle with its identification number. The Material Properties command can also be selected by double-clicking the mouse within the material. The Material Properties command will display the following dialog window.



Select the material type by clicking the mouse button on the property name in the list at the left of the screen. The information corresponding to the material type you select is displayed in the boxes to the right of the screen, where they can be changed, if desired. Changes made to the property name field will also appear in the property name list.

Property data must be entered in the indicated units. FEHT allows properties to be entered either as numerical constants or as user-defined functions of potential (*e.g.*, temperature), position and/or time. See the Entering Variable Properties and Boundary Specifications section of this chapter for additional information concerning variable properties. (Constant property values, but not equations, are automatically converted in heat transfer problems when the unit system is changed with the **Scale and Size** command (**Setup** menu).) All

property data are saved with the problem definition when the **Save** or **Save As** command (**File** menu) is used.

Materials may be Distributed, Lumped or Anisotropic. Anisotropic materials are those in which the thermal conductivity differs in the X and Y directions. Lumped materials have a uniform temperature. Click in the Type box to toggle the choice. The pattern associated with a material can be changed by pressing the mouse button in the pattern box to the right of the property Type field. This will cause a pattern palette to appear showing the possible patterns. To select a pattern, hold the mouse button down while moving to position the cursor over the desired pattern in the palette. The pattern color is selected in the same manner by pressing the mouse button in the color palette. Patterns may be displayed or hidden with the **Show (Hide) Patterns** command (**Display** menu).

### Generation, Volume Charge Density, Bernoulli Constant

**Generation** becomes accessible for heat transfer problems after one or more materials have been selected. (To select a material, click the mouse button on a point within the material, but not on an internal line or node.) Internal generation is entered on a per unit volume basis which, for Cartesian coordinates, assumes unit depth of material into the screen. Internal generation may be expressed in an equation as a function of temperature, position and/or time, as described in the Entering Variable Properties and Boundary Specifications section at the beginning of this chapter.

**Volume Charge Density** replaces the **Generation** in the **Specify** menu for electrostatics problems. The command is accessible after one or more materials have been selected. The volume charge density [Coul/m<sup>3</sup>] may be entered as a function of voltage, spatial position, and/or time. In Cartesian coordinates, unit depth of material into the screen is assumed.

**Bernoulli Constant** replaces the **Generation** in the **Specify** menu for potential flow problems. The command is accessible after one or more materials have been selected. The Bernoulli Constant command allows the user to change the zero default value of the Bernoulli constant which is used in the calculation of the pressure contours. The governing equation is

$$P + \frac{1}{2} \rho (u^2 + v^2) = BC$$

where

P is the local pressure [Pa]

$\rho$  is the fluid density [kg/m<sup>3</sup>]

BC is the Bernoulli constant [Pa]

u and v are velocities [m/s] related to the local stream function by

$$u = \frac{\partial y}{\partial x} \quad v = \frac{\partial y}{\partial y}$$

**Boundary Conditions** is used to set the values of node potentials (*e.g.*, temperatures, voltages, etc.) and material boundary conditions. This command becomes accessible after one or more nodes or border lines have been selected.

### Lines

Boundary conditions may be specified for lines which form the outline of a material, but not for element lines. Select all of the lines which are to have the same boundary conditions by clicking on the center of each line. (Use the Zoom and/or Hide Node Positions command (Display menu) to make the selection process easier for closely packed lines.) Then select the Boundary Conditions command (or double-clicking the mouse button on the last line) to bring up the Boundary Conditions dialog window for lines. The appearance of the Boundary Conditions dialog box will differ depending on the subject choice. The dialog box shown below is applicable for heat transfer problems.

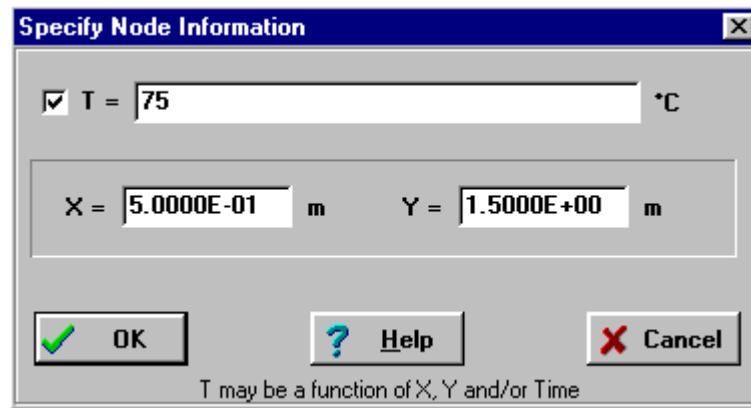
Option	Value	Unit
<input type="checkbox"/> Temperature		°C
<input type="checkbox"/> Heat Flux		W/m <sup>2</sup>
<input checked="" type="checkbox"/> Conv. Coef	5.4+0.4*T	W/m <sup>2</sup> -K
Fluid Temp.	20+sin(Time/20)	°C

Boundary information may be a function of Time and/or T

Boundary information may be entered as numerical constants or as equations involving the potential and/or time as shown above. Multiple boundary specifications can be made by entering information in the appropriate edit boxes. To remove a specification, click the check box to the left of the name.

### Nodes

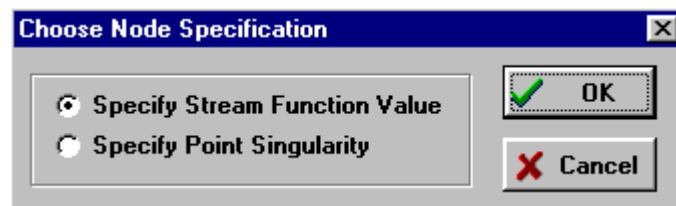
To enter boundary information for a node, place the cursor on the node and click the mouse button causing it to flash and then select the Boundary Conditions command. Alternatively, double-click the mouse button with the cursor positioned on the node. A dialog box will appear, as shown below, in which the potential at the node may be entered. The potential at the node may be entered as a numerical constant or as an equation involving time, and/or spatial position, as explained at the start of this chapter. To remove the specification, click the mouse button in the check box.



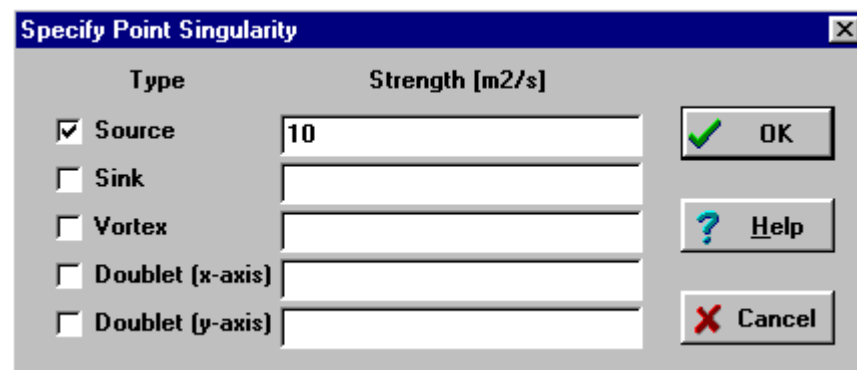
If a single node is selected, its coordinates can also be changed with this command. The X and Y (Cartesian) or R and Z (Cylindrical) coordinates will be displayed in edit boxes where the values can be changed. However, the new coordinates will not be accepted if it causes lines to cross.

### Point Singularities

Point singularities can be specified for nodes in Potential Flow problems. Selecting the Node Information menu item will provide the following dialog menu.



If Specify Point Singularity is chosen, the Point Singularity dialog window appears. One or more singularities of the types indicated in the dialog can be specified (or unspecified) for each selected node by clicking in the check box to the left of the singularity type and entering the value of the strength. The entered values must be numerical constants.



Type	Singular Solution
Source	$\psi = S/(2\pi) \theta$
Sink	$\psi = -S/(2\pi) \theta$
Vortex	$\psi = -S/(2\pi) \ln r$
Doublet (x-axis)	$\psi = -S/r \sin(\theta)$
Doublet (y-axis)	$\psi = -S/r \cos(\theta)$

where

$S$  is the specified singularity strength [ $m^2/sec$ ]

$$\theta = \arctan[(Y-Y_0)/(X-X_0)]$$

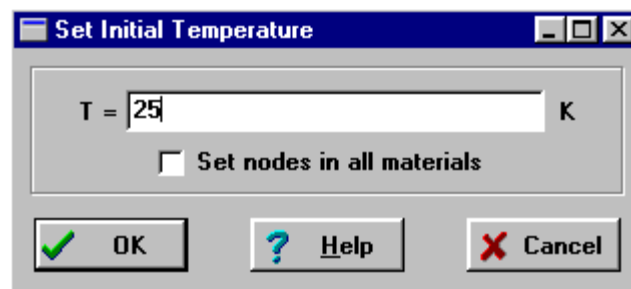
$$r = \sqrt{(X-X_0)^2 + (Y-Y_0)^2}$$

$X_0$  and  $Y_0$  are the coordinates of the singular point

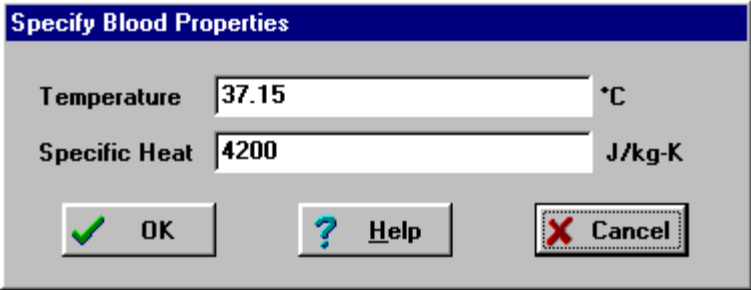
**Guess Conditions (Temperatures, Voltages, Head Values)** is accessible for heat transfer, 2-D fins, electric current and porous flow problems only if **Steady-State** has been specified in the **Setup** menu and one or more nodes or materials have been selected. This command allows the a starting or guess potential (temperature, voltage, or head) to be specified. This guess potential is used only when properties and/or boundary conditions are functions of the potential.

