Lawrence Berkeley National Laboratory

Title

THERM 2.0: a PC Program for Analyzing Two-Dimensional Heat Transfer through Building Products

Author

Windows and Daylighting Group

Publication Date

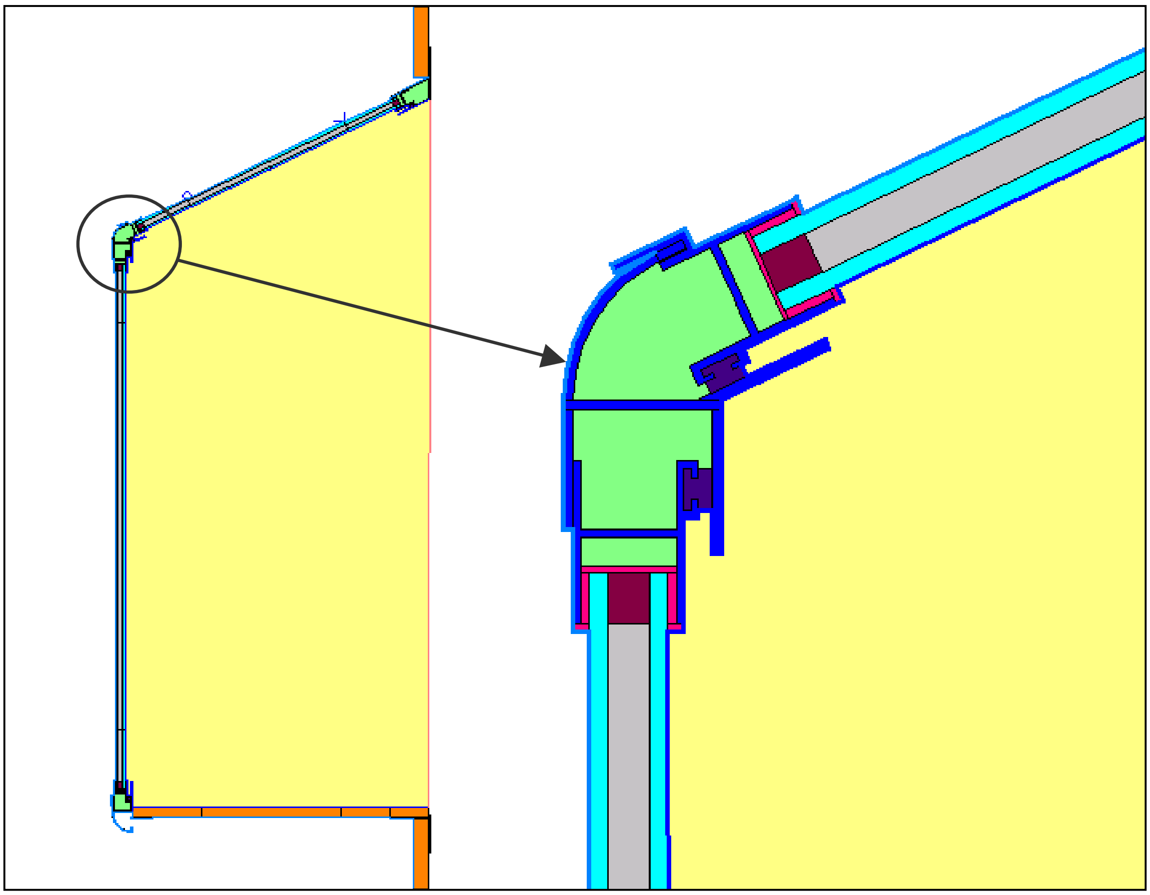
1997-12-08

LBL 37371 Rev. 2
June 1998

Program Description

A PC Program

THERM 2.0



for Analyzing Two-Dimensional Heat Transfer Through Building Products

Windows and Daylighting Group
Building Technologies Department
Environmental Energy Technologies Division
Ernest Orlando Lawrence Berkeley National Laboratory
Berkeley CA 94720 USA

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technology, State and Community Programs, Office of Building Systems of the U.S. Department of Energy under Contract No. DE-AC03-76SF00098.

LBL-37371 Rev. 2
TA-338

THERM 2.0: Program Description

A PC Program for Analyzing the Two-Dimensional Heat Transfer Through Building Products

Elizabeth Finlayson, Robin Mitchell, and Dariush Arasteh
Windows and Daylighting Group
Building Technologies Department
Environmental Energy Technologies Division
Lawrence Berkeley National Laboratory
Berkeley, California 94720
http://windows.lbl.gov/software/software.html

Charlie Huizenga
Center for Environmental Design Research
University of California
Berkeley, California

Dragan Curcija
Department of Mechanical Engineering
University of Massachusetts
Amherst, Massachusetts

June 1998

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technology, State and Community Programs, Office of Building Systems of the U.S. Department of Energy under Contract No. DE-AC03-76SF00098.

Table of Contents

1. INTRODUCTION

1.1. Overview

THERM is a state-of-the-art, Microsoft Windows™-based computer program developed at Lawrence Berkeley National Laboratory (LBNL) for use by building component manufacturers, engineers, educators, students, architects, and others interested in heat transfer. Using THERM, you can model two-dimensional heat-transfer effects in building components such as windows, walls, foundations, roofs, and doors; appliances; and other products where thermal bridges are of concern. THERM's heat-transfer analysis allows you to evaluate a product's energy efficiency and local temperature patterns, which may relate directly to problems with condensation, moisture damage, and structural integrity.

THERM's two-dimensional conduction heat-transfer analysis is based on the finite-element method, which can model the complicated geometries of building products. The program's graphic interface allows you to draw cross sections of products or components to be analyzed. To create the cross sections, you can trace imported files in DXF or bitmap format, or input the geometry from known dimensions. Each cross section is represented by a combination of polygons. You define the material properties for each polygon and introduce the environmental conditions to which the component is exposed by defining the boundary conditions surrounding the cross section. Once the model is created, the remaining analysis (mesher and heat transfer) is automatic. You can view results from THERM in several forms, including U-factors, isotherms, heat-flux vectors, and local temperatures.

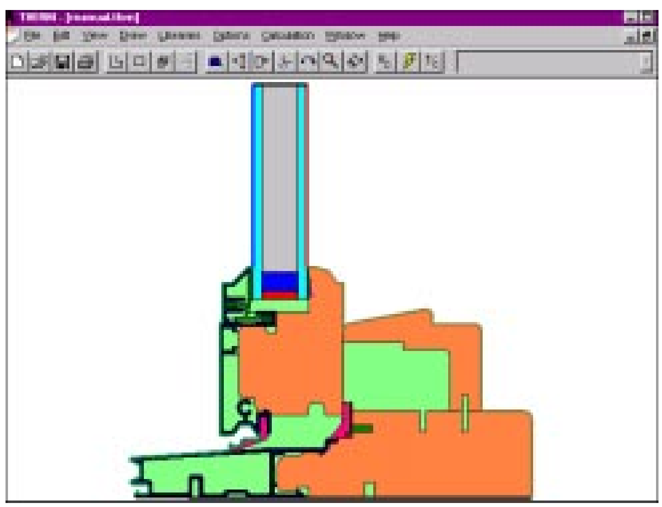


Figure 1-1. Sample THERM window cross section

This version of THERM includes several new technical and user interface features; the most significant is a radiation view-factor algorithm. This feature increases the accuracy of calculations in situations where you are analyzing non-planar surfaces that have different temperatures and exchange energy through radiation heat transfer. This heat-transfer mechanism is important in greenhouse windows, hollow cavities, and some aluminum frames.

THERM is a module of the WINDOW+5 program under development by LBNL. WINDOW+5 is the next generation of the WINDOW software series[1,2] and is being developed for the Microsoft Windows™ operating environment. THERM's results can be used with WINDOW's center-of-glass optical and thermal models to determine total window product U-factors and Solar Heat Gain Coefficients. These values can be used, in turn, with the RESFEN program, which calculates total annual energy requirements in typical residences throughout the United States.

Update information, future releases, and program information about THERM and the other software tools from the Windows and Daylighting Group at LBNL mentioned above can be found on the World Wide Web at URL: http://windows.lbl.gov. To obtain a copy of THERM, WINDOW, or RESFEN, fax or email your address and phone number to Software Request, Fax: 510 486-4089, email: PLRoss@lbl.gov. If you have questions or problems about using the program, email Thermhelp@lbl.gov.

1.2. New THERM Features

This version of THERM significantly upgrades the calculation and drawing features of the previous program.

1.2.1. New General Features

- 32-bit program compilation allows multitasking and greater detail in models,
- Report of simulation results can be printed,
- THERM files can be exported to a format that WINDOW 4.1 can read,
- Temperature is displayed at cursor,
- Calculation manager can run batch simulations.

1.2.2. New Drawing Features

- Automatic cavity fill feature,
- Drawing can be flipped and rotated to any angle,
- Improved void and overlap detection,
- "Undo" of one level when drawing.

1.2.3. New Calculation Features

- View-factor radiation modeling accounts for self viewing effects. This is particularly useful for modeling projecting products such as greenhouse windows, skylights, and certain aluminum windows (e.g., sliders) that "see" themselves.
- Advanced view-factor radiation modeling for window frame cavities accounts for self-viewing elements in cavities.
- Condensation Index modeling option uses local (as opposed to average) convection correlations in glazing cavities, as well as the advanced view factor radiation model, resulting in more accurate local temperatures than in previous version.
- "Tape measure" calculates average temperatures across any line segment.

1.3. THERM Components

THERM has three basic components:

- a graphic user interface that allows you to draw a cross section of the product or component for which you are performing thermal calculations.
- a heat-transfer analysis component that includes: an automatic mesh generator to create the elements for the finite-element analysis, a finite-element solver, an optional error estimator and adaptive mesh generator, and an optional view-factor radiation model.
- a results displayer.

1.3.1. The THERM Graphic User Interface

THERM has standard graphic capabilities associated with the Microsoft Windows(TM) operating system. For example, THERM allows you to use:

- Both mouse and cursor operations;
- Standard editing features, such as Cut, Copy, Paste, Select All, and Delete;
- A toolbar to access frequently used commands and short-cut keys;
- Multiple windows so several projects can be open concurrently.

THERM has powerful drawing capabilities that make it easy to model the geometry of the cross section of a building component.

You can use two methods to draw a cross section:

- Trace an imported DXF or bitmap file.
- Draw the geometry based on a dimensioned drawing.

The following thermal properties must be defined for the cross section geometry:

- Material properties (broken into two classes: solids and cavities) of each component of the cross section,
- Boundary conditions at the external edges of the cross section.

1.3.2. Heat-Transfer Analysis

THERM uses two-dimensional (2D) conduction and radiation heat-transfer analysis based on the finite-element method, which can model the complicated geometries of fenestration products and other building elements. (A complete discussion of the solution method is available in Finlayson 1995[4] and Curcija et al. 1995[10] as well as in Appendix C of this manual). This method requires that the cross section be divided into a mesh made up of nonoverlapping elements. This process is performed automatically by THERM using the Finite Quadtree method[5]. Once you have defined the cross section's geometry, material properties, and boundary conditions, THERM meshes the cross section, performs the heat-transfer analysis, runs an error estimation, refines the mesh if necessary, and returns the converged solution.[4] See Appendix C for more details.

1.3.3. Results

The results from THERM's finite-element analysis of a fenestration product or building component can be viewed as:

- U-factors,
- isotherms,
- color-flooded isotherms,
- heat-flux vector plots,
- color-flooded lines of constant flux,
- temperatures (local and average, maximum and minimum).

The rest of this manual explains how to get started using THERM:

- how to install the program,
- how to draw the cross-section geometry,
- how to define the thermal properties,
- how to perform the thermal analysis,
- how to view the results,
- how to print a report,
- how to export files to WINDOW.

2. INSTALLATION

2.1. Computer Requirements

First, make sure your computer system meets these specifications:

- IBM-compatible 80486 or higher with a math coprocessor. A Pentium-class computer is preferable for speed.
- At least 16 MB of random access memory (RAM). 32 MB or more of RAM is preferable for optimum operation.
- Microsoft Windows 95™, or Windows NT™. (Note: If you are running THERM under Windows NT™, make sure that the Windows 95 checkbox in the Options / Preferences, Preferences tab is *unchecked* after you have installed the program).
- WINDOW 4.1 (must be installed to create glazing systems).
- Hard disk drive with at least 5 megabytes of available disk space.
- Printer supported by Microsoft Windows 95™, or Windows NT™ (serial, parallel, or shared over a network).

2.2. Installing the Program

1. Put the program installation diskette in the appropriate floppy drive.
2. From the **Start** button in Microsoft Windows 95™ or Windows NT™, choose **Run**.

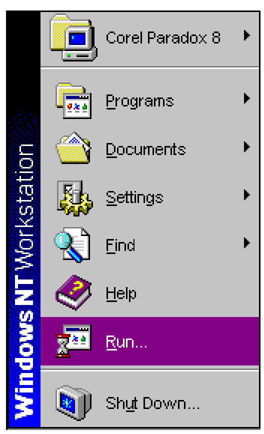


*Figure 2-1. Choose **Run** from the Microsoft Windows 95™ or NT™ **Start** button.*

3. When the Run screen appears, type

a:\setup.exe

Click on the **OK** button.

Substitute the appropriate floppy drive name if the THERM diskette is not in drive "**a**"

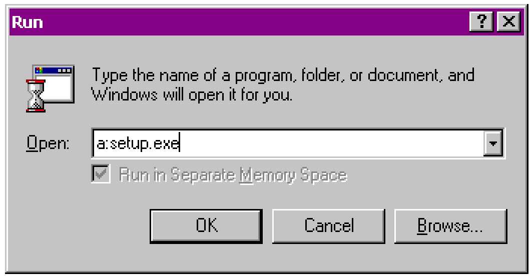


Figure 2-2. Run the SETUP.EXE file on the THERM diskette.

A small **Setup** window will appear saying that THERM setup is preparing the InstallShield® Wizard. Wait for this window to disappear; then, proceed to the next step.

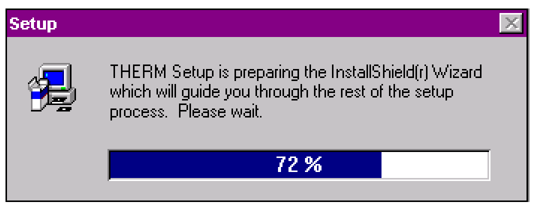


*Figure 2-3. The **Setup** window*

4. The THERM **Welcome** window will appear. From this window, continue the installation by clicking on the **Next** button, or cancel the installation by clicking on the **Cancel** button. Each installation window has a **Back** and a **Cancel** button, so you can return to previous steps or stop the installation at any time.

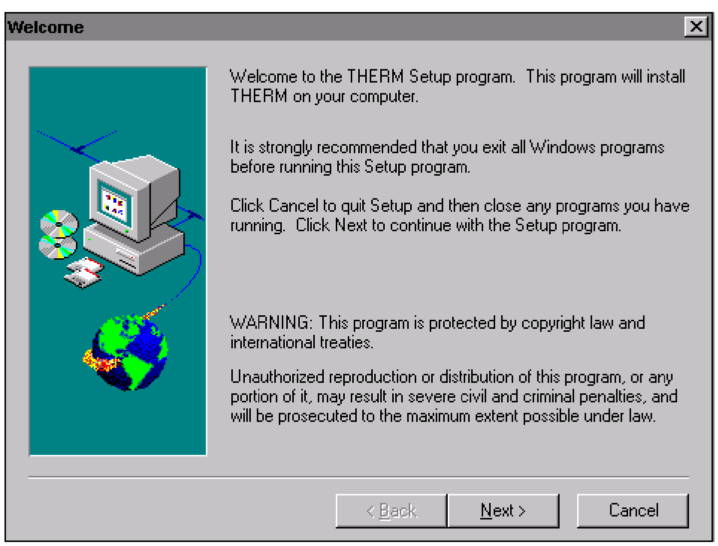


*Figure 2-4. The THERM **Welcome** window*
Click on the Next button to install the program.

5. Setup will display the **Software License Agreement** screen. You must click on the **Yes** button and agree to the terms of the license in order to continue with the installation.

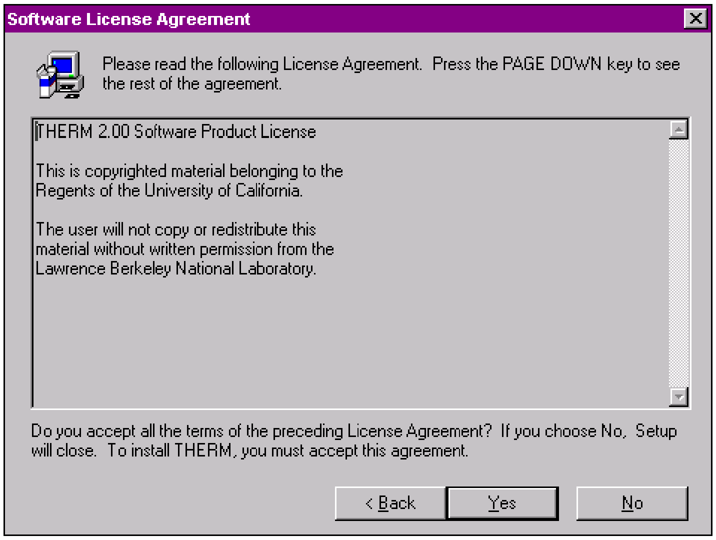


*Figure 2-5. The initial THERM **Setup** window.*
*Click on the **Continue** button to install the program.*

6. In the **User Information** screen, type your name and company into the input boxes; then, click on the **Next** button.

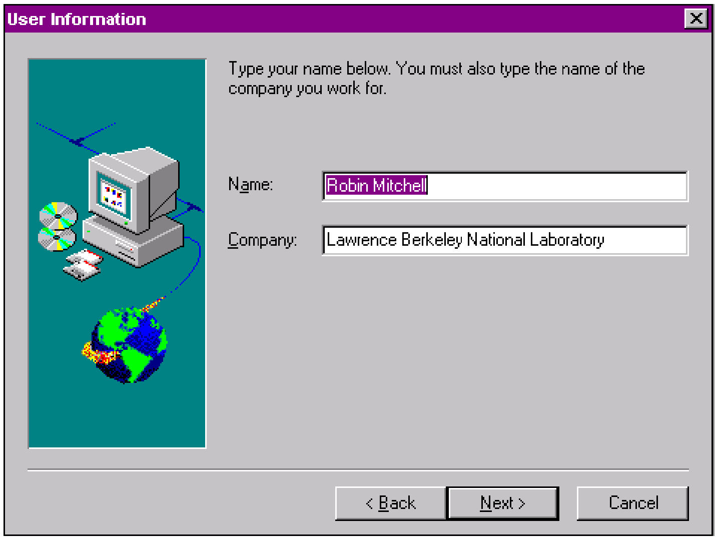


*Figure 2-6. Type your name and company in this **Setup** screen and click on the **Continue** button.*

7. Setup will automatically install THERM in the **Program Files** directory on your **C:** drive. If you want to change the drive location and/or the name of the directory, you can click on the **Browse** button to specify a different drive. When you have finished, click on the **Next** button.

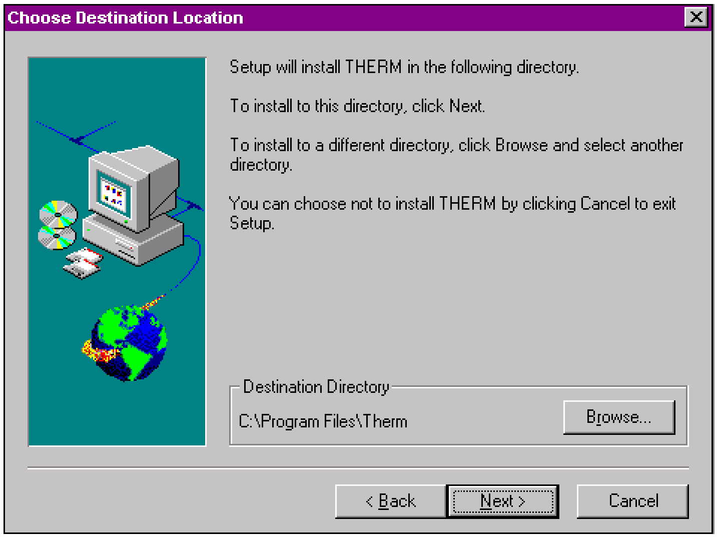


Figure 2-7. Default location for the THERM installation is C:\Program Files\Therm; you can change the location using ***Browse****.*

8. In the **Select Program Folder** screen, setup will automatically default to a folder called THERM; you can specify a different folder.

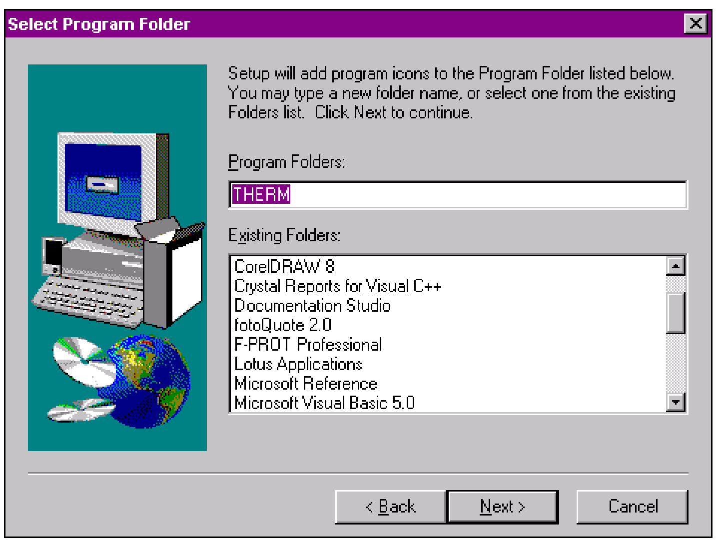


Figure 2-8. Default Program Folder is THERM

9. Setup will display all the settings you have specified; if you want to change any settings, you can click on the **Back** button. Note that the **Setup Type** is always set to **Complete**.

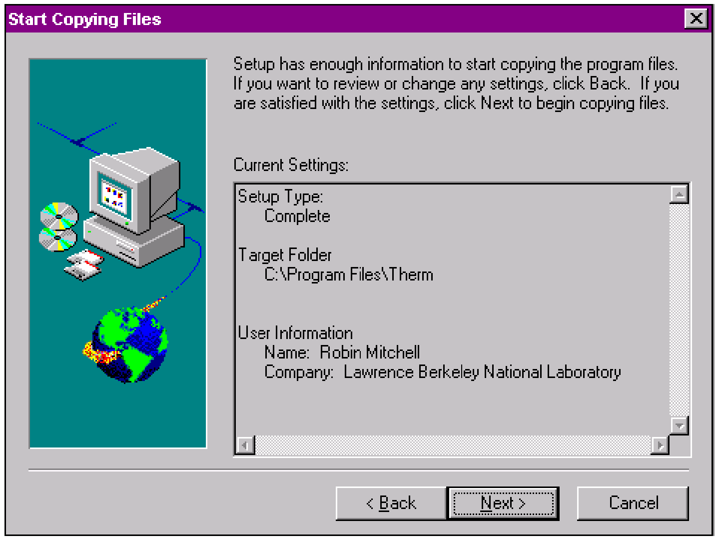


Figure 2-9. Setup verifies all the installation settings.

10. Setup will decompress the installation files and copy the program files to the directory specified in the previous screen. Setup will also create all the subdirectories and associated files needed for THERM to run. Setup will ask for the necessary disks as it progresses through the installation.

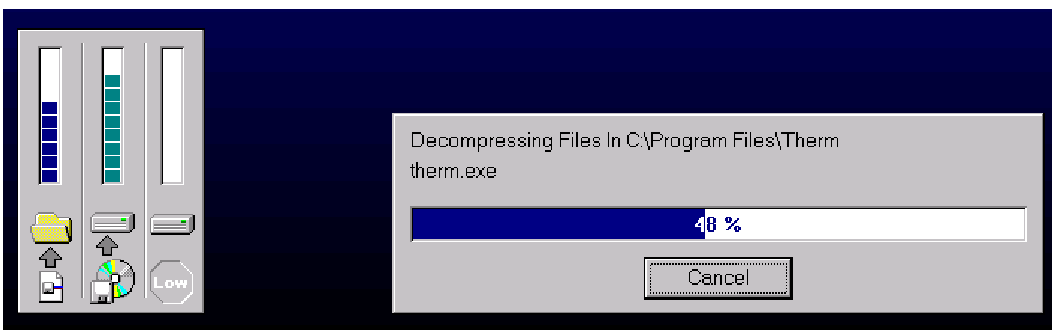


Figure 2-10. Setup will decompress and copy the program files to the specified location.

11. Setup gives you the option of viewing the README file. When you are ready, click on the **Finish** button to complete the installation.

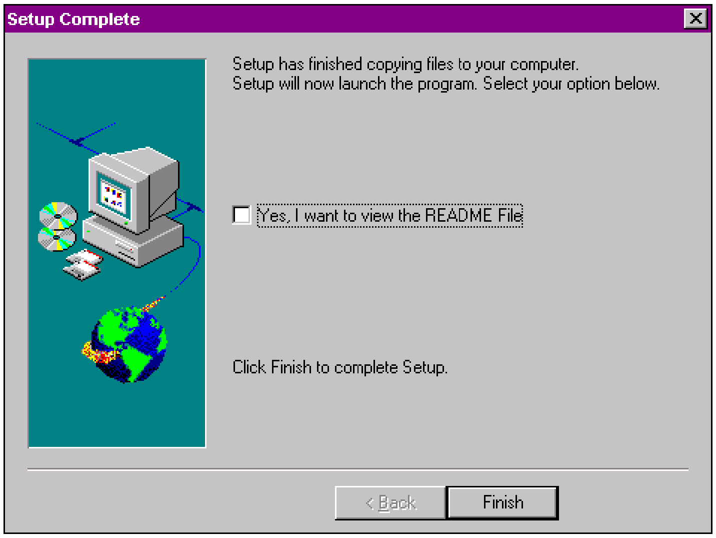


*Figure 2-11. You can view the README file or click on **Finish** to complete the installation.*

12. Setup will automatically make an entry for THERM in the Programs list, accessible from the **Start** button.

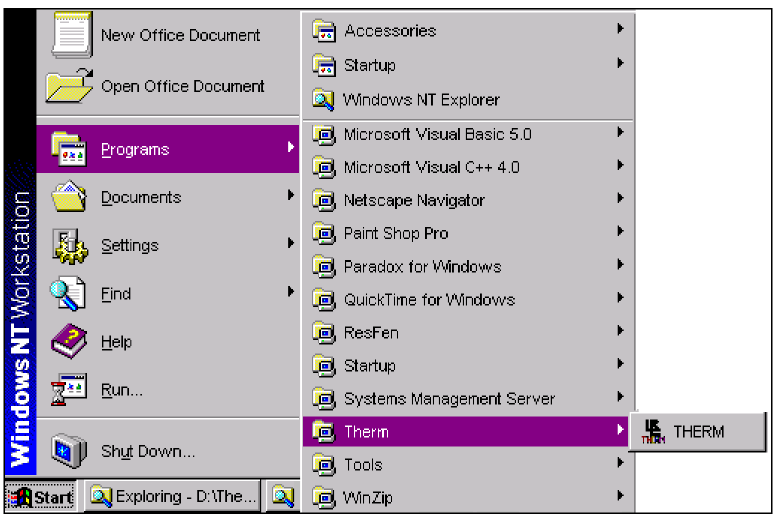


*Figure 2-12. THERM can be run from the **Start Programs** list.*

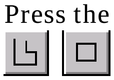
3. QUICK START

This chapter is intended to provide a summary of the steps for drawing a cross section and calculating the results. Each section references places in this manual that explain procedures in more detail.

The examples shown are for typical window cross sections. However, all of THERM's features (except for the Condensation Index Model which only applies to glazing systems) can be used to model two-dimensional representations of products.

3.1. Draw Cross Section and Select Materials
(for detailed instructions, see Chapter 5, "Drawing Cross-Section Geometry")

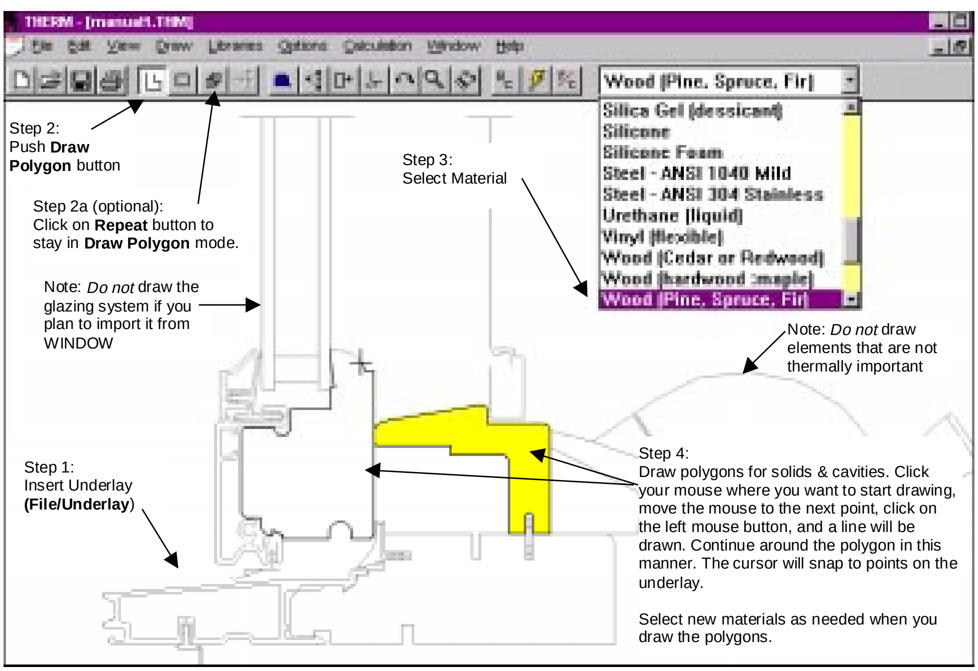
1. If tracing an underlay, bring in a DXF or bitmap file (using **File/Underlay**); to draw the cross section geometry yourself, go to the next step.

2. Press the **Draw Polygon** or **Draw Rectangle** toolbar button.


3. Select the material for the element you are drawing from the pull-down list in the upper right of screen. (Type the first few letters of the material to scroll through the list without using the mouse).

4. If using an underlay, trace the cross-section elements with your mouse, or work from a dimensioned drawing using THERM's step function (type a value to set the step size; when you press an arrow key the cursor will move by the step distance in the direction of the arrow key; see Section 5.5, "Drawing Using the Keyboard"). THERM recognizes the vertices of the DXF file and will snap the lines being drawn to the underlay. Select new materials as needed before you draw each component, and the materials will be automatically assigned to that component. Draw all components except the glazing system, which will be inserted later.

   If the cursor is snapping to lines or points inappropriately, zoom in (click on the right mouse button) to reduce cursor sticky distance (**Shift-right** mouse click zooms out), or use **Draw/Snap Settings**.



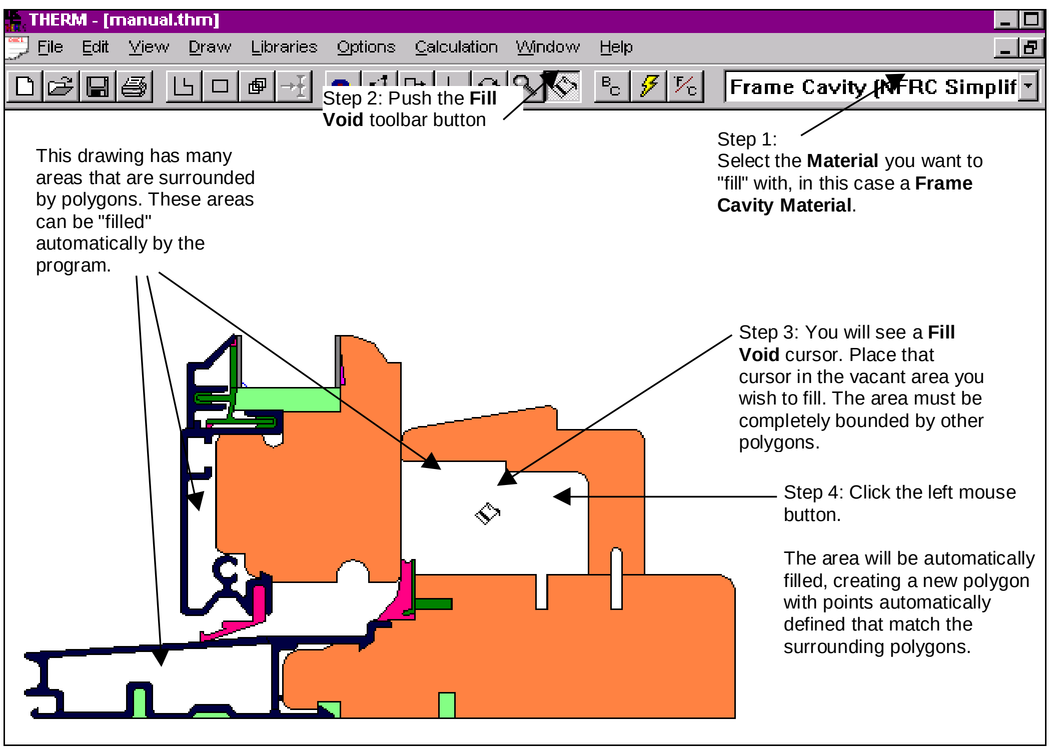
3.2. Drawing Features

There are several powerful drawing features in THERM, including fill void; insert, move and delete points; move, cut, copy, and paste polygons; and flip and rotate the entire cross section. See Section 5.6 "Editing Polygons" for an in-depth discussion of these features.

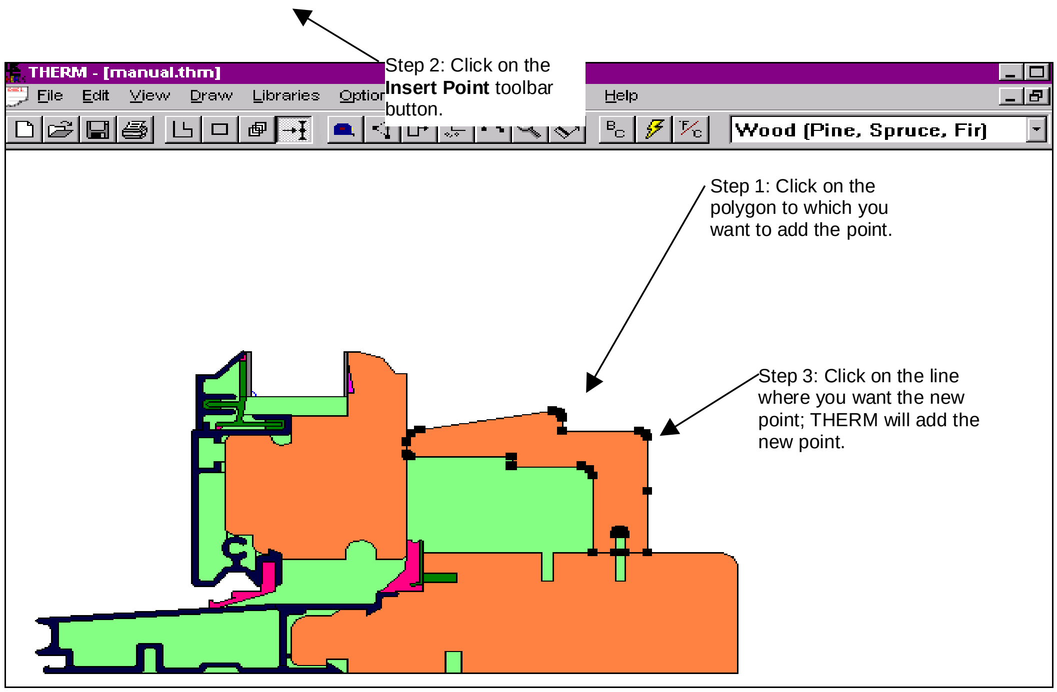
1. **Fill Void feature:** An area of the drawing that is completely surrounded by polygons can be automatically "filled" by THERM, using the **Fill Void** toolbar button.

   To fill an area, first select the **Material** you want to "fill" with, such as the **Frame Cavity (NFRC Simplified)** material shown in the figure below. Then push the **Fill Void** toolbar button, and a **Fill Void** cursor will be displayed. Put the cursor inside the area to be filled and press the left mouse button. (If the area to be filled is smaller than the cursor, zoom in by pressing the right mouse button). THERM will automatically create a polygon that fills the area.

   For THERM to perform its calculation, the cross section must not contain any voids or overlaps (see Section 6.3.2, "Finding Voids and Overlaps"). THERM drawing features that are designed to minimize unintentional creation of voids and overlaps as you are drawing include the fill void feature, the "snap" feature (see Section 5.3.3, "Snap Settings") and the dynamic overlap checking as you draw polygons.



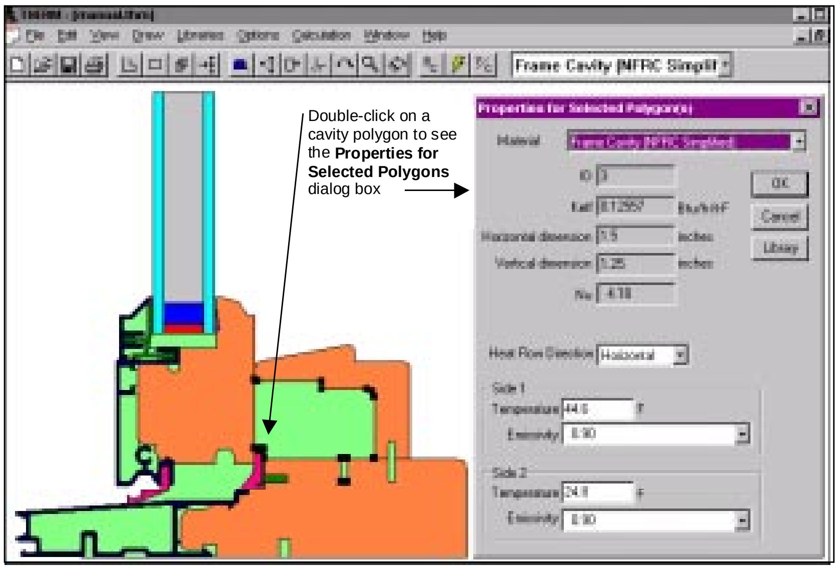
2. **Insert Points**: If you need to insert new points into a polygon, use the following steps:
   - Click on the polygon to which you want to add the point
   - Click on the **Insert Points** toolbar button
   - Move the mouse cursor over the location on the polygon where you want to add the point.
   - Click the left mouse button. A new point will be added.



3.3. Materials
(see Section 5.10, "Assigning Materials After Drawing Polygons" and Section 5.11, "Defining New Materials")

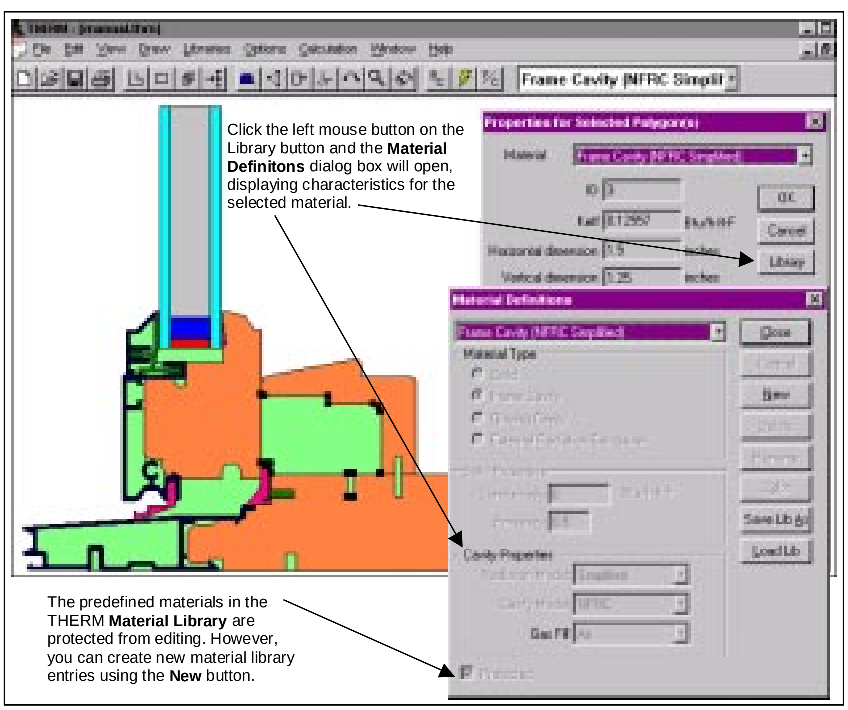
Materials can be specified as you are drawing; they can also be changed after polygons have been created.

1. Double-click on a polygon; the **Properties for Selected Polygon(s)** dialog box will be displayed. Depending on the type of material selected for the polygon (**Solid**, **Frame Cavity**, or **External Radiation Enclosure**), different values will be displayed in the dialog box.



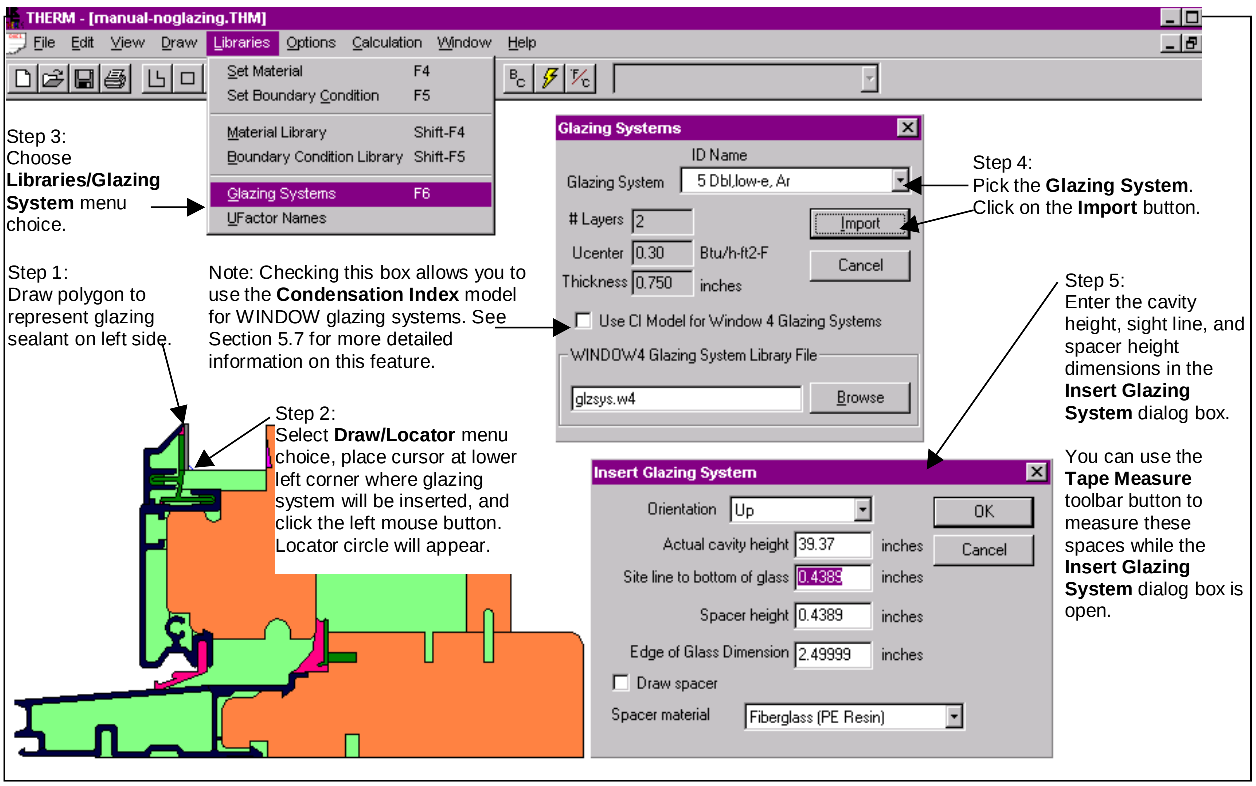
2. From **Properties for Selected Polygon(s)**, you can press the **Library** button, and the **Material Definitions** dialog box will be displayed. From **Material Definitions**, you can view all the entries in the **Material Library** in order to see more detailed characteristics of each material.

3. The THERM material library contains predefined materials that cannot be edited; however, from the **Material Definitions** dialog box you can create new material library entries, using the **New** button.



3.4. Insert Glazing System (see Chapter 5, "Drawing Cross-Section Geometry")

1. On the left-hand side of the glazing unit, draw a rectangle to represent the sealant between the glazing and the frame.
2. Use the **Locator**, accessed from **Draw/Locator**, to define the lower left corner of the glazing system to be inserted.
3. Select the **Libraries/Glazing System** menu choice to access the **Glazing Systems** dialog box, and choose a glazing system previously created in WINDOW 4.
4. From the **Glazing System** dialog box, use the pull-down list to select a glazing system from the WINDOW4 (glzsys.w4) library; click on the **Import** button. You can use the **Browse** button to change to a glazing system library in another directory.
5. The **Insert Glazing System** dialog box will appear. Enter all information in the input fields, in particular the cavity height (if you are using the Condensation Index Model), sight line, and spacer height dimensions. Leave the **Draw Spacer** box unchecked if you want to paste in a spacer from another THERM file. Press the **OK** button.
6. THERM will insert the glazing system automatically. If you get a message that the glazing system won't fit, adjust the frame components as needed.
7. Add another rectangle for the sealant to fill in any gap between the right side of the glazing system and the frame.



3.5. Adding a Custom Spacer

THERM's **copy** and **paste** drawing features allow you to have libraries of components that can be used in many different cross sections. You could, for example, store a spacer in a separate THERM file that can be copied into any cross section you are drawing. You can also make use of the **flip** and **drawing locator** features during this process.

1. Open the THERM file containing the spacer. In the example shown, the file contains half of a spacer.
2. Use the **Draw/Locator** menu choice and click your left mouse button on the upper left corner of the spacer.
3. Use the **Edit/Select All** menu choice to select all the polygons in the spacer.
4. Use the **Edit/Copy** menu choice or **Ctrl-C** to copy all the selected polygons into memory in standard Microsoft Windows™ style.
5. Use the **File/Open** menu choice to open the cross section into which you want to copy the spacer.
6. Use the **Draw/Locator** menu choice to position the locator in the upper left corner of cavity where the spacer will go. The location of the Locator in this file should match the position where you placed the locator on the spacer; the program will match up the two locator positions when you paste the spacer.
7. Use the **Edit/Paste** menu choice or **Ctrl-V** to paste the spacer into the second cross section. The program will place the copied spacer by matching the two locator positions. Now you have positioned your half spacer in the cross section. To complete the spacer, continue with the following steps.

8. Use the **Window** menu to select the spacer file.

9. In the spacer file, use the **Draw/Flip/Horizontal** menu choice to flip the spacer; this new shape will form the second half of the spacer when pasted into the cross section. (THERM applies the flip and rotate functions to the entire cross section).

10. Use the **Draw/Locator** menu choice and click your left mouse button on the upper right corner of the spacer.

11. Use the **Edit/Select All** menu choice to select all the polygons in the spacer geometry.

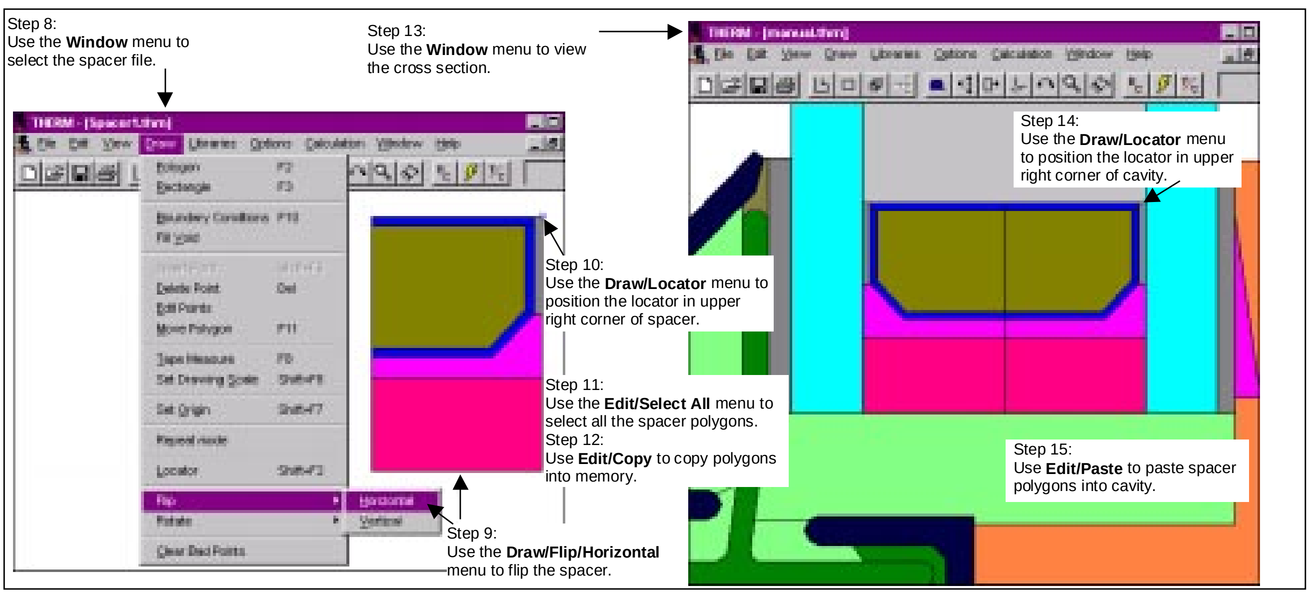
12. Use the **Edit/Copy** menu choice or **Ctrl-C** to copy the "flipped" spacer polygons.

13. Use the **Window** menu to view the cross section into which you already pasted the first half of the spacer.

14. Use the **Draw/Locator** menu choice to position the locator in the upper right corner of the cavity to which you want to match the upper right corner of the spacer.

15. Use the **Edit/Paste** menu choice or **Ctrl-V** to paste the spacer into the cross section. The program will place the copied spacer by matching the two locator positions. Now the second half of the spacer is in position.

16. Use the **File/Save** menu choice to save the cross section containing the new spacer.

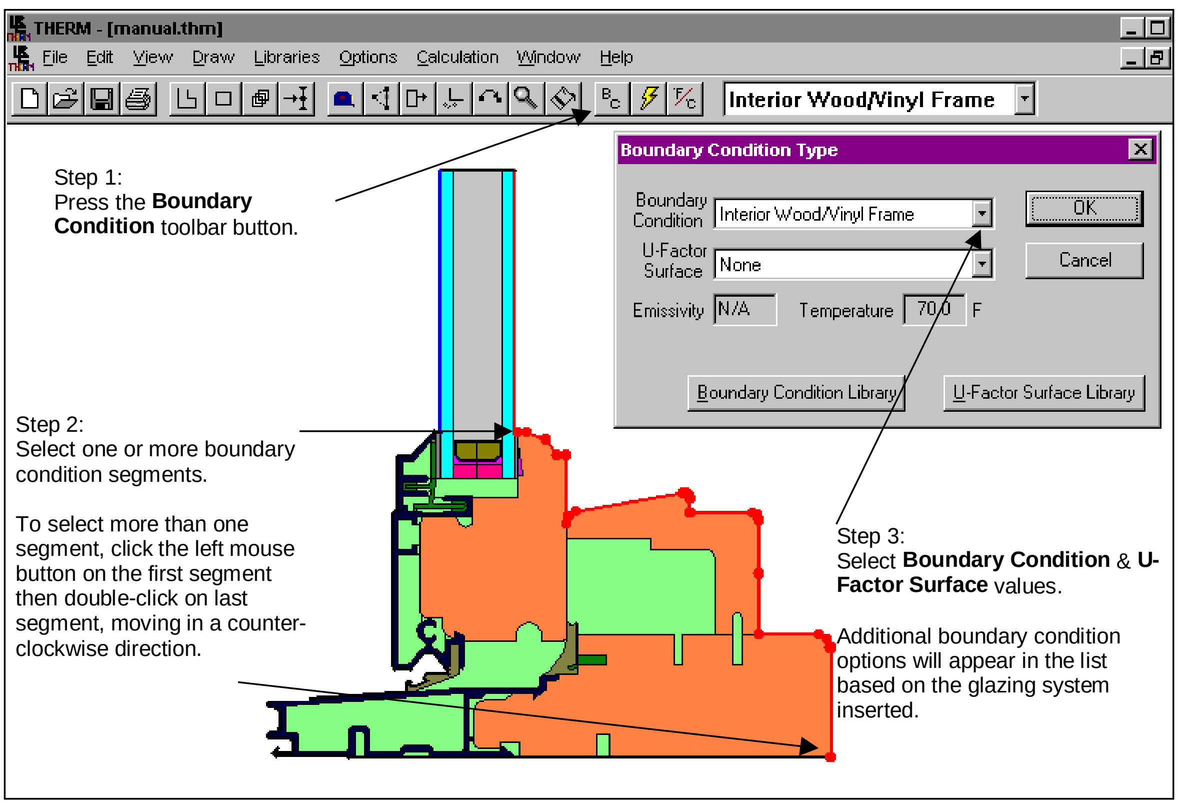


3.6. Specify Boundary Conditions (see Section 6, "Defining Boundary Conditions")

1. Press the **Boundary Conditions** toolbar button. The external boundary is automatically drawn, indicated by a thick line. (A default boundary condition of **Adiabatic** is assigned).

2. To change the default boundary condition, select a boundary segment (double-click the left mouse button on a boundary segment) or multiple contiguous boundary segments (hold the **Shift** key down, click the left mouse button on the first segment and the last segment, moving in a counter-clockwise direction, and then press **Enter**; use the **Ctrl** key in the same manner to select multiple non-contiguous boundary segments). You must have at least two non-adiabatic boundary conditions for the program to perform a simulation.

3. The **Boundary Condition Type** dialog box will appear. Pick the appropriate **Boundary Condition** and **U-Factor Surface** choices for each boundary segment.



3.7. Run Simulation and View Results (see Chapter 7, "Calculating Results")

1. Press the **Calculation** toolbar button. 

2. THERM automatically generates a finite-element mesh to perform the calculation. The status of the calculation is displayed in the status bar at the bottom of the THERM window.

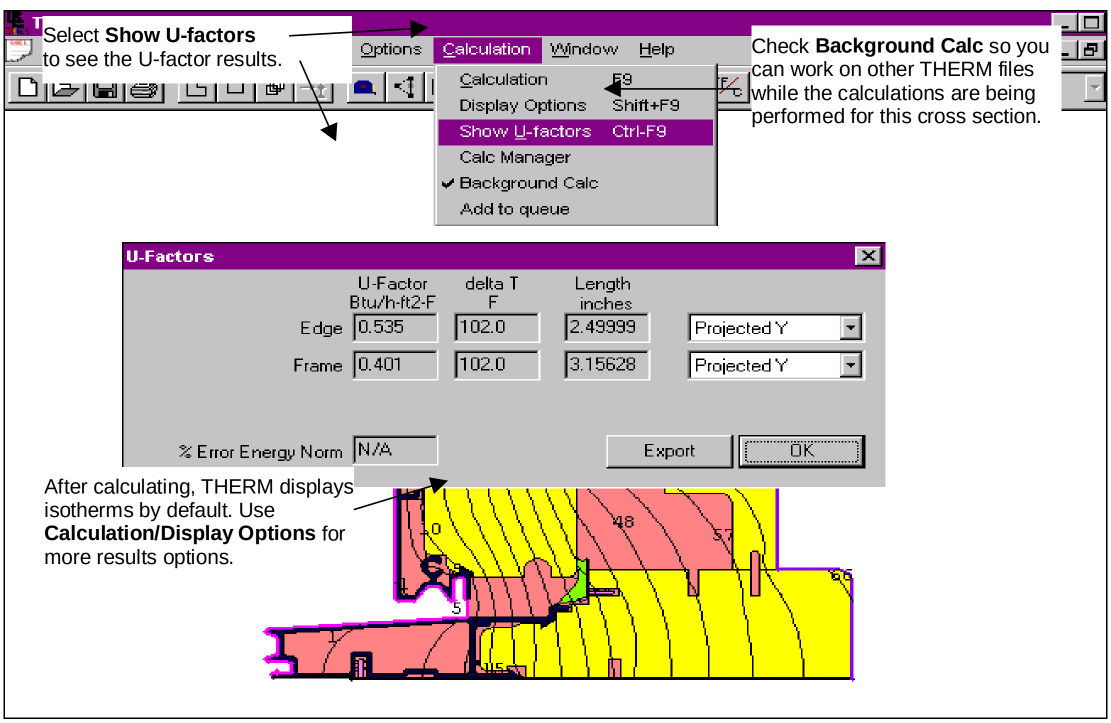
3. Results will be displayed as isotherms on the cross section when the calculation is completed.

4. From the **Calculation** menu, select **Display Options**.

5. Select the desired graphic display option and check the **Draw Results** box.

6. Select the **Calculation/Show U-factor** menu choice to see the U-factor values. If U-Factor values are blank, then you need to assign **U-Factor Surface** labels to the surfaces of interest (double-click on the boundary segment to assign these labels.)

Other graphic results can be accessed using the **Calculation/Display Options** menu choice.

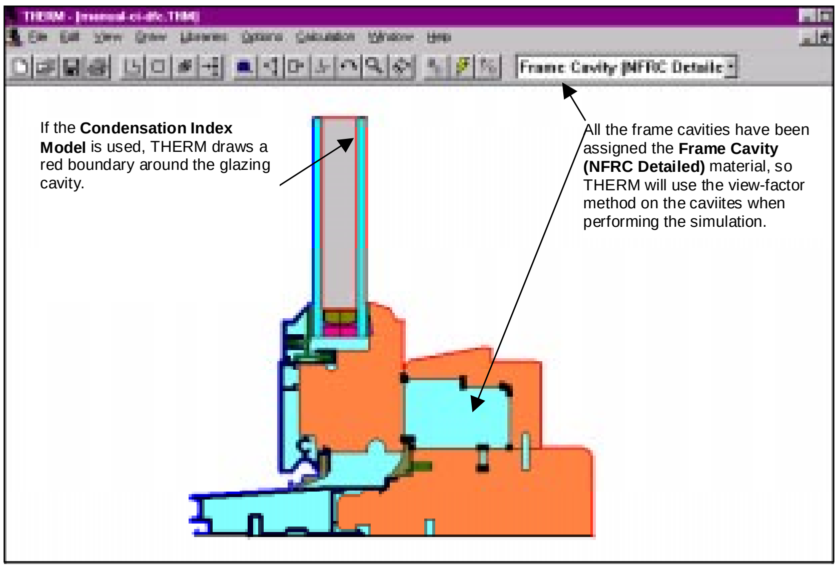


3.8. THERM's Advances in Modeling

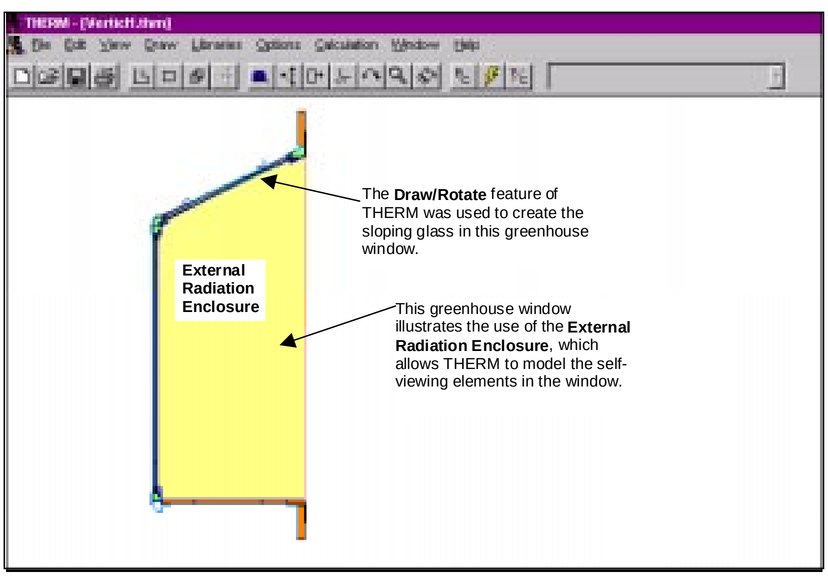
Several features in THERM represent advances in thermal modeling. These include the Condensation Index Model for glazing cavities and the view-factor radiation model for frame and glazing cavities and external radiation enclosures.

1. Assigning frame cavities a material defined with the **Detailed Frame Cavity** model allows the program to use the view-factor method when performing the simulation. See Section 5.11, "Defining New Materials," for a more detailed discussion of this feature.

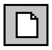
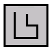
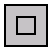
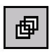
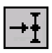
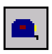
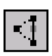
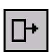
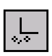
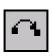
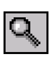
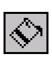
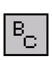
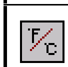
2. If you use the **Condensation Index Model** for an imported WINDOW glazing system, the program will use local film coefficients and a detailed radiation model to give more accurate local temperatures. THERM draws a bold red line around the glazing cavity boundaries to indicate that the **Condensation Index Model** is in effect. See Section 5.9, "Inserting a Glazing System," for more details about this feature.



3. The THERM file below illustrates a greenhouse window cross section in which the **Draw/Rotate** feature has been used to tilt the upper glazing. In addition, the interior area enclosed by the window is defined as an **External Radiation Enclosure**, so the program will model the window using the view-factor radiation methodology. See Section 5.11.4 for more information.



3.9. Toolbar Reference Table

Button	Description	Menu / Choice	Short cut
	New THERM file	File / New	Ctrl+N
	Open existing THERM file	File / Open	Ctrl+O
	Save THERM file	File / Save	Ctrl+S
	Print THERM file	File / Print	Ctrl+P
	Draw Polygon	Draw / Polygon	F2
	Draw Rectangle	Draw / Rectangle	F3
	Repeat Mode This button can be used with the Draw Polygon, Draw Rectangle, Insert Point, Tape Measure, Edit Points, Move, and Fill Void buttons to repeat those functions.	Draw / Repeat Mode	
	Insert Point	Draw / Insert Point	Shift F6
	Turn on Tape Measure	Draw / Measures	F8
	Edit Points	Draw / Edit Points	
	Move polygon or rectangle	Draw / Move	F11
	Set drawing origin	Draw / Set Origin	Shift F7
	Change Snap Settings	Draw / Snap Settings	F12
	Turn on Zoom to enlarge view of drawing	View / Zoom	*Zoom to fit: F7* *Zoom In*: right mouse button *Zoom Out*: shift right *Center:* Ctrl-right mouse button
	Fill void	Draw / Fill Void	
	Define Boundary of cross section	Draw / Boundary Conditions	F10
	Begin Simulation	Calculation / Calculation	F9
	Switch Unit Systems between SI and IP	Options / Switch Units	

3.10. Other Shortcuts

Description	Menu/Choice	Shortcut
Material Library	Libraries / Material Library	Shift-F4
Boundary Condition Library	Libraries / Boundary Conditions	Shift-F5
Set Material	Libraries / Set Material	F4
Set Boundary Condition	Libraries / Set Boundary Condition	F5
Snap to point or polygon within sticky distance of the cursor		Space bar
Snap to the last point drawn		End
Decrease the step size by a factor of 10.		Ctrl-Arrow key

4. GETTING STARTED

4.1. The Main Screen

The **Main Menu** of the THERM program is shown below. The menu choices allow you to access different program functions.

The **Main Screen**, which is displayed when THERM starts, is where you create and edit cross sections, select the properties (e.g., materials and boundary conditions) associated with the elements of the product you are analyzing, and perform thermal calculations.

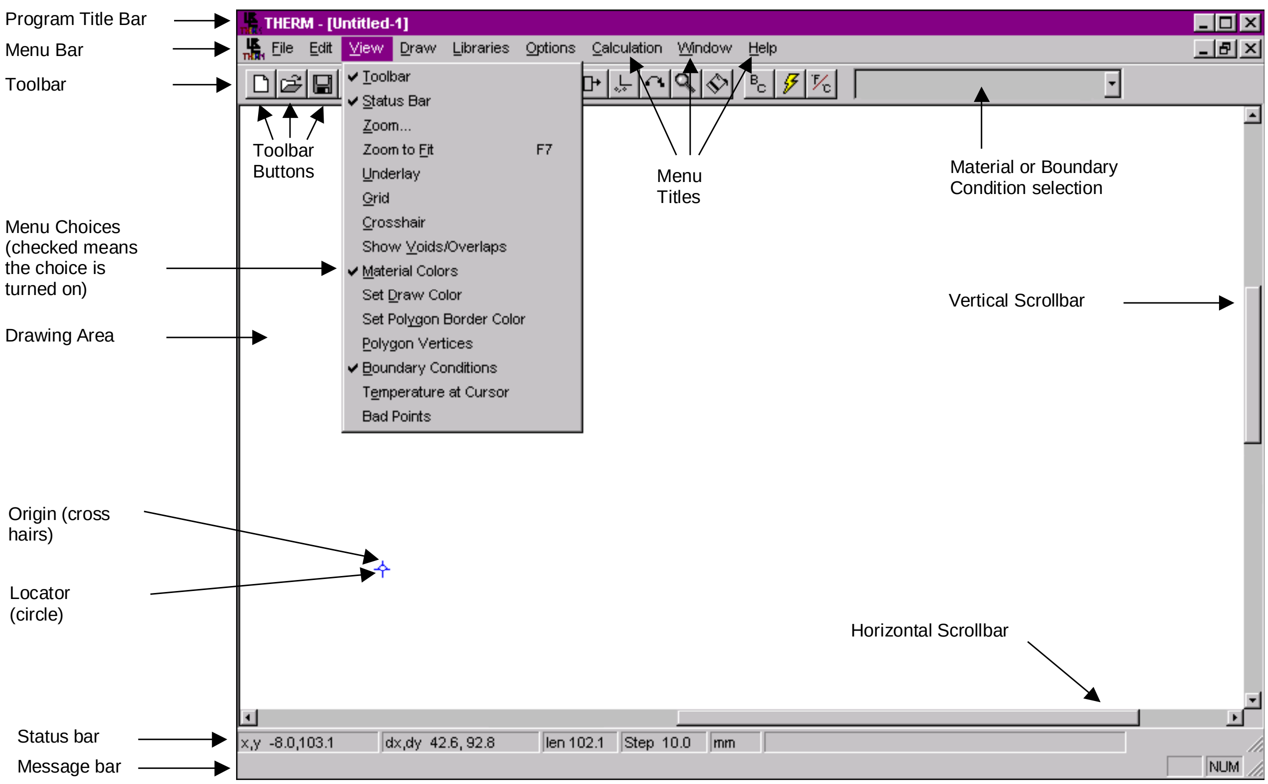


*Figure 4-1. THERM **Main Screen** components*

The **Main Screen** consists of the following components:

- Program Title Bar: This shows the program name as well as the name of the currently active drawing file in square brackets. New files are named Untitled followed by a sequence number until you save them with another file name.
- Menu Bar: The Main Menu is below the Program Title Bar and is the way to access all THERM menus. From the Main Menu, you can set the drawing options, assign materials and boundary conditions, and run calculations. You can use a mouse to activate a menu by clicking the left mouse button once on the desired menu title. You can use the keyboard to activate a menu by holding down the **Alt** key and typing

the menu's underlined character. For example, pressing **Alt-F**, and then typing **N** would access the **File/New** menu choice.

- Menu Choices: Every menu option on the Main Menu offers associated menu choices that appear when you click the mouse on a Menu title. You can activate the choices on the menu list by clicking on them using the left mouse button. You can also use the keyboard to activate a menu choice by typing the underlined character or using the arrow keys to move through the choices.

- Toolbar: The toolbar contains buttons for the most common activities in the program, such as Save File and Draw Polygon. A button is activated by placing the mouse cursor on it and clicking the left mouse button. Each toolbar button corresponds to a menu choice, so you can use either a mouse or a keyboard to make your selection (see Section 3.9, "Toolbar Reference Table").

- Drawing Area: The large white space below the toolbar on the Main Screen is for drawing the geometry of the cross section you wish to analyze.

- Origin: The cross hairs in the drawing area indicate the 0,0 point of the drawing.

- Locator: The small circle at the center of the origin cross hairs is the Locator. It can be used to indicate a location for a drawing component, such as a glazing system imported from WINDOW or a spacer copied from another THERM file. By default, the Locator and Origin are in the same position when you start a new drawing. However, the Locator can be moved to any position in the drawing area using the Draw / Locator menu choice.

- Cursor Feedback Bar: The cursor feedback bar displays information about the coordinates of your drawing cursor. This is explained in more detail later in this section.

- Message Bar: The message bar at the bottom of the Main Screen displays information about the currently selected menu or toolbar button.

4.1.1. Cursor Feedback Bar

The **Cursor Feedback Bar** is the top row of information at the bottom of the **Main Screen**. This bar displays information about the cursor position.

The **x,y** information gives the absolute **x** and **y** coordinates of the cursor relative to the origin (the small cross hairs). The **dx, dy** information gives the **x** and **y** coordinates of the cursor relative to where you most recently began drawing; if you place the cursor anywhere in the drawing window and click the left mouse button, the **dx,dy** values go to **0,0**. These values will change as you move the mouse, showing the coordinates relative to the position where you most recently began drawing. The **len** information gives the distance traveled since the last click of the left mouse button. This length is reset to **0** every time the left mouse button is clicked.

The **Step** information displays the current **Step Size**, which is defined by typing a number with the keyboard. The **Step Size** is the distance traveled by the cursor for every keystroke with the arrow keys. For example, if the **Step** value is set to 10 mm, then pressing the right arrow once will move the cursor 10 mm.

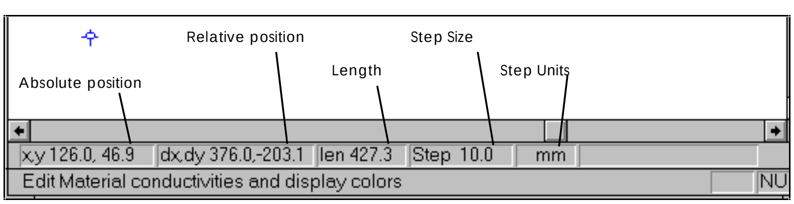


*Figure 4-2. **Cursor Feedback Bar** notations*

4.2 Main Menu

Figure 4.3 shows THERM's main menu choices.

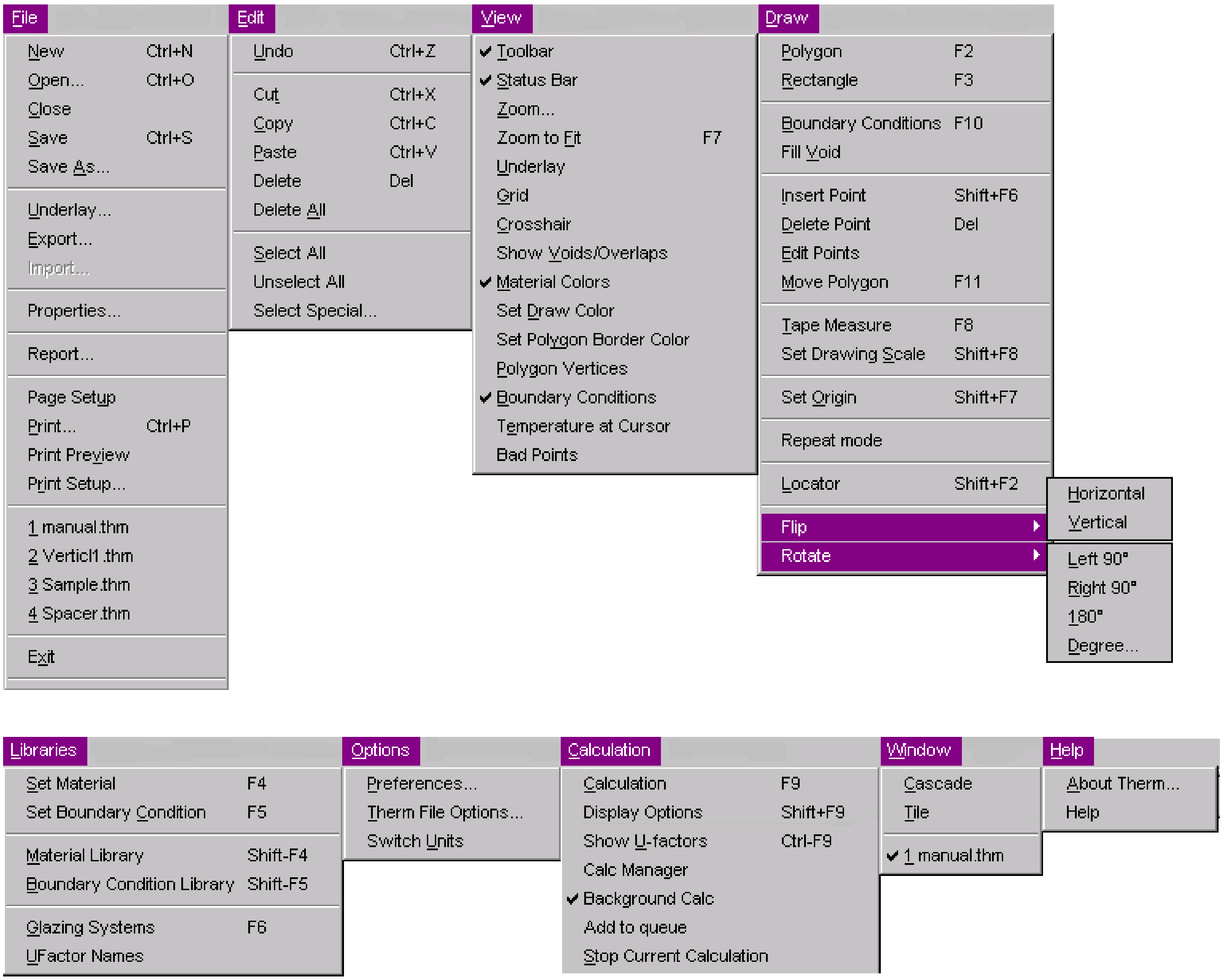


*Figure 4-3. THERM **Main Menu** choices*

To use the keyboard for selecting menu options, press **Alt** plus the underlined character to select the menu; then, type the underlined letter of the menu choice. For example, to choose the **New** choice under the **File** menu, type **Alt F**; then, type **N**.

File	Used to start a THERM drawing file, edit an existing file, and save work on open files. Also used to access printing options, and to import an underlay file.
Edit	Used to modify the components in a currently open THERM drawing.
View	Used to change the view options such as to zoom in on the drawing, view the underlay, and change the position of the drawing origin.