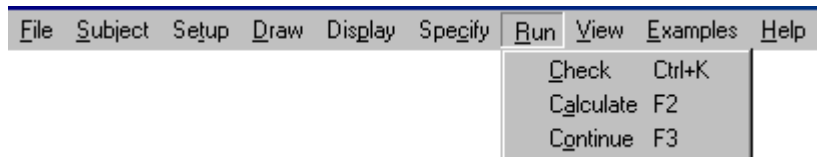
**Initial Conditions (Temperatures, Voltages, Head Values)** is accessible for heat transfer, 2-D fins, electric current and porous flow problems only if **Transient** has been specified in the **Setup** menu and one or more nodes or materials have been selected. This command allows the potential (temperature, voltage, or head) to be specified at time 0. To specify the initial potential of one or more nodes or lumped materials, click the mouse on the node causing it to flash, execute the Initial Conditions command and enter the potential value.

The initial potential values of all nodes within a material may be specified by selecting the material and then executing the Initial Conditions command. If the Set node in all materials checkbox is selected, then the initial values of all nodes in all materials will be set at the specified value.



**Blood Properties** is applicable only for Bio-Heat Transfer problems. The following dialog window will appear in which the temperature and specific heat of blood can be specified.



**Run Menu**

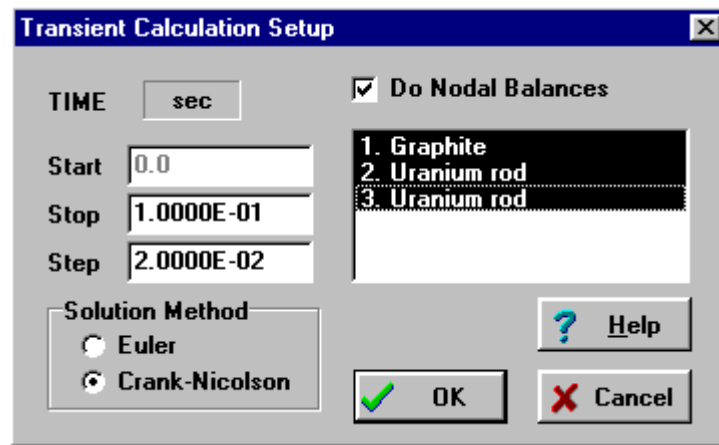
**Check** will review the problem definition and report any apparent errors. The problem will first be checked to ensure that all distributed materials are divided into triangular elements. Lines of non-triangular elements will be marked with a small square. Use the **Auto Mesh** or **Element Lines** command (**Draw** menu) to correct the problem. The properties of each material will be examined next to see that they are all defined. Materials with undefined properties will be made to flash. **The Material Properties** command (**Specify** menu) can be used to specify properties. The boundary conditions will be checked next to ensure that all boundary conditions are properly specified. Boundaries found to be in error will be marked with a small square and an explanatory note in the information window at the top of the screen. Corrections are made with the **Boundary Conditions** command (**Specify** menu). For heat transfer problems, convective boundaries may not be specified between two distributed materials. For transient problems, nodes with unspecified temperatures will be marked with a box centered about the node. Error messages will be shown in the information window at the upper right.

**Calculate** initiates the calculations. The problem definition is first checked as described under the **Check** command. Any error in the problem definition aborts the calculations. For steady-state problems, calculations begin directly after the checking process is completed. A dialog box will appear indicating that the calculations are in progress. The number of unknown potentials and the bandwidth of the matrix used in the calculations will be displayed. The calculations can be aborted by clicking the mouse in the Abort button. When the calculations are completed, the elapsed time will be displayed and the Abort button will become a Continue button. Click the Continue button to remove the dialog box.

If no errors are found for transient problems, the dialog window shown below will appear in which the stop time, time step and solution method are specified. The times may be entered in either seconds or hours. Click the mouse in the box with the time unit to toggle to the other unit. The Crank-Nicolson solution method provides greater stability than the Euler method and, for finite-elements (unlike finite differences), no computational penalty. The Euler method is provided as an option so that the effect of different calculation methods on results and stability can be investigated. A check box is provided for nodal balances. The nodal balances are required if the **Nodal Balances** or **Energy Flows vs Time** windows are to be used to view the transient output. If the Do Nodal Balances check box is enabled, one or more material outlines for which the balances will be done can be



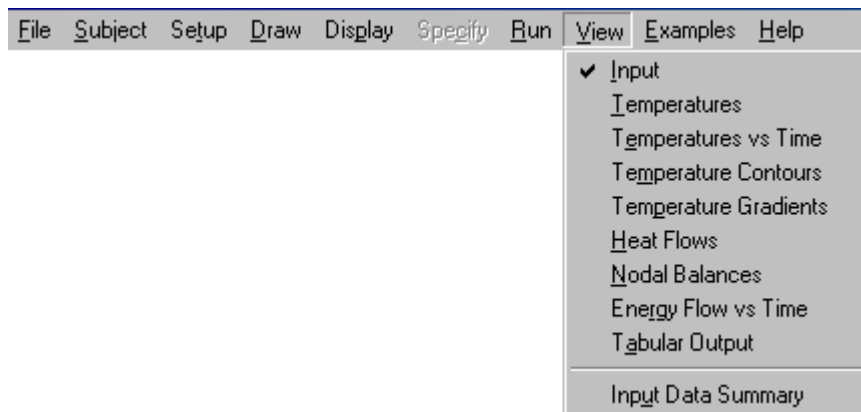
selected from the list. Selecting Nodal Balances is optional because it requires additional storage and more computational effort. A detailed discussion of the use of nodal balances is provided in the **View** menu section and in Chapter 3.



**Continue** allows a transient simulation to be restarted using, as the initial conditions, the temperatures calculated in a previous steady-state or transient calculation. This command can be used to investigate the effect of time-dependent boundary conditions or varying property values. The **Continue** command is accessible only for transient problems and only after a steady-state or transient simulation has been previously run. After issuing this command, the transient setup dialog window shown above will appear in which the stop time and time step can be entered. The start time will be set to the stop time of the previous calculation (or zero for a preceding steady-state calculation.) The conditions at this start time will be set to those calculated at the end of the previous calculation.

Another good use for the Continue command is that it enables you to start with very small time steps to keep oscillations small and then later increase the time step to make more rapid progress (with less oscillations) in the transient solution.

### View Menu



The FEHT program is centered around nine main windows. Only the Input window is used for entering the problem description. The other eight windows output results in a variety of forms. Each command in this menu will bring the corresponding window to the front. A description of the functions and capabilities of for each window are provided in this section.

**Input** causes the Input Window to be the active window. The problem description is developed in the Input Window which always appears when a problem is started. Information particular to the Input window is as follows.

1. The scale of the problem is set with the **Scale and Size** command. The coordinates of the cursor position are displayed in the upper left information window.
2. Each material is represented by a separate outline composed with a series of straight lines developed with the **Outlines** command in the **Draw** menu. The properties of each material are specified with the **Material Properties** command in the **Specify** menu. There is an upper limit of 52 frames, but the number of lines and nodes is limited only by available memory.
3. The **Paste from Clipboard** command in the **File** menu allows a graphical object composed in any other Windows application to be displayed on the screen. This graphical object can be repositioned and resized if the **Size/Move Template** command in the **Draw** menu is selected. Then the object can be traced to make the process of outlining materials easier. The use of a template is particularly convenient for circular object. The **Hide Template** command in the **Display** menu can be used to remove the template display when it it no longer needed.
4. Each distributed material must be discretized into triangular elements using the **Element Lines** and **Reduce Mesh** commands in the **Draw** menu. The number of triangular elements is limited only by available memory.

5. The location of an existing node can be changed in two ways. After selecting the Reposition Nodes command in the Display menu, the node can be dragged to a new location by holding the mouse button down with the cursor positioned on the node while moving it to its new location. Alternatively, the exact coordinates of the node can be entered by first selecting the node and then choosing the Bound. Conditions command in the Specify menu. In either case, the node cannot be moved to position such that it causes existing lines to cross.
6. Properties or boundary conditions can be specified only after a node, line, or material is selected. A selected object will flash. To select an object, click the mouse on that object. FEHT first checks to see if the mouse click is on a node, then on a line, and then on a frame. You can hide the nodes and/or lines using the **Show/Hide Nodes** and **Show/Hide Lines** commands in the **Display** menu to make the selection process easier. Alternatively, you can zoom a section of the drawing to a larger size using the **Zoom** command in the **Display** menu.
7. Nodes and lines for which boundary conditions have been specified are shown in bold. The feature can be disabled with the **Hide Bound. Conditions** command in the **Display** menu.
8. Selection of multiple nodes, lines, or groups is simplified by the **Group** command in the **Draw** menu. When the objects are grouped, selecting any one of the objects within a group causes all of the object in the group to be selected. For example, you may wish to group all boundaries which have the same boundary conditions. To override the group selection, press and hold the Option key while selecting the object.
9. The name of the current file appears in the window title bar.

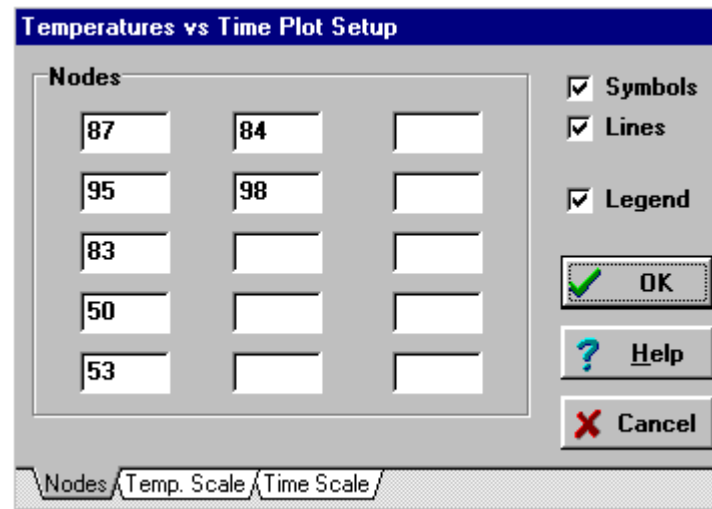
**Potentials** displays the calculated potentials applicable to the selected subject. The menu item in the **View** menu will be renamed depending on the subject, e.g, **Temperatures, Voltages, Magnetic Potentials, Stream Function Values, Head Values**.

For steady-state problems, this command will immediately display the potential at each node in a small circle centered about the node. The cursor position in the scale used to construct the drawing is shown in the information window at the upper left, just below the menu bar. The potential at the cursor position will be displayed in this window when the mouse button is depressed. Note that FEHT only calculates temperatures at the nodal positions. The temperatures at other points are estimated by linear interpolation from the temperatures of triangular element enclosing the cursor. The potentials of closely spaced nodes can more easily be seen in an enlarged drawing obtained with the **Zoom** command in the **Display** menu.

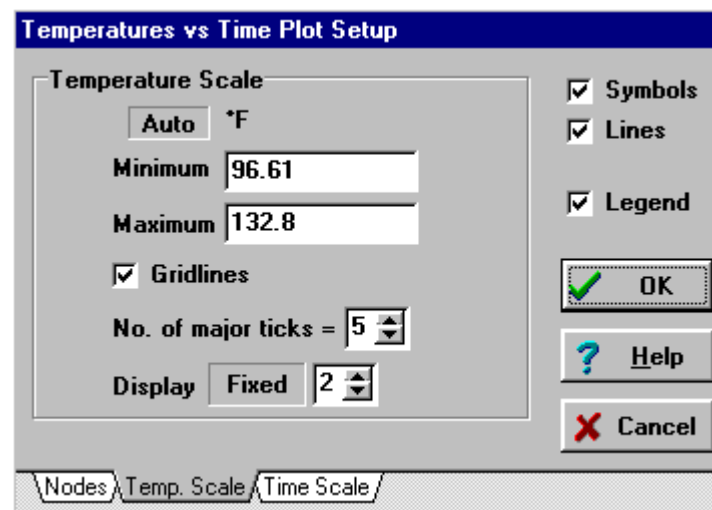
For transient problems, a dialog window will first appear with a scroll bar to set the time for which the potentials are to be displayed. The potentials at the selected time will appear when the dialog window is dismissed. The time corresponding the displayed potentials is shown in the center information window below the menu bar. Clicking in this window causes time to be incremented by one timestep and the potentials at this new time to be displayed.

**Potentials vs Time** window is applicable only for transient problems. The menu item in the **View** menu will be renamed **Temperature vs Time, Voltages vs Time, or Head vs Time**, depending on the subject. The images shown below are for a heat transfer problem in which the potential is temperature.

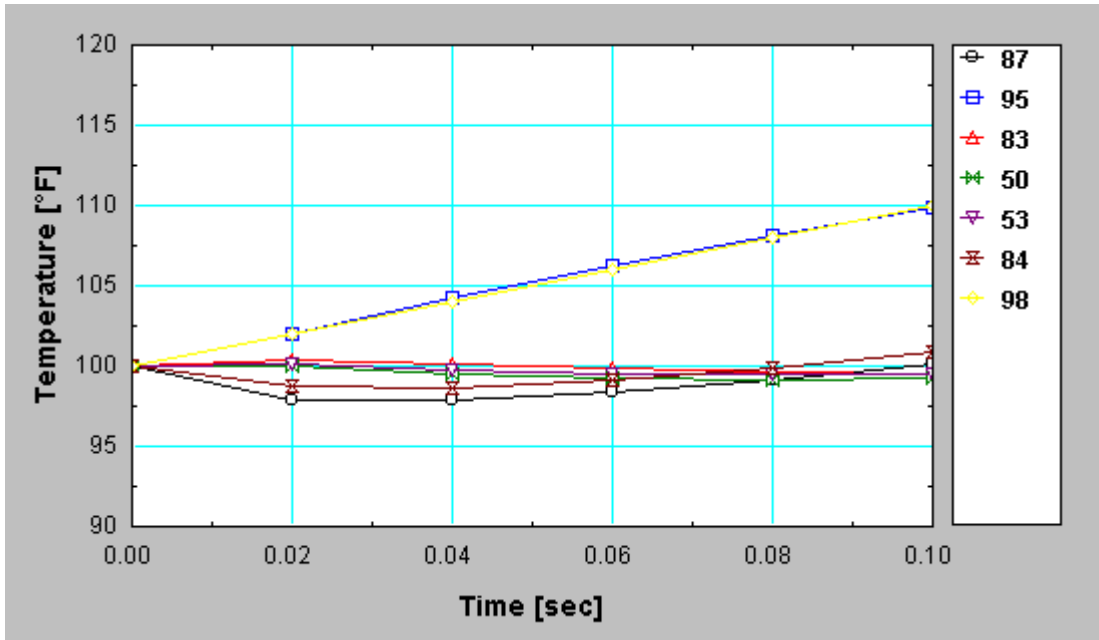
The following plot setup dialog window will appear in which 1 to 15 node numbers can be entered. (Recall that node numbers can be displayed with the **Show Node Numbers** command in the **Display** menu.) The simplest way to enter the node numbers is to select the nodes you wish to plot before selecting the Potentials vs Time command. The appearance of plot symbols, plot lines and a plot legend are controlled by the check boxes at the right.



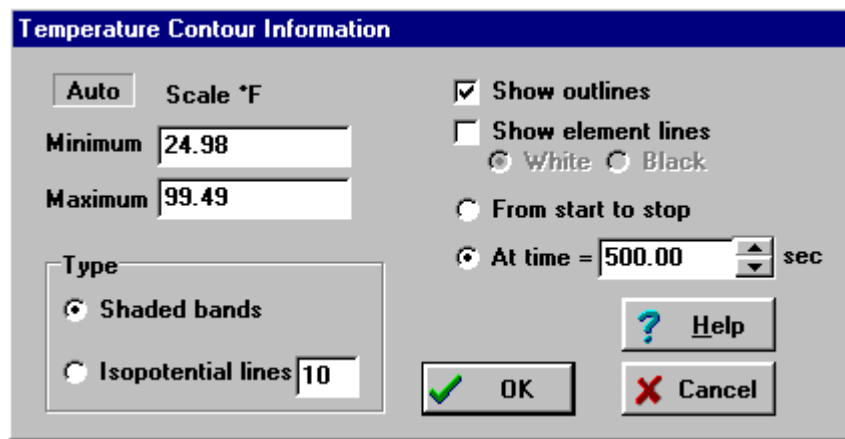
Attributes of the X and Y-axis scales can be viewed or changed by clicking on the appropriate tab at the bottom of the dialog window. The default values for time are the start and stop times used in the calculations. Click on the Temp. Scale tab to set the minimum and maximum temperatures for the potential ordinate. The scale may be set by the user or set automatically by clicking the mouse button in the User/Auto box. The Auto option fills in the minimum and maximum potential values for the nodes and time period you have selected. Changes can then be made to these temperature limits. The Display fields control the appearance of the axis numbers.