Draw	Used to access the drawing options, such as to draw a rectangle or polygon, move objects, and insert points.
Libraries	Used to access the **Materials**, **Boundary Conditions**, and **Glazing Systems** libraries. From this menu, you can add new library entries, edit existing entries, and import **Glazing Systems** from WINDOW 4.
Options	Used to access some of the THERM program options, such as the measuring units (SI or IP), drawing options, and simulation options.
Calculation	Used to access the finite-element calculation, which simulates the thermal performance of the cross section. Also used to access the **Calc Manager** and define the format in which you wish to view the results.
Window	Used to access each of the open drawings in THERM and display them either as tiled or cascaded. This menu show the names of all open drawings.
Help	Used to access the THERM **Help** file, which contains the information in this manual, as well as the **About Therm** choice, which contains information about the program version and authors.

4.2.1. Pull-down Lists

A pull-down list displays choices for an input value. To display the choices, use the left mouse button to click the arrow to the right of the input box. You can select an item from any pull-down list using one of the following methods:

- Click on the arrow of the pull-down list: This will cause the first set of items in the list to show. A vertical scroll bar will appear on the list if there are more items than can be displayed in one view.

- Type the first few letters of the item: If you type the first letter of the item, the first option that starts with that letter is chosen. If you add the next letter of the item, the program will go to the option in which these first two letters appear. When the correct item it highlighted, you can select it by clicking on Enter.

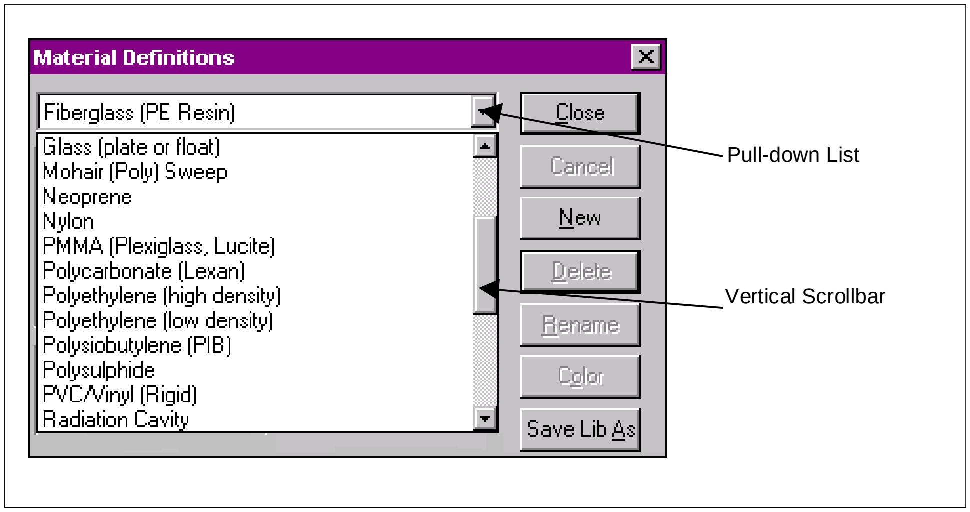


Figure 4-4. Example of a pull-down list

4.2.2. THERM Files

You should save each cross section that you draw as a THERM file, using the **File/Save** menu or the **Save File** toolbar button. THERM will automatically add a THM extension to the file name you type. Two sample files, SAMPLE.THM and SPACER.THM, are included in the installation diskettes. You can open these, or any other THERM file, using the **File/Open** menu choice, or the **Open File** toolbar button. A list of files with a THM extension will be displayed, and you can select the file you want.

You can also start a new file using the **File/New** menu choice or the **New File** toolbar button. You will see a blank drawing area, and you can begin drawing. When you save the file, using the **File/Save** menu choice, THERM will automatically assign a THM extension to the name.

THERM allows you to have multiple THM files open at one time. You can see a list of all opened files using the **Window** menu. You can close each open THM file independently, using the **File/Close** menu option. You can close all the files, and still have the THERM program active. However, the **File** menu becomes reduced to a few choices, such as **New** or **Open**, when there are no files open.

You can see a list of the last four open THERM files at the end of the **File** menu. You can open one of these files by either clicking your mouse on the file name or typing **Alt-F** and the number of the file.

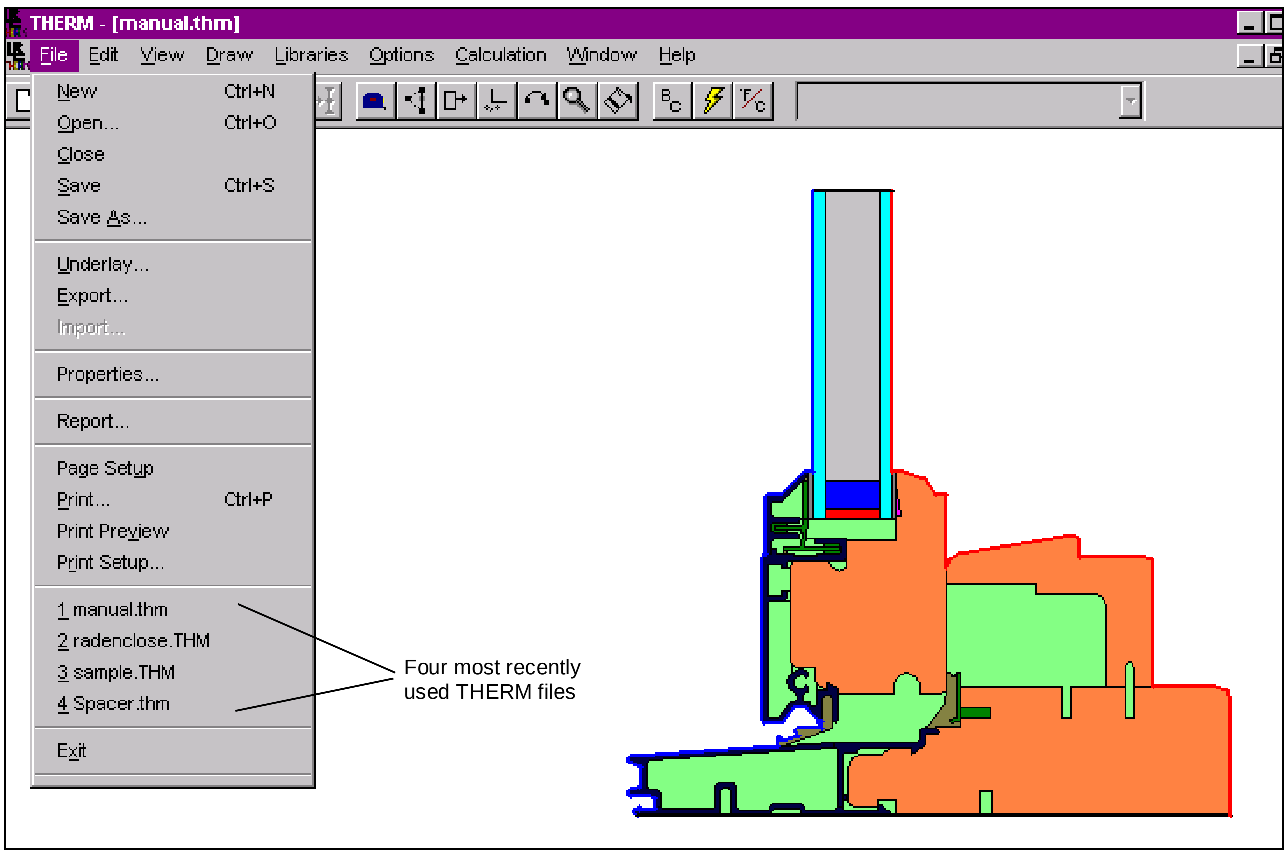


*Figure 4-5. Last four open THERM files are shown at the end of the **File** menu.*

5. DRAWING CROSS-SECTION GEOMETRY

5.1. Overview of Drawing a Cross Section

You can start drawing a cross section in THERM using either of two approaches:

- Import a DXF file or a bitmap file as an underlay. Then, use the DXF file autoconvert feature or trace the underlay using a mouse.
- Work from a dimensioned drawing and use the “step size” function and arrow keys to specify the cross-section dimensions and draw the elements.

In general, the subsequent steps are the same whether you trace an imported file or start from a dimensioned drawing:

1. Draw polygons and rectangles to represent the cross-section geometry (for more details, see Sections 5.4-5.5). When modeling windows, we recommend that you import a glazing system from the WINDOW program (see Section 5.9), but you can also represent the glazing systems with individual rectangles (recommended only for advanced users).
2. Assign materials to polygons and rectangles as you draw (recommended) or after the drawing is completed (see Section 5.10). Use the THERM default material library to model solids or frame cavities and to define new materials as necessary (see Section 5.11).
3. Add, delete, or move polygons and points as necessary (see Section 5.6).
4. Insert a glazing system created in WINDOW. Glazing systems from WINDOW are imported into THERM with all material properties, including the effective conductivity of the glazing cavity and boundary conditions, automatically defined (see Section 5-9). Use of the Condensation Index Model requires that the glazing system be imported from WINDOW (see Section 5-9).

When you have finished these steps you are ready to go to Chapter 6, "Defining Boundary Conditions."

The following sections explain in detail how to draw cross sections with or without an underlay.

5.2. Importing a DXF or Bitmap File as an Underlay

THERM reads and interprets DXF and bitmap formatted files, which simplifies the process of creating thermal models.

- **DXF Files:** A DXF (drawing exchange format) file is an export format supported by many CAD and graphics programs. When you export DXF files from another program, it is best to keep them simple, eliminating elements such as text, shading, and dimensions that are not needed for the thermal model. Section 5.2.3 provides suggestions on how to make a DXF file for use with THERM. A DXF filter function (see Section 5.2.2) in THERM can be used to restrict what entities (points, polylines, and arcs) are included in the underlay. Using an imported DXF file has the following advantages:
  - **Autoconvert:** Closed polylines and curves are automatically converted into polygons.
  - **Snap to Underlay:** When tracing an underlay, the cursor will automatically snap to the underlay vertices, making drawing fast and accurate. Section 5.3.3 discusses **Snap Settings**, which govern how THERM decides where to snap.

- **Bitmap Files:** THERM can also import bitmap files (files with a BMP extension), made, for example by scanning a drawing or capturing a screen from another drawing tool. However, BMP files do not contain information about points or lines, so THERM cannot perform autoconvert or "snap" to the bitmap underlay as with DXF files. It is often easier to work from a dimensioned drawing than to try to get the correct dimensions by tracing a bitmap underlay.

Use the following steps to import a DXF file to use as an underlay for your THERM drawing:

- Select **Underlay** from the **File** menu.
- Select the DXF or bitmap file name from the **Underlay** dialog box.

5.2.1. Select Underlay from the File menu

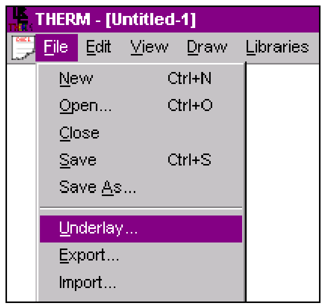


*Figure 5-1. To import a DXF file, pick the **Underlay** option from the **File** menu.*

5.2.2. Select DXF or Bitmap File

1. The **Underlay** menu choice will bring you to the **Underlay** dialog box.

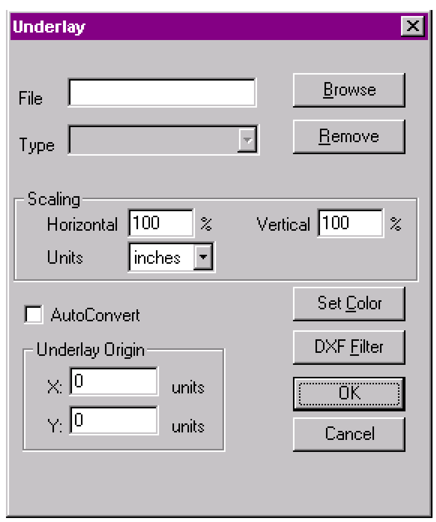


*Figure 5-2. The **Underlay** dialog box*

2. Click on the **Browse** button to see all files in your current working directory with a file extension of *.DXF or *.BMP.

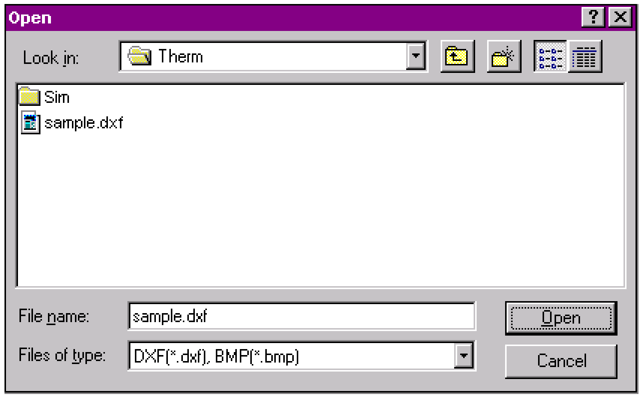


*Figure 5-3. Click on the **Browse** button to get to the **Open** dialog box.*

3. Highlight the desired file and press the **OK** button, which will bring you back to the **Underlay** dialog box.

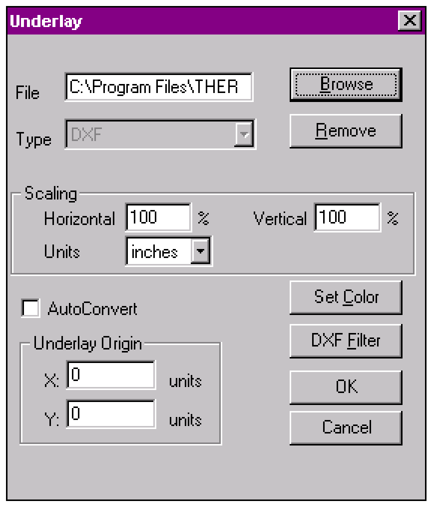


*Figure 5-4. Select the DXF file using the **Underlay** dialog box.*

Options you may want to set in the **Underlay** dialog box include:

- **Scaling:** Scaling inputs allow you to change how THERM scales the imported underlay in both horizontal and vertical directions. This feature is usually used only for BMP files. If you change the horizontal value, the vertical value will automatically change to match it, but you can then change the vertical value and the horizontal value will not change. The default values are 100% in both directions. For DXF files, use the program default value of 100% unless the DXF file is not 1:1. Make sure the **Units** value is set to match the system that the DXF file was drawn in. The default is **IP**, so if you are using an underlay created in SI units, make sure you switch this value.

- **Autoconvert:** This feature converts closed polylines and curves into polygons. If a DXF file is exported from a drawing made up of closed polylines, THERM can automatically convert the drawing into polygons. The polygons will have the currently selected material property. If the DXF file is exported from a drawing made up of lines or open polylines, THERM will not be able to convert the drawing to polygons. You can always try to use the **Autoconvert** feature by checking the box and seeing what happens. If THERM does not find any polylines, it will not generate any polygons. Make sure to check the polygons generated by THERM. A common problem is overlapping polygons; polygons are sometimes completely hidden underneath other polygons. Use **View/Show Voids/Overlaps** (see Section 5.3.5) after using the **Autoconvert** feature to check for overlapping polygons. See Section 5.2.3 for a more complete discussion on how to make DXF files that THERM can correctly autoconvert.

With autoconverted DXF files, curves are converted into a series of straight line segments at a degree specified by the user. The default value for this division is that every 15° of arc is converted into a line segment. This value can be changed using the **Options/Preferences** menu choice, by entering a different value in the **Arc to Polygon conversion** input box on the **Drawing Options** tab. It is recommended that you do not over specify curves.

- **Underlay Origin:** This box indicates the underlay's x and y coordinates relative to the THERM origin. These coordinates represent the position of the lower left corner of the underlay; their values default to 0,0.
- **Set Color:** This button allows you to set the color of the underlay as it will appear in your drawing area. If you have a color monitor, this feature allows you to differentiate the underlay from other drawing components.
- **DXF Filter:** This button leads to another dialog box where you can indicate which elements you want THERM to import from the DXF file. This feature is used to eliminate unnecessary elements in the DXF file.

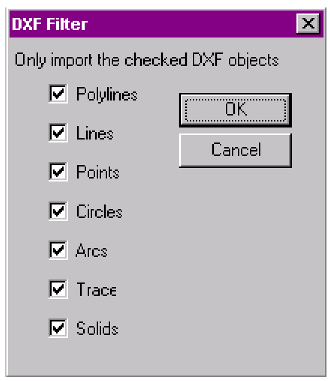


*Figure 5-5. The **DXF Filter** dialog box*

When you import an underlay to THERM, the program will zoom the drawing to fit in the drawing area.

The following two figures show a DXF file drawn with polylines that THERM can interpret. Figure 5.6 shows the underlay without **Autoconvert**, and Figure 5.7 shows the same underlay imported with **Autoconvert** turned on. The underlay in Figure 5.6 can be traced when drawing the polygons; in Figure 5.7, the polygons already exist.

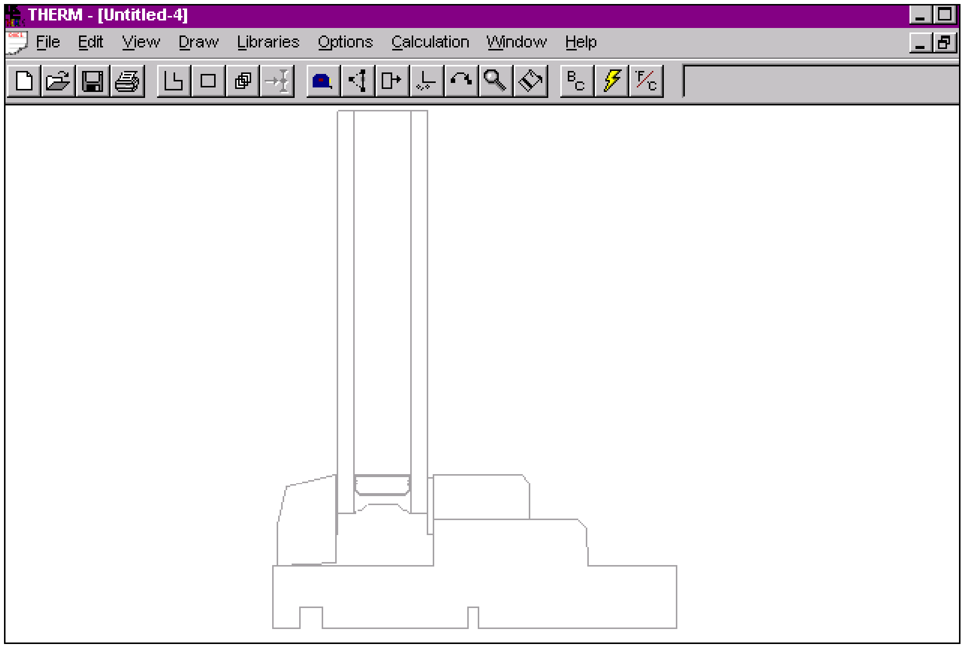


Figure 5-6. A DXF file imported as an underlay with ***Autoconvert*** *turned off, so no polygons are drawn*

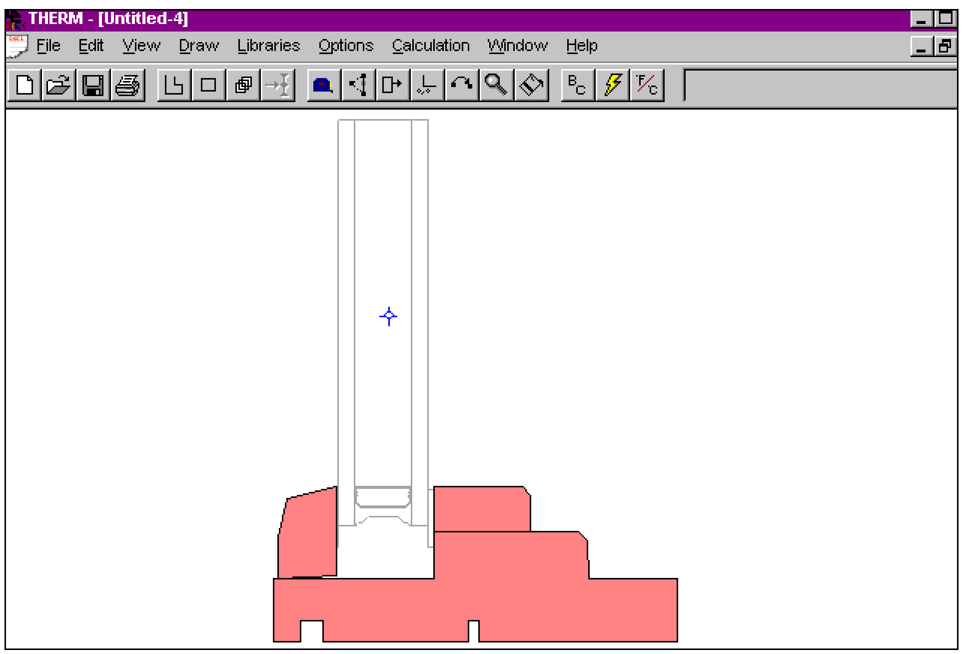


Figure 5-7. A DXF file imported as an underlay with ***Autoconvert*** *turned on, so closed polylines are converted into polygons*

THERM only allows one DXF file at a time to be associated with a model. If you want to use multiple DXF files, first remove the existing file using the **File/Underlay** menu choice, and press the **Remove** button. You can then import a new underlay.

Once you have finished working with a DXF file, you can use the **View/Underlay** menu choice to hide the underlay; the underlay view is off if there is not a check mark to the left of the menu choice. Although you cannot see the underlay, it is still associated with the model until you remove it using the method discussed above.

5.2.3. DXF File Conversion

The DXF file underlay feature in THERM allows you to create simulation models quickly and accurately. With this feature, you can export a drawing created in a computer drafting program using a DXF file format that can be read by THERM and automatically converted into polygons.

You can take many steps when creating your drawing and DXF file to simplify the process of using the DXF file with THERM. These modifications can be made after a drawing is created in the drawing program but will be most efficient if they are incorporated into standard drawing practices.

When you make your drawings:

- Put dimensions and shading on different layers from the main drawing. When you export the DXF file, turn off these layers, so the underlay will not be complicated with information that is not needed for the thermal analysis.

If you want to use the THERM DXF **Autoconvert** feature:

- Use *closed polylines* when creating your DXF file. THERM can autoconvert DXF files that have the cross-section elements drawn as closed polylines. A drawing can be converted to closed polylines by using commands such as **Boundary** in AutoCAD™. This feature works very well with straight-line drawings but is somewhat unpredictable with polylines that contain curves.
- You can use AutoLISP programs that clean up drawings by creating closed polylines and deleting duplicate entities. In AutoCAD™, this can also be done by hand using the **Trim**, **Extend** and **Delete** features.
- Modify the cross section to break up places where a polyline intersects itself. THERM does not draw regions that have multiple connections (i.e., donuts). Make this modification while the object snap is set to ensure that these line segments intersect existing lines. Line segments can be drawn on a separate layer that can be turned off when the drawing is used for non-THERM purposes.
- Test the DXF file import into THERM; if THERM has difficulty picking up some details of the drawing, exploding the DXF file may help. Unfortunately, once closed polylines are exploded, they cannot be autoconverted.

When you Export the DXF file:

- Only export the elements in the DXF file that are needed for the thermal analysis. For example, in AutoCAD™, you can use the “object” option of the **dxfout** command and select only the items that you want to export. This eliminates extraneous elements in your DXF files, such as extra views of the drawing or elements that were deleted but remain in the database.
- DXF files are often in the form of assembly drawings, so snap-in pieces are usually not placed accurately, and deformable pieces are usually drawn as overlapping entities in their nondeformed state. These features make an assembly drawing difficult to use in THERM. It is easier to create a THERM model if you provide the parts unassembled in one or several DXF files and then assemble them. You can make use of the **Flip** and **Rotate** features in THERM to create multiple cross sections using the DXF files. The placement of these pieces can also be corrected within the drawing program that generates the DXF file.

If you have unexpected results when you export a DXF file, consider the following possibilities:

- Sometimes the DXF file comes into THERM with long vertical lines that do not show up on the original drawing. When this happens, it is best to reimport the DXF file into the drafting program, cut and paste it into a new drawing, and export it again.
- Computer-generated drawings often contain entities on top of other entities. This can occur when a drawing is exploded on top of itself. When files like this are autoconverted, they arrive as overlapping polygons in THERM. Duplicate entities need to be removed before drawings are exported as DXF files. AutoLISP routines are available that perform this task, but it is time consuming. The overlapping entities can be deleted easily in THERM. If you delete one too many layers you can reinstate a layer using THERM's **Undo** feature. The **View/Show Voids/Overlaps** feature is very useful for detecting overlapping entities; however, multiple layers of overlaps may be difficult to detect because the first overlapping region will be displayed as blue, the second one of top of it will be displayed as white, the third on as blue, and so on.

5.3. Drawing Features

It is important to understand the following THERM features before starting your drawing.

5.3.1. Sticky Distance

The sticky distance is 3mm (1/8") when a drawing is at 100% scale, which corresponds to about half of one side of the cursor cross hair. As you zoom in or out, the sticky distance remains constant. This means that you can zoom in to do finer detail because the sticky distance will get smaller relative to your drawing as you zoom in and larger relative to your drawing as you zoom out. Sticky distance only applies to mouse clicks, not to numeric cursor positioning (see Section 5.5, "Drawing with the Keyboard" for a discussion of numeric cursor positioning).

5.3.2. Drawing Preferences

Use the **Options/Preferences** menu choice and go to the **Drawing Options** tab to change the following drawing settings:

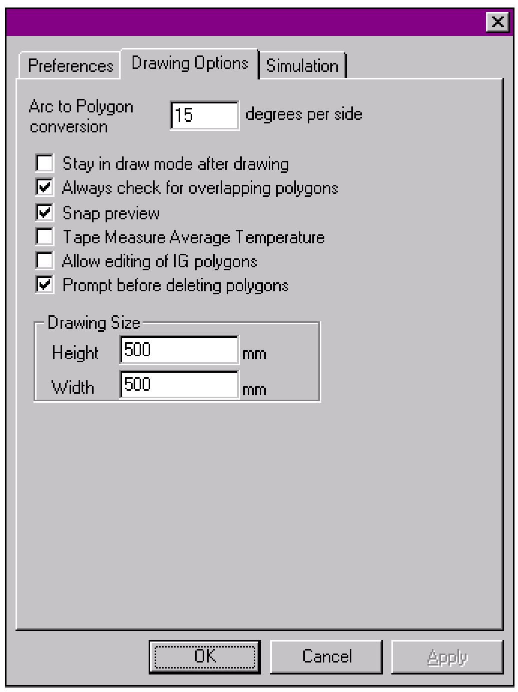


Figure 5-8. Setting the drawing preference options

- *Arc to polygon conversion:* This option tells THERM how to translate an arc in a dxf file into a polygon in a THERM file. The smaller the value, the more line segments THERM will construct when translating the arc. The default setting of 15° is recommended.
- *Stay in Draw mode after drawing:* This option turns the **Repeat Mode** on by default.
- *Always check for overlapping polygons:* This option tells THERM to check for overlapping regions each time you edit or draw a polygon. (Note: This checking procedure sometimes reports and overlap when there isn't one. If you repeatedly get an error message saying you have overlapping polygons, you may want to temporarily turn this feature off by unchecking the box.) The program will check for overlaps at simulation time; you can also check the drawing after it is finished using the **View/Show Voids/Overlaps** menu choice.

- *Snap Preview:* This option affects how the cursor reacts to movement of the mouse. **Snap Preview** causes the cursor to snap to any point that is within sticky distance. For complicated drawings this can be very time consuming because the program must check every point to see if it is within sticky distance. This option only affects what you see as you move the mouse, but does not affect snapping when you draw.

- *Tape Measure Average Temperature:* This feature enables display of the average temperature over a straight line segment measured by the tape measure. This feature is only available for a THERM file with current and available results.

- *Allow editing of IG polygons*: This checkbox allows you control whether a glazing system imported from WINDOW can be edited. If you activate this feature, you can then insert and delete points in a glazing system. Use this feature with extreme care. The default value is unchecked, meaning the feature is not activated.

- *Prompt before deleting polygons:* This feature determines whether the program prompts you before deleting polygons. The default setting is to ask for confirmation before deleting polygons. If this box is unchecked, the program will not ask for confirmation before deleting a polygon. Under this circumstance, a polygon that is inadvertently deleted can be restored using **Edit/Undo**.

- *Drawing Size:* This option displays the overall size available for the THERM drawing. These dimensions are modified automatically and will rarely need to be changed manually.

5.3.3. Snap Settings

THERM has several “snap” settings that can be used to make drawing easier. These settings instruct the drawing cursor to automatically go, or snap, to certain points, lines, or angles. How THERM decides where to snap is determined in part by the sticky distance of the cursor. If the cursor is snapping to points that you don't want, use the zoom feature to zoom in and decrease the sticky distance. (See Section 5.3.1 "Sticky Distance" for more details).

Use the **Snap Settings** tab under the **Options/THERM File Options** menu choice to control the “Snap” features in THERM.

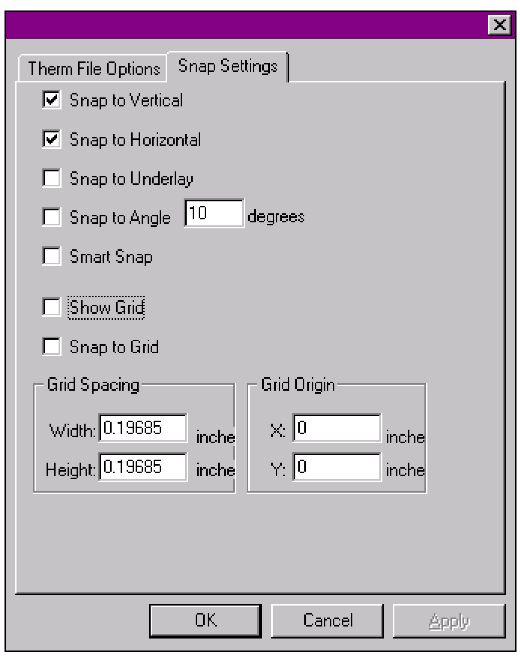


*Figure 5-9. Setting the **Snap Settings** options, accessed from the **Options/Therm File Options** menu*

- *Snap to Vertical:* If your drawing cursor is within the sticky distance of drawing a vertical line, the program will snap the cursor to the exact vertical position.
- *Snap to Horizontal:* If your drawing cursor is within the sticky distance of drawing a horizontal line, the program will snap the cursor to the exact horizontal position.
- *Snap to Underlay:* If your cursor is within the sticky distance of an underlay point as you trace a DXF underlay, the program will snap the cursor to that point. If two points on the underlay are both within the sticky distance to the cursor, the cursor may snap to the wrong point. Zooming in will decrease the sticky distance and allow the cursor to snap to the desired point.
- *Snap to Angle:* Much like **Snap to Horizontal** & **Vertical**, this feature allows the program to snap the drawing cursor to a defined angle.
- *Smart Snap:* This feature extends edges of polygons past the end points of polygons and checks to see if the point that you want to insert is close to one of these lines. This option is useful for lining up edges when there is a gap between them. The **View/Crosshair** menu choice, which turns the drawing cursor into a cross hair that fills the drawing area, can also be used for this purpose.
- *Show Grid:* This option allows you to control whether the grid shows on the drawing area.
- *Snap to Grid:* This feature forces the program to snap the drawing cursor to grid points. If you use this feature, you should also check the "Show Grid" option.
- *Grid Spacing:* This option allows you to change the spacing of the drawing grid. The default grid spacing is 5 mm for both width and height. The minimum grid space is 0.3 mm; a smaller spacing is not visible on most monitors.
- *Grid Origin:* By default, the origin of the grid points falls on the drawing origin; by changing the x and y values in the **Grid Origin** box, you can specify a different origin for the grid points.

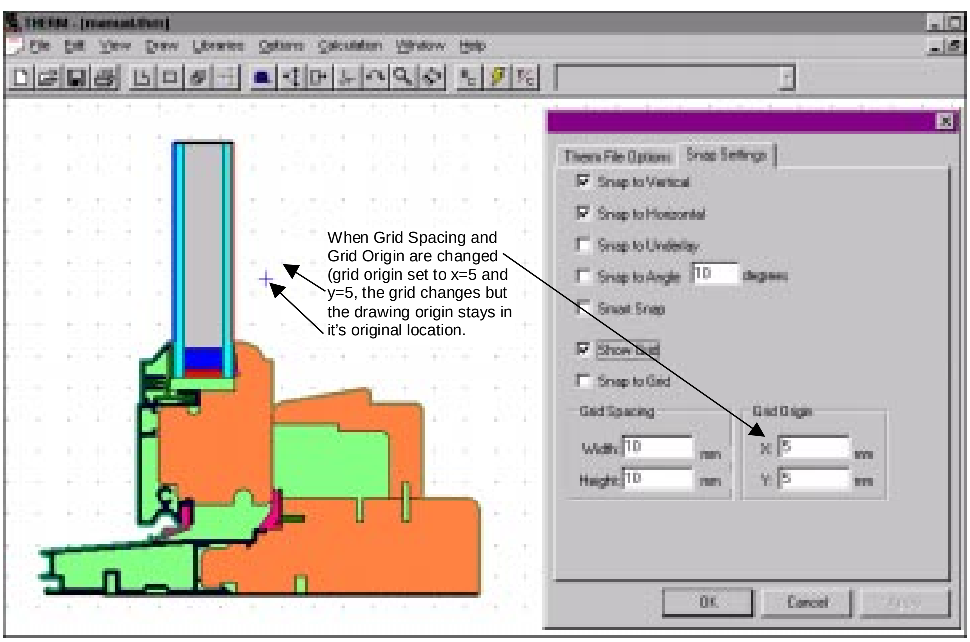


Figure 5-10. Changing the grid origin relative to the drawing origin

Snapping occurs according to a defined priority sequence. The list below shows the priority the program will use to determine what to stick the cursor to.

Snap Priority:

The snap priority is largely determined by what Snap Settings have been checked. The Snap Settings are discussed in the previous section. Assuming that the **Snap Settings** are turned on, THERM will snap points and polygons in the following priority order:

- *Polygon point:* The highest priority is to stick to a point on another polygon. This feature is always enabled. If the cursor is sticking to a point that you don't want it to stick to, you need to zoom in (click your right mouse button).
- *Underlay point* (for DXF only because bitmaps don't have defined points): The next level of priority is to snap to underlay points. This option can be turned on and off. See the previous discussion of **Snap Settings**, *Snap to Underlay*.
- *Grid point:* The third level of priority is to snap to a grid point. You can define the grid origin and spacing, and the "snap to grid" feature can be turned on and off. In the previous discussion of Snap Settings, see *Snap to Grid.*
- *Polygon line:* The fourth level of priority causes the cursor to stick to a line at the point perpendicular to where the mouse clicks if the cursor is within the sticky distance of that line. This feature cannot be turned on and off.
- *Smart Snap:* The fifth level of priority is enabled if **Smart Snap** is turned on. In the previous discussion of **Snap Settings**, see *Smart Snap.*
- *Snap to Horizontal/Vertical:* The sixth priority is to snap so that an exactly horizontal or vertical line segment is created. In the previous discussion of **Snap Settings**, see *Snap to Vertical* and *Snap to Horizontal.*
- *Snap to Angle:* The last level of priority is to snap to a defined angle. In the previous discussion of **Snap Settings**, see *Snap to Angle.*

5.3.4. Zoom

The zoom feature makes a drawing appear larger or smaller. Click the **Zoom** toolbar button or use the **View/Zoom** menu choice; a dialog box will appear with different percentage enlargements and reductions.

- This box will show the current zoom percentage. You can change this value to a specific percentage.
- The cursor becomes a magnifying glass. Place the magnifying glass over the center of the area to be magnified and click the left mouse button.

You can also use the mouse to zoom as you are drawing:

- *Zoom in* (Right mouse click): click the right mouse button to zoom in (make the drawing larger by a factor of 2). The program will zoom in, using the cursor position as the center of the newly zoomed screen.
- *Zoom out* (Shift right mouse click): hold the **Shift** key down and click the right mouse button to zoom out (make the drawing smaller).
- *Zoom to Fit* (Ctrl right mouse click): hold the **Ctrl** key down and click the right mouse button; the program will zoom the drawing to the largest scale at which the entire drawing still fits in the view window.
- *Scrolling:* You can scroll your drawing both horizontally and vertically while zoomed in.
- *Maximum Zoom:* You can zoom your drawing to 50,000%.

*NOTE: The drawing features in THERM were developed using a 32-bit operating system, Microsoft Windows NT™. THERM also runs on Windows 95™ and Windows 98™, which use 16-bit drawing functions. The zoom feature is one of the few places where this difference is apparent. If you are using Windows 95™ or Windows 98™ and you experience distortion of your drawing when zooming at a high level, make sure that the Windows 95 box is checked under **Options/Preferences**. Even with this box checked, you may still see some display distortion of your drawing. If this happens zoom out (Shift key + right mouse click) until the drawing appears accurate.*

5.3.5. Voids and Overlaps

In order for the automatic mesh generator to create a mesh there can be no voids or overlapping regions in your drawing. There are many drawing features that help you to create a valid model as you draw, such as snapping to points, filling voids, checking for overlapping and invalid polygons.

There is also a view option, **View/Show Voids/Overlaps**, that allows you to easily identify large voids and overlaps. This feature draws the model in white and displays all overlaps and voids in blue (the area outside the boundaries of the model will also be displayed in blue), and allows you to discover problems before you define the boundary conditions. It is also a useful feature to fix similar problems after autoconverting a DXF file; depending on how the DXF file was defined, overlapping polygons may be created during the autoconvert process. See Section 5.2.3, "DXF File Conversion" for more details. Figure 5-11 shows an example of this view option.