The plot, shown below, can be moved or resized. To move a plot, press and hold the mouse button down anywhere within the plot rectangle, while moving the cursor to a new location. To resize the plot, press and hold the mouse button down on the lower right corner of the plot. The cursor will be changed into a cross and the plot will resize as the mouse is moved. Double-clicking the mouse anywhere in the Plot Window will cause the plot setup dialog window to reappear so any of the plot attributes can be changed.



**Contours** produces a contour plot of the appropriate potential (Temperature, Voltage, Magnetic Potential, Stream Function, Pressure, Head) for the selected subject. The following dialog window will appear in which the minimum and maximum potentials must be entered. This may be done by the user or automatically by clicking the mouse button in the User/Auto box in preceding the scale. The Auto option will determine and display the minimum and maximum values.

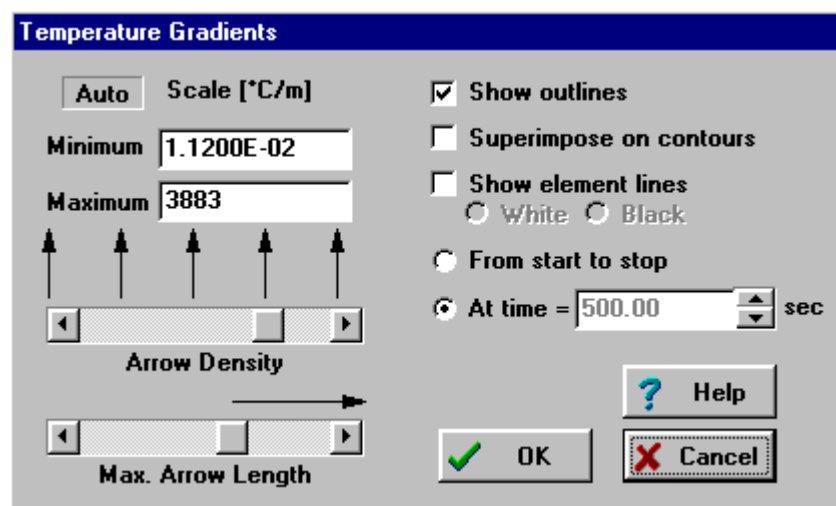


Shaded bands or isopotential lines can be selected by clicking the appropriate button at the bottom left of the dialog box. Shaded band contour plots display the variation in the potential using five to seven different colors. Isopotential line plots draw lines of constant potential. The number of lines must be between 2 and 100 and is specified by the user. In both types of contour plots, the value of the potential at the cursor position will be displayed in the upper left information window by pressing and holding the mouse button.

Element lines can be made to appear in either white or black by selecting Show element lines check box or by selecting Show Element Lines check box menu. The outline of the various materials will be shown if the Show Outlines check box is selected.

For transient problems, the contours may be plotted from the start to the stop times as a movie or at a selected time by clicking the appropriate button. The selected time is set with the arrow controls to the right of the time. Each click on the arrow will increment the time by one time step. The time is also shown in the Time box at the top - middle of the screen, just below the menu bar. Clicking the mouse button in the leftmost information window below the menu bar will cause the plot to be redrawn for the next time step.

**Gradients Window** will be titled appropriately for the selected subject. Examples are **Temperature Gradients, Current Density, Flux Density, Volumetric Flux**. In all cases, this window displays the temperature gradients (for heat transfer problems) or the appropriate flux density vector. The direction of the vector quantity is indicated by an arrow and the length of the arrow shaft is proportional to its magnitude. The magnitude of of the vector at the cursor position can be displayed in the upper right information window by pressing and holding the mouse button. The number of arrows and the maximum arrow length are specified in the Gradients dialog window, which appears as follows for heat transfer problems.



The minimum and maximum magnitudes can be set by the user or automatically by clicking in the User/Auto box. The Auto option will determine and display the minimum and maximum magnitudes. The outline of the various materials will be shown if an X appears in the Show Outlines check box. The X appears or disappears by clicking the mouse button within the box. Similarly, the element lines will be displayed is the Show Element Lines check box is selected. A third check box allows the heat fluxes to be

superimposed on the potential contours. The center of the arrow vectors are placed at equidistant locations throughout the distributed materials. The Arrow Density scroll bar controls the distance between the arrow centers. The length of the flux arrow corresponding to the maximum magnitude can be adjusted by the scroll bar. For transient problems, the temperature gradients may be plotted from the start to the stop times as a movie or at a selected time by clicking the appropriate button. The time for which the gradients are to be shown is set with the scroll bar. After the plot has been displayed, the gradients for the next time step can be automatically drawn by clicking in the center information window, just below the menu bar.

**Flows Window** changes its name depending on the selected subject. The possible choices are **Heat Flows, Current, Charge, Magnetic Flux, or Volumetric Flow**. In each case, the direction and magnitude of appropriate flow normal to any selected line will be displayed. This command will cause the screen to be redrawn with node positions hidden and all element lines shown. Click the mouse button on the line for which the flow is to be determined; an arrow will appear showing the direction of the flow normal to the line. The magnitude of the flow normal to the line appears in the information window at the top of the screen. The total flow from two or more connected lines may be found by selecting the lines in a clockwise or counter-clockwise manner. You may wish to group the lines to make the selection process simpler using the **Group** command (**Draw** menu). The total flow for all selected lines is also displayed in the information window. The sign convention is that the flow for the first-selected line is positive.

#### Note

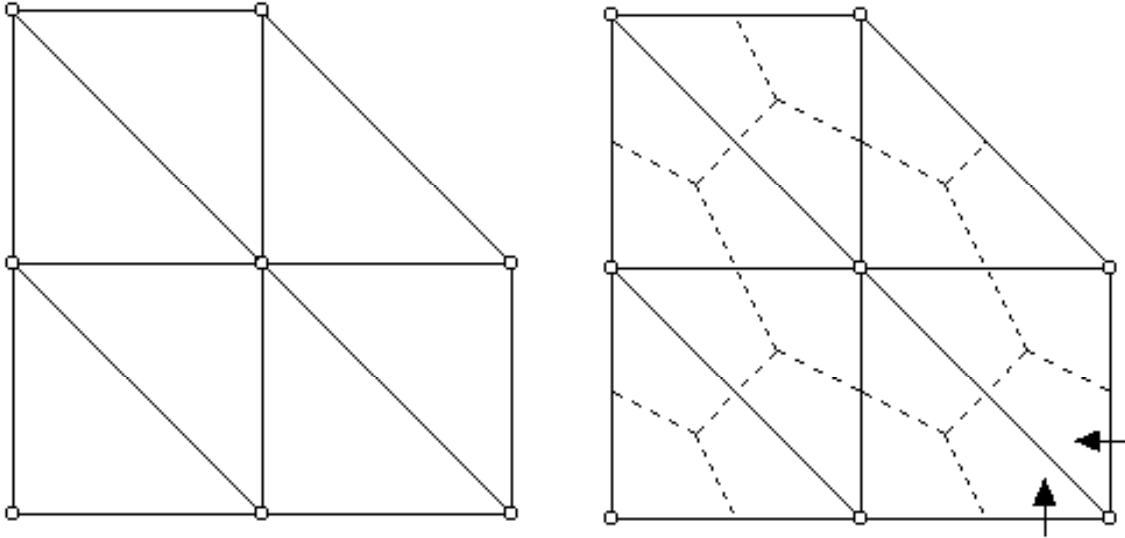
Because the finite-element method used in FEHT is not optimized for calculating gradients at the boundaries of elements, the flow estimate tends to be inaccurate for constant potential or unspecified boundaries. Convective boundaries (for Heat Transfer) provide accurate estimates since, in this case, the heat flow calculation is made using the convection equation and does not rely on the temperature gradient. A convective boundary can be made to appear as a constant temperature boundary by selecting a large convection coefficient and a fluid temperature equal to the known boundary temperature. An alternative solution for constant temperature or unspecified boundaries is to use the information in the **Nodal Balances** window, as described below.

**Nodal Balances** displays an estimate of the total flow of energy, current, mass, etc. (as appropriate to the subject) into a selected nodal system. The Nodal Balances menu item is available only after the calculations are completed.

The figure on the left shows a material outline discretized into triangular elements. The

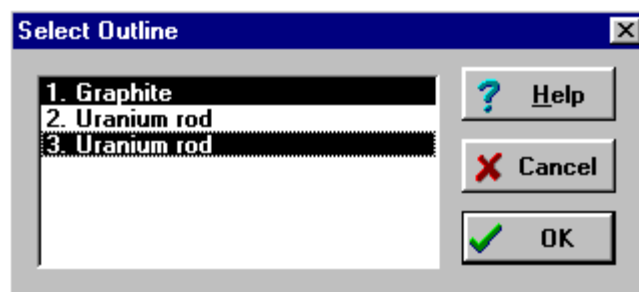


figure on the right superimposes the nodal system boundaries. The calculations in FEHT could be viewed as balances on each nodal system, as explained in Chapter 3. After clicking on a node, the total flow of energy, current, mass *into* that node from outside the solid material (or from outside the outline for which nodal balances calculations are done) will be displayed in the information window at the upper right.



The total flow into an interior node, such as the center node in the figure above, will always be zero. However, clicking on a node on a boundary, such as the node on the lower right will produce the flow into that node across the boundary as represented by the arrows. The total flow across a boundary can be determined by clicking, in succession, on the nodes which form the boundary. In a heat transfer problem, for example, selecting all of the nodes bounding a material will display the total energy into the material. The energy across a boundary can also be obtained with the Heat Flows command (or its equivalent for other subjects). However, the Nodal Balances command provides more accurate results when the boundary has specified potential or is unspecified. Energy balances will always check when using nodal balances

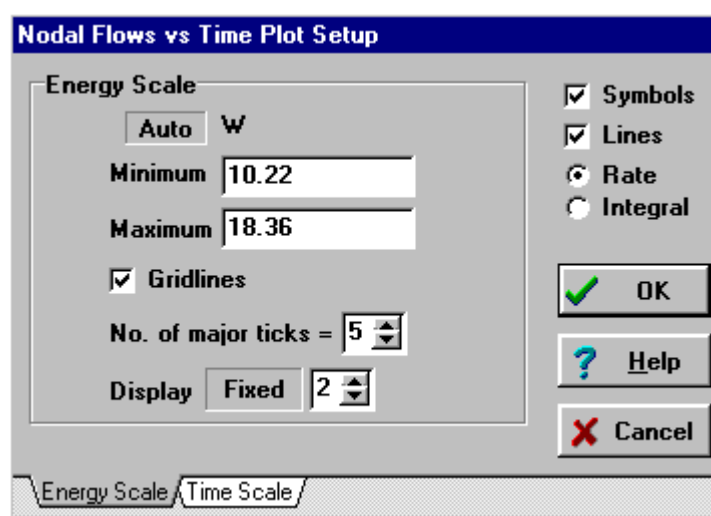
When a steady-state problem has more than one outline, the Nodal Balances command will first display the following dialog window in which the outlines for the nodal balances are chosen. One or more outlines can be chosen by clicking on the outline name. Clicking a second time deselects that outline. The nodal system boundaries will be drawn only for those outlines which are selected. The flow of energy, current, mass, etc. between two materials can be found by selecting one of the two material outlines. Nodal balances on the boundary nodes between the two materials will give the flow across the boundary nodes into the selected material outline.



The nodal balances for each node and outline are stored for steady-state problems. For transient problems, the storage of the balances at each time step could require excessive storage. Consequently, the nodal balances are done for one set of selected outlines if the Do Nodal Balances check box in the transient Calculate dialog window is enabled.

**Flow vs Time (Energy Flow vs Time Window, Current vs Time, Volumetric Flow vs Time)** produces a plot of the total flow into one or more selected nodes as a function of time. This command is applicable only for transient problems and only if the Do Nodal Balances control in the Transient Solution Setup dialog window is selected (displayed with an X in the box). To use this command, first select the node or nodes for which the energy flow is to be plotted. Then select this command to bring up the following dialog window in which the scales for the ordinate and abscissa can be specified.

The plot controls are identical to those for the Potentials vs Time plot, with the exception of the Rate and Integral radio buttons. If the Integral radio button is selected, the integral of the flow quantity will be plotted, rather than the calculated rate value.



Clicking the OK button will dismiss the dialog window and display the plot. The plot can be relocated and resized as explained for the **Potentials vs Time** window. Double-clicking

the mouse button on the Flows vs Time window will bring up the setup dialog window so that the appearance of the plot can be changed.

**Tabular Output** will display a table of calculated results. The table, similar in appearance to a spreadsheet, provides a row for each node. For steady-state problems, each row lists the node number, the node coordinates, the node potential (e.g., temperature) and the nodal balance (NB). For transient problems, the node potentials and nodal balances at each time step are displayed in successive columns with a # symbol preceding each step number.

Specified potential values appear in bold font.

The format of the data in the columns is initially set to a default setting. If you wish to change the format, click on the header cell of the column you wish to change. A small dialog window will appear in which you can change the display format. For transient problems, the display format you select for the potentials or balances at one time step can be applied to all time steps by checking the box below the display format setting.

Buttons at the top of the window allow you to print or copy the selected section of the table. The buttons are disabled until a selection is made. To select cells, click the mouse in the upper left corner of the desired selection. Press the Shift key down and then click in the lower right cell, scrolling if necessary. Selected cells will display in inverse video. Click the Select All button at the top of the table to select all cells in the table. If the data are copied to clipboard by clicking the Copy button, they can be pasted into any spreadsheet or word processor application.

**Input Data Summary** produces a summary report of the input information. The summary provides the horizontal (X or R) and vertical (Y or Z) coordinates of each specified node having a specified potential, a list of the boundary conditions, and the property data for each material. The information in this window can be copied with the Copy to Clipboard command.

## Help Menu

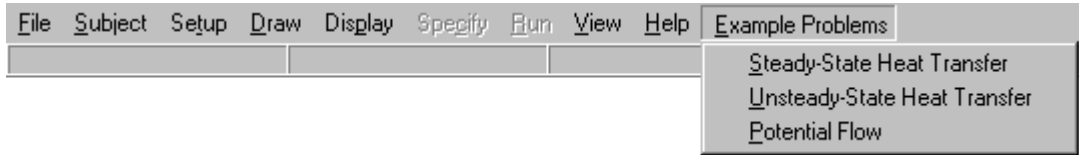


**Help Index** will activate the Help processor which provides specific information on the use of FEHT. The Help processor will open to the FEHT Information index which lists the subjects for which help is available. Clicking on the subject opens the Help window to information for that subject. Help can also be accessed in two other ways. Pressing the F1 key while in FEHT will bring up help information specific to the window or dialog which is foremost. Pressing the Shift-F1 keys will cause the cursor to turn into the Help cursor. Select a menu item with the Help cursor to obtain specific explanatory information for that menu command.

**Using Help** shows information provided by the Windows Help processor on how to use the features in the Help program.

**About FEHT** will bring up the FEHT header window. This window indicates the version of your FEHT program and your registration number. This information will be needed in any correspondence with F-Chart Software.

**f-Chart Web Site** will start your default browser and connect you to the f-Chart web site. You may wish to visit this web site periodically to see if updates have been posted to the FEHT program.

*Examples Menu*

The Example Problems menu is a user-defined menu that is designed to allow easy access to FEHT files. It can have any name. It is created with an ASCII file identified with the filename extension (.TXB). When FEHT reads a textbook index file, it creates the menu at the far right of the menu bar, as shown above. The format of the textbook index file is quite simple. The menu shown above was created with the following textbook index file.

## Example Problems

1

FEHT

Finite Element Analysis Program

F-Chart Software

Reserved

&gt;Steady-State Heat Transfer

Cinder block brick|examples\cinder.FET|no.HLP|no.bmp

Fuel rods|examples\FuelRod.FET|no.Hlp|no.bmp

Radiation boundary|examples\Holman38.fet|no.hlp|no.bmp

&gt;Unsteady-State Heat Transfer

Transient|examples\Transient.FET|no.Hlp|no.bmp

Freezing water|examples\Freezing.FET|no.Hlp|no.bmp

Step change in wall|examples\Plaster.fet|no.hlp|no.bmp

&gt;Potential Flow

Flow in a venturi|examples\Venturi.fet|no.hlp|no.bmp

The first line in the file is the menu title. This title is the name of the menu which will appear in the menu bar to the right of the Help menu. The following line is a version number used internally by FEHT. The following three lines provide information about the problem set. This information will be displayed whenever any menu item is selected from this menu. A fourth line containing the word "reserved" is provided for possible future use. The following lines contain a menu item name (preceded by an identifying > character) and then one or more problem descriptions. The menu item name is the name that will appear in the menu list.

Each problem description line contains four pieces of information, separated by a | character. The first item is a descriptive name for the problem, which may be up to 128 characters. The second item is the filename for the FEHT program file. This filename may be partially or fully

qualified with directory information. However, in most cases, directory information is not necessary and it should not be included. The third item is an optional help file which is to be associated with this file. The help file can be an ASCII text file, a Windows HLP file produced by a Help formatting program, or an HTML file readable by a browser program. By convention, the help files have a .HLP or .HTM filename extension. If no help file is available, enter NO.HLP for this field. The final item is the filename for a figure associated with the problem.

The .TXB file should be placed in the same location (i.e., floppy disk, subdirectory, or folder) with all of the referenced EES program and help files. When the user selects a command from the this menu, a dialog window will appear showing a list of the descriptive names of the problems for that menu item. The user can then select a name from the list and the file associated with that problem will be opened.



---

# *The Finite Element Method*

---

The finite-element method, like the finite-difference method, is a numerical scheme to approximate the solution to a partial differential equation such as that governing the conduction of heat in a solid material. The purpose of this chapter is to present a brief summary of the finite-element method used in FEHT to provide a better understanding of the program. More detailed explanations can be found in references 1-3.