Voids and overlaps that are smaller than a pixel at the zoom level of the model when the **View/Show Voids/Overlaps** feature is selected may not show up even after subsequent zooms. THERM has a feature that can detect small voids and overlaps that is activated later in the modeling process. See Chapter 7 for more details.

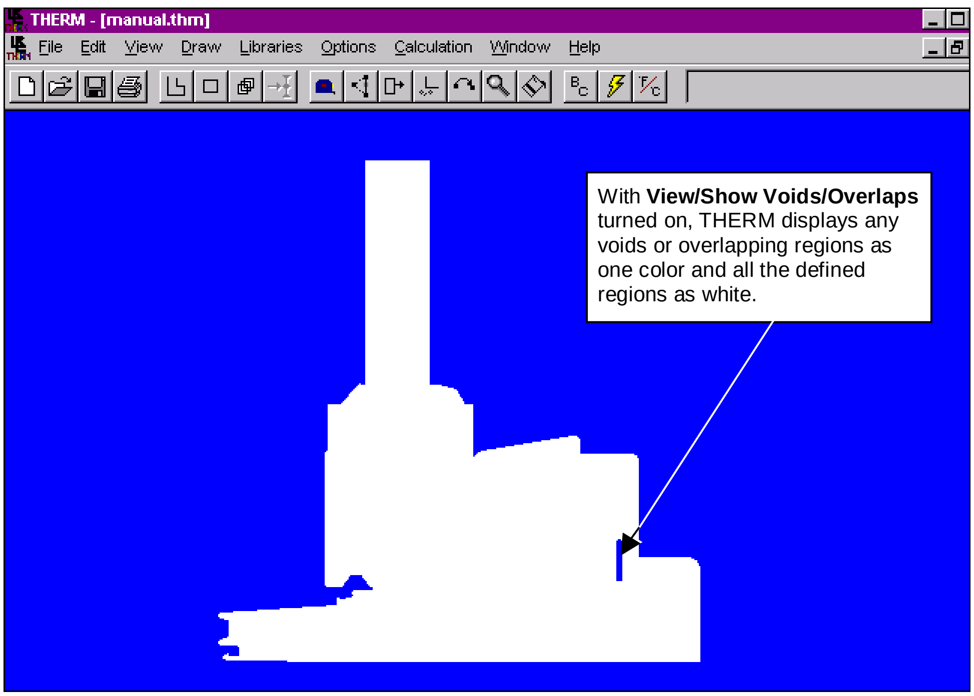


*Figure 5-11. Use the **Draw/Show Voids/Overlaps** menu to view voids.*

5.3.6. Other Drawing Features

Other drawing features that may help you create the cross-section geometry are:

- *Draw Repeat Mode:* This feature, accessed from the **Draw/Repeat Mode** menu choice or the **Draw Repeat Mode** toolbar icon, keeps you in the drawing mode you are currently in. For example, if you are drawing polygons with the **Repeat Mode** feature on, your cursor stays in "draw polygon" mode, so you don't have to continually reselect the **Draw/Polygon** menu choice or toolbar button.

- *Material Colors:* The **View/Material Colors** menu choice allows you to turn the polygon colors on and off.

- *Set Draw Color:* The **View/Set Draw Color** menu choice allows you to select a color for the drawing line when drawing polygons. This feature helps you keep track of where you are when drawing a complicated polygon. The default color is black.

- *Set Polygon Border Color:* The **View/Set Polygon Border Color** menu choice allows you to select a color for the borders of the polygons. The default color is black.

- *View Polygon Vertices:* The **View/Polygon Vertices** menu choice displays the vertices of all the polygons in the model. This feature can be very useful when drawing polygons that share a common complicated edge.

- *Select Materials:* You can select materials as you draw each polygon; when you are in "draw polygon" or "draw rectangle" mode, pick the material for the component you are about to draw from the material list in the right-hand section of the toolbar. This material will then be assigned to the polygon. You can also assign materials after the polygons have been drawn.

- *Set Drawing Scale:* This feature allows you to convert drawings to full scale (1" = 1"). Changing the drawing scale changes all of the dimensions in the drawing. *It should not be used to resize individual parts.* THERM will allow imported glazing systems to be rescaled, but this is not recommended because the properties of the glazing system are dependent on the dimensions of the geometry as defined in WINDOW. In this version of THERM, these properties are not corrected when the glazing system is rescaled.

1. Select the **Draw/Set Drawing Scale** menu choice.

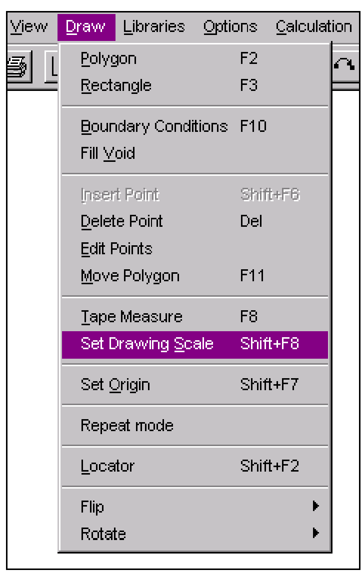


*Figure 5-12. Select **Set Drawing Scale** from the **Draw** menu.*

2. A small tape measure will appear. Put the tip of the arrow on the tape measure at the point from which you want to start measuring and press the left mouse button. Then move the tape measure arrow tip to the point where you want to finish measuring, and press the left mouse button (the cursor will snap to points within the sticky distance). Type in the actual line length value.

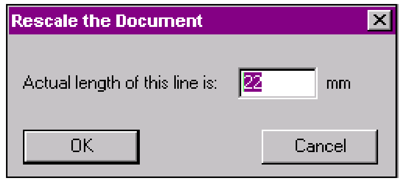


Figure 5-13. After measuring the distance, type in the dimension to which you want that distance to be set.

5.4. Drawing Using a Mouse

This drawing discussion assumes use of a mouse; this is the technique most likely to be used when tracing an underlay. See the discussion following ("Drawing Using a Keyboard") for methods to use the keyboard for drawing, which may be faster and more accurate than using a mouse if you have a dimensioned drawing.

For this example, we are tracing an underlay and will draw many polygons to represent the model. The figure below shows numbered steps for creating a polygon.

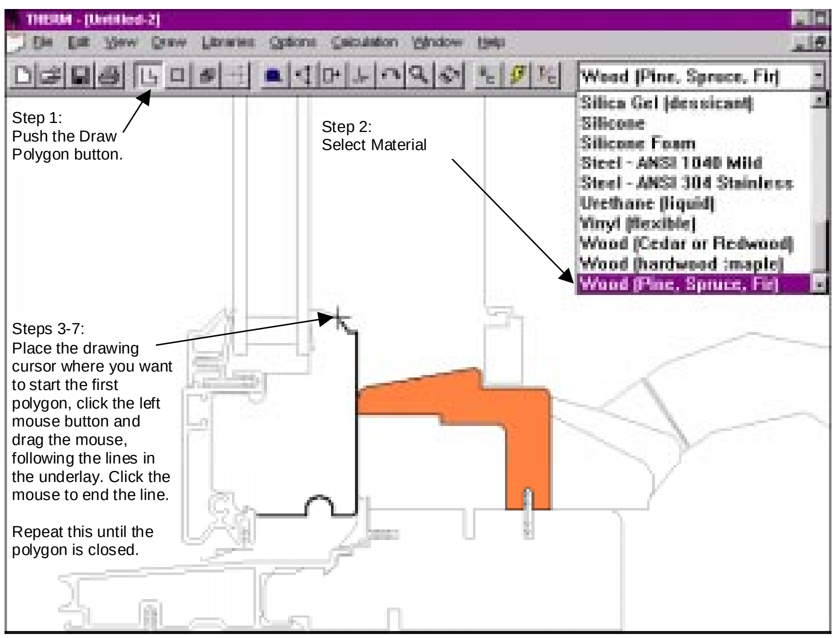


Figure 5-14. Steps for drawing cross-section geometry

1. To draw the first polygon, press the **Draw Polygon** toolbar button or use the **Draw/Polygon** menu item.
2. Pick the material for the polygon from the material list in the right-hand section of the toolbar.
3. Place the drawing cursor where you want to start the first polygon.
4. Click the left mouse button to set the starting point for the first line. (THERM will snap to the nearest vertex if you are using a DXF file).

5. Move the mouse to the right and a line will follow the mouse. Click the left mouse button when you get to the end point for the line, and THERM will draw the line. A point is automatically created at each end of this new line.

6. Continue drawing the edges of the polygon in the same manner, moving the cursor to the next line end point, and clicking the left mouse button to define that point.

7. When you only have one line left to draw, you can double-click the left mouse button, click on the first point, or press **C** to close the polygon.

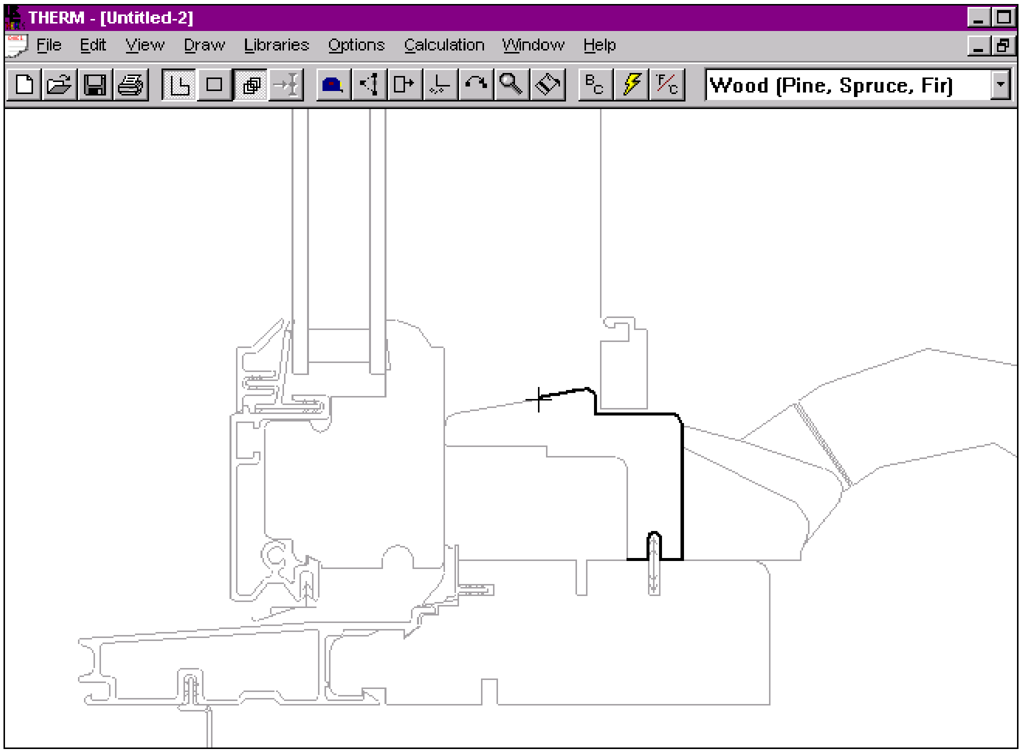


Figure 5-15. Drawing a polygon by tracing an underlay element

If you find that the program is "snapping" lines to points that you don't want, try zooming in. If you are still having problems, you may need to turn off some of the options in **Options/THERM File Options** menu, under the **Snap Settings** tab.

After you have drawn the first polygon, use the same procedure to draw all the other polygons required to provide an accurate representation of the cross-section. For THERM to perform its calculation, a cross section made out of multiple polygons must not contain any voids or overlaps. (See Sections 5.3.5, "Voids and Overlaps", and 6.3, "Error Detection in THERM" for more information). The "snap" feature (see Section 5.3.3, "Snap Settings") as well as THERM's dynamic checking as you draw are intended to minimize unintentional creation of voids and overlaps.

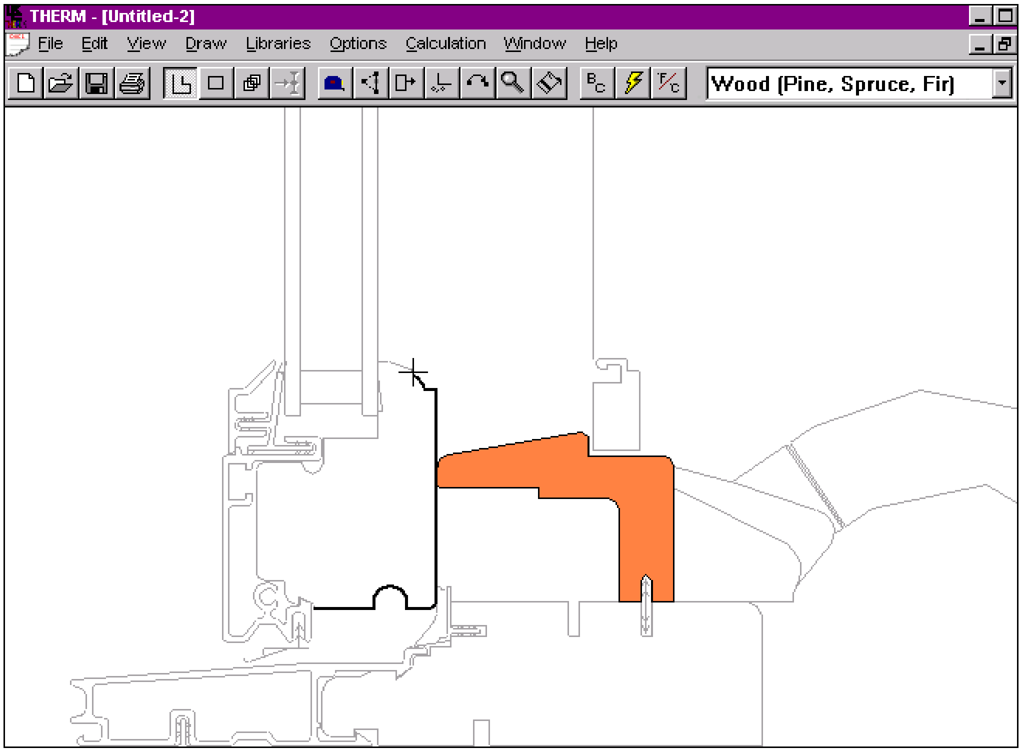


Figure 5-16. The first polygon is complete; start the next polygon.

5.4.1. Suggestions for Drawing Cross Sections

You will be able to successfully draw complicated cross-sections in THERM if the following guidelines are adhered to. These guidelines apply to the drawing of extrusions and how to best represent arcs and curves.

- **Extrusions:** Currently THERM does not draw multiply connected regions or "donuts". This limitation is most noticeable when drawing components created from extrusions (or pulltrusions). Here are some suggestions to aid in the creation of THERM models for extruded products.
  - The standard method for drawing an extrusion in a CAD program is to draw the outside edge of the extrusion and then draw the cavities. Creating a model in THERM uses a different logic. First draw the extrusion and then fill the cavities using the automatic fill void feature.
  - THERM can handle very complicated polygons but it is easier to keep track of what you are doing if you use many less complicated polygons. It is easy to forget where you started drawing the polygon if it is too complicated.
  - Complicated intersections between two pieces of the same material in intimate thermal contact can be simplified without effecting the thermal performance of the cross section.
  - When breaking up extrusions you should avoid introducing detail into the cross section to do this:
    - Make the breaks at vertices that already exist in the drawing.
    - Avoid creating acute angles.
    - Avoid creating very small rectangles.

The next two figures give an example of how the above drawing principles can be applied to an extruded product.

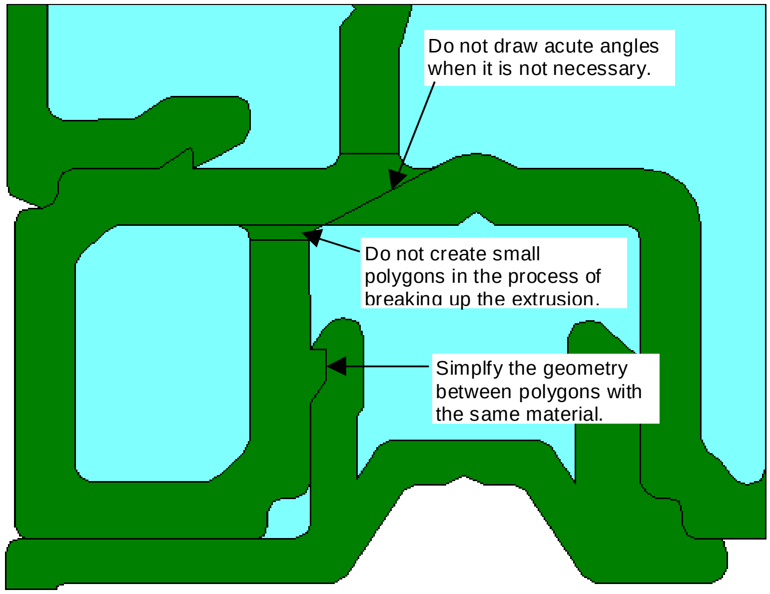


Figure 5-17. This cross section illustrates drawing mistakes when breaking up an extrusion.

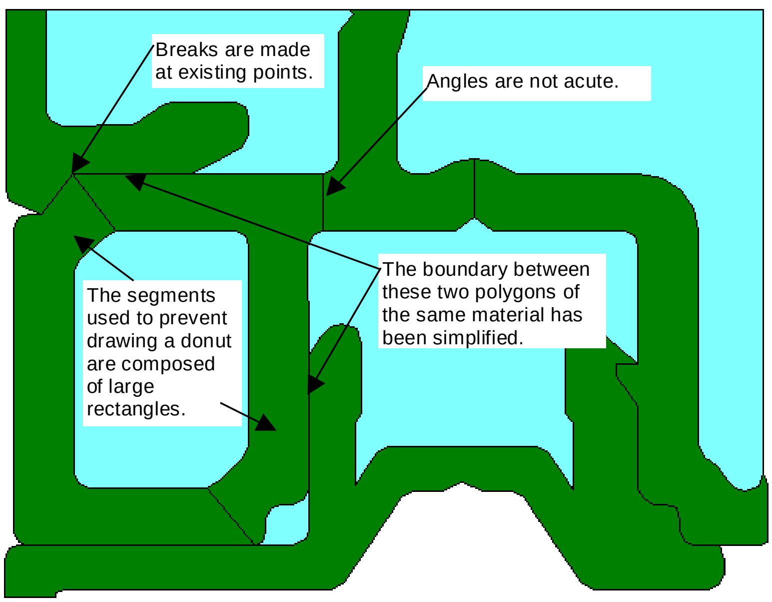


Figure 5-18. This cross section illustrates a better way to draw the extrusion geometry in Figure 5-17.

- **Arcs and curves:** The default value converts arcs into line segments every 15 degrees. This level of detail is appropriate for curves with a radius greater than 3mm. If the curve has a radius greater than 1 mm and less than 3 mm then breaking up the arc every 30 degrees (or combining two segments on the DXF file underlay) is adequate. If the radius of the curve is less than 1mm then every 45 degrees (or combining three segments on the DXF file underlay) is sufficient. These guidelines are based on an attempt to keep the difference between the approximated curve and the real curve small, approximately 0.1 mm. This distance can be checked using the tape measure. Using many points to model curves in THERM may create problems for the mesher. The guidelines listed above should result in a good representation of the geometry for the purposes of thermal analysis.

5.5. Drawing Using the Keyboard (Numeric Cursor Positioning)

If you are working from a dimensioned drawing, it may be easier and more accurate to use the keyboard and THERM's numeric cursor positioning rather than using the mouse. Numeric cursor positioning allows you to move the cursor to either an absolute or relative location by typing a number and pressing an arrow key.

Follow the steps below to draw polygons with the keyboard:

1. Type **Alt-D**: Shows the **Draw** menu.
2. Type **P** or **R**: With the **Draw** menu showing, type **P** for Polygon or **R** for Rectangle. The cursor will appear on the drawing area. (These functions can also be accessed with the mouse by clicking the appropriate toolbar button).
3. Move the cursor to the starting point of the drawing: There are several different methods for moving the cursor to where you want to start:
    - If you want to start drawing at the origin, press the **Home** key.
    - If your starting point is not at the origin, type the x coordinate; the **Step Size** dialog box will appear. Select **Absolute** (**Relative** is the default), and press the right arrow key; then type the y coordinate followed by the **Up Arrow** key, and the cursor will be positioned at those coordinates.
4. Press **Enter**: Press the **Enter** key to start drawing (the equivalent of clicking the mouse button).
5. Type the first dimension (a number): Type the first line length for the polygon you are going to draw. The **Step Size** dialog box appears, and the step size value is set to the dimension you typed. *Do not* press **Enter** and do not click on the **OK** button with the mouse. (The **OK** button is used when you are setting the step size for future use.)

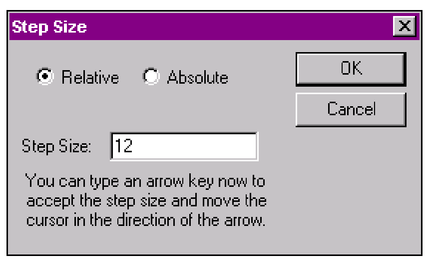


*Figure 5-19. Type a number to bring up the **Step Size** dialog box.*

6. Press the **Up**, **Down**, **Left**, or **Right Arrow**: Press the appropriate arrow key (i.e., the right arrow to draw a line to the right); a line of the specified step size will be drawn in the direction of the arrow.
7. Press **Enter** to commit the line: Press **Enter** to commit the line to the drawing. The new line will appear; a vertex has been established where the cursor is positioned.
8. Type the next dimension (a number).
9. Press the appropriate arrow key.
10. Press **Enter** to commit the line.
11. Continue drawing lines until you have only one line remaining to complete the polygon.
12. Press **C** to **Close**: Press **C** to draw the last segment of the polygon. The program will automatically draw a line from the point where the cursor is resting to the first point drawn for the polygon.

Other conventions to know when using the keyboard to draw:

- ***Enter** starts drawing:* Pressing the **Enter** key to start drawing a polygon will cause the cursor to snap (according to the snap settings). This makes it easy to start drawing from a point that you have already defined.

- *Mouse-driven commands:* Mouse-driven commands work even when using the keyboard; there are several short-cut keys designed for keyboard drawing. (See Sections 3.9 and 3.10 for more details about short cuts).

- *Absolute Locations:* Absolute locations can be entered by using the **Absolute** setting in the **Step Size** dialog box. For example, to move the cursor to x=10 and y=5, click on the **Absolute** button, type **10**, press the up arrow, type **5**, and press the right arrow.

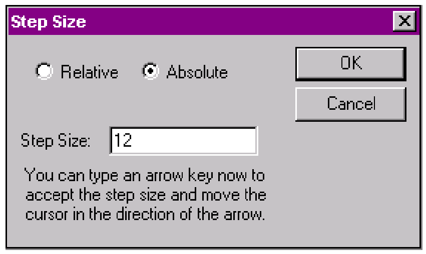


*Figure 5-20. Type the step value to bring up the **Step Size** dialog box, and click on the **Absolute** button.*

- *Ctrl + arrow key:* If you press the **Ctrl** key while moving the arrow keys, the cursor will move 10% of the specified step size. For example, if the step size is set to 1 mm, pressing the **Ctrl+Right Arrow** keys will move the cursor 0.1 mm to the right. This feature can be very useful when extreme accuracy is needed to place a point or line.

- *Space bar*: Press the **Space bar** if you want to snap to the point or polygon that is within the sticky distance of the cursor.

- *End:* While you are drawing a polygon, you can press the **End** button to get back to the most recent previous point you drew.

- *Esc:* If you draw a point by mistake, press the **Esc** button and it will be deleted. If you press **Esc** again, a dialog box will give you the choice of deleting the point previous to the last point you just deleted (this can be done several times, but the video display tends to get garbled), deleting the polygon, or canceling the **Escape** command.

5.6. Editing Polygons

Once a polygon has been drawn, it can be edited. For example, the entire polygon can be moved; points can be added, deleted or moved; etc. Because a rectangle is one type of polygon, any reference in this manual to polygons also applies to rectangles.

5.6.1. Selecting a Polygon

Several different techniques can be used to select a polygon:

- *Left Mouse Click:* Put your mouse cursor over the polygon you want to select and click the left mouse button. You can tell that a polygon has been selected because all the points on it will be displayed as squares.

- ***Ctrl** or **Shift** left mouse click:* If you want to select multiple polygons, you can hold either the **Ctrl** or **Shift** keys down as you click your mouse on the polygons to be selected.

- *Tab:* You can use the **Tab** key to select any polygon in the drawing; each time you press **Tab**, the program will select the next polygon. By pressing **Tab** multiple times, you can "tab around" to all the polygons in your drawing. This is sometimes a good way to select very small polygons that you can't easily select using the mouse.

- *Selection box:* You can draw a "selection box" around a polygon or a group of polygons by clicking and holding down the left mouse button in the drawing area and dragging the mouse in a diagonal motion so that a dashed box is drawn around the polygon(s) that you want to select. Release the mouse button, and the polygons surrounded by the dashed box will be selected.

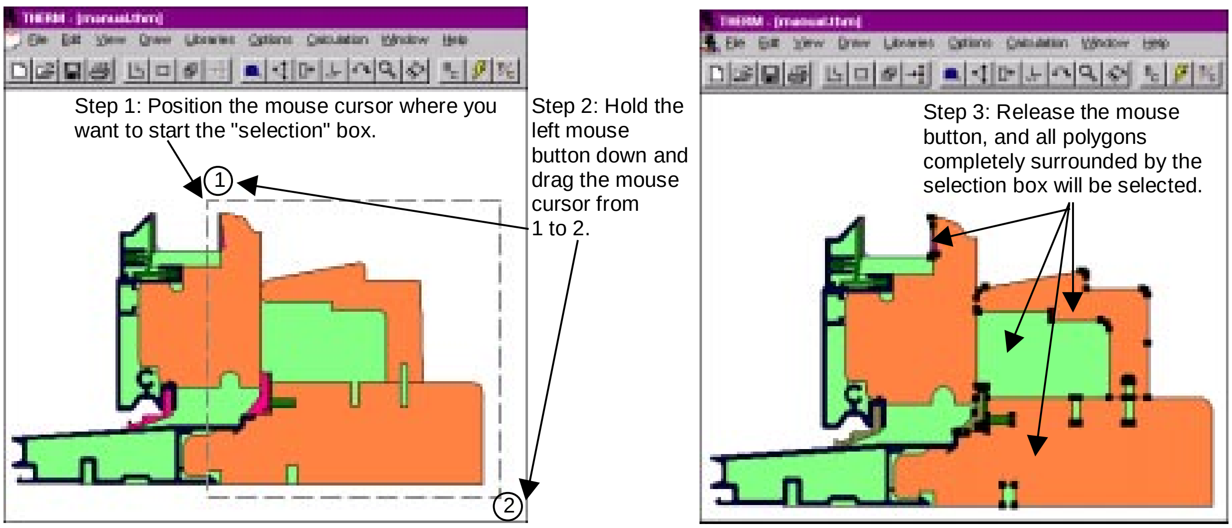


Figure 5-21. Selecting polygons using the selection box

- *Select Special:* You can use the **Edit/Select Special** menu choice to select polygons or boundary conditions of a certain type or ID. This menu choice brings up a dialog box that allows you to specify the type of object you want to select. This is an efficient way to change the materials of many identical objects with one command.

  When you do a calculation, you will occasionally get an error message saying that there is a bad polygon in the model. This message returns a polygon ID number. Using **Select Special** allows you to quickly locate that polygon in the model. See Chapter 7 for information about fixing invalid polygons.

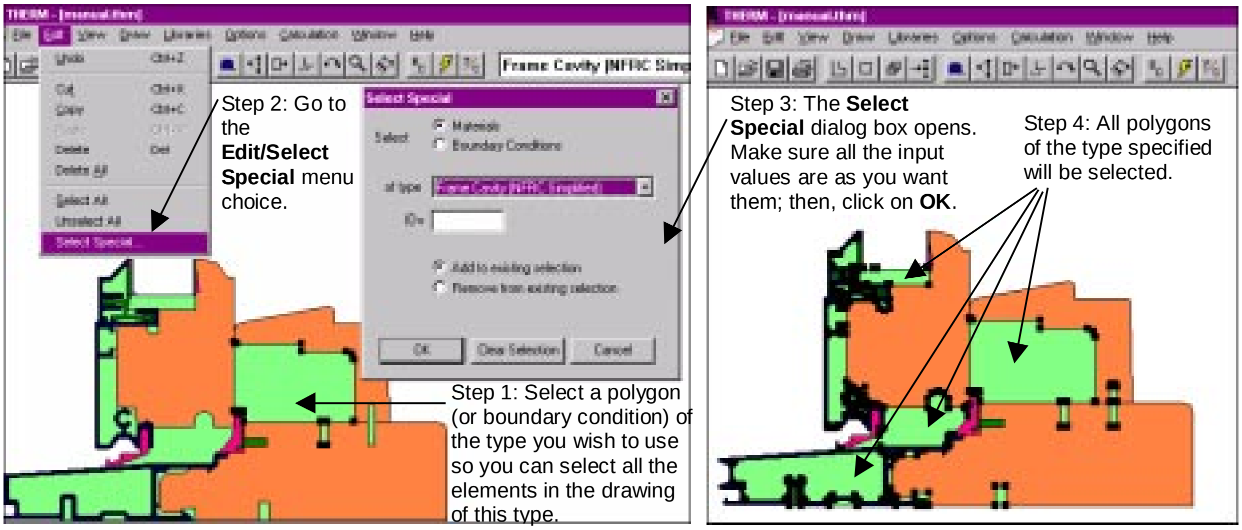


*Figure 5-22. Using **Select Special** for selecting polygons*

5.6.2. Moving a Polygon

To move a polygon:

1. Click the mouse on the **Move** toolbar button, or use the **Draw/Move** menu choice.

2. Place the mouse cursor inside the polygon that you want to move. Click and hold down the left mouse button as you move the mouse to the new location; release the mouse button. The polygon will move to the new location. Any snap settings in effect during the move will apply to the moved polygon.

When moving a polygon, you can hold the **Shift** key and the left mouse button down together as you drag the mouse and move the object; in this situation, THERM constrains the movement to the horizontal and vertical directions.

It is recommended that you use the drawing locator feature when moving multiple polygons.

1. Place the locator (select the **Draw/Locator** menu choice or press **Shift F2**, and click the left mouse button) on a point on one of the polygons to be moved; this point will be used to place the polygon in its new location.

2. Select all of the polygons to be moved, using any of the methods discussed in Section 5.6.1.

3. Cut (**Ctrl-X**) the selected polygons.

4. Place the locator (use **Shift F2** and click the left mouse button) at the location to which you want the point selected on the polygon in Step 1 to move. (The locator can be moved with the keyboard **Step Size** functions. It will stick to existing points within the sticky distance.)

5. **Paste** (**Ctrl-V**) the selected polygons. If no overlaps are created, the polygons will be pasted in the new location.

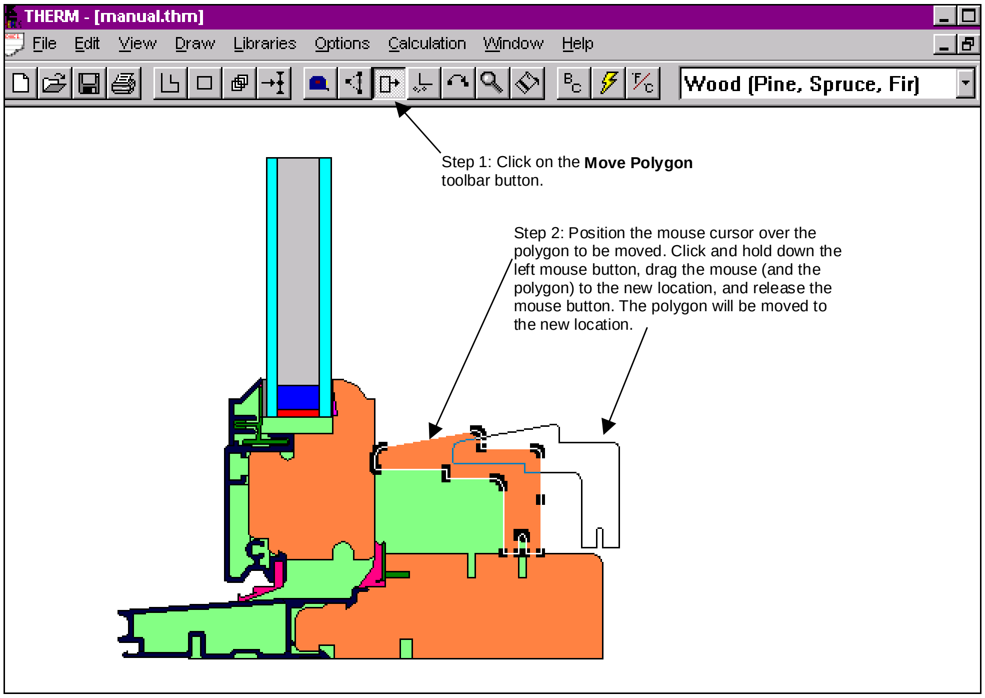


Figure 5-23. Moving a polygon

5.6.3. Deleting a Polygon

To delete a polygon:

1. Select the polygon (click the left mouse button while the cursor is inside the polygon) that you want to delete.

2. Press the **Delete** key on your keyboard.

3. A message will appear asking if you want to delete the selected polygon. If you do, press **Enter** to accept the default “Yes” response. (*Note:* You can disable this question in the **Drawing Options** tab accessed from the **Options/Preferences** menu, by unchecking the "Prompt before deleting polygons" checkbox.)

4. The polygon will disappear. (If you delete a polygon by mistake, use the **Edit/Undo** menu choice or **Ctrl-Z** to have the program reinstate the polygon).

5.6.4. Moving a Point

To move a point on a polygon:

1. Click the mouse on the **Edit Points** toolbar button or use the **Draw/Edit Points** menu choice.

2. Select the polygon (click the left mouse button while the cursor is positioned inside the polygon) so that the vertices are displayed.

3. Move the cursor to the point to be moved.

4. A double-arrow cursor is displayed when you are within sticky distance of the point.

5. Use the **Step Size** and **Arrow keys** to move the point, or click and drag the point with the left mouse button.

6. The lines attached to the point are pulled along with it.

When moving a point, you can hold the **Shift** key and the left mouse down together as you drag the mouse to move the point; in this situation, the program constrains movement to the horizontal and vertical directions.

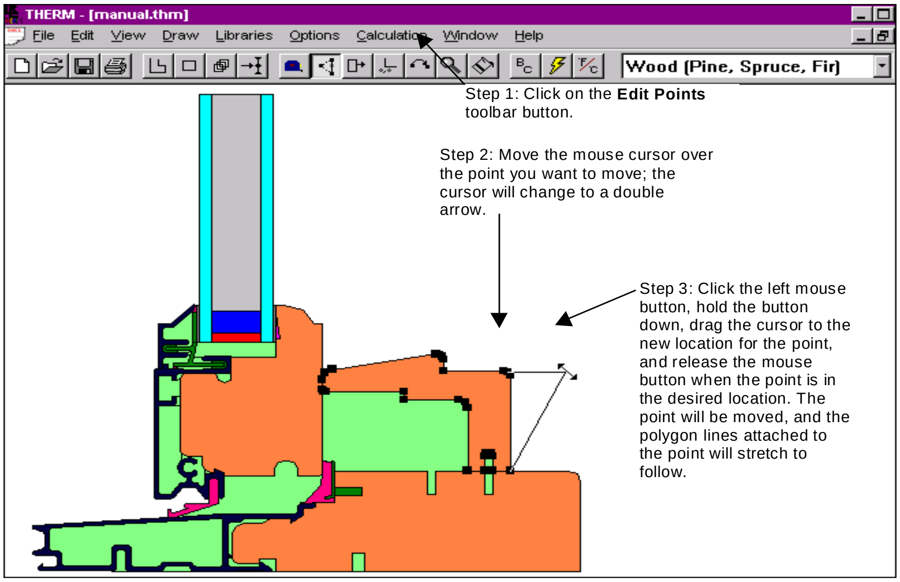


Figure 5-24. Moving a point on a polygon

5.6.5. Moving a Line

To move a line on a polygon:

1. Select the polygon (by clicking the left mouse button) so that the points are displayed.

2. Press the **Edit Point** toolbar button, or use the **Draw/Edit Point** menu choice.

3. Move the mouse cursor over the line you want to move; a four-point arrow cursor appears when you are in a position that allows you to move a line. (Note that you need to be outside the sticky distance of any nearby points in order to see the four-point arrow; use the zoom in feature by clicking the right mouse button if you are sticking to points).

4. When the four-point arrow cursor appears, hold down the left mouse button and drag the line to the new location, or use the arrow keys to move the line a specified distance. When the line is in place, release the mouse button, which will cause the line to move and the polygon to be resized.

When you are moving a line, you can hold the **Shift** key and the left mouse down together as you drag the mouse to move the points; and the program will constrain the movement to the horizontal and vertical directions.