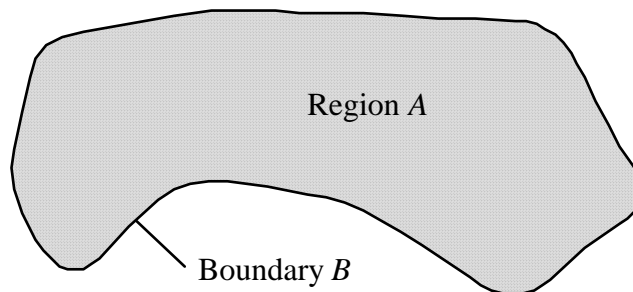
The governing partial differential equation for two-dimensional conduction heat transfer can be written in Cartesian coordinates as: (See references 4-8 for the analogous relations in other subject areas.)

$$\frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \dot{q} - \rho c \frac{\partial T}{\partial t} = 0 \quad (1)$$

where

- T        is temperature
- x        is the horizontal position coordinate
- y        is the vertical position coordinate; m
- k        is the material thermal conductivity
- c        is the material specific heat
- $\dot{q}$       is the internal generation rate per unit volume
- $\rho$       is the material density
- t        is time

The objective is to satisfy (1) in a specified area *A* enclosed by boundary *B* subject to specified boundary conditions (convection, specified heat flux inflow and specified temperature) and an initial condition.





In the finite-element method, the partial differential equation is transformed into an integral expression. Numerical approximation of the integral then provides an approximate solution. The transformation is made by multiplying (1) by an arbitrary function  $f(x,y)$  and integrating over area  $A$ .

$$\iint_A f(x,y) \left[ \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \dot{q} + \rho c \frac{\partial T}{\partial t} \right] dx dy = 0 \quad (2)$$

The exact solution,  $T(x,y,t)$ , would cause the bracketed term to be zero for all  $x$ ,  $y$ , and  $t$ . An approximate solution will be sought that will cause the integral to be zero at selected points within  $A$ , although this solution may not necessarily satisfy the equation at all points within  $A$ .

The solution to (2) begins by integrating the first two terms by parts and rearranging to obtain

$$\frac{d}{dt} \iint_A f \rho c T dx dy + \iint_A \left[ \frac{\partial f}{\partial x} k \frac{\partial T}{\partial x} + \frac{\partial f}{\partial y} k \frac{\partial T}{\partial y} \right] dx dy = \iint_A f \dot{q} dx dy + \int_B f q_n'' ds \quad (3)$$

where  $q_n''$  is the local heat flux into area  $A$  along the boundary  $B$ .

$B$ , the boundary line over which the last integral in (3) is evaluated can be broken into

$$B = B_c + B_s + B_a + B_t \quad (4)$$

where

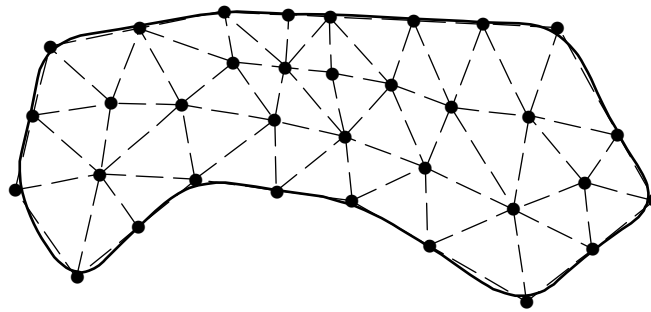
- $B_c$  is the convective portion of the boundary for which  $q_n'' = h (T_\infty - T)$
- $B_s$  is the portion of the boundary with specified heat flux so that  $q_n'' = q_s''$
- $B_a$  is the adiabatic portion of the boundary for which  $q_n'' = 0$
- $B_t$  is the specified temperature portion of the boundary

Expanding the last integral in (3), the equation can be rewritten to explicitly consider the boundary conditions.

$$\begin{aligned} \frac{d}{dt} \iint_A f \rho c T dx dy + \iint_A \left[ \frac{\partial f}{\partial x} k \frac{\partial T}{\partial x} + \frac{\partial f}{\partial y} k \frac{\partial T}{\partial y} \right] dx dy = \\ \iint_A f \dot{q} dx dy + \int_{B_c} f h (T_\infty - T) dS + \int_{B_s} f q_s'' ds + \int_{B_t} f q_n'' ds \end{aligned} \quad (5)$$

The function  $f$  is still to be chosen. At this point,  $f$  will be chosen to have a value of zero along a boundary of specified temperature so that the last integral in (5) is identically zero. No approximations have been made thus far.

In the finite-element approximation used in FEHT, nodes connected by straight-line segments are used to approximate boundary  $B$ . The actual boundary is replaced by a polygon approximation. Additional nodes also connected by straight-line segments, along with the boundary nodes, are used to discretize region  $A$  into a set of triangular elements. The temperature within each triangular element is assumed to vary linearly between the temperatures at the three vertices (nodal points) for that element. Since the temperature varies linearly within an element, isotherms will be parallel, straight lines and the temperature gradient will be uniform within each element. This continuous, piecewise-linear, temperature profile is an approximation to the actual temperature profile. The problem then is to determine the temperature at each nodal point.



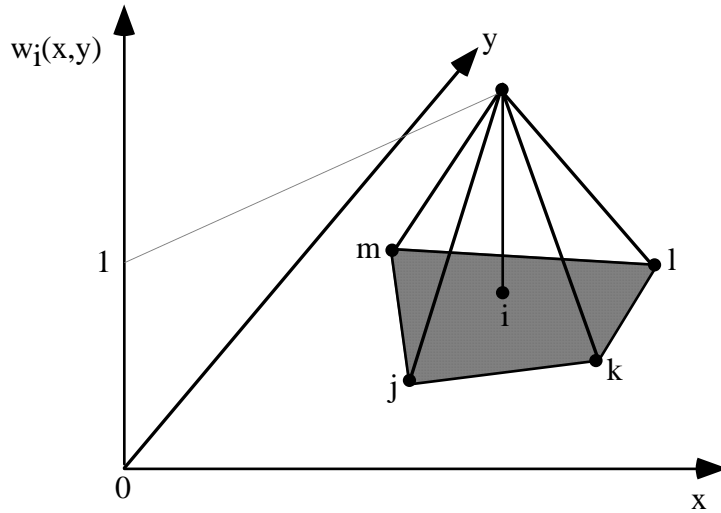
The solution  $T(x,y,t)$  will thus be approximated as a vector of  $n$  temperatures where  $n$  is the number of triangle vertices (nodes). Since the temperature at point  $x,y$  within any triangular element is assumed to vary linearly between the three vertex temperatures, the temperature at any point within a triangular element can be represented by

$$T(x,y,t) \approx w_1(x,y) T_1(t) + w_2(x,y) T_2(t) + \dots + w_n(x,y) T_n(t) = \mathbf{w}^T(x,y) \mathbf{t}(t) \quad (6)$$

where

$T_i(t)$  is the temperature of triangle vertex  $i$  at time  $t$

$w_i(x,y)$  is a 'weighting' function for node  $i$  that is 1 at node  $i$ , 0 at all other nodes, and varies linearly from 1 at node  $i$  to 0 along the element boundaries opposite to node  $i$  as shown in the figure below.



The matrix (bold) notation appearing in (6) and following equations is to be interpreted as

$$\mathbf{w}^T = [w_1 \quad w_2 \quad \dots \quad w_n] \quad \mathbf{t} = \begin{bmatrix} T_1 \\ T_2 \\ \cdot \\ \cdot \\ T_n \end{bmatrix}$$

The weighting functions are known functions of  $x$  and  $y$  found for each element by solving three simultaneous algebraic equations of the form

$$T^{(e)} = C_1^{(e)} + C_2^{(e)} x + C_3^{(e)} y \tag{7}$$

for the  $C_1^{(e)}$ ,  $C_2^{(e)}$ , and  $C_3^{(e)}$  in terms of the temperatures at the element vertices. The weighting function for node  $i$  for  $x, y$  positions within element  $e$  composed of nodes  $i, j$ , and  $k$  is

$$w_i^{(e)}(x, y) = \frac{1}{(x_{ij}y_{jk} - x_{jk}y_{ij})} [x_j y_k - x_k y_j - x y_{jk} + x_{jk} y] \tag{8}$$

where  $x_{ij}$  is shorthand notation for  $x_j - x_i$ . The weighting functions in the assumed temperature distribution, (6), are known. The  $n$  nodal temperatures must still be determined.