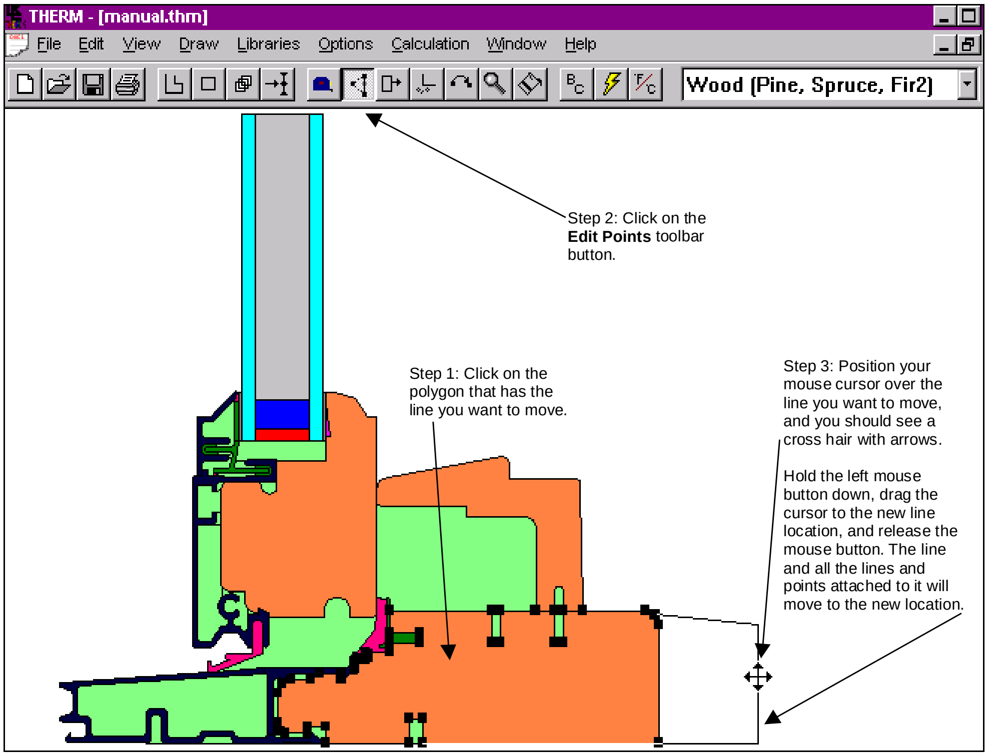


*Figure 5-25. Move a line by pressing the **Edit Point** button and placing the "move line" cursor over the line to be moved; hold the left mouse button, and move the mouse in the desired direction.*

5.6.6. Adding a Point

To add a point on a polygon:

1. Move the mouse cursor over the polygon that is to contain the new point, and press the left mouse button. This selects the polygon so that the points of the polygon are displayed.
2. Press the **Insert Point** toolbar button, or use the **Draw/Insert Point** menu choice (**Shift-F6**).
3. Move the mouse cursor to the point on the polygon edge where you want to add the new point. When the mouse cursor is properly positioned, click the left mouse button, and the new point will be displayed. It is also possible to use the keyboard input method (see Section 5.5) to move the cursor to the location where you want the point to be inserted, and then use the **Draw/Insert Point** menu choice. The **Insert Point** command does not recognize the snap priorities. If you are using the keyboard input method to insert a point at a set distance from an existing point, move the cursor near that point, and press the spacebar so the cursor snaps to that point. This will ensure that the point is inserted the exact distance specified by the keyboard input method.

A common cause of the message saying the program cannot create overlapping regions is when you try to insert a point within the sticky distance of an existing point. In this situation, zoom in to the drawing by clicking the right mouse button.

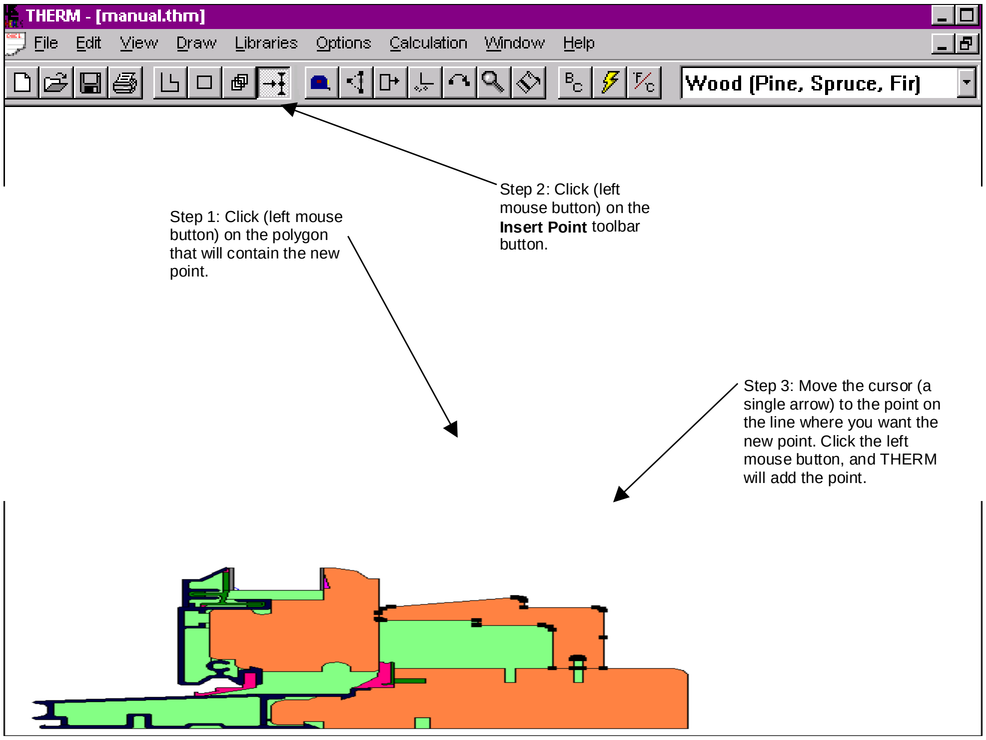


*Figure 5-26. Add a point by pressing the **Insert Point** button and clicking where the point is to be inserted.*

5.6.7. Deleting a Point

To delete a point on a polygon:

1. Click the mouse on the **Edit Points** toolbar button, or use the **Draw/Edit Points** menu choice.
2. Select the polygon (click the left mouse button) so that the points are displayed.
3. Move the cursor to the point to be deleted (get within sticky distance of the point).
4. A double-arrow cursor is displayed.
5. Press the **Delete** key on your keyboard, and the point will be removed. If you delete a point by mistake, use the **Edit/Undo** menu or **Ctrl-Z** to undelete it.

5.7. Flip and Rotate

THERM has two drawing features that allow you to manipulate the entire cross section, Flip and Rotate. You can use these functions to create sloped cross sections, as well as to use the same components multiple times in different orientations. Keep in mind that:

- Complicated drawings may take a few seconds to flip.
- If the drawing isn't visible in the drawing area, use **Ctrl-Right** mouse button to center it.

5.7.1. Flip

From the **Draw** menu, select the **Flip** choice, and then either the **Horizontal** or **Vertical** options. Figure 5-28 shows the result of horizontally flipping the cross section in Figure 5-27.

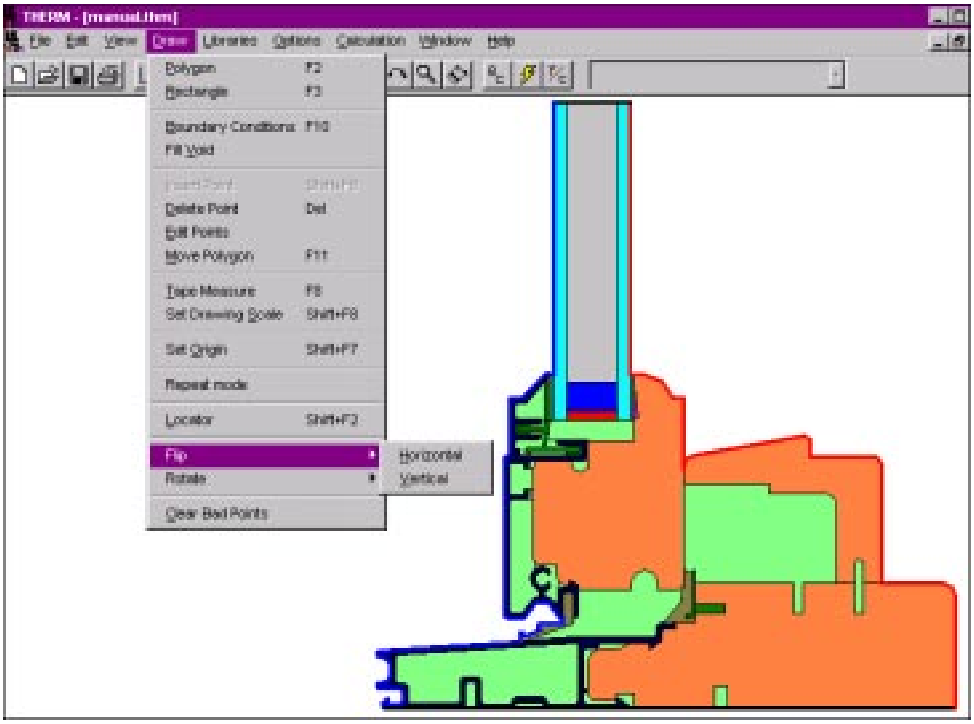


*Figure 5-27. Select **Flip** from the **Draw** menu.*

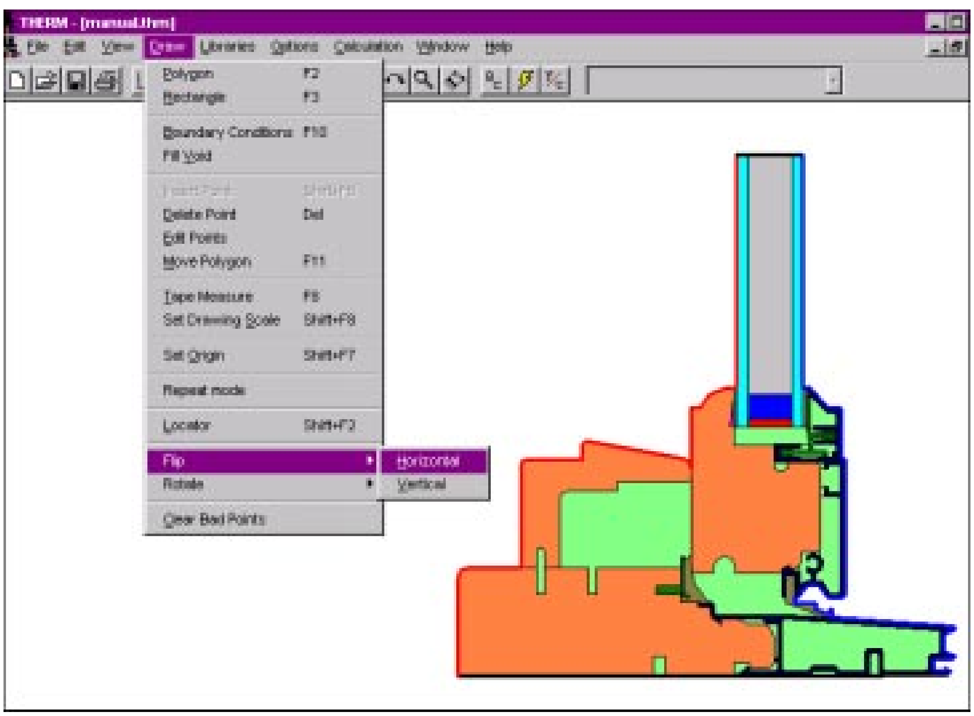


Figure 5-28. The cross section has been flipped in the horizontal direction.

5.7.2. Rotate

From the **Draw** menu, select the **Rotate** choice, and then either the **Left 90°, Right 90°, 180°** or **Degree** options. Choosing **Degree** displays a dialog box which allows you to enter the exact degrees of rotation. Figure 5-30 shows the result of horizontally flipping the cross section in Figure 5-29.

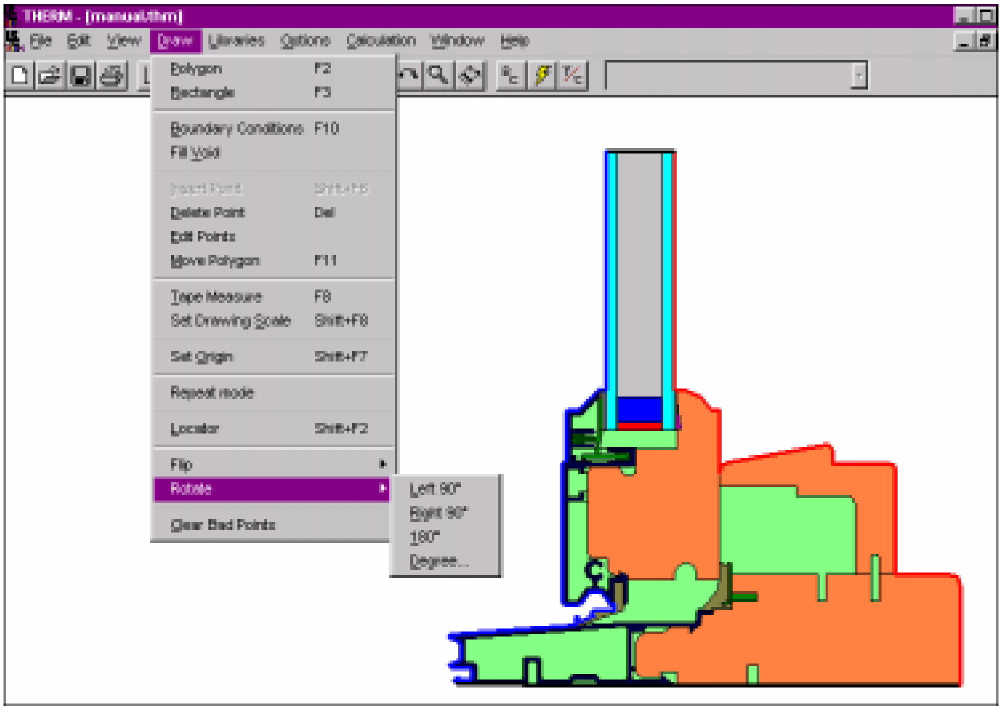


*Figure 5-29. Select **Rotate** from the **Draw** menu.*

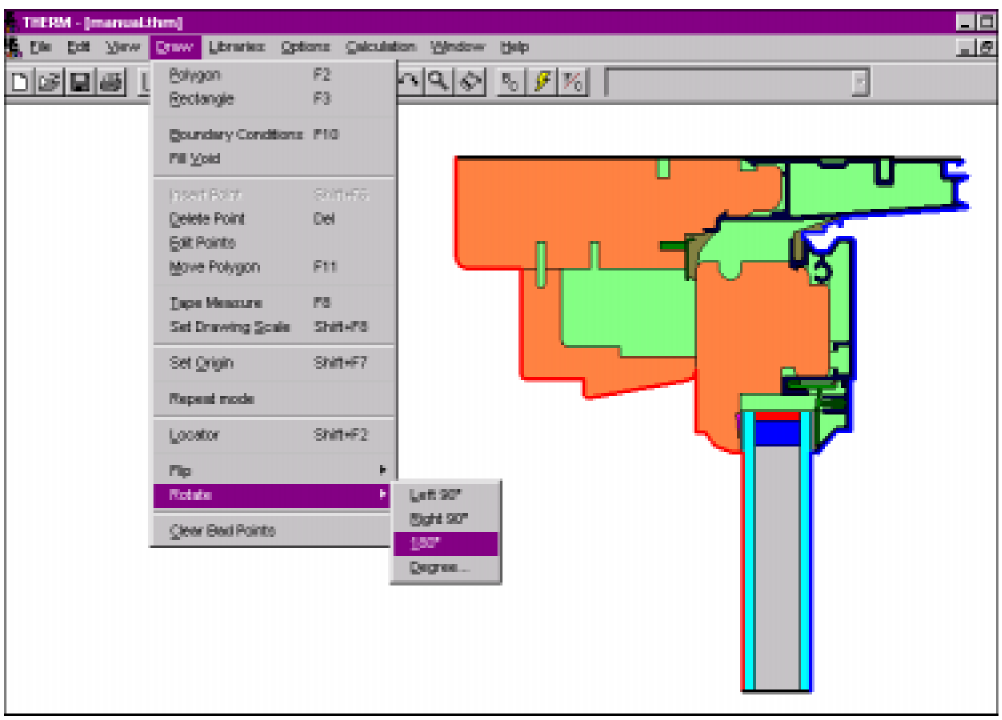
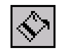


Figure 5-30. The cross section has been rotated 180 degrees.

5.8. Fill Void

THERM has a drawing feature that allows you to create a polygon by "filling" a void or undefined area in the cross section that is completely surrounded by other polygons. When this polygon is created, THERM automatically creates all the necessary points to match the points on the surrounding polygons. This feature eliminates the need for you to draw these potentially complicated polygons.

Figure 5-31 contains a cross section which has an undefined area completely surrounded by polygons, which is a perfect candidate for using the fill void feature. To create a polygon for this cavity, follow these steps:

- Click the left mouse button on the **Fill Void** toolbar button. 
- The drawing cursor will change to the **Fill Void** cursor. 
- Place the **Fill Void** cursor inside the undefined are that you want to fill with a polygon.
- Click the left mouse button, and THERM will draw a polygon that fills the area and contains points matching all the vertices on the adjacent polygons, as shown in Figure 5-32.
- You may need to select the newly created polygon and change the material. See Section 5.10, "Assigning Materials After Drawing Polygons".

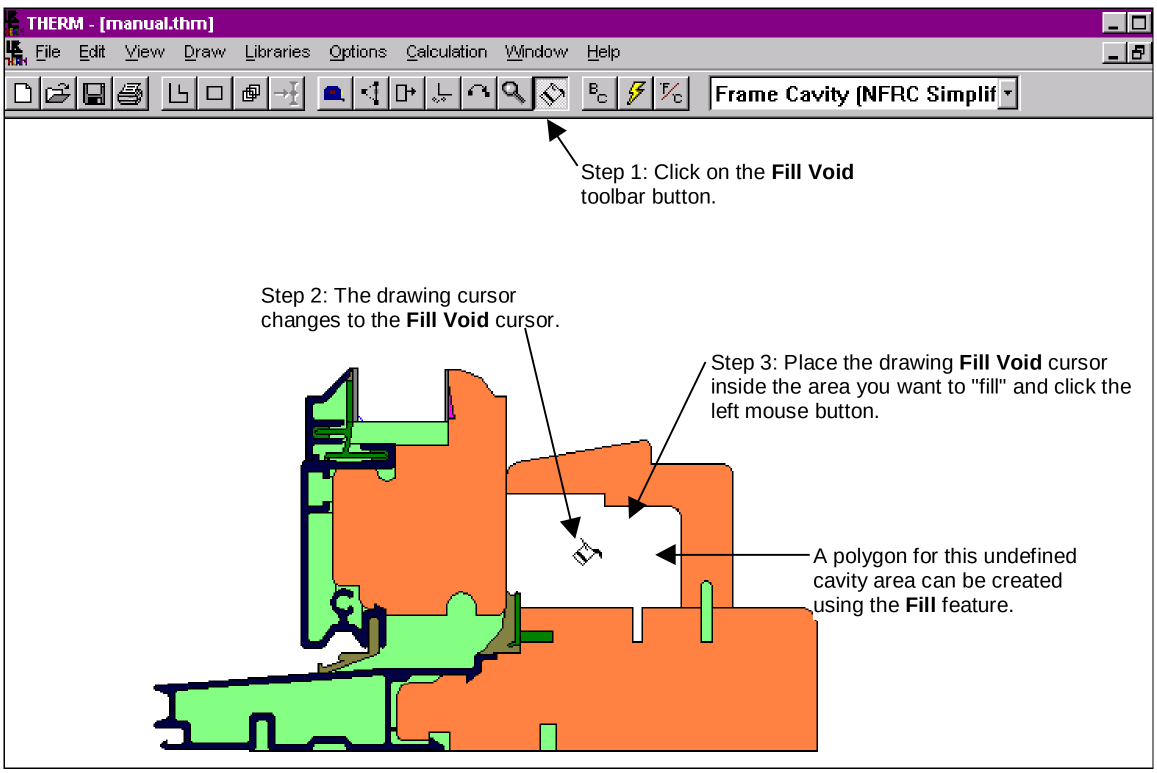


*Figure 5-31. Click on the **Fill Void** toolbar button, place the drawing cursor inside the void to be filled, and click on the left mouse button.*

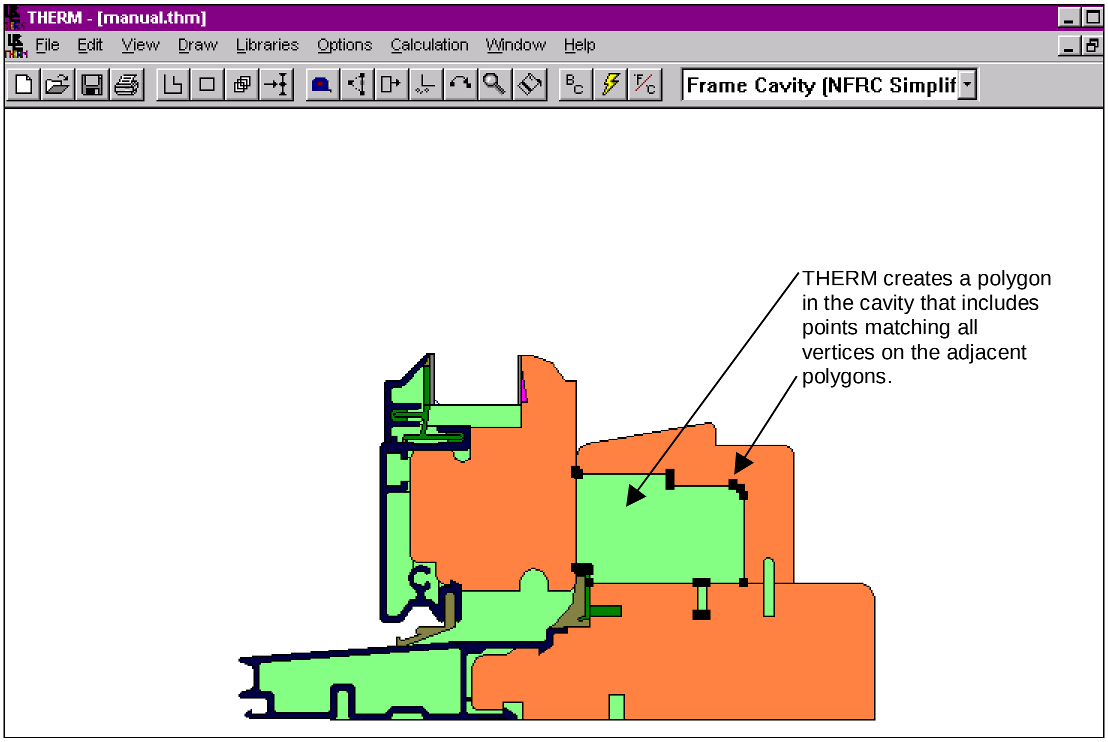


Figure 5-32. The ***Fill Void*** *feature creates a polygon in the cavity containing points matching all vertices on the adjacent polygons.*

5.9. Inserting a Glazing System

When you have drawn the nonglazed elements of your cross section, you are ready to add a glazing system. If you used **Autoconvert** for a DXF file, you should delete any polygons that may have been automatically drawn by THERM for the glazing system. Although it is possible to draw the glazing system from individual components and assign materials to them, it is much easier to insert a glazing system from the WINDOW library. Material properties and boundary conditions are predefined if you use this feature. A sample glazing system library (glzsys.w4) is included with THERM. Additional glazing systems can be created using the WINDOW program.

5.9.1. Inserting a WINDOW4 Glazing System

THERM can read the **Glazing Systems Library** created by WINDOW4 and automatically insert glazing systems into your cross section drawing if you do the following:

1. Specify the location for the glazing system to be inserted by selecting the **Draw/Locator** menu choice.

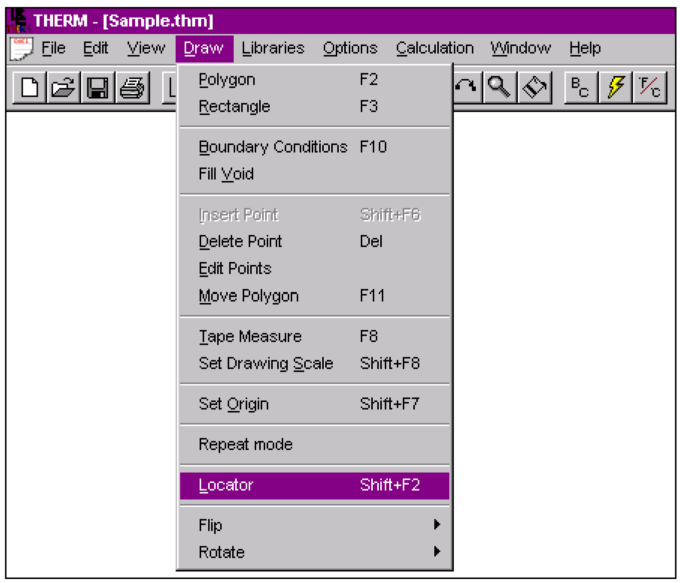


*Figure 5-33. Select **Locator** from the **Draw** menu.*

2. Click with the left mouse button on the location for the lower left-hand corner of the inserted glazing system; a small circle will appear at the selected location.

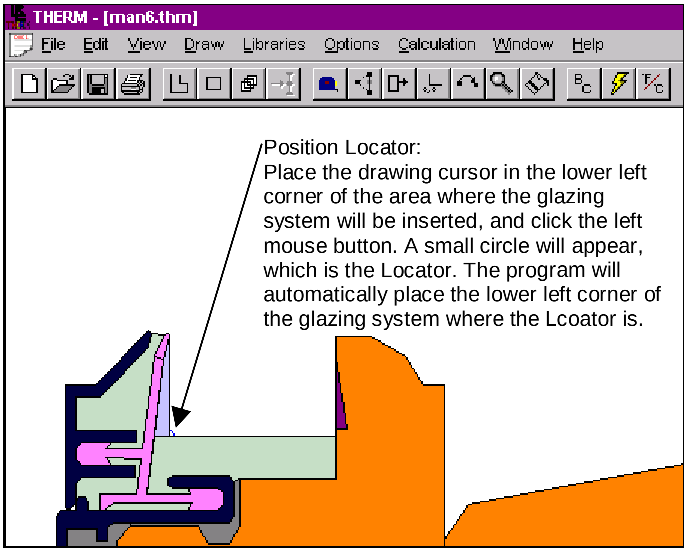


Figure 5-34. A small circle indicates the location of the lower left corner of the glazing system to be inserted.

3. Select the **Libraries/Glazing Systems** menu choice.

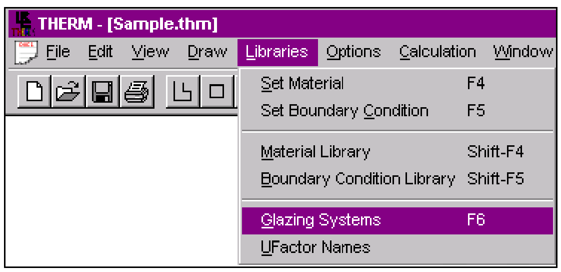


*Figure 5-35. Select the **Glazing Systems** choice from the **Libraries** menu.*

4. From the **Glazing Systems** dialog box, shown below, specify the path by which the program can find the **Glazing System Library** file, GLZSYS.W4. Use the **Browse** button to find GLZSYS.W4 files; all **Glazing Systems** defined in the specified GLZYSYS.W4 library will appear in the **Glazing System** pull-down list.

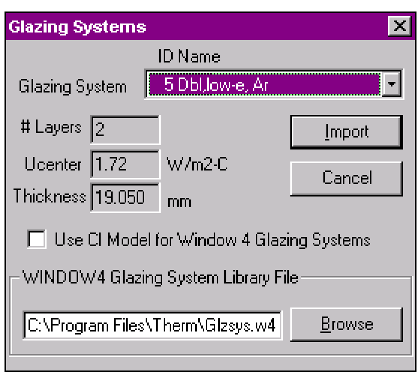


*Figure 5-36. Select a WINDOW4 glazing system from the **Glazing Systems** dialog box.*

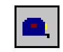
5. Pick the desired **Glazing System** from the pull-down list. If you want THERM to use the Condensation Index (CI) Model when simulating the cross section, check the **Use CI Model for Window 4 Glazing Systems** check box. If you do not check this box when you import the glazing system and later decide to

do a CI calculation, you may select this model under **Options/Therm File Options**. If you select this option, you will get U-factors based on the CI algorithms which are slightly different from the algorithms in WINDOW4.1; these U-factors are not consistent with current NFRC U-factor procedures.

6. Press the **Import** button.

7. Another dialog box will appear, asking for the position of the glazing as well as dimension information.

While the **Insert Glazing System** dialog box is open, it is possible to use the **Tape Measure** toolbar button to go to the drawing, determine the dimensions needed, highlight the dimension and press **Ctrl-C** to copy the value from the **Tape Measure** results box, and then press **Ctrl-V** to paste the value into the **Insert Glazing System** input boxes.



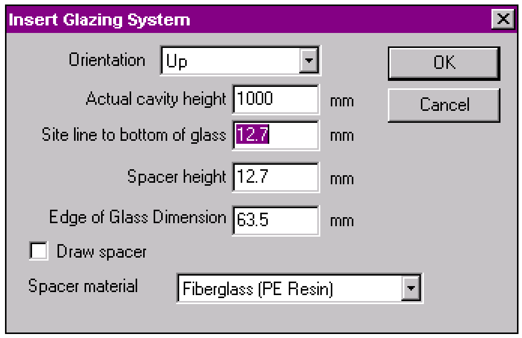


Figure 5-37. Specify the location and dimensions for the inserted glazing system.

Orientation — Orientation of the glazing system. **Up** means the spacer will be at the bottom of the drawing. **Down** turns the glazing system upside down, placing the spacer at the top of the drawing. This orientation can be changed using the flip and rotate features once the glazing system has been imported. **Default:** Up.

Actual cavity height — The total height of the glazing system cavity. **Default:** 39.37 inches (IP); 1,000 mm (SI). This value is only important when doing a CI calculation.

Site line to bottom of glass — Distance from the frame sight line to the bottom of the glass, which forms the demarcation point between the frame and the edge. For NFRC window modeling purposes, the sight line is the demarcation point between the frame and the edge U-factor calculations. **Default:** 0.5 inches (IP); 12.7 mm (SI).

Spacer height — Distance from the top of the spacer to the bottom of the glazing system. This dimension determines how THERM will draw the generic spacer or how much space to leave if the **Draw spacer** box is not checked. **Default:** 0.5 inches (IP); 12.7 mm (SI).

Edge of Glass Dimension — Dimension of glazing that is used to calculate the impact of the frame "edge effects." **Default:** 2.5 inches (IP); 63.5 mm (SI).

Draw spacer — If this box is checked, the program will draw a rectangle of the chosen material when the glazing system is imported. If it is unchecked, the program will leave the spacer area blank, and you can draw your own spacer or paste in a spacer from another THERM file (see Section 5.12, "Creating a Component Library" later in this section). **Default:** unchecked.

Spacer material — If the **Draw spacer** box is checked, this pull-down choice will determine the material of the rectangle that the program draws to represent the spacer.

The drawing below illustrates the dimensions being defined in the **Insert Glazing System** dialog box. These dimensions are used by the program to determine how much area to leave for the spacer, and to determine the demarcation point between the frame and the edge.

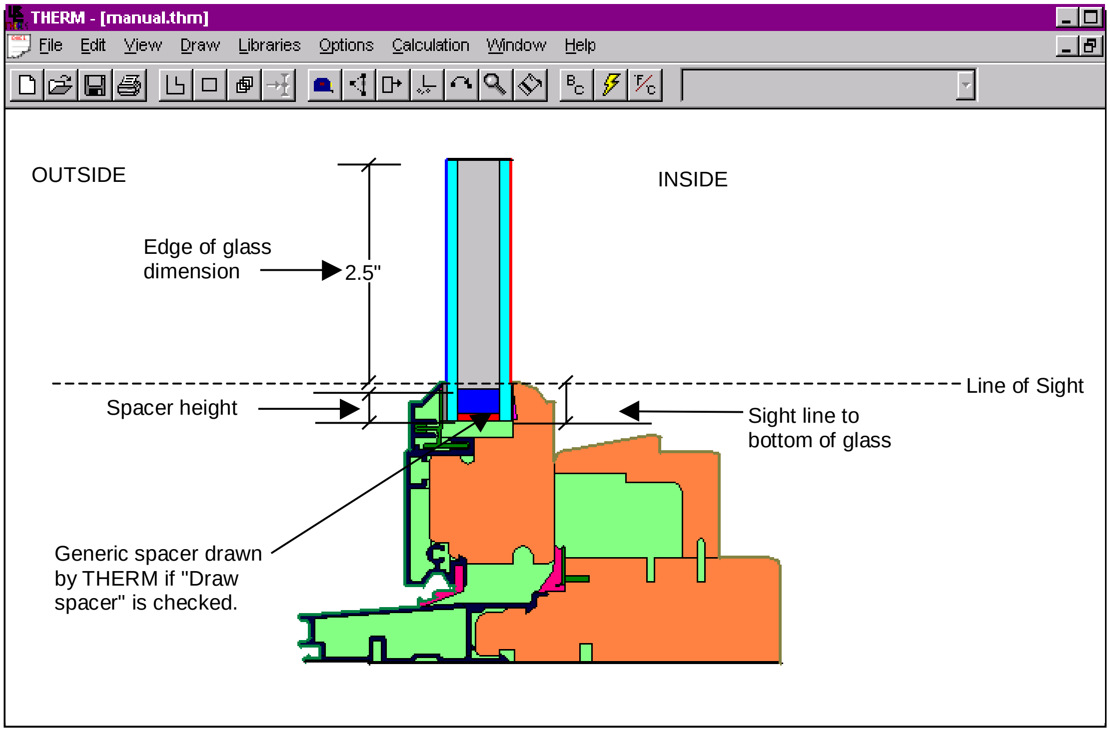


*Figure 5-38. Illustration of dimension definitions used in the **Insert Glazing System** dialog box*

8. THERM will insert the glazing system, as shown below, including the appropriate number of glazings and cavities. If the **Draw spacer** box was unchecked, an opening is left for a spacer, which must be defined separately. You can draw rectangles to represent the spacer or paste a spacer from another THERM file into this drawing (see Section 5.12, "Creating a Component Library" for more details.)

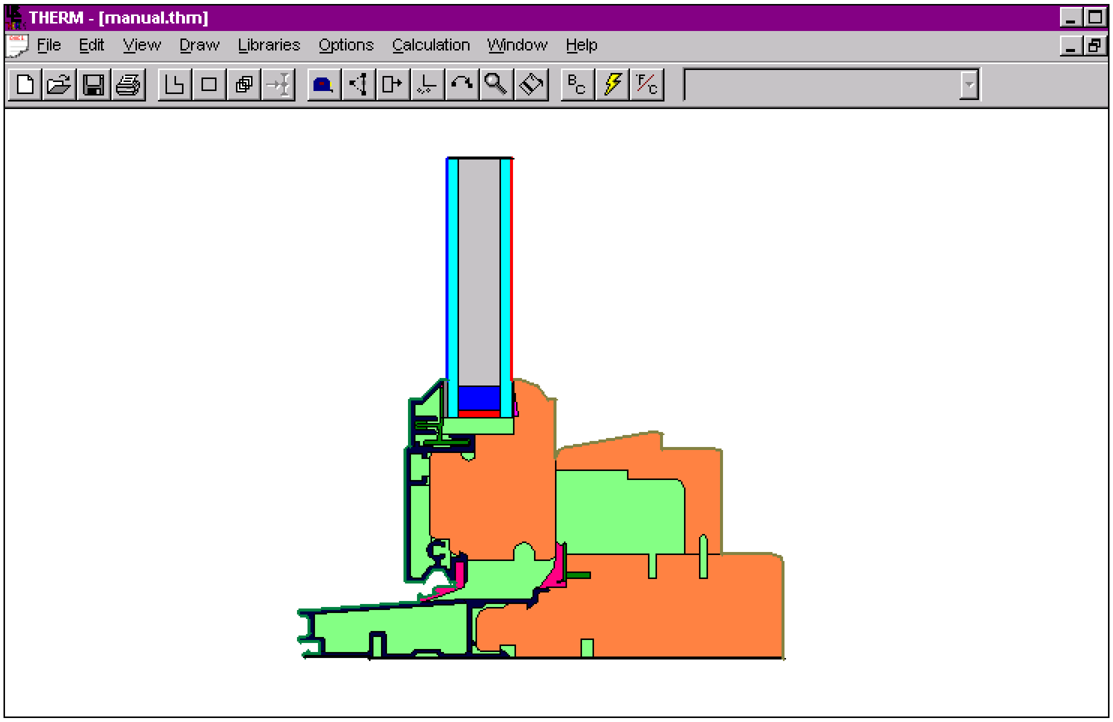


Figure 5-39. THERM inserts the glazing system in the drawing based on the position of the Glazing Locator.

9. You may need to move the glazing system after it is inserted (see Section 5.6, "Editing Polygons"). The glazing layers and cavities are grouped together and can be moved as a unit. Although it is possible to separate these layers (if you turn on the **Allow editing of IG polygons** from the **Options/Preferences** menu choice, on the **Drawing Options** tab), it is recommended that you do not do this. Instead, to change the glazing system, you should delete the existing glazing system (using the same technique as for deleting any other polygon except that all glazing system polygons will be deleted as a group) and import a new glazing system from WINDOW4. Although it is possible to create a glazing system in THERM by drawing polygons and assigning materials, this is only recommended when you are modeling a nonstandard product because the process is time-consuming and difficult to do correctly.

10. If you checked the **Use CI Model for Window 4 Glazing Systems** box when importing the glazing system, you will see red boundary conditions in the cavity between the layers of glass (for multipane systems). These are drawn automatically by the program.