The next step is to substitute the temperature distribution given by (6) into (5) to obtain

$$\frac{d}{dt} \iint_A f \rho c \mathbf{w}^T \mathbf{t} \, dx \, dy + \iint_A \left[ \frac{\partial f}{\partial x} k \frac{\partial \mathbf{w}^T}{\partial x} \mathbf{t} + \frac{\partial f}{\partial y} k \frac{\partial \mathbf{w}^T}{\partial y} \mathbf{t} \right] dx \, dy = \iint_A f \dot{q} \, dx \, dy + \int_{B_c} f h (T_\infty - \mathbf{w}^T \mathbf{t}) \, ds + \int_{B_s} f q_s'' \, ds \quad (9)$$

For a single  $f(x,y)$ , (9) will be a single ordinary differential equation involving the  $n$  unknown temperature functions  $T_1(t), T_2(t), \dots, T_n(t)$ . Equation (9) is still not solvable in that there are  $n$  unknown temperatures but only one equation. However,  $f(x,y)$  can be any function of  $x$  and  $y$  which satisfies the constraint that  $f(x,y) = 0$  along a boundary of specified temperature. There are an infinite number of functions which meet this constraint. We will choose  $n$  such independent  $f$  functions to be used in (9). The functions can be represented in matrix notation.

$$\mathbf{f}(x,y) = \begin{bmatrix} f_1(x,y) \\ f_2(x,y) \\ \cdot \\ \cdot \\ \cdot \\ f_n(x,y) \end{bmatrix}$$

The functions,  $\mathbf{f}(x,y)$ , are arbitrary. Different choices for these functions will, in general, result in different approximate solutions. The Galerkin [2-3] method is to set  $f_i(x,y)$  to be identical to  $w_i(x,y)$ . With this definition of  $\mathbf{f}$ , rearrangement of (9) results in

$$\left[ \iint_A \mathbf{w} \rho c \mathbf{w}^T \, dx \, dy \right] \frac{d\mathbf{t}}{dt} + \left[ \iint_A \left[ \frac{\partial \mathbf{w}}{\partial x} k \frac{\partial \mathbf{w}^T}{\partial x} + \frac{\partial \mathbf{w}}{\partial y} k \frac{\partial \mathbf{w}^T}{\partial y} \right] dx \, dy \right] \mathbf{t} + \left[ \int_{B_c} \mathbf{w} h \mathbf{w}^T \, ds \right] \mathbf{t} = \iint_A \mathbf{w} \dot{q} \, dx \, dy + \int_{B_c} \mathbf{w} h T_\infty \, ds + \int_{B_s} \mathbf{w} q_s'' \, ds \quad (10)$$

The integrals in (10) may be evaluated by summing over all elements. For example, for the area integrals

$$\iint_A [\dots] dx dy = \sum_e \iint_{A^{(e)}} [\dots] dx dy \quad (11)$$

The integration for each individual element is done analytically since the form of  $\mathbf{w}$  for any element is the simple linear function given in (8). Integration of (10) results in a system of  $m$  ordinary differential equations where  $m$  is equal to  $n$  minus the number of specified temperature nodes. These equations are solved by the Cholesky decomposition to obtain the temperatures as functions of time. For steady-state problems, the time derivative term in (10) is set to zero to result in a system of  $m$  algebraic equations which are solved to obtain the  $m$  unknown steady-state temperatures.

The system of differential equations resulting from the approximations described above may be written using matrix notation as

$$\mathbf{C} \dot{\mathbf{t}} + (\mathbf{K} + \mathbf{H}) \mathbf{t} = \mathbf{g} + \mathbf{h} + \mathbf{q} \quad (12)$$

where

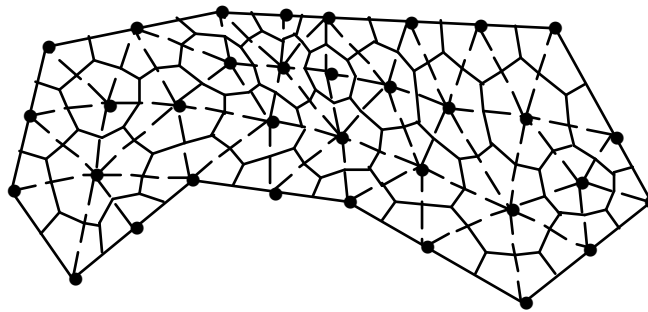
- $\mathbf{t}$  = temperature vector (contains the temperatures at each nodal point)
- $\dot{\mathbf{t}}$  = time derivative of  $\mathbf{t}$  (contains the time derivatives of each nodal temperature)
- $\mathbf{C}$  = capacitance matrix (involves density and specific heat of the material)
- $\mathbf{K}$  = conduction matrix (involves thermal conductivity of the material)
- $\mathbf{H}$  = convection matrix (involves the specified heat-transfer coefficient on the boundary)
- $\mathbf{g}$  = generation vector (involves the specified energy generation per unit volume of the material)
- $\mathbf{h}$  = convection vector (involves the specified heat-transfer coefficient and ambient temperature on the boundary)
- $\mathbf{q}$  = heat-flow vector (involves the specified heat flux on the boundary)

In developing (12) it is assumed that at any instant of time, the density, thermal conductivity and specific heat are uniform within each triangular element, although they can differ for each element. The values of the convection coefficients, fluid temperatures and boundary heat flux are taken to be uniform along any given boundary segment but can be different for each boundary segment. In determining the numerical values in  $\mathbf{C}$ ,  $\mathbf{K}$ ,  $\mathbf{H}$ ,  $\mathbf{g}$ ,  $\mathbf{h}$  and  $\mathbf{q}$ , FEHT calculates the contribution from each element and boundary segment and adds them. An element formed by connecting nodes  $i$ ,  $j$  and  $k$  will add contributions to rows  $i$ ,  $j$  and  $k$  in columns  $i$ ,  $j$  and  $k$  in  $\mathbf{C}$  and  $\mathbf{K}$  and to rows  $i$ ,  $j$  and  $k$  in  $\mathbf{g}$ . A boundary segment between nodes  $i$  and  $j$  will add contributions to rows  $i$  and  $j$  in columns  $i$  and  $j$  in  $\mathbf{H}$  and to rows  $i$  and  $j$  in  $\mathbf{h}$  and  $\mathbf{q}$ .

If there were no nodes having specified temperatures, (12) could now be solved for the nodal temperatures in  $\mathbf{t}$ . If there are nodes whose temperatures are specified (*e.g.*, nodes along a boundary whose temperature is specified), modifications to (12) must be made prior to solving to ensure that  $\mathbf{t}$  will contain the correct values of the specified nodal temperatures. When running FEHT, the number of unknown nodal temperatures is reported.

For transient problems, FEHT can approximate the solution with either an Euler or a Crank-Nicolson step-by-step solution. Since the finite-element capacitance matrix  $\mathbf{C}$  is not diagonal (unlike the usual finite-difference formulation), a finite-element Euler solution will not be explicit. Therefore there is no computational penalty for using the more-accurate Crank-Nicolson method. Crank-Nicolson is the default setting in FEHT for transient problems. For steady problems  $\dot{\mathbf{T}} = 0$  and (12) reduces to a system of algebraic equations. FEHT uses Cholesky-square-root decomposition to obtain a direct (*i.e.*, non-iterative) solution of algebraic equations (including the solutions of the algebraic equations required for each step of a transient problem).

For the purpose of understanding some of the features of FEHT, it is convenient to also think in terms of a set of nodal systems in addition to the set of triangular elements. These nodal systems are constructed by connecting the centroid of each triangular element to the midpoint of each of the three sides of the triangular element.



The entries in the vectors in the finite-element equation (12) may be shown to have energy significance for these nodal systems. Rearranging of (12) produces

$$\mathbf{C} \dot{\mathbf{t}} + \mathbf{K} \mathbf{t} = \mathbf{g} + (\mathbf{h} - \mathbf{H} \mathbf{t}) + \mathbf{q} \tag{13}$$

where

$\mathbf{C} \dot{\mathbf{t}}$  = vector containing the rate of energy storage in each nodal system

$\mathbf{K} \mathbf{t}$  = vector containing the total rate of conduction out of each nodal system to surrounding nodal systems

$\mathbf{g}$  = vector containing the energy generation rate within each nodal system

$\mathbf{h} - \mathbf{H} \mathbf{t} =$  vector containing the specified convection into each nodal system  
 $\mathbf{q} =$  vector containing the specified heat flow into each nodal system

The finite-element equations represented by (13) may be viewed as being derived from an energy balance on each nodal system. The right-hand side of (13) is the sum of the energy generated within each element  $\mathbf{g}$  plus the sum of the specified convection  $\mathbf{h} - \mathbf{H} \mathbf{t}$  and the specified heat flow  $\mathbf{q}$  into each element. This energy inflow to each element must either be conducted out to surrounding elements  $\mathbf{K} \mathbf{t}$  or be stored within the element  $\mathbf{C} \dot{\mathbf{t}}$ .

To understand how to calculate heat flows across the boundary of the region, it is convenient to rearrange (13) by moving  $\mathbf{g}$  to the other side of the equation to give

$$\mathbf{C} \dot{\mathbf{T}} + \mathbf{K} \mathbf{T} - \mathbf{g} = (\mathbf{h} - \mathbf{H} \mathbf{T}) + \mathbf{q} \quad (14)$$

The left-hand side of (14) now involves energy storage, conduction and generation which all occur within the region. The right-hand side of (14) involves the energy inflows at the boundary due to specified convection heat flows and specified heat fluxes. A vector  $\mathbf{q}_o$ , whose entries are the energy inflows to each nodal system *from outside the region into each of the nodal systems*, may be defined such that

$$\mathbf{q}_o = \mathbf{C} \dot{\mathbf{t}} + \mathbf{K} \mathbf{t} - \mathbf{g} \quad (15)$$

and

$$\mathbf{q}_o = (\mathbf{h} - \mathbf{H} \mathbf{t}) + \mathbf{q} \quad (16)$$

If there are no specified nodal temperatures, either (15) or (16) may be used to calculate the heat flow,  $\mathbf{q}_o$ . For any interior node or for any node on an adiabatic boundary, the corresponding entry in  $\mathbf{q}_o$  will be 0 because there is no energy flow into these nodal systems from outside the region. For a specified-temperature node there will usually be an unspecified energy inflow (or outflow), in addition to specified convection and specified heat flux, that is required to maintain the node at its specified temperature. In this case, (16) cannot be used to compute the corresponding entry in  $\mathbf{q}_o$  because the unspecified energy inflow is not included with the specified convection and heat-flux inflows. However, (15) can still be used to calculate  $\mathbf{q}_o$  because the total inflow plus the generation must still be equal to the storage plus the conduction out of the element (into surrounding elements in the region). If you select **Heat Flows** from the **View** menu and then click on a boundary segment, the heat flow reported is based on (16). If you select **Nodal Balances** from the **View** menu and then click on a node, the boundary heat inflow that is reported is calculated using (15). Energy balances will always check when using nodal balances (15).