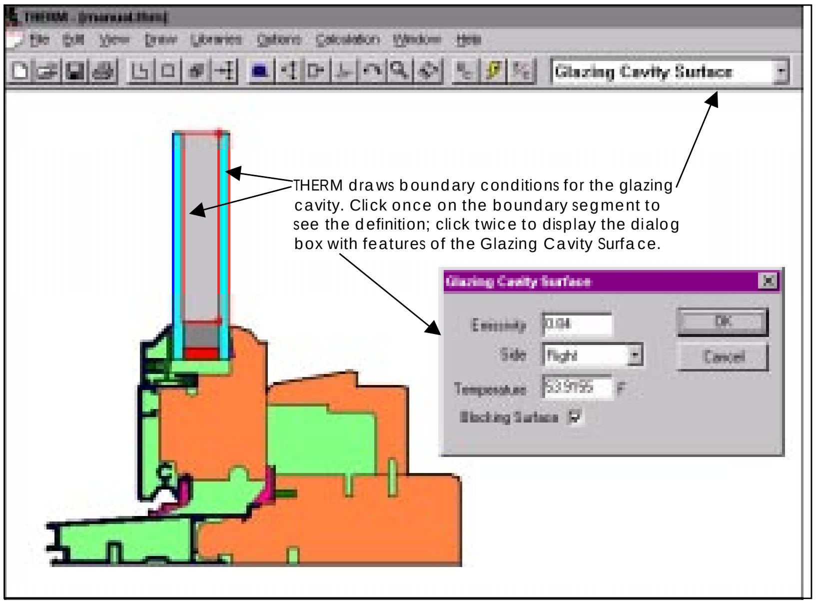


Figure 5-40. Condensation Index procedure for glazing cavities

The **Glazing Cavity Surface** dialog box appears when you double-click on a glazing cavity boundary condition created using the Condensation Index feature of THERM. The fields are the following:

Emissivity The emissivity of the surface.

Side The side of the surface relative to the cavity. The choices are:

- Adiabatic
- Left
- Right
- Open

Temperature The temperature of the surface.

Blocking Surface This check box is used to indicate whether the surface is blocking radiation transfer. The default value in the program is that all surfaces are blocking. Unless you are worried about calculation speed (under default conditions, calculation time may be longer), it is easiest and most accurate to keep the default on.

5.9.2. Condensation Index

The Condensation Index (CI) Model provides an alternative method for determining heat transfer through glazing systems. It was developed to improve the accuracy of local temperature modeling which is important in predicting where condensation will occur in a fenestration product. Glazing systems imported from WINDOW4 use an effective conductivity model to account for conduction, convection, and radiation. The convection model is based on a single value determined from the average behavior of the entire glazing cavity. The traditional radiation model assumes infinite parallel plates of constant temperature within the component being modeled[(2)]. These assumptions have been shown to yield acceptable results for the average heat loss (U-factor), but they are inadequate for predicting local temperatures[(13)]. The condensation index model included in THERM[(14,15)] uses local film coefficients and a detailed radiation model. This more detailed model has been shown to give more accurate local temperature results then the effective conductivity model. The condensation index model calculates U-factors based on the CI algorithms which are slightly different from those in WINDOW 4.1; these U-factors are not consistent with current NFRC U-factor procedures. It is currently limited to modeling glazing cavities with pure gas (air, argon, or krypton). This restriction will be eliminated in future versions of the program.

- **Turn on the Condensation Index Model.** The Condensation Index Model can be activated in two different ways:
  - *Turn it on for all imported glazing systems:* By checking the **Use CI Model for Window 4 Glazing Systems** checkbox in the **Therm File Options** choice of the **Options** menu, as shown in Figure 5-41, you set this option for all glazing systems that are imported into THERM.
  - *Turn it on for each glazing system as you import it:* By checking the **Use CI Model for Window 4 Glazing Systems** checkbox on the **Glazing System Library** dialog box when you import a system from WINDOW, as shown in Figure 5-42, you can specify for individual systems whether to use the CI Model.

  If the Condensation Index Model is not checked when the glazing is first imported, it can be changed before the final calculation in the **Therm File Options** dialog box; checking the **CI Model** checkbox at that time will change the model of a glazing system already imported into THERM. This way, you can do a calculation for U-factor and then do a condensation index calculation without redrawing the model.

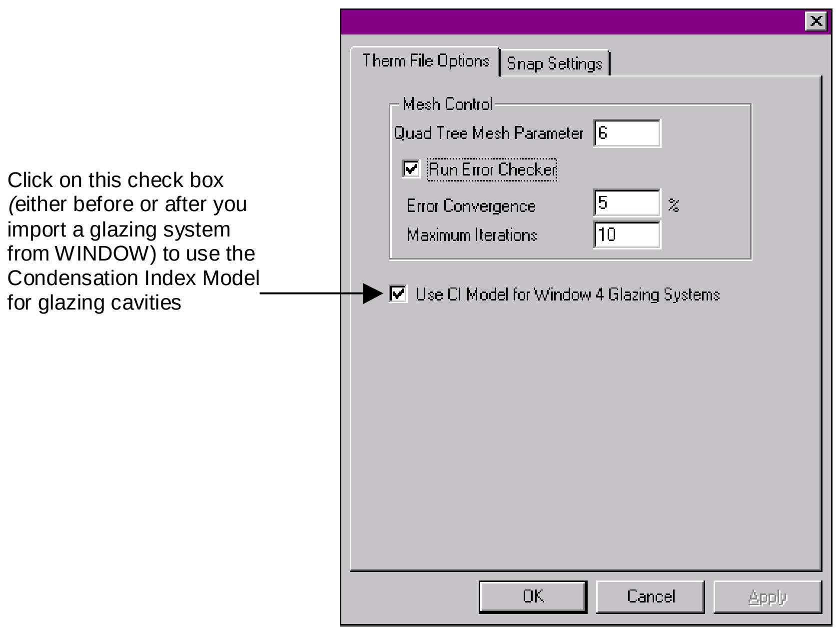


Figure 5-41. Check box for turning on the Condensation Index feature for glazing cavities

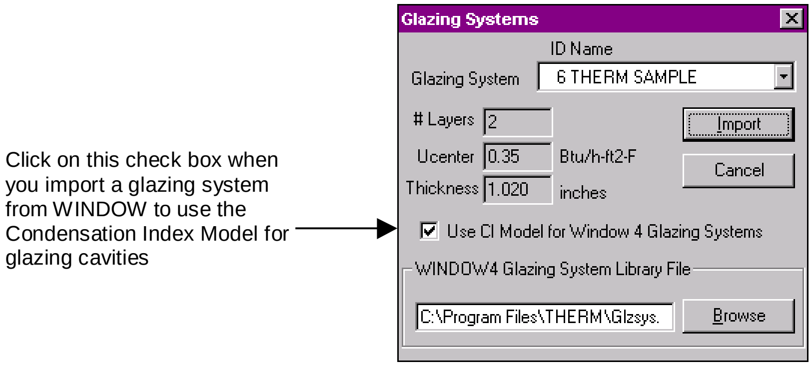


Figure 5-42. Check box for turning on the Condensation Index from the Glazing System Library when importing a system from WINDOW.

- **Import a glazing system from WINDOW4**. The glazing system appears with the glazing cavity surrounded by a red line. This is an easy way to tell whether an effective conductivity or condensation index model is being used. The condensation index model does not treat the glazing cavity as a solid material but as an additional boundary condition. This boundary condition is automatically defined when the glazing system is imported. Double-click on this boundary to make sure it is correct for your model.

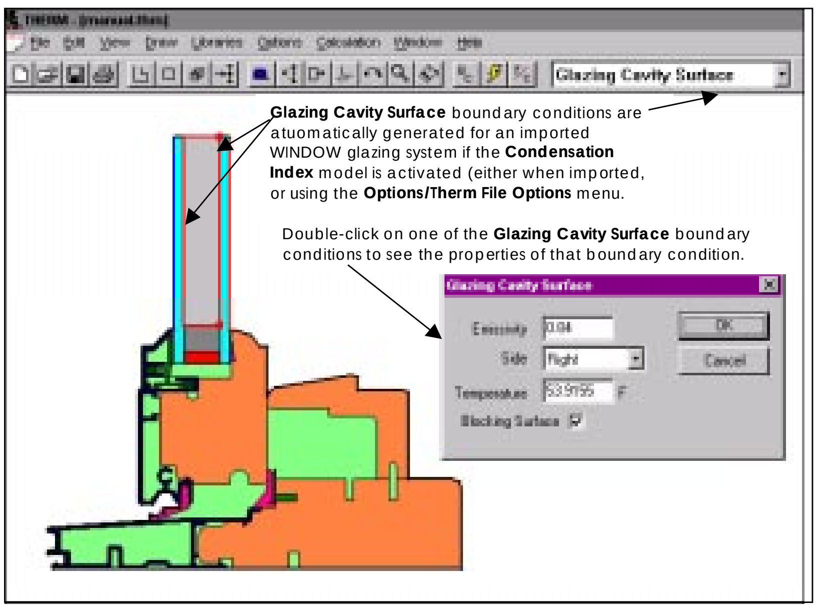


Figure 5-43. Condensation Index procedure for glazing cavities

The **Glazing Cavity Surface** dialog box appears when you double-click on a glazing cavity boundary condition created using the Condensation Index feature of THERM. The fields are the following:

Emissivity — The emissivity of the surface. This value comes from the WINDOW glass library which must be in the same subdirectory as the glazing system library; otherwise, a default value of 0.84 is used. This value can be overridden.

Side — The side of the surface relative to the cavity. The choices are:

- *Adiabatic:* assigned to the bottom of the glazing system.
- *Left*: assigned to the left surface of the glazing cavity.
- *Right:* assigned to the right surface of the glazing cavity.

- *Open:* assigned to the top of the glazing cavity. When this condition is chosen, THERM sets the temperature equal to the average glazing cavity temperature.

Temperature The temperature of the surface. These values are obtained from the WINDOW glazing system library. The temperature of the **Open** boundary is the average glazing cavity temperature. There is no temperature shown for the **Adiabatic** surface because it is not used in the calculation.

Blocking Surface This check box is used to indicate whether the surface is blocking radiation transfer. The default value in the program is that all surfaces are blocking. See Section 6.4.1, "Frame Cavities with Detailed Radiation Model" for a definition of blocking surfaces. For a typical glazing cavity with parallel glazing layers, the spacer is the only component that could provide a blocking surface. Changing the default on a surface that is actually nonblocking may provide a modest increase in calculation speed. Labeling a blocking surface as nonblocking will result in incorrect results. It is recommended that you use the program default value, which causes the program to determine whether or not the surface is blocking.

5.9.3. If the WINDOW4 Glazing System Doesn't Fit

When you try to insert the W4 glazing system, you may get the following message:

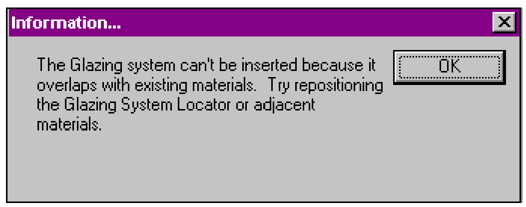


Figure 5-44. Message displayed if the Glazing System cannot be inserted

Either reposition the Locator, or move elements of the cross section to allow the glazing system to fit. In addition, you can obtain the width of the glazing from WINDOW4 and edit the cross section to fit. Glazing systems are usually separated from the frame on both sides by tape or sealant. The best way to insert a glazing system is to draw a polygon representing the sealant on one side. This polygon helps to locate the glazing system. Once the glazing system is imported, draw a polygon to represent the sealant on the other side of the glazing system. These sealant polygons are small, so you will probably need to zoom in to draw them.

Another possible reason that the glazing system doesn't fit is due to a round off discrepancies. The thickness of the glazing system in WINDOW 4.1 is calculated to three decimal places in either SI or IP units. However, the tape measure gives lengths to four decimal places in IP units and three decimal places in SI units. If you are having trouble fitting the glazing system, you may want to try measuring the space for the glazing system with the tape measure in SI units to eliminate the possible round-off error.

5.9.4. Inserting Multiple WINDOW4 Glazing Systems

You can place more than one glazing system in a drawing. When you add systems, the program asks if you want to replace the existing system or add another.

5.9.5. Information about the Glazing System

You can view information about the inserted glazing system by double-clicking your mouse on any part of the glazing system. The following window will appear:

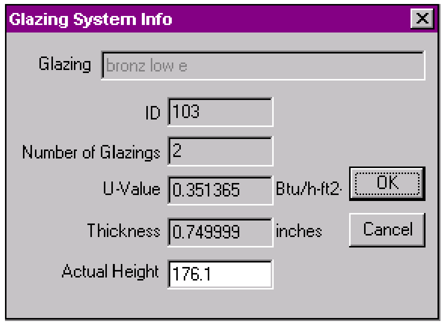


Figure 5-45. Double-click on the inserted glazing system to see information about the system.

5.10. Assigning Materials After Drawing Polygons

Material properties must be specified for each polygon. While you are drawing, THERM assigns the most recently used material. You can select the material as you are drawing or change the material assignments after the polygons have been drawn.

There are two different classes of materials, solids, and cavities. Solids have a specified thermal conductivity. A frame cavity model is used to calculate the effective conductivity for cavities commonly found in aluminum, vinyl, and fiberglass window frames. The glazing cavity in an insulated glazing unit is treated as a solid with an effective conductivity that accounts for conduction, convection, and radiation (see Section 5.9.2, "Condensation Index" for an alternative way of treating the gas cavity). It is recommended that the glazing cavity be modeled by importing a glazing system from WINDOW4.

After the polygons are drawn, you can assign a different material to them by using one of the methods discussed below. You can select multiple polygons that all need to be assigned the same materials using the selection techniques discussed in Section 5.6.1, "Selecting a Polygon."

5.10.1. Use the Material pull-down list to the right of the toolbar:

1. Select the polygon by placing your mouse inside the object and clicking the left mouse button.
2. The vertices of the selected polygon are indicated as *squares.*
3. Select the appropriate material from the list in the upper right corner of the toolbar. The choices in the list show all the entries in the **Material Library**[16]. A fast way to move through the list is to type the name or even just the first few letters of the material you want. For example, typing **Wood** would put you on the first instance of **Wood** in the Material List, **Wood (Cedar or Redwood)**. You could then use the down arrow key to get to **Wood (Pine, Spruce, Fir)**. You can then press **Enter** or click on the material with your left mouse button, and the selected material will be assigned to the polygon.

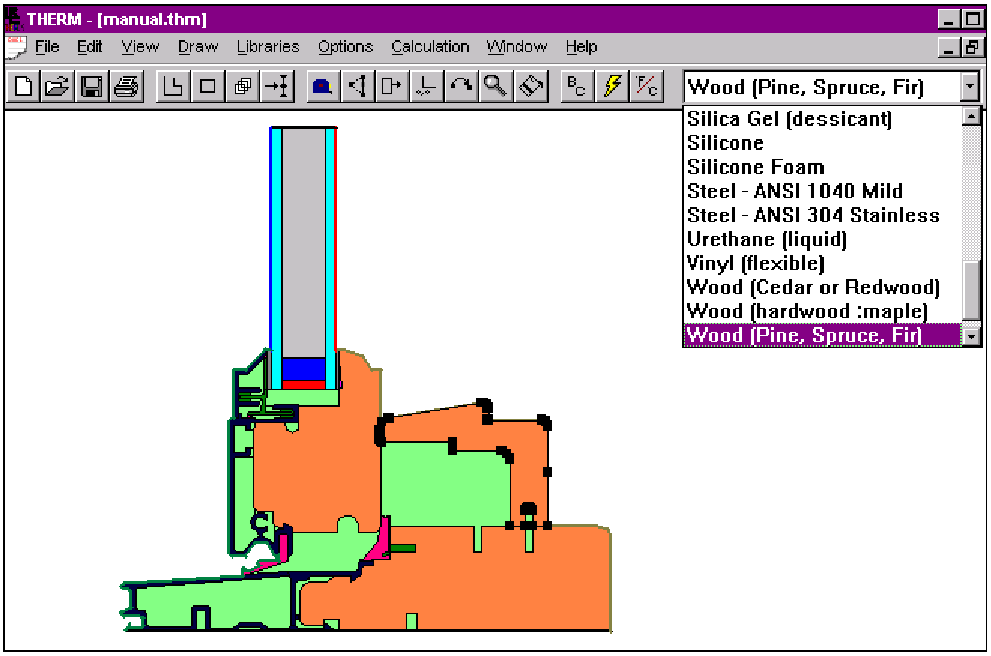


Figure 5-46. You can select polygons after they are drawn and assign materials to them.

5.10.2. Access the Properties for Selected Polygon(s) dialog box:

Another way to assign a material to a polygon is to double-click on the polygon you want to change, or, if the polygon is already selected, press **Enter**, use the **Library/Set Material** menu choice, or press **F4**. This causes the **Properties for Selected Polygon(s)** dialog box to appear, as shown in the following figure.

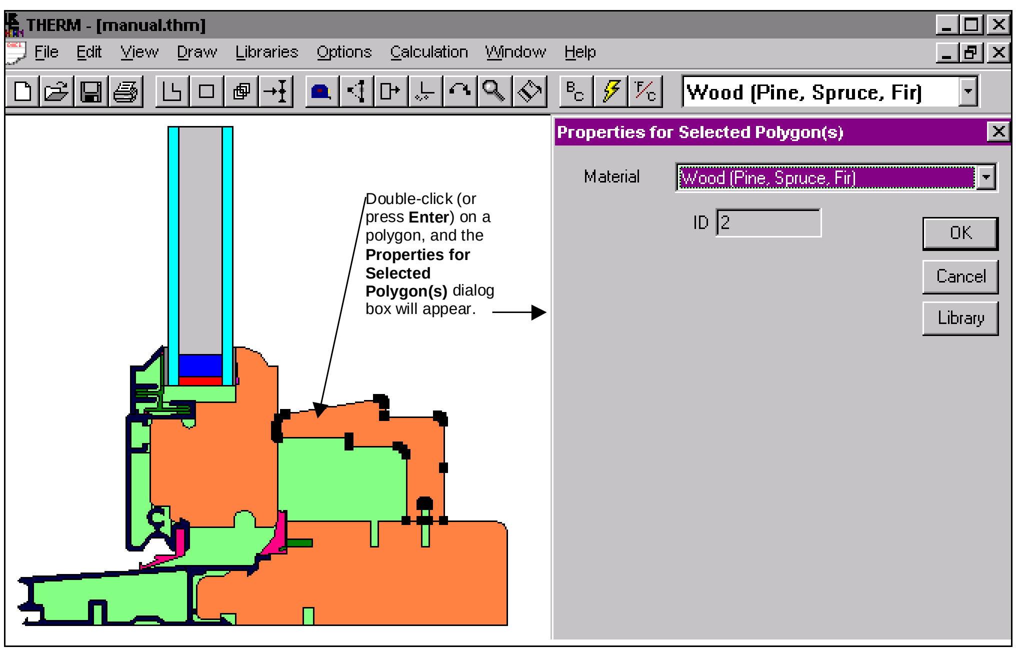


Figure 5-47. Material properties for solid material are accessed by double-clicking mouse on polygon.

The advantage of this method is that you can access the **Material Library** by using the **Library** button if you want to add a new material or change the properties of a material.

The information displayed in the **Properties of Selected Materials** dialog box is dependent on the type of material, which is discussed in more detail in Section 5.11, "Defining New Materials." However, all material types will have the following fields:

- *Material:* a pull-down list, which shows the currently defined material for the selected polygon. With this pull-down list, you can change the material for the selected polygon. This list contains the same choices as the list on the right side of the toolbar.
- *ID:* a field below the Material pull-down list, which is a unique number that is assigned to every polygon in the cross section. This number is useful for identifying specific objects in the THERM reports and when using the **Select Special** feature (discussed in Section 5.6).

5.10.3. Solid Materials

For solid materials, the **Material** name and **ID** number are the only values displayed; however, if you press the **Library** button, you can access the **Material Definitions** dialog box for that material and see the **Conductivity** and **Emissivity**.

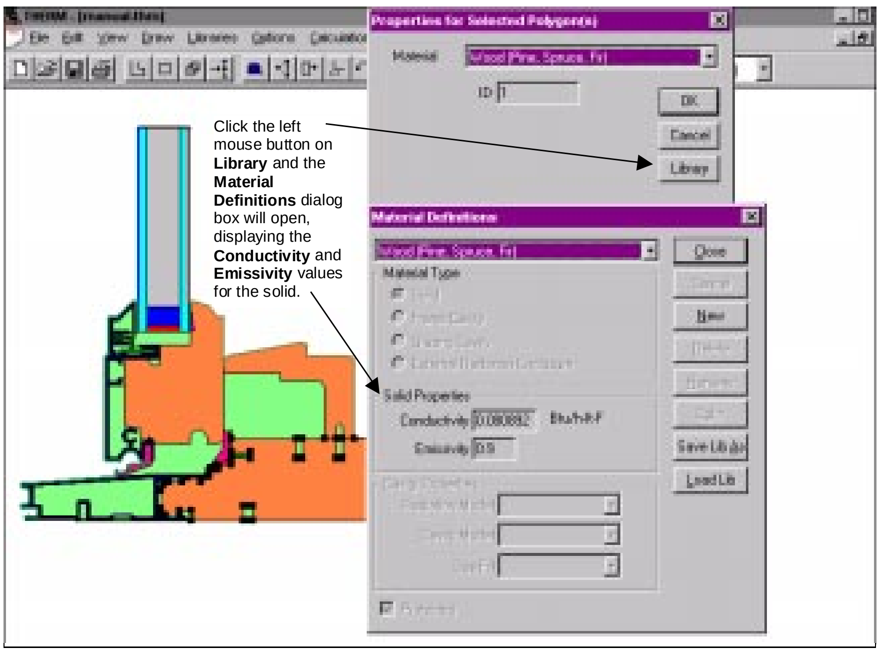


Figure 5-48. ***Material Definitions*** *dialog box accessed from the* ***Library*** *button shows the* ***Conductivity*** *and* ***Emissivity*** *for* ***Solids****.*

5.10.4. Frame Cavities

For cavities, several values are listed in addition to the material name and ID number:

- **Keff**, the effective conductivity, and **Nu**, the Nusselt number, which are calculated by THERM based on the model choices specified in the **Material Library** and cannot be edited.
- The **Vertical** and **Horizontal dimensions** are the actual dimensions of a rectangular cavity or the effective dimension determined by the **Cavity Model** selected in the **Material Library** for nonrectangular cavities. These values can only be edited if the **User Dimensions Cavity Model** is selected.

- The **Heat Flow Direction** can be selected from a pull-down menu. The three choices are **Horizontal**, **Up** and **Down**. This choice determines what correlation will be used to calculate the convective film coefficient. **Horizontal** heat flux assumes that the governing temperature difference is between the vertical walls of the cavity. Heat flow **Up** assumes that the governing temperature difference is between the horizontal walls of the cavity and that the lower wall is warmer than the upper wall. Heat flow **Down** assumes that the governing temperature difference is between the horizontal walls of the cavity and that the upper wall is warmer than the lower wall.

- The **Temperatures** are used to calculate the convective film coefficient in all cavity models and the radiative conductance in the simplified radiation model. For horizontal heat flow these would be the average temperatures of the vertical sides of the cavity. These values can be edited.

- The **Emissivities** appear for the simplified radiation model only. These are the emissivities of the surfaces perpendicular to the heat flow direction, i.e., the vertical sides of the cavity for horizontal heat flow. These values can be edited either by selecting a predefined value from the pull-down menu or by typing in a value.

See Section 5.11, "Defining New Materials" for a more detailed explanation of how to use this information.

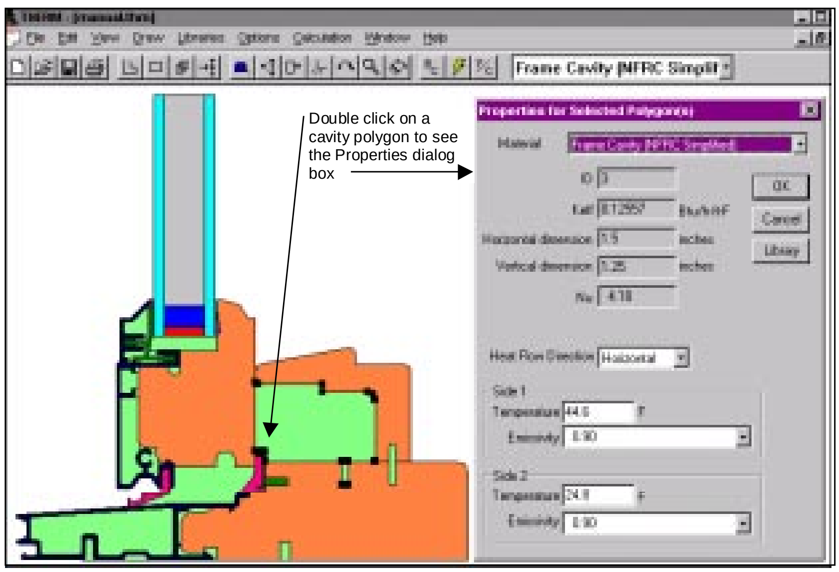


Figure 5-49. Material properties for cavities accessed by double-clicking mouse on polygon.

5.11. Defining New Materials

There are four material types in THERM:

- Solid
- Frame Cavity
- Glazing Cavity (not currently implemented)
- External Radiation Enclosure.

THERM comes with a set of predefined materials that cannot be edited (the input values are disabled). Materials can be added to the **Material Library** using the following technique.

1. Open the **Material Library** by selecting the **Library/Material Library** menu choice, and the **Material Definitions** dialog box is displayed. You can use the pull-down list at the top of **Material**

Definitions to view the characteristics of any of the materials, but because the predefined materials are protected from editing (they are grayed out), you cannot change any of the values, as shown in Figure 5-50. However, you can make new materials in this library, as explained in the following steps.

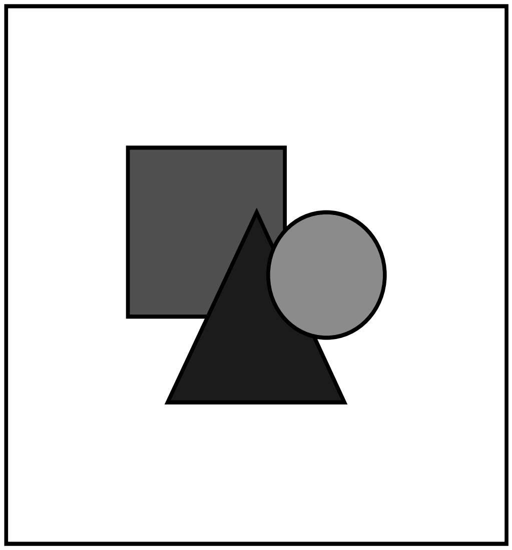


*Figure 5-50. The default **Material Library** that comes with THERM has only "Protected" materials that cannot be edited.*

2. To make a new material, click on the **New** button.
3. You will be asked for the name of the new material. Type a name that is not already in the library, and click on the **OK** button.

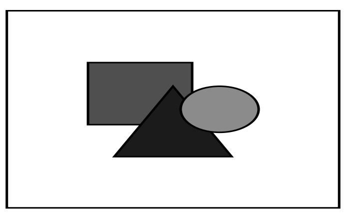


Figure 5-51. Type a unique name for the new material.

4. Now you are presented with the same **Material Definitions** dialog box, but all the fields are available for you to edit. Different fields are available for editing depending on the **Material Type** that you choose, i.e., solid, cavity, or external radiation enclosure. These different types are discussed in detail in the following sections. Enter the input values as appropriate.
5. Click on the **Color** button to select a color for your material other than the default blue.
6. When you have defined the characteristics of the material, click on the **Save Lib As** button in order to save the material permanently to the default library (named **material.lib**) so that it can be used again. (If you forget to save it, you will be asked when you close THERM if you want to save the changes you made to the library unless you turn this feature off using **Options/Preferences/Prompt for saving libraries on program exit**). See the discussion in Section 5.11.5, "Making Custom Material Libraries" for details about having material libraries other than the default.

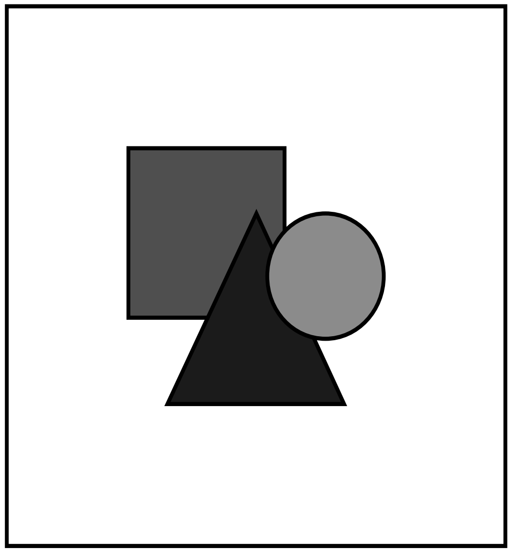


*Figure 5-52. To define a new material, click on the **New** button; a nonprotected **Materials Definition** dialog box will appear, allowing you to define the characteristics of the new material.*

5.11.1. Solid Materials Type

For the **Solid** material type, the relevant input values are found in the **Solid Properties** group box on the **Material Definitions** dialog box, which contains the following values:

Conductivity	**Units:** W / m-K (SI); Btu / h-ft-°F (IP)
Emissivity	**Default value:** 0.9

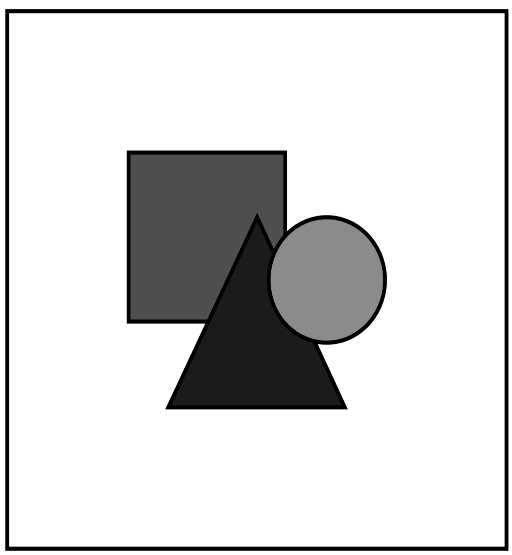


Figure 5-53. Material Library definition for solids

For the predefined materials, these values are not editable. However, if you click on the **New** button and make a new material, you will be able to edit these values, as discussed previously in Section 5.11, "Defining New Materials".

5.11.2. Frame Cavity

There are two materials in the default THERM library defined with the **Material Type** of **Frame Cavity**. These materials have the following characteristics (the **Radiation** and **Cavity Model** definitions are discussed at the end of this section):

- *Frame Cavity (NFRC Simplified):* **Radiation Model:** Simplified

  Cavity Model: NFRC
- *Frame Cavity (NFRC Detailed):* **Radiation Model:** Detailed

  Cavity Model: NFRC.

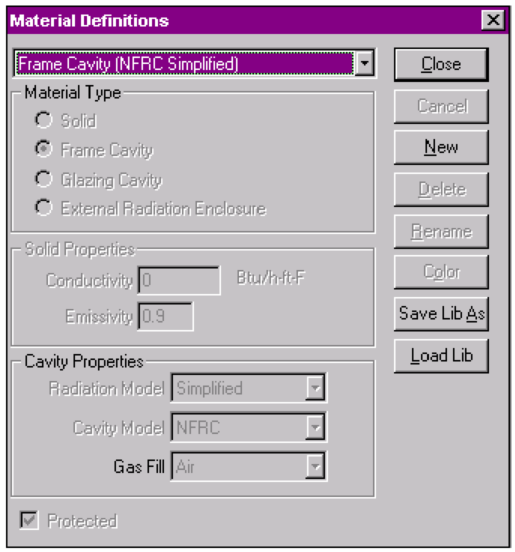


*Figure 5-54. Default Frame Cavity (NFRC Simplified) protected **Material Library***

If you want to make new frame cavities, follow the instructions above for making new materials; specify the new input values in the **Cavity Properties** group box in the **Material Definitions** dialog box. It is not necessary to create a new material for each frame cavity. The following discussion explains the values needed for many of these inputs.

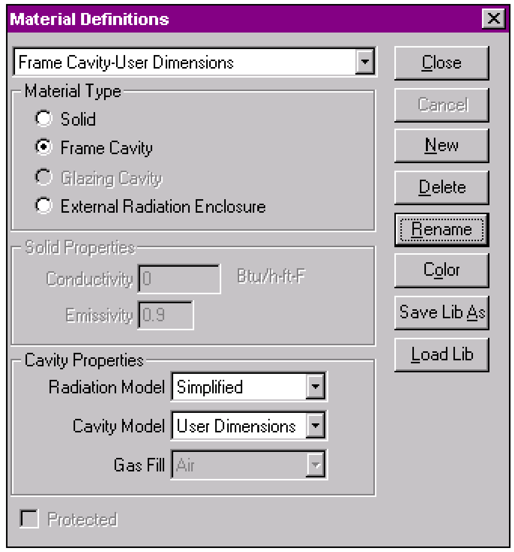


Figure 5-55. Creating a new frame cavity entry in the ***Material Library***

For a frame cavity material, you need to specify the type of **Radiation Model** and **Cavity Model** the program should use. The most common case for NFRC compliance will be the **Simplified Radiation Model** and the **NFRC Cavity Model**.

Radiation Model The radiation model to be used by THERM.

- **Simplified:** uses the method in ASHRAE SPC 142P[16], modeling radiation effects as an effective conductance based on the temperature and emissivity of the two parallel faces perpendicular to the heat flow. For nonrectangular cavities, this conductance is determined using the dimensions of the approximate rectangle in the **Cavity Model** (see below) and a curve fit model. The default values for the temperatures in the frame cavity are 7 °C and –4 °C. The default emissivities are 0.90 for both surfaces. The temperatures and emissivities of each frame cavity can be edited in the **Properties for Selected Polygons** dialog box, which you can access by double-clicking on the cavity.

- **Detailed:** does an element-by-element view-factor calculation taking into account the temperature and emissivity of each element; these THERM algorithms are based on FACET[12]. The detailed model reads the emissivity of the surrounding material from the material library. These emissivities can be overridden by double-clicking on the red boundary that appears around the cavities once the boundary conditions have been defined. For more discussion about the detailed model, see Section 6.5.1, "Frame Cavities with Detailed Radiation Model" and Appendix C, "Theoretical Background."

Cavity Model The cavity model to be used by THERM. This model accounts for convection as well as the way the nonrectangular cavities are approximated.

- **NFRC:** In this model, the effective conductivity is calculated based on a rectangle that fully encloses the cavity. Convection correlations are based on the *Handbook of Heat Transfer Fundamentals*[11]; the specific correlations used can be found in ASHRAE SPC 142P[16].

- **CEN (unventilated):** This model calculates the effective conductivity based on a rectangle with the same aspect ratio and area as the cavity being analyzed. This is a European standard. When this Cavity Model is used with the **Simplified Radiation**

Model, THERM uses the CEN version of the radiation model, which is slightly different than the SPC 142 method.

- **CEN (slightly ventilated):** This model is also part of the European standard. It is to be used for cavities that have an opening to the environment of greater than 2 mm and less than 10 mm.
- **User dimension:** In this model, the user can define the rectangular dimensions for a nonrectangular shape; the effective conductivity is based on the SPC 142P method.

The effective conductance of frame cavities is obtained from standard correlations[11]. The material **Frame Cavity NFRC (Standard)** assumes a horizontal heat flow. For a rectangular cavity, the governing temperature difference is assumed to be between the vertical walls of the cavity. An average emissivity for these surfaces needs to be determined. **Horizontal** heat flux assumes that the governing temperature difference is between the vertical walls of the cavity. Heat flow **Up** assumes that the governing temperature difference is between the horizontal walls of the cavity and that the lower wall is warmer than the upper wall. Heat flow **Down** assumes that the governing temperature difference is between the horizontal walls of the cavity and that the upper wall is warmer than the lower wall. When drawing window models, take care that the heat flow in the cavities is defined according to the conditions that will apply when the cross section is positioned in the window. In some cases, the vertical and horizontal dimensions will have to be switched (or the cross section rotated) for these definitions to be correct.

After a polygon has been assigned a frame cavity material, you can view the cavity dimensions as well as the effective conductivity and Nusselt number of the cavity by double clicking on the polygon. (This is discussed in more detail in Section 5.10, "Assigning Materials After Drawing Polygons.")

5.11.3. Glazing Cavity

This material type is not currently being used. Glazing cavity properties should be obtained from WINDOW, which occurs automatically when you use imported glazing systems.

5.11.4. External Radiation Enclosures

The external radiation enclosure feature uses the detailed radiation view-factor model to accurately simulate the radiant exchange between the building component being analyzed and its surroundings. This radiation view-factor model takes into account the extent to which the component “sees” or radiates to itself. In thermal modeling of fenestration products, radiation accounts for half of the heat loss on the room-side surface. The simplified method assumes that the component only has radiant exchange with its surroundings. This assumption does not hold true for projecting fenestration products such as greenhouse windows and skylights and may also be problematic for sliding windows with deep channels.

The radiation enclosure is one of the standard classes of material included in the material library. Radiation enclosures are bounded by the model and by boundary conditions of specified temperature. Radiation enclosures are unique because every point on the bounding surface of the model must be included in the radiation enclosure. (For all other materials, THERM can add any missing points.) It is possible to draw the radiation enclosure by hand, making sure a vertex exists on the radiation enclosure polygon for every vertex on the adjacent polygon. The fill void feature, however, can simplify this procedure. The following steps explain how to use this feature to create radiation enclosures:

1. Draw a temporary polygon that surrounds the outside of the radiation enclosure.

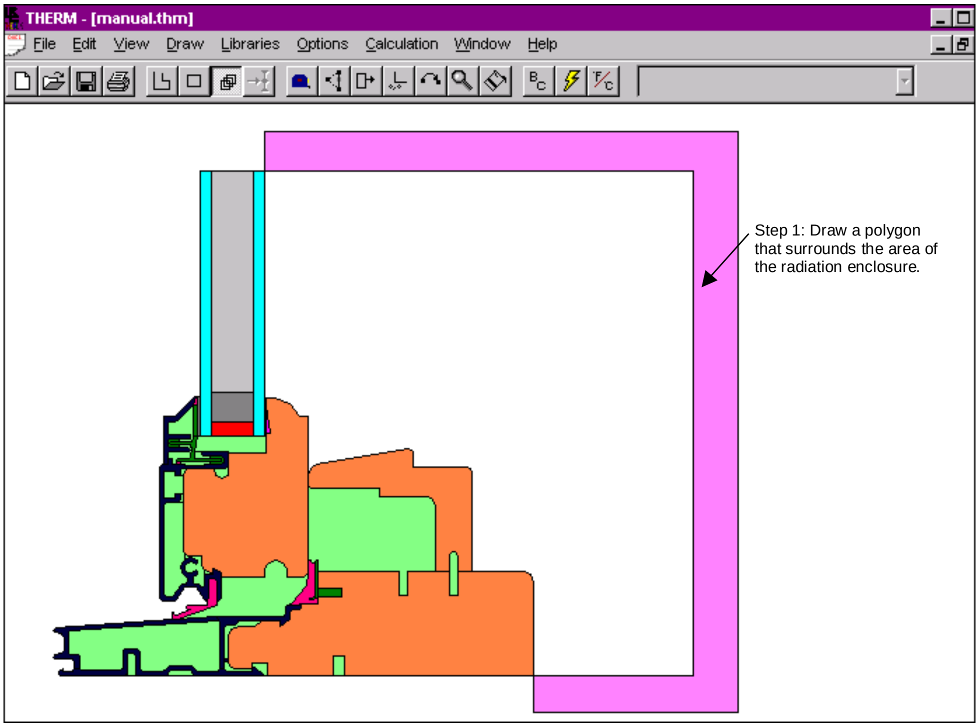


*Figure 5-56. Draw a temporary polygon around the area that will become the **External Radiation Enclosure**.*

2. Click on the **Fill Void** icon on the toolbar.

3. Place the **Fill Void** cursor inside the undefined area surrounded by the temporary polygon, and click the left mouse button. The fill void feature will create a polygon with points defined that match the points on all adjacent polygons.

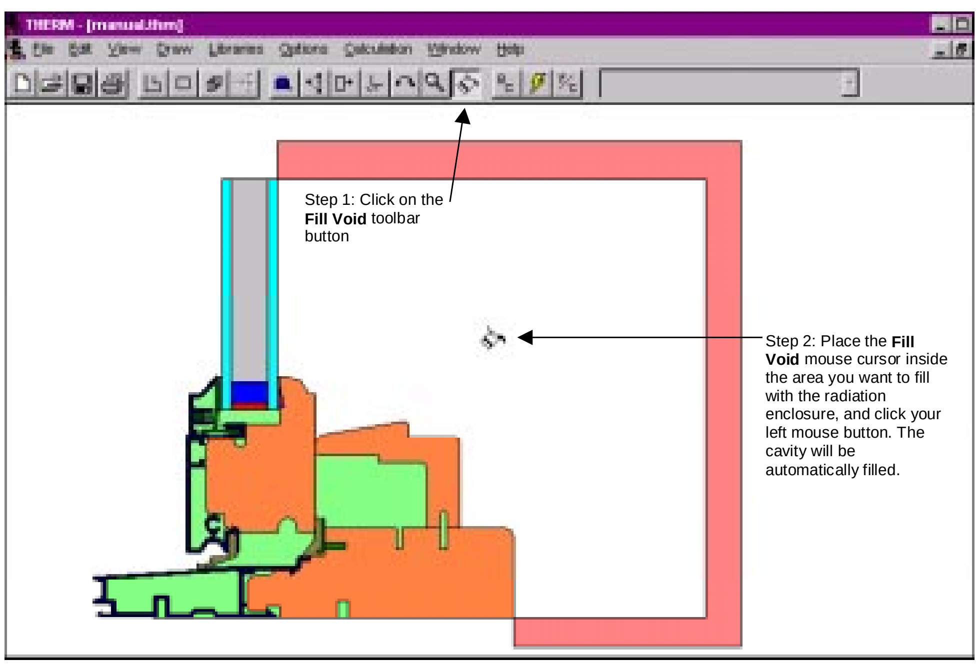


*Figure 5-57. Use the **Fill Void** toolbar button to create the **External Radiation Enclosure** polygon.*

4. Select the polygon that is created by using a single click of the left mouse button, as shown in Figure 5-58. Assign the material of the radiation enclosure polygon from the **Materials** pull-down list on the right side

of the toolbar, or from the **Libraries/Material Library** menu choice. Select the predefined material in the **Material Library** called **Radiation Enclosure** which has the **Material Type** of **External Radiation Enclosure**, shown in Figure 5-59. Any material of this type will have special boundary conditions automatically assigned to it; these are called **Radiation Surface**. See Chapter 6, "Defining Boundary Conditions" and Section 7, "Calculating Results" for more information about **External Radiation Enclosures**.

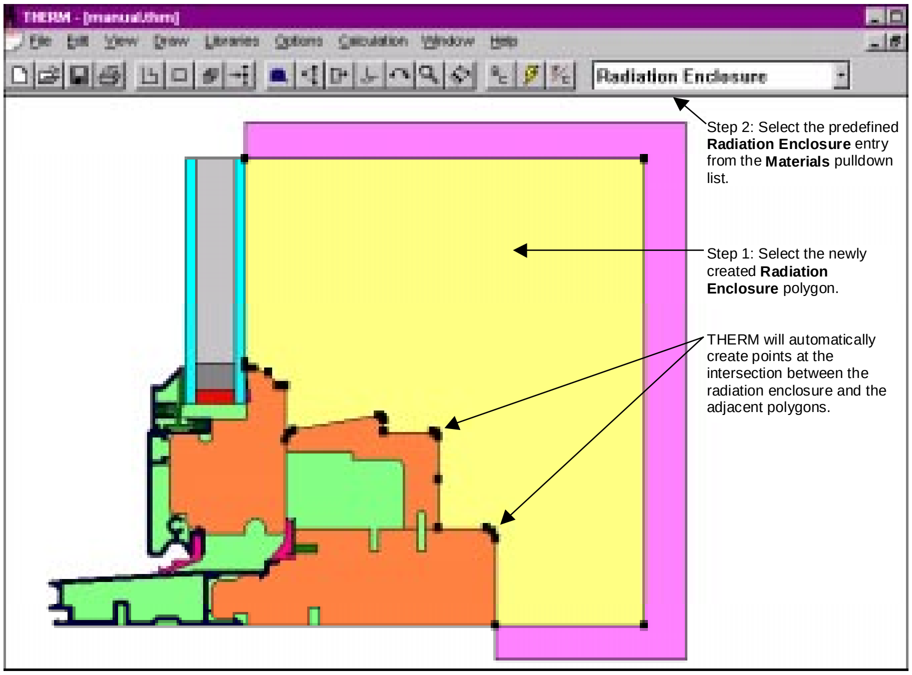


*Figure 5-58. Assign the predefined material **Radiation Enclosure** to the Radiation Enclosure polygon.*

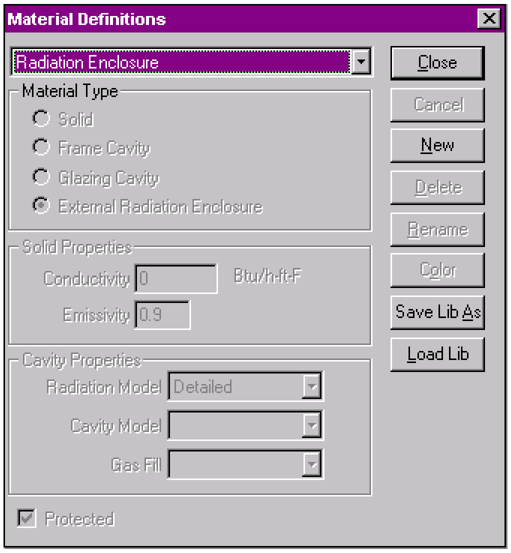


*Figure 5-59. The predefined **Radiation Enclosure** material can be used for the **External Radiation Enclosure** polygons*

5. Delete the temporary polygon used to surround the radiation enclosure. If a radiation enclosure is completely surrounded by other materials (either solids or cavities), it will be considered a void in the model. This is an incorrect use of the radiation enclosure feature; instead use the detailed frame cavity model for cavities surrounded by other materials.

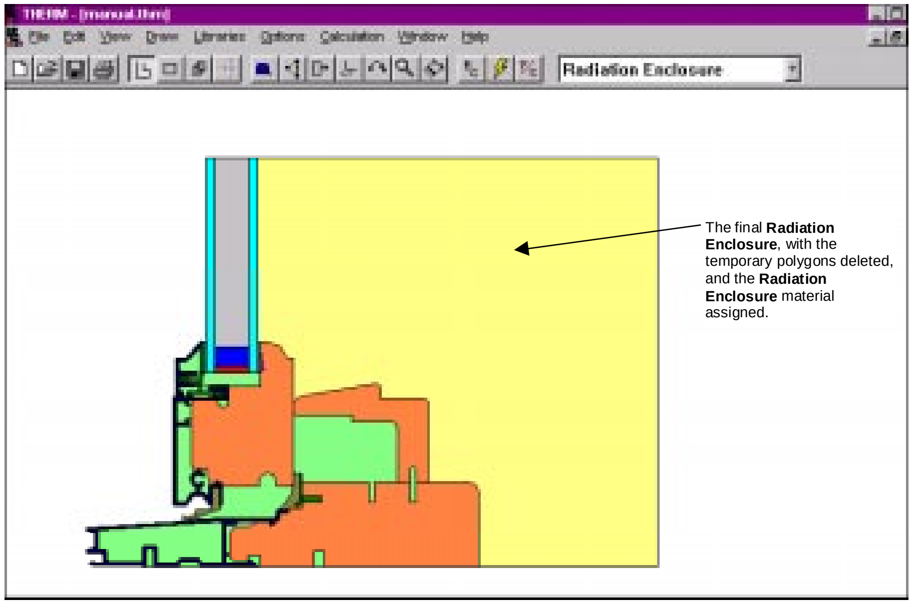


*Figure 5-60. The completed **Radiation Enclosure (the temporary surrounding rectangle deleted)**.*

6. Complete the analysis by defining boundary conditions (see Section 6.5.2, "External Radiation Enclosures" for special information about boundary conditions for this situation) and calculating the results (see Chapter 7).

5.11.5. Making Custom Material Libraries

The default THERM library is called "material.lib" and is located in the THERM installation directory. The default library can be customized by adding new material entries, or other custom material libraries can be created, using different names and different directories if desired. Use the **Save Lib As** button in the **Material Definition** dialog box to create custom libraries and load them using the **Load Library** button in the same dialog box.

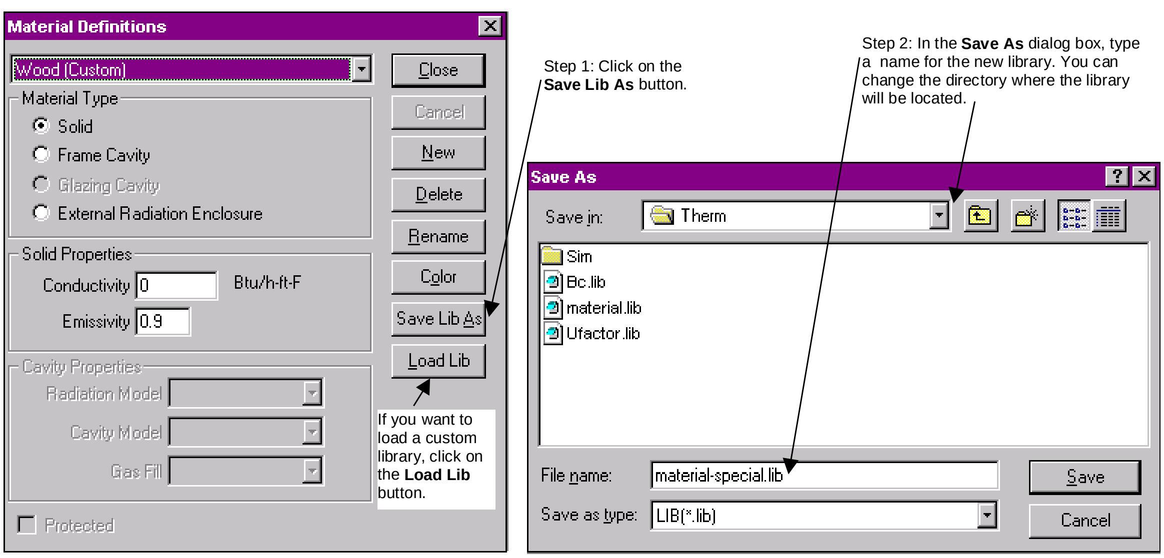


Figure 5-61. Material properties for cavities accessed by double-clicking mouse on polygon.

5.12. Creating a Component Library

In modeling windows, you will probably find that there are some components you need to draw again and again. THERM allows you save these frequently used components as separate files from which you can copy and paste. This is useful when the separate components that make up a window cross section are provided as separate drawings. It is also useful for drawing small, finely detailed objects, such as spacers.

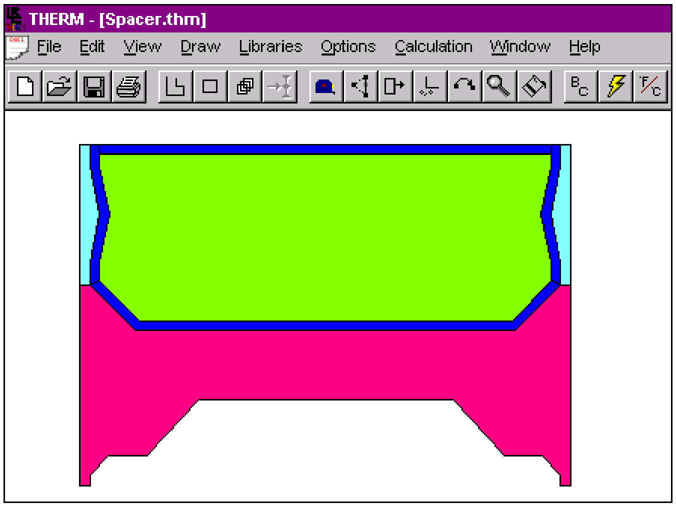


Figure 5-62. THERM file containing just a spacer

To cut and paste components between THERM files:

1. Open both a current working THERM file and the THERM file containing a component you wish to copy and paste.

2. In the working file, place the locator (**Shift F2**) on the point where the matching point on the component will be placed.

3. Use the Window menu to switch to the component file, and select the portions of the component that you wish to copy and paste into the working file. A single THERM file, for instance, could contain all the spacers from a single manufacturer. Use the **Edit/Select All** menu to include all the features in the component file, or use the other polygon selection methods to select specific elements of the component (see Section 5.6.1, "Selecting a Polygon" for more details.)

4. Place the locator (use **Shift F2** and click the left mouse button) at the point which will match the location in the working file.

5. Press **Ctrl-C** to copy the selected polygons into the Windows™ clipboard.

6. Using the **Window** menu, switch to the working file.

7. Press **Ctrl-V** to paste the selected polygons into the working file. THERM will position the pasted polygons into the working file by matching the positions of the locator points defined earlier for the component file and the working file.

5.13. Importing a FRAME™ F30[3] File

THERM can import FRAME™ F30[3] files. The program brings these files in as rectangles. In order to perform a calculation on a FRAME™ F30 file, insert a glazing system, and assign materials, boundary conditions, and U-factor tags. F30 files are created for use with a finite-difference program and are limited to rectangularized geometry, and are limited in the number of available rectangles used to define a cross section, often resulting in over simplification of cross sections. In addition, cavities which have been divided up into rectangles must be redrawn in THERM. THERM is a finite-element program that allows for geometry to be represented by general polygons. Use of F30 file is recommended for comparison purposes only.

Check the THERM website (go to the "Software" section of windows.lbl.gov) for updates about FRAME file conversions.

6. DEFINING BOUNDARY CONDITIONS

6.1. Overview of Boundary Conditions

Chapter 5 describes procedures for drawing a cross-section geometry and assigning materials to components, which are the first steps toward performing a heat-transfer analysis. The next step is to define the boundary conditions for the cross section. The procedures for defining boundary conditions are not dependent on how the cross section was drawn.

The steps for assigning boundary conditions are summarized below and explained in detail in the sections that follow.

- Click on the **Boundary Conditions** toolbar button (or use the **Draw/Boundary Conditions** menu choice or the **F10** keyboard shortcut). This causes THERM to draw the boundary conditions for the cross section. Under most circumstances, boundary conditions are drawn automatically around the outside of the completed cross section (see Section 6.5, "Special Boundary Condition Cases" for exceptions). THERM moves around the model counterclockwise and indicates the boundary by a heavy line.
- The program checks for characteristics in the geometry that will cause a problem in the heat-transfer simulation. These include undefined voids, overlapping polygons (see Section 6.3.2, "Finding Voids and Overlaps"), and polygons that are not contiguous. If problems are detected, THERM will issue a message. You may need to resolve these problems before the boundary conditions can be defined. When you have fixed any problems, click on the **Boundary Conditions** toolbar button again.
- Reassign boundary conditions as necessary. The boundary is actually composed of many small boundary segments that default to the condition of **Adiabatic** unless you have imported a glazing system from WINDOW in which case THERM assigns predefined boundary conditions to the glazing. You can change the boundary conditions of each segment of the cross section using the procedure described below in Section 6.2.3.
- Define U-factor tags if desired. These tags allow you to specify a group of boundary segments as one unit for which a single U-factor is to be calculated.

When these steps are complete, you have finished assigning the boundary condition definitions and can now perform the heat-transfer calculations for your cross section and then view the results (see Chapter 7, "Calculating Results," for a detailed discussion of the calculation procedure.)

6.2. Assigning Boundary Condition Definitions

The general method for defining boundary conditions is explained below. The example given does not involve any detailed or comprehensive modeling conditions for frames or glazing cavities, which are described in Section 6.5, "Special Boundary Condition Cases."

6.2.1. Generate Boundary Conditions

1. Make sure you save your file, then click on the **Boundary Conditions** toolbar button or use the **Draw/Boundary Conditions** menu choice.

2. The program draws an exterior boundary around all the cross section components, indicated by a heavy bold line, as shown in the following figure. This line is actually composed of many boundary segments, each of which is individually assigned a boundary condition.

3. A default boundary condition of **Adiabatic** is automatically assigned to all segments except those on glazing systems imported from WINDOW, which have predefined boundary conditions. The **Adiabatic** boundary condition is defined by a black color. In the predefined Boundary Condition Library that is included with THERM, boundary conditions for segments exposed to a cold environment are indicated with shades of blue; conditions for segments exposed to a warm environment are indicated by shades of red.

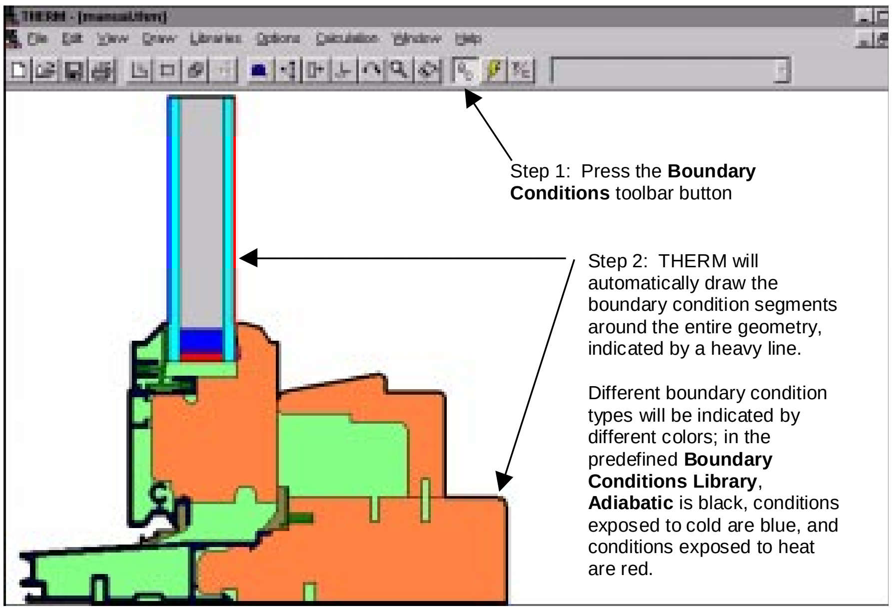


*Figure 6-1. Click on the **Boundary Condition** (BC) toolbar button; THERM will automatically draw the boundary condition segments, indicated by a heavy line.*

6.2.2. Automatic Check of Geometry Definition

When THERM draws boundary conditions, it checks the cross-section geometry to make sure that there are no problems that will cause the thermal calculation to fail. If the program finds such problems, it displays messages describing the problems; the boundary conditions may not be drawn until you correct the problems. Problems that cause the program to issue boundary condition error messages are:

- Noncontiguous polygons, i.e., polygons that are not touching any other polygon in the cross section. The error message will say, "There are materials that are outside of the Boundary Conditions."

- Regions inside the cross section that are not defined as a polygon are classified as voids. Polygons whose edges overlap an adjacent polygon are classified as overlapping. The error message will say, "The geometry contains voids or overlapping regions. The edges surrounding these regions will be highlighted in red. You must fix this problem before simulating." (THERM usually detects this problem as you are drawing, but it does not catch every instance.)

If you get these error messages when you push the **Boundary Conditions** toolbar button, you will have to correct the problems before simulating. See Section 6.3, "Error Detection in THERM" later in this chapter for more information about correcting these problems.

6.2.3. Reassign Boundary Conditions

THERM comes with a predefined **Boundary Conditions Library** that can be used to assign boundary condition to each segment. It is also possible to add new definitions to the **Boundary Conditions Library**, which is discussed in detail in Section 6.4, "Defining New Boundary Conditions". In most cases, you will want to change the boundary condition **Adiabatic** that THERM assigns automatically. THERM cannot perform the simulation without at least two nonadiabatic boundary condition, and will issue a message if this condition is not met. To reassign boundary conditions:

1. Select one or more boundary segments.

   To select ***one*** segment for assigning boundary conditions, place your mouse cursor over the segment and click the left mouse button once. You will see that the segment you have selected is indicated by rounded endpoints.

   To select ***multiple*** boundary segments:

   - For *contiguous* segments, select the first segment (by placing your cursor on the segment and clicking the left mouse button once), hold the **Shift** key down, move your cursor ***in a counter-clockwise direction*** to the last segment, and, keeping the **Shift** key down, click the left mouse button on the last segment and then press **Enter** (or, keeping the **Shift** key down, double click the left mouse button on the last segment) to select the series of segments and display the **Boundary Condition Type** dialog box.

     OR

   - For *noncontiguous* boundary segments, hold the **Ctrl** key down and click on multiple segments, then press **Enter** on the last segment (or, hold the **Ctrl** key down and double-click the left mouse button on the last segment) in order to access the **Boundary Condition Type** dialog box.

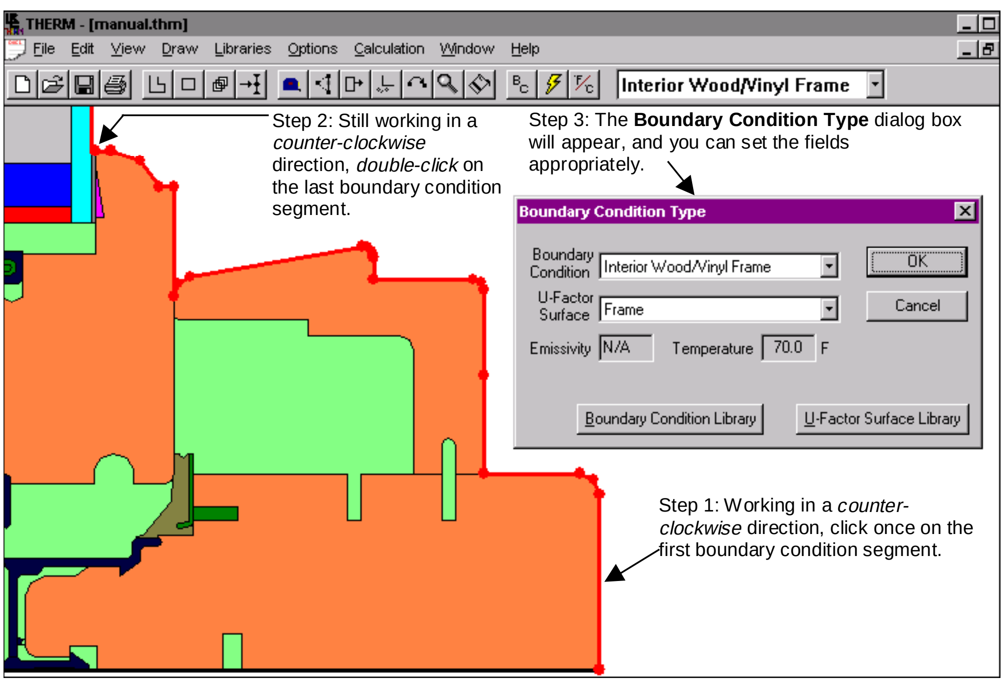


Figure 6-2. Select multiple boundary segments by pressing down the shift key and clicking the mouse on segments as you move in a counter-clockwise direction, OR click on the first segment and then double-click on the last segment, in a counter-clockwise direction; all segments in between will be selected.

2. When you have the selected boundary segments, you can use two methods for reassigning boundary conditions to them:

   - Select the boundary condition definition from the pull-down list on the right-hand side of the toolbar; this list shows the entries in the **Boundary Condition Library**.

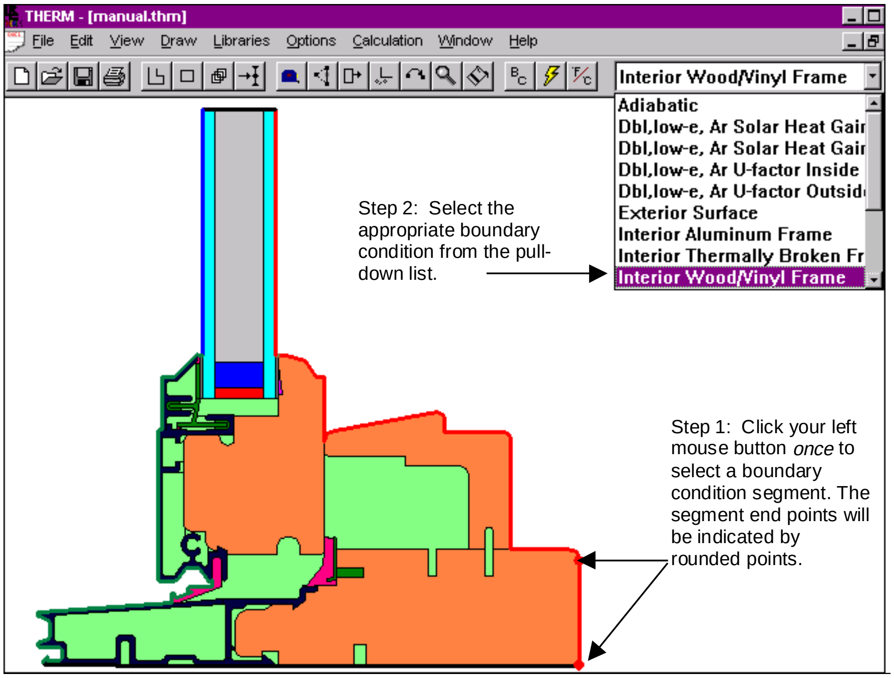


Figure 6-3. One method for defining the boundary condition properties is to click on one or more boundary segments and pick the desired boundary condition definition from the pull-down list.

OR

   - Press **Enter** after you have selected the boundary condition segment(s), which accesses the **Boundary Condition Type** dialog box. This method has the advantage of allowing you to change the **U-factor Surface** tags (see Section 6.2.4, "Define U-factor Surface Tags") and also accessing the **Boundary Condition Library** in addition to selecting the boundary condition. (Alternatively, you can select the boundary segment with one left mouse click; and then select the **Library/Set Boundary Condition** menu choice to access the **Boundary Condition Type** dialog box, or select the **Library/Boundary Condition Library** menu choice to access the **Boundary Conditions** library dialog box.)

     Select the appropriate values from the pull-down lists for **Boundary Condition** and **U-Factor Surface**. The choices in the **Boundary Condition** list show all the entries in the **Boundary Condition Library**, and the choices in the **U-Factor Surface** list show all the entries in the **U-Factor Surface Library**. These libraries can be accessed directly from this dialog box using the two buttons at the bottom of the box. If you have imported a glazing system, the boundary conditions for that glazing system will also appear in the **Boundary Condition** list (the glazing system name appears as a prefix to the title) for both **U-factor** and **Solar** conditions.

     You can access the **Boundary Condition Library** (shown in the following figure) to view the properties associated with a boundary condition by clicking the **Boundary Condition Library** button at the bottom of the **Boundary Condition Type** dialog box. The predefined boundary conditions that are provided with THERM cannot be edited, but you can add new boundary conditions to the library (see Section 6.4, "Defining New Boundary Conditions" later in this chapter).

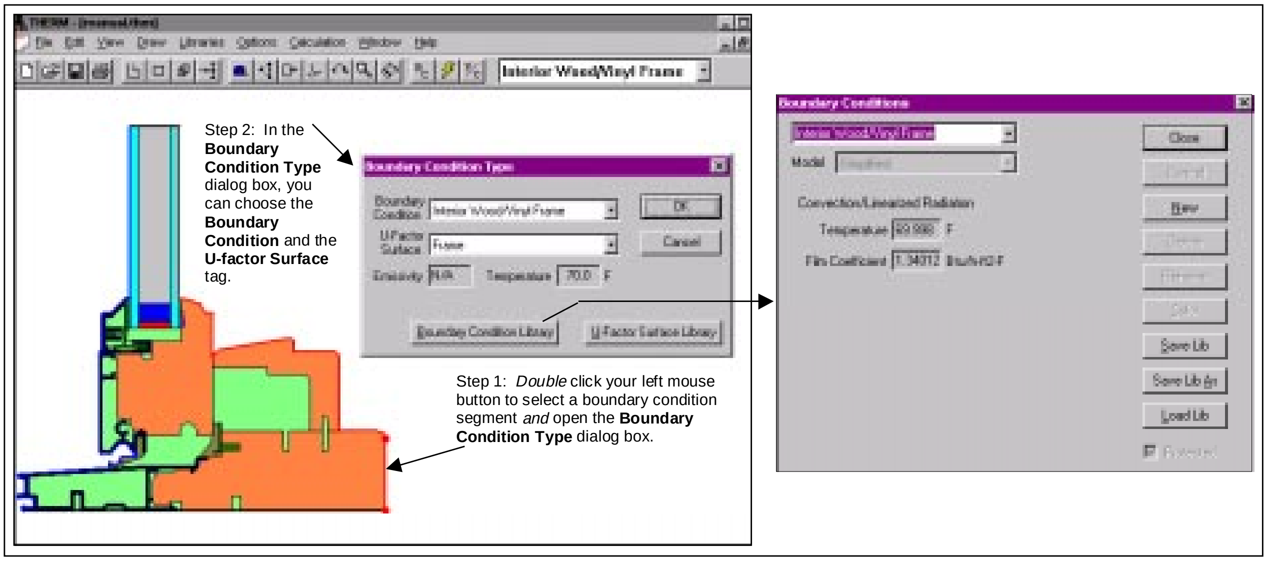


Figure 6-4. Another method for assigning boundary conditions is to double-click on a segment to access the ***Boundary Condition Type*** *dialog box where you can select the* ***Boundary Condition*** *as well as the* ***U-factor Surface*** *tag name.*

Glazing systems that are imported from WINDOW have predefined boundary conditions; THERM assumes that the outside surface is on the left unless you flip the glazing system horizontally after it is imported. Four boundary conditions are imported with the glazing system:

- *U-factor Inside Film, U-factor Outside Film:* calculated under environmental conditions equivalent to ASHRAE winter.
- *Solar Heat Gain Inside Film, Solar Heat Gain Outside Film:* calculated under environmental conditions equivalent to ASHRAE summer.

These boundary conditions show the WINDOW glazing system name as a prefix. For example, the name of an inside glazing system boundary condition would be "Dbl, low-e, Ar U-factor Inside Film" if the name of the glazing system imported from WINDOW was "Dbl, low-e, Ar."

3. Continue to define all the boundary conditions. When you are finished your cross section should look similar to the following figure. You can see boundary conditions more clearly if you turn off the material colors (go to the **View** menu and uncheck the **Material Colors** choice; then, you will see only the polygon outlines, and the boundary conditions appear prominent). Similarly, you can turn off the display of the boundary conditions by unchecking the **Boundary Condition** choice in the **View** menu.

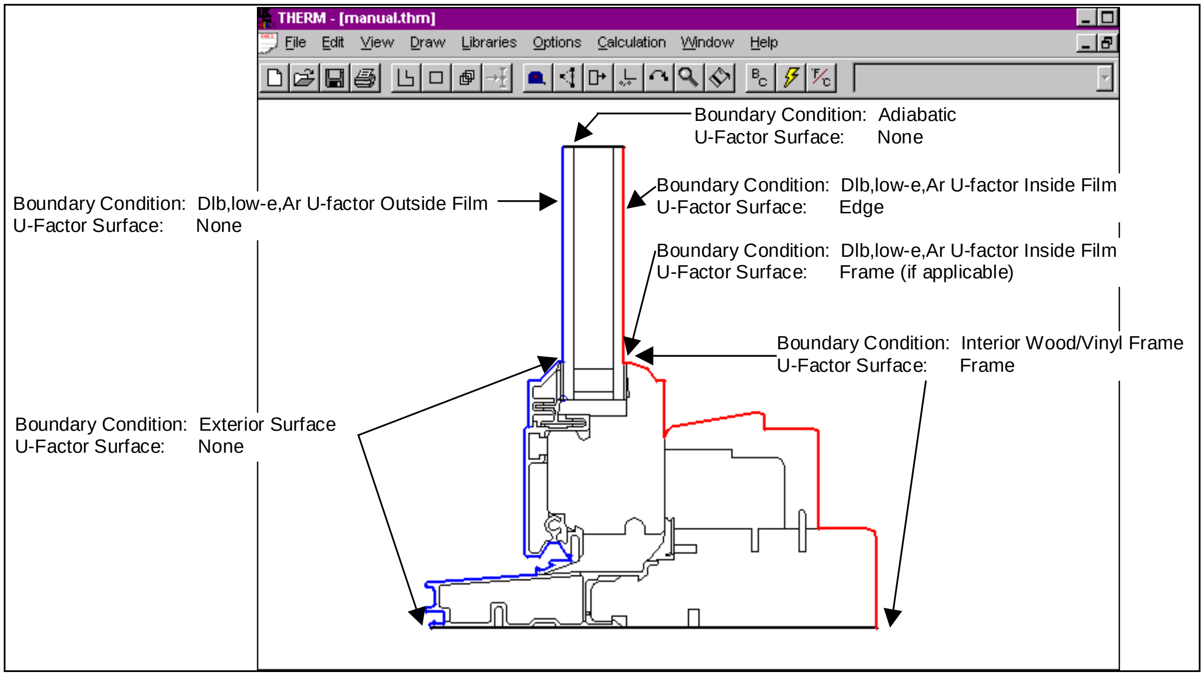


Figure 6-5. Final boundary condition configuration for a simplified model

6.2.4. Define U-factor Surface Tags

U-factor surface tags are used to label and group boundary segments for the program's U-factor calculation. Without these tags the program will not calculate U-factors. The U-factor is a measure of the heat transfer through a cross section under specific environmental conditions. The U-factor calculation integrates the heat flux for the boundary segment (or group of segments that have been given the same tag), divides that flux by the projected length of the segment(s) and the defined temperature difference, and displays a U-factor. A single U-factor tag can be assigned to a group containing as many segments as you wish. You can create as many tags or groups as you wish. A U-factor will be calculated for each unique tag name. U-factors can be calculated for a group of boundary segments even if individual segments within the group were defined with different boundary conditions.

It is common practice in the analysis of building components to define the U-factor on interior (room) boundary surfaces. Interior and exterior surfaces are often separated by adiabatic boundaries. The U-factor of adiabatic boundaries (those having no heat flow) will be zero, so there is no need to tag these surfaces. If U-factors are desired for the exterior boundary surfaces, these surfaces should be tagged separately from interior surfaces. THERM assumes that no heat is stored in the cross section, so all energy that enters the cross section on the interior surface leaves through the exterior surface. If the full length of the interior and exterior surfaces are given the same **U-factor Tag**, the resulting U-factor will be zero.

The choices for the **U-factor Tag** pull-down list are the values that come with THERM:

- *None:* Use **None** for adiabatic boundary segments or any segment where a U-factor result is not desired.
- *Frame:* Use **Frame** for interior frame boundary segments and for glazing boundary segments that are between the frame and the sight line.
- *Edge:* Use **Edge** for all the glazing system boundary segments that are above the sight line.

You can make new values in the **U-factor Names Library** by clicking on the **U-factor Surface Library** button on the **Boundary Condition Type** dialog box. Click on **Add** and type in a unique **U-factor Surface Tag** name, and the program will add that name to the **U-factor Names Library**.