**Guidelines for Choosing a Time Step for Transient Problems**

For conduction transients using either the Euler or the Crank-Nicolson method for stepping ahead in time, there is a *critical time step*. For time steps greater than the critical time step, the solution will contain numerically induced oscillations. These oscillations will be unstable in an Euler solution but will be stable in a Crank-Nicolson solution. Since the Crank-Nicolson oscillations are always stable, the oscillations are most noticeable at the start of a transient and diminish as time increases and the correct steady-state solution is obtained. A Crank-Nicolson solution with some oscillations is often considered to be satisfactory.

At the start of a transient solution, *FEHT* makes an *estimate* of the critical time step and reports the ratio of the user's time step to the *estimated* critical time step. If the estimate of the time step were exact, a ratio greater than 1.0 would mean an Euler solution would be unstable and a Crank-Nicolson solution would contain stable oscillations. However, the estimate of the critical time step is always less than or equal to the exact critical time step. This means that the ratio reported by *FEHT* will usually be greater than 1.0 when an Euler solution becomes unstable and oscillations appear in a Crank-Nicolson solution.

Since a Crank-Nicolson solution is more accurate than an Euler solution (with the same time step), is always stable and (unlike finite differences) requires no more computational effort at each time step, the Crank-Nicolson solution is preferred for finite elements.

When running a transient problem for the first time, you often have no idea of a reasonable time step to use. In this case, specify any time step you want and specify a stop time that is about ten times the time step. Run the transient and note the value of the ratio of your time step to the estimated critical time step. Select a few nodes and examine a temperature-time plot for these nodes. Nodes near boundaries where step changes in boundary conditions are imposed will exhibit the most severe oscillations. The plot should give you an idea of the severity of the oscillations and an idea of whether the stop time is close to steady state or not. Armed with this additional knowledge, run the transient again. Choose a time step such that the ratio of your time step to the estimated critical time step is about 5 or 10. Adjust the stop time if the previous stop time seemed to be too short or too long but you may still want to take only a few steps in the solution. Run the solution again and examine the new temperature-time plot. Readjust the time step and stop time and run again as needed to obtain an acceptable solution.

There are two factors that will reduce the estimated critical time step (and therefore the time step for an acceptable solution). Smaller elements will reduce the critical time step. One mesh reduction will decrease the critical time step by about a factor of four (problem dependent). Increasing the heat-transfer coefficient will eventually reduce the critical time step. Therefore, modeling a step change in boundary temperature by a step in fluid temperature with a large heat-transfer coefficient is not recommended.



The initial condition affects the severity of the oscillations. One way to approach a problem that has severe oscillations when using a “large” time step is to reduce the time step for a few steps and then *continue* the transient with the “large” time step. For example, start the transient with a time step that is one-tenth of the “large” time step and take ten steps. Then *continue* the solution using the “large” time step. The initial condition of the continued solution will produce less severe oscillations in the continued solution than if the “large” time step had been used for the entire transient.

Factors that have no bearing on the critical time step are: fluid temperature, generation and specified heat flux.

### References

1. Myers, G.E.: Analytical Methods in Conduction Heat Transfer-Second Edition, AMCHT Publications, Madison, WI, 1998, Chapter 9
2. Dhatt, G. and Touzot, G., The Finite Element Method Displayed, Wiley-Interscience, 1984.
3. Lapidus, L. and Pinder, G.F., Numerical Solution of Partial Differential Equations in Science and Engineering, John Wiley, 1982.
4. Cheng, D.K., Field and Wave Electromagnetics, Addison-Wesley, 1985.
5. Stremmer, F.G. et al., “Numerical Solutions and Mapping of Electrostatic Fields Using the Apple Macintosh Computer,” IEEE Trans. on Educ., February, 1990.
6. Pennes, H.H., “Analysis of Tissue and Arteroid Blood Temperatures in the Resting Human Forearm”, *J. Appl. Physiology*, Vol. 1, pp. 93-122, 1948.
7. Bowman, H.F., Cravalho, E.C., and Woods, M., “Theory, Measurement, and Application of Thermal Properties of Bio Materials”, *Annual Review Biophys, BioEng*, 4, pp. 43-80, 1975.
8. Fox, R.W. and McDonald, A.T., *Introduction to Fluid Mechanics*, Wiley, 3rd Edition, 1985.

---

## *Hints for using FEHT*

---

Listed below are a number of suggestions which simplify the use of FEHT. Some of these suggestions are described in the manual, but they are easily missed.

1. It is best to set the boundary conditions before beginning the discretization process. Discretizing necessarily breaks the boundary lines into smaller pieces so that more selections are needed to set the boundary conditions after discretization.
2. The **Group** command (**Draw** menu) simplifies the selection process for lines and nodes. It is helpful to group all boundary lines or nodes which have the boundary or initial conditions. Once they are grouped, all of the selected items in the group will be selected if you click on any one. The Group command is in effect only for the window in which it was created. Press the Ctrl key to override the group selection.
2. The **Zoom** mode enlarges a chosen section of the screen and thereby facilitates the placement or selection of nodes or element lines in tight places. When constructing a new outline or element lines, you may wish to scroll a hidden part of the screen into view. You can't click on the scroll bars because that would interrupt the drawing process. However, you can use the arrow keys to scroll.
3. To set the cursor at a specific X,Y position during drawing operations, it is easiest to first move to the correct X value. Then hold down the Shift key to lock the X position of the cursor while you move to the correct Y value. The Ctrl key locks the Y cursor position. If you wish to exactly specify the coordinates of a node, double-click the mouse on the node in the Input Window. A dialog window will appear in which the coordinates of the nodes can be entered
4. Some of the patterns used to identify materials visually interfere with the construction of element lines or obscure the template which is being traced. Use the **Hide Patterns** command to temporarily hide the patterns.
5. Use the **Save** command often as you prepare the problem definition. You should always save your work before using the **Reduce Mesh** or **Delete** commands (**Draw** menu).
6. To prepare the problem definition of systems with complicated shapes, it may be

advantageous to construct the drawing with a drawing programs or scan it in. The drawing can then be copied to the clipboard with the application's **Copy** command. After this is done, the drawing can be imported into FEHT with the **Paste From Clipboard** command (**File** menu) and moved or resized as needed. Trace over the drawing while using the **Outline** command (**Draw** menu) and then use **Hide Template** to remove the template from view.

7. The **Heat Flows** command provides an estimate of the heat flow across any line you select. Because the finite-element method used in FEHT is not optimized for calculating temperature gradients at element boundaries, the estimate tends to be inaccurate for constant temperature or unspecified boundaries. Specified heat flux boundaries are, of course, not a problem since the heat flow is known. Convective boundaries also provide accurate estimates since, in this case, the heat flow calculation is made using the convection equation and does not rely on the temperature gradient. A convective boundary can be made to appear as a constant temperature boundary by selecting a large convection coefficient and a fluid temperature equal to the known boundary temperature. Alternatively, the **Nodal Balances** window provides accurate estimates of the energy flows across any material boundary.
8. The Center Drawing and Size to file screen options of the **Scale and Size** command are handy when switching between machines with differing screen sizes.
9. The **Continue** command can be used to allow variable time step sizes in a transient calculation. Improved accuracy will result when smaller time steps are used during periods of rapid temperature change.
10. See the section entitled Guidelines for Choosing a Time Step in Chapter 3 for optimizing your specification of the time step used in transient problems.

---

## *Program Information*

---

FEHT limitations regarding problem size are as follows: An error message will be issued if any of these limitations are exceeded.

Maximum number of nodes	12000 (commercial version)
Maximum number of line segments	available memory
Maximum number of material outlines	52
Maximum number of triangular elements	available memory

FEHT will internally renumber nodes so as to minimize the bandwidth of the problem using the algorithm of Collins.

Collins, R.J. "Bandwidth Reduction by Automatic Renumbering," International Journal for Numerical Methods in Engineering, Vol. 6, pp. 345-356, (1973).

The calculation section of FEHT has been adapted from FEM2D, a FORTRAN program developed by Glen Myers, Dept. of Mechanical Engineering, University of Wisconsin - Madison.

FEHT was developed with **Borland's Delphi**.