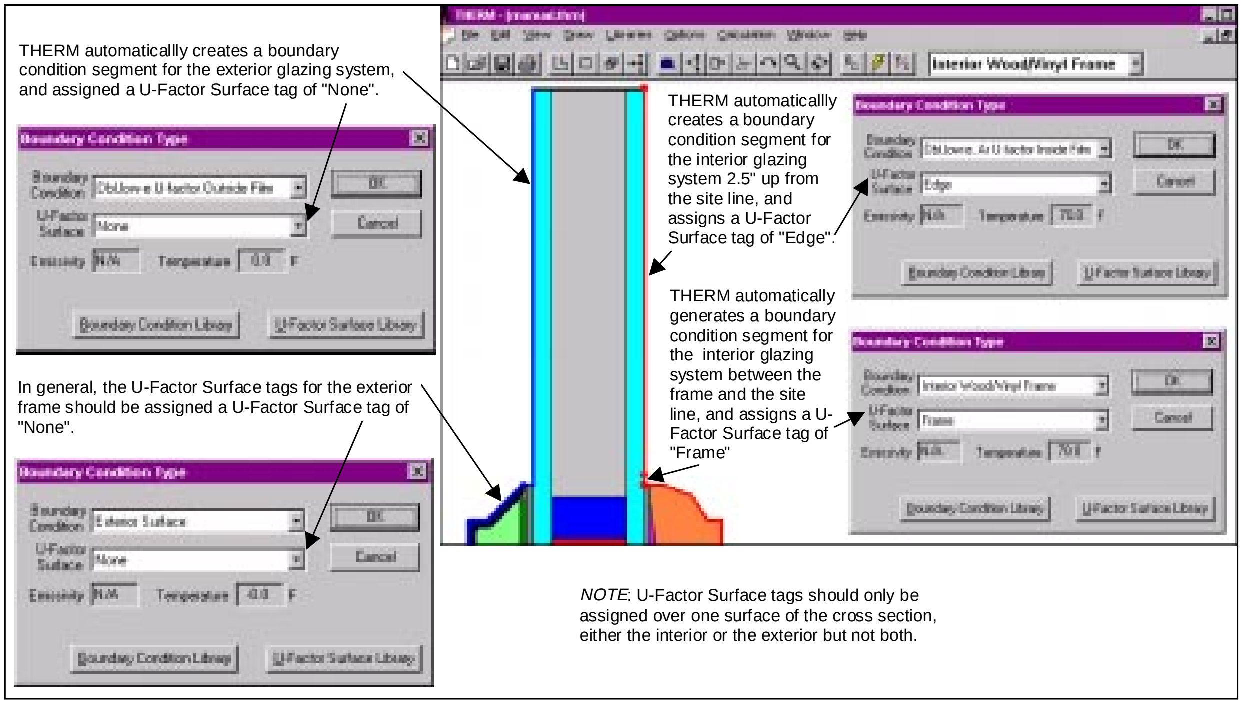


Figure 6-6. Assigning U-Factor Surface tags to different boundary segments. THERM automatically makes two boundary segments on the interior surface of the glazing system, one from the frame to the sight line (with a U-Factor Surface tag of "Frame") and one from the sight line up to 2.5" of the glass (with a U-Factor Surface tag of "Edge").

6.3. Error Detection in THERM

Defining the boundary conditions invokes several features in THERM which detect errors made during the drawing process. The error detection features are intended to help you fix questionable (or “bad") points and find small voids and overlaps in the geometry so that the automatic mesh generator can create a well formed mesh. Other error detection features in THERM that identify invalid polygons and polygons which cause the automatic mesh generator to fail are discussed in Section 7.3.

THERM's error-detection algorithms rely on the definition of two minimum allowable distances between two adjacent points in a cross section:

- **Floating point tolerance:** The distance between two points at which THERM will automatically merge the points, set to 0.01 mm.

- **Checking tolerance:** The distance between two points at which THERM will mark the points as bad, set to 0.1 mm.

When checking for geometry errors, THERM does the following:

- First, THERM checks the polygon edges because the mesher needs exactly matching points on any polygons that are adjacent to each other. If a point exists on the edge of a polygon but not on the edge of an adjacent polygon, and if the adjacent edge is less than the floating point tolerance (0.01 mm), THERM automatically adds a point to the adjacent edge and merges the points so that there is an identically placed point on both polygons. If the distance between the adjacent edges is greater than the floating point tolerance, the program does nothing, and an unmatched edge is created resulting in either a void or an overlapping region, as shown in Figure 6.7.

- Next, THERM checks to see if any points are closer together than the checking tolerance (0.1 mm), and marks such points with red circles, indicating they are "bad" points. It is important for you to check and correct these marked areas before the simulation continues.

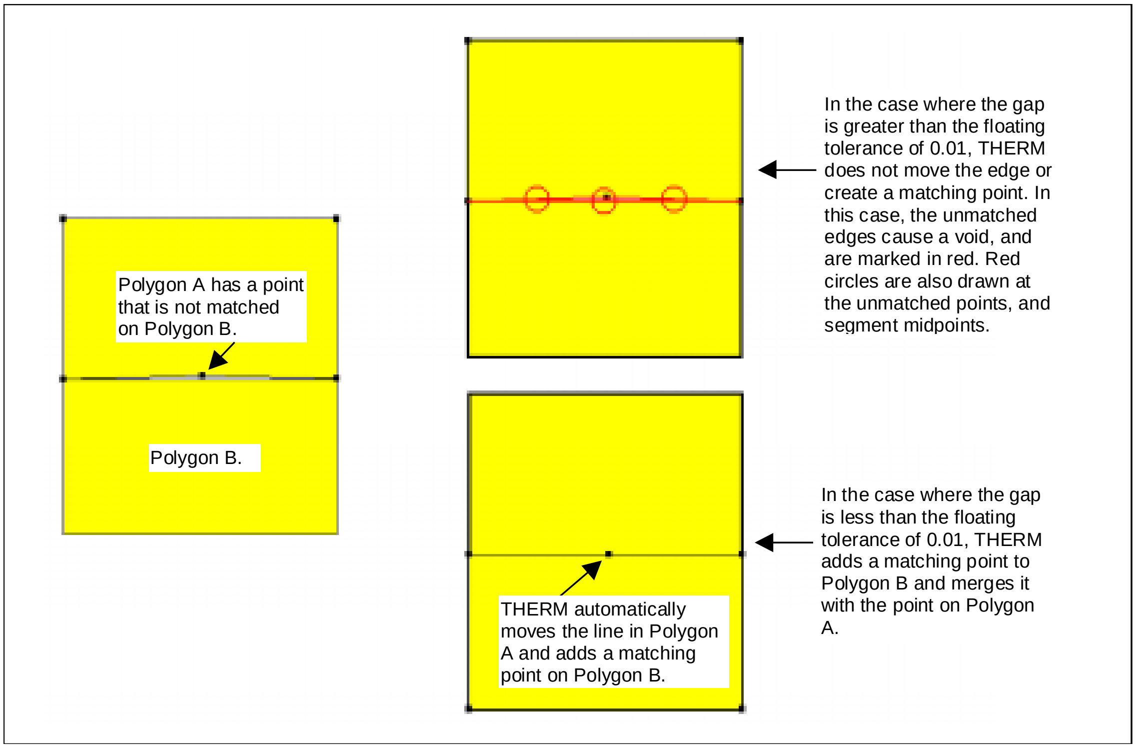


Figure 6-7. The size of the void in a drawing will determine how THERM will handle the error checking.

6.3.1. Finding and Fixing Bad Points

As discussed above, THERM marks points as bad if they are separated by a distance of less than the checking tolerance (0.1 mm). If these points are mistakes or flaws in the model they should be fixed but in some cases they may be intentional details that should be left alone. Because you may want to keep these points in the model, THERM draws red circles around the points and displays a message, shown in Figure 6-8, which gives you the option of having the program either mark the points so that you can evaluate them or automatically adjust them:

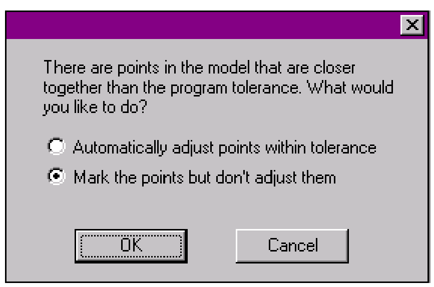


Figure 6-8. This message may appear when you create the boundary condition

- *Mark the points but don't adjust them:* This is the default option and it is recommended that you choose this option for your first attempt at assigning the boundary conditions. It allows you to check the circled points and either correct the points manually or leave them in the model if you want to keep them.

  Occasionally points exist on the edges of both adjacent polygons, but they are not exactly aligned. In this case the program adds matching points to the edges of each polygon resulting in points on both polygons that are very close together (which may be detected as bad points). In this case, you need to delete one of the points, remembering to delete the point on both polygons. This can be done by selecting one polygon, holding down the shift key, selecting the other polygon and then using the **Draw/Edit Points** menu choice to delete the point from both polygons in one step. If you do not delete the point on both polygons THERM will automatically recreate it when you reassign the boundary conditions. Be careful in choosing which point to delete that you select the one that will allow the geometry of the model to be accurately represented.

If you choose to have the program mark the bad points, you may want to improve the clarity of the display by turning off the material colors. Go to the **View** menu and uncheck the **Material Colors** choice. Then you will see only the outlines of the polygons, and the circled points will become very obvious. You can zoom in (click the right mouse button) on the circled area to see if you need to fix the points.

Sometimes there will be a red circle in an area that looks fine. If you select the vertices attached to that point one at a time and move them a short distance (less than the sticky distance) from the point and release the point, the vertices will snap to a common point and the problem point will be removed. If your model only contains areas like this you can use the Automatically Adjust Points within Tolerance options without negative consequences.

If there are fine details containing points less than 0.1 mm apart that you want to preserve in your model, you can turn off the display of the red circles using the **View/Bad Point** command. However, even though you can't see the circles they are still there. If you want to remove the red circles, they can be cleared using the **Draw/Clear Bad Points** command.

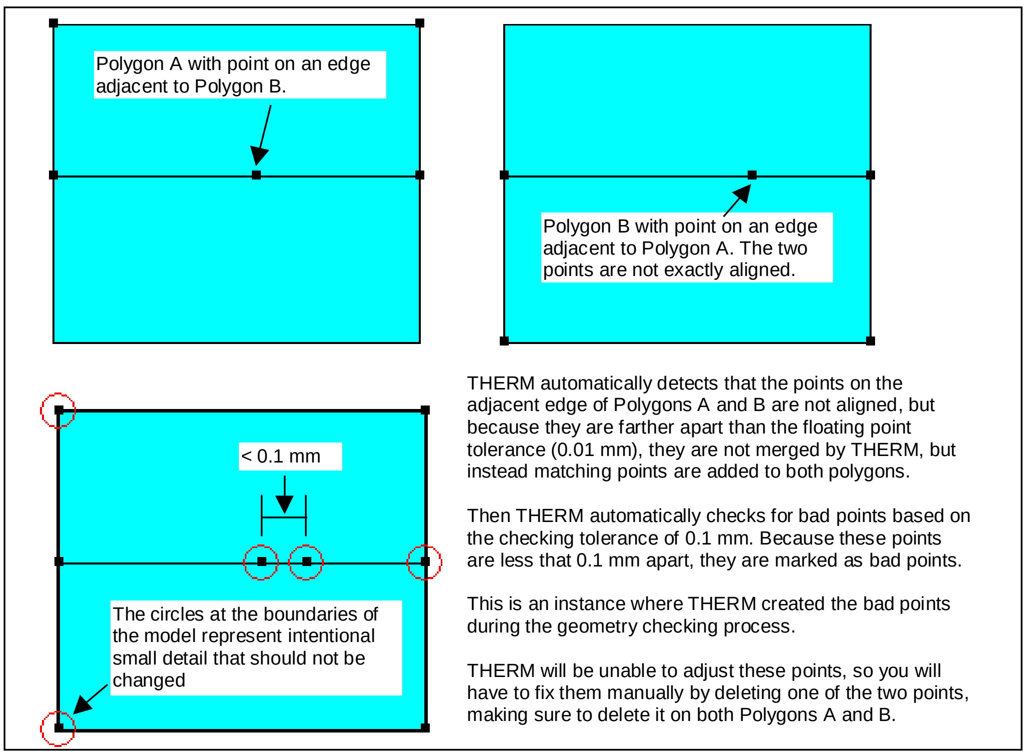


Figure 6-9. An instance where THERM created bad points in the process of checking for matching points on adjacent polygons.

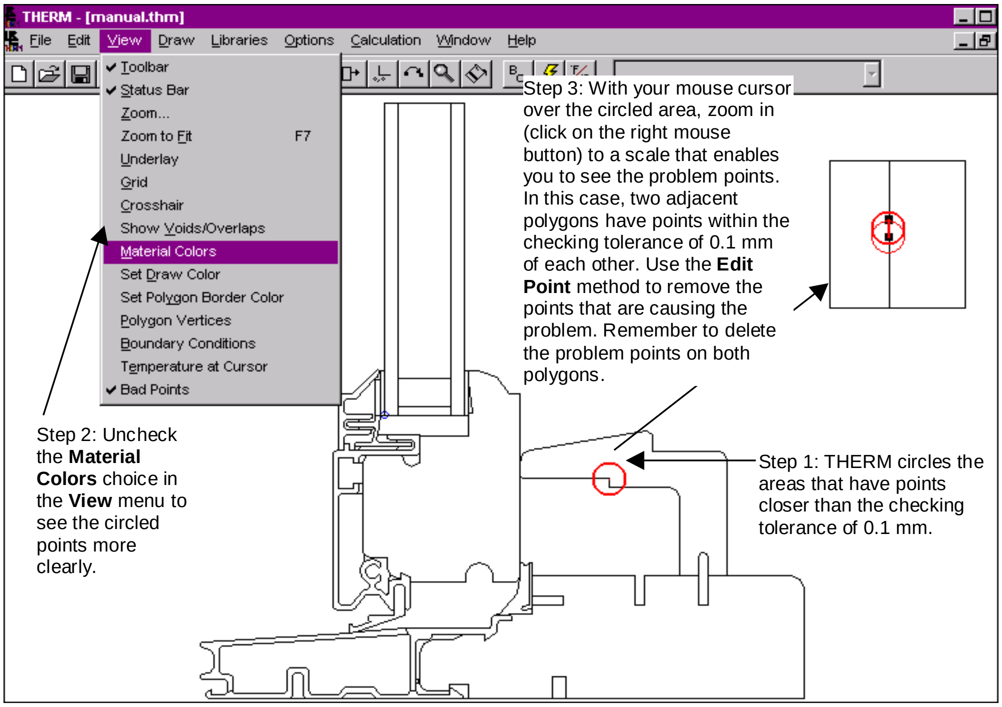


*Figure 6-10. To view the circled "bad" points more clearly, "uncheck" the **Material Colors** choice from the **View** menu.*

- *Automatically adjust points within tolerance:* When this option is chosen, THERM will automatically merge any two points that are within the checking tolerance of 0.1 mm. In cross sections with fine detail the automatic fix feature can make unwanted changes in your model. It is strongly recommended that you save your work before using this feature. Once this option is selected it cannot be undone with the undo command.

  Sometimes when using the automatic adjust option you will get a message saying that THERM cannot fix all the bad points, as shown in Figure 6-11. The unfixed points will be marked and you must fix them before the simulation continues.

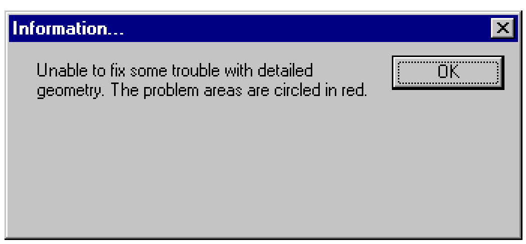


Figure 6-11. This message may appear if you select the automatic adjustment option and THERM cannot correct the problem.

The existence of bad points does not keep THERM from assigning the initial boundary conditions. When you are adjusting the model to correct bad points, if you do not make changes to the segments that make up the boundary, the boundary conditions will not be deleted and you can continue with the simulation process without reassigning the boundary. This is the recommended way to proceed as it ensures that you have correctly fixed all problems in the model. It is recommended, however, that you do recreate the boundary conditions to ensure that you have correctly fixed all problems in the model. If you do recreate the boundary conditions make sure to delete the existing red lines and circles using the **Draw/Clear Bad Points** menu option. THERM will remember the boundary condition assignments so you will not need to redefine the boundary segments.

6.3.2. Finding Voids and Overlaps

Defining the boundary conditions also causes THERM to check the model for small voids and overlaps that are caused by unmatched polygon edges as discussed in Section 6.3, "Error Detection in THERM." (Large voids and overlaps can be detected using the **View/Voids and Overlaps** featured discussed in Chapter 5). An unmatched edge is an edge on a polygon that does not have a matching edge on an adjacent polygon, which indicates that there is either a void or an overlap in that area. THERM marks small voids and overlaps by drawing the unmatched edge in red with a circle at its midpoint and displaying the warning message as shown in Figure 6-12. These areas must be fixed before the simulation process can continue.

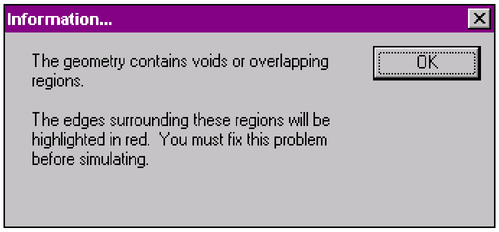


Figure 6-12. THERM issues a warning if it detects voids or overlapping regions when trying to define boundary conditions.

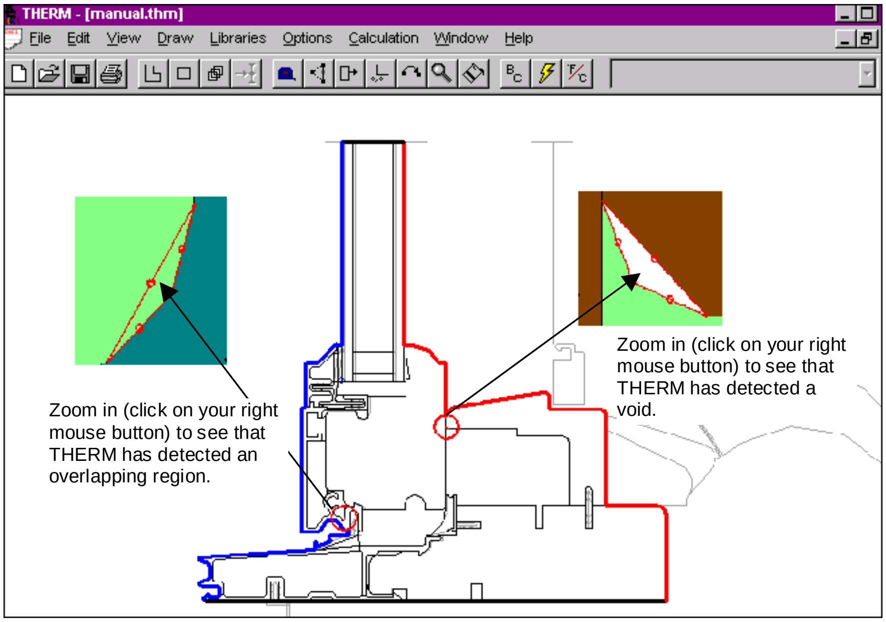


Figure 6-13. Voids and overlaps are indicated by circles once the boundary conditions have been assigned.

6.4. Defining New Boundary Conditions

THERM includes a Boundary Conditions Library with a set of predefined boundary conditions that are protected from editing (the input values are disabled). If you do not find the boundary condition you need in the **Boundary Condition** pull-down list, you can add it to the **Boundary Condition Library** using the following technique.

1. Open the **Boundary Condition Library** by selecting the **Library/Boundary Condition Library** menu choice; the **Boundary Conditions** dialog box will be displayed. The predefined boundary conditions cannot be edited, but you can use the pull-down list at the top to view the characteristics of any of the entries.

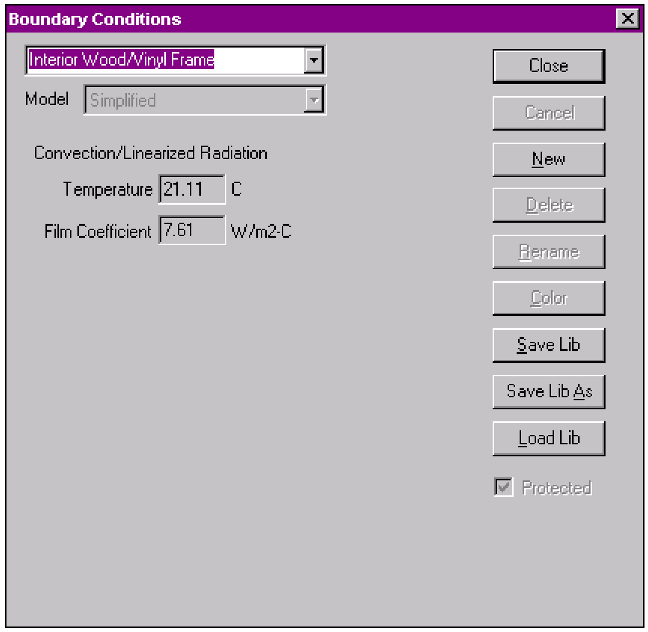


Figure 6-14. The default Boundary Condition Library that comes with THERM has boundary condition definitions that cannot be edited.

2. To make a new boundary condition, click on the **New** button.

3. You will be asked for the name of the new boundary condition. Type a name that is not already in the library and click on the **OK** button.

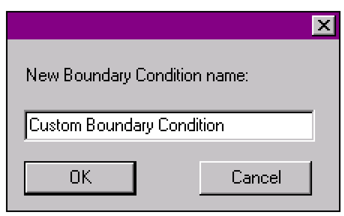


Figure 6-15. Type a unique name for the new boundary condition.

4. Click on the **Color** button to change the color from the default value of blue.

5. Now you are presented with the same **Boundary Conditions** dialog box, but all the fields are available for you to edit, so you can create your custom boundary condition. Different fields are available for editing depending on the **Boundary Condition Model** that is chosen, i.e., simplified, comprehensive, or radiation enclosure surface. These different types are discussed in detail in the following sections. Fill out the input values as appropriate.

6. When you have defined the characteristics of the boundary condition, click on the **Save Lib** button; the boundary condition will be saved permanently to the default library (named **bc.lib**). If you forget to save, THERM will ask if you want to save when you exit the program (unless you turn this feature off using **Options/Preferences/Prompt for saving libraries on program exit**).

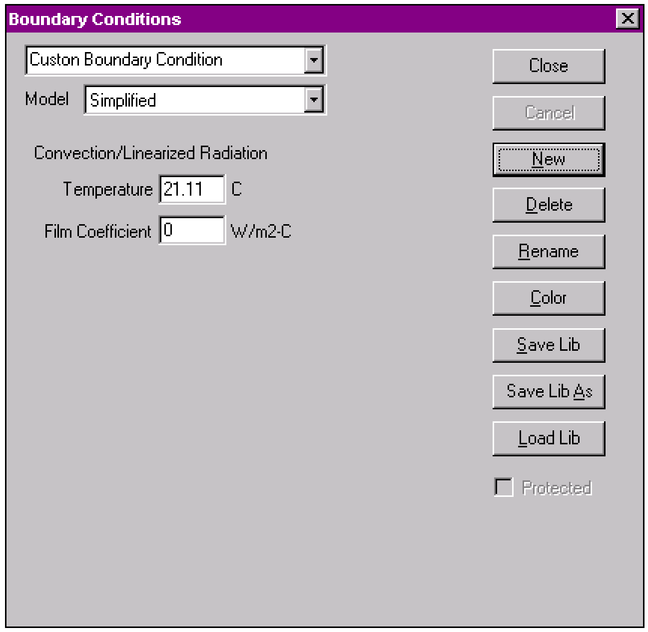


*Figure 6-16. To define a new boundary condition, click on the **New** button, and the **Boundary Conditions** dialog box will appear, allowing you to define the characteristics of the new boundary condition.*

In the **Boundary Conditions** dialog box, you can set the **Model** pull-down list to **Comprehensive**, and the input values that will be available are the following:

Convection Define the convection values for:

- *Temperature* -- surrounding temperatures for convective heat transfer
  Units: °F (IP); °C (SI)

- *Film coefficient* -- film coefficients to use for convective heat transfer
  Units: Btu / h-ft^2-°F (IP); W / m^2-°C (SI)

Constant Heat Flux Used to model a constant heat flux on the surface, such as solar gain

- *Flux* -- constant flux value
  Units: Btu / h-ft^2 (IP); W / m^2 (SI)

Radiation Used to define the radiation component of the model

- *Enclosure Model:* use this if you are defining boundary conditions for the cross-section elements that "see" the external radiation enclosure.

- *Black-Body Radiation:* use this if you want to model radiation exchange with a black body as an alternative to drawing a radiation enclosure. For this case you have to define the values for view factor and temperature (Ti).

Constant Temperature Used to define the temperature at the surface being specified

- *Temperature*: Units: °F (IP); °C (SI)

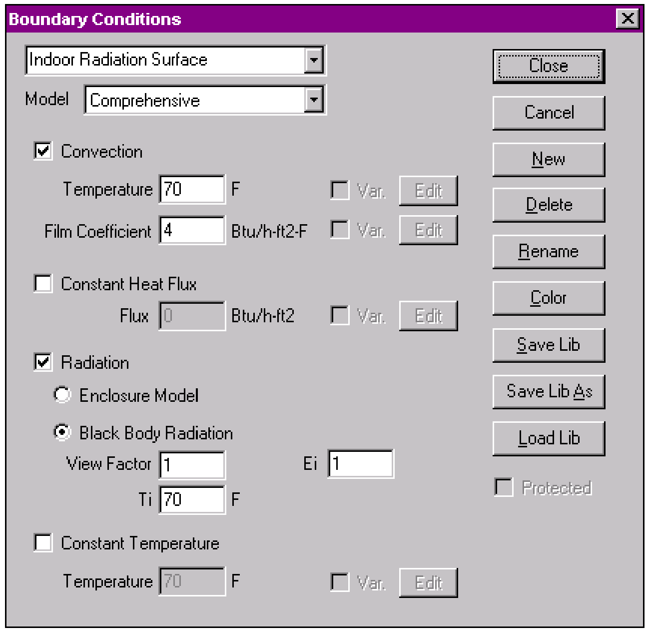


Figure 6-17. Sample boundary condition library entry for a comprehensive model

6.5. Special Boundary Condition Cases

There are two special boundary condition cases in THERM, for the following conditions:

- Detailed frame cavity radiation model
- External Radiation Enclosure

6.5.1. Frame Cavities with Detailed Radiation Model

If you used a detailed radiation model within a frame cavity, then, when you create the boundary conditions for that cavity, THERM will automatically highlight the cavity surfaces in red and label them **Frame Cavity Surface** (shown in the following figure).

Sometimes frame cavities are broken up into multiple cavities to better model convection heat transfer. If this is done with a detailed frame cavity model THERM treats the sub-cavities individually for convection purposes but merges them for radiation purposes. The red boundary will show this.

If you change the cavity to another type of material after the boundary conditions are drawn, you must redefine the boundary conditions.

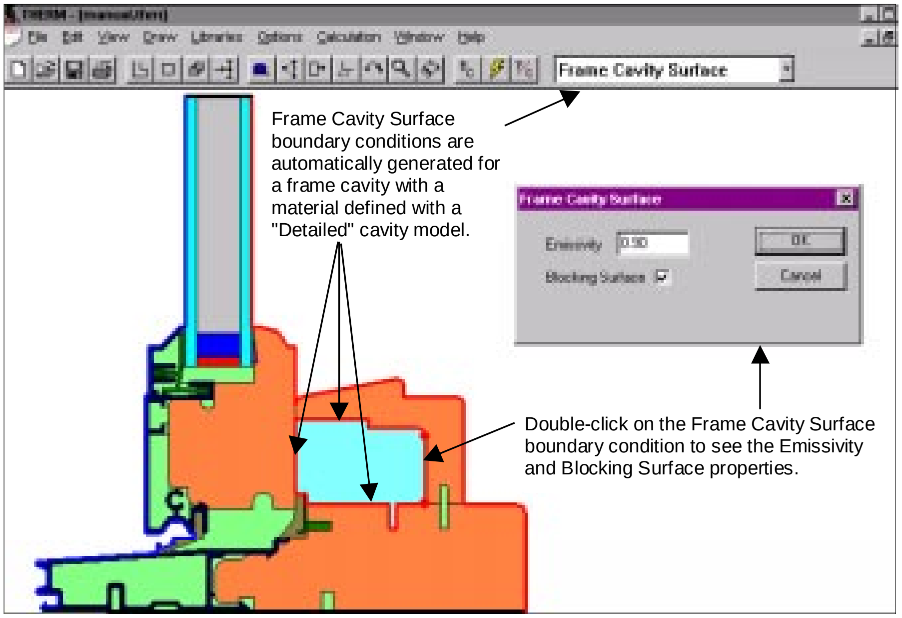


Figure 6-18. When the frame cavity is assigned a material with a Detailed cavity model, THERM automatically assigns boundary conditions to the cavity.

To see the **Frame Cavity Surface** properties, double-click on one of the red segments; a dialog box will appear with the following values:

Emissivity — This value is the emissivity of the frame cavity surface, automatically determined by THERM based on the emissivity of the surrounding materials as defined in the **Material Library**. You can override the definition by changing the value in this input box. If the emissivity value is changed in the **Material Library** after the boundary conditions are drawn, the change will not take effect until the boundary conditions are redrawn (by clicking on the **Boundary Conditions** toolbar button or pressing **F10**.)

Blocking Surface — This check box indicates whether the surface that you are defining should be checked automatically by the program to determine if it blocks radiation. A radiation blocking surface is a surface that interferes with the radiation that would otherwise travel between surfaces, as shown in the following figure. Checking this box does not automatically make it a blocking surface, it just tells THERM to determine whether or not the surface is blocking. If the box is not checked, the program will not make this determination. **Default:** Checked, which means that all surfaces will be checked by THERM to see if they are blocking. This default is recommended because it guarantees that the radiation is modeled correctly. However, for simple rectangular surfaces, there is no need to check for blocking surfaces.

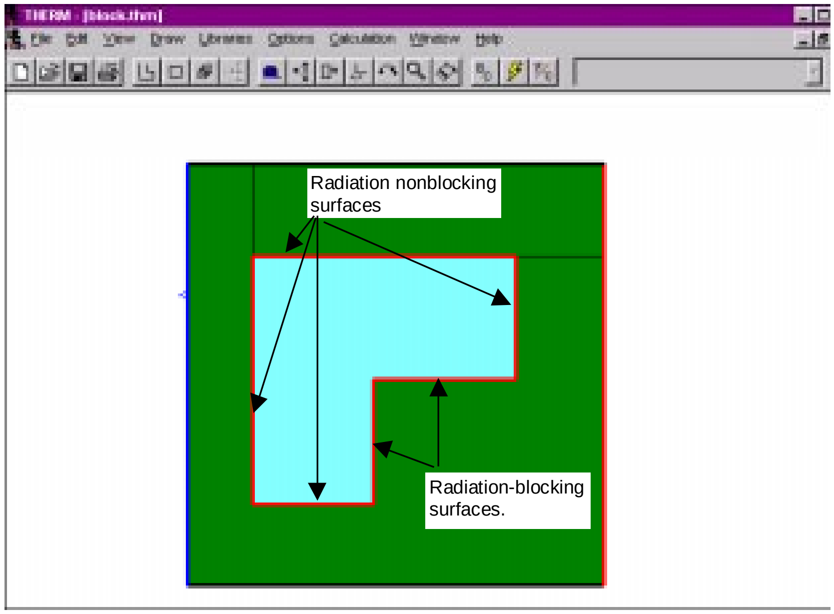


Figure 6-19. An example of radiation-blocking surfaces

6.5.2. External Radiation Enclosures

If you assign a radiation enclosure material to a polygon, then, when you create the polygon's boundary conditions, THERM will automatically assign boundary conditions named **Radiation Surface** (shown in the following figure).

To see the **Radiation Surface** properties, double-click on one of the boundary conditions; a dialog box will appear showing the emissivity and temperature, as well as a checkbox indicating whether the surface is blocking radiation.

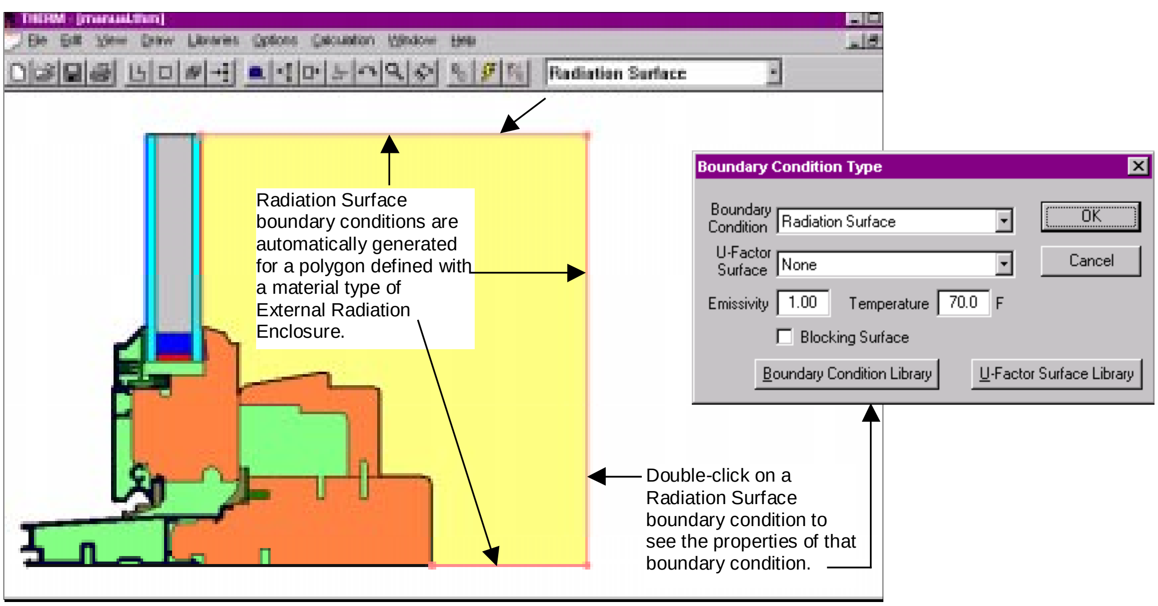


Figure 6-20. THERM automatically assigns Radiation Surface boundary conditions to the external radiation enclosure surfaces.

In order for the program to model an external radiation enclosure, it may be necessary to change the boundary conditions of the cross section elements that "see" the enclosure, in this case the interior frame and glazing assembly elements.

Changing these boundary conditions requires you to first create custom **Boundary Condition Library** records (see Section 6.4, "Defining New Boundary Conditions") that have the following definitions, as shown in Figure 6-21:

Model Set to **Comprehensive**.

Convection Define the appropriate temperatures and film coefficients for each element type. Make sure that the film coefficient only accounts for convection and does not contain a radiation component. (The film coefficients in the simplified boundary condition are a combination of convective and radiative effects.)

Radiation Set to **Enclosure Model**.

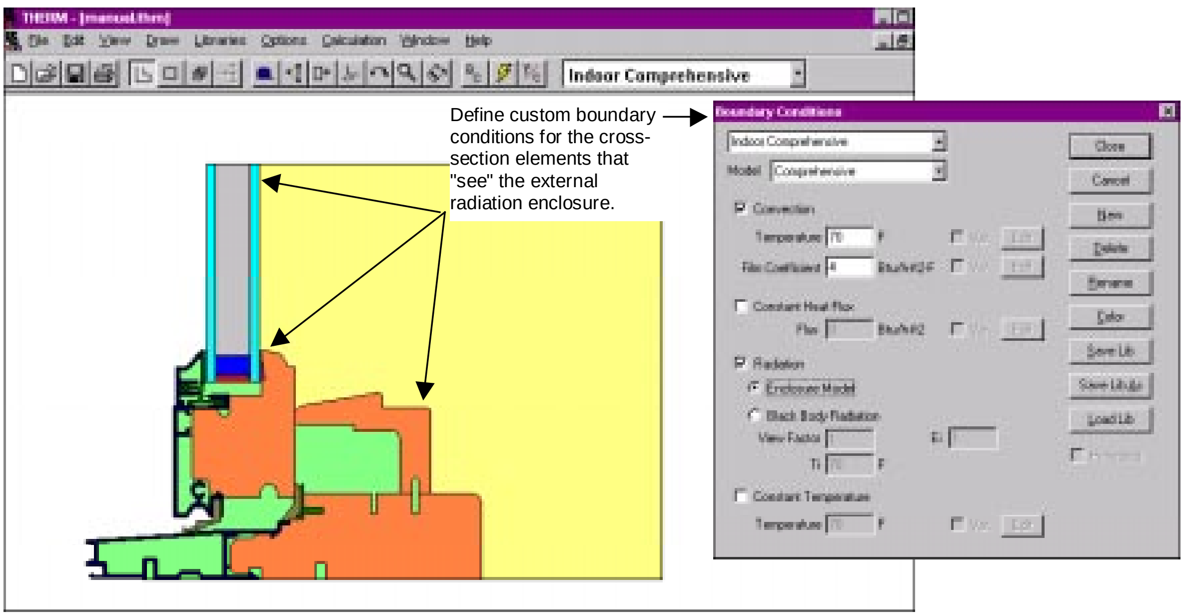


*Figure 6-21. Define new boundary conditions for the elements that "see" the radiation enclosure with the **Comprehensive** model.*

When you have defined the new boundary condition in the library, you need to assign it to the appropriate boundaries, as shown in Figure 6-22.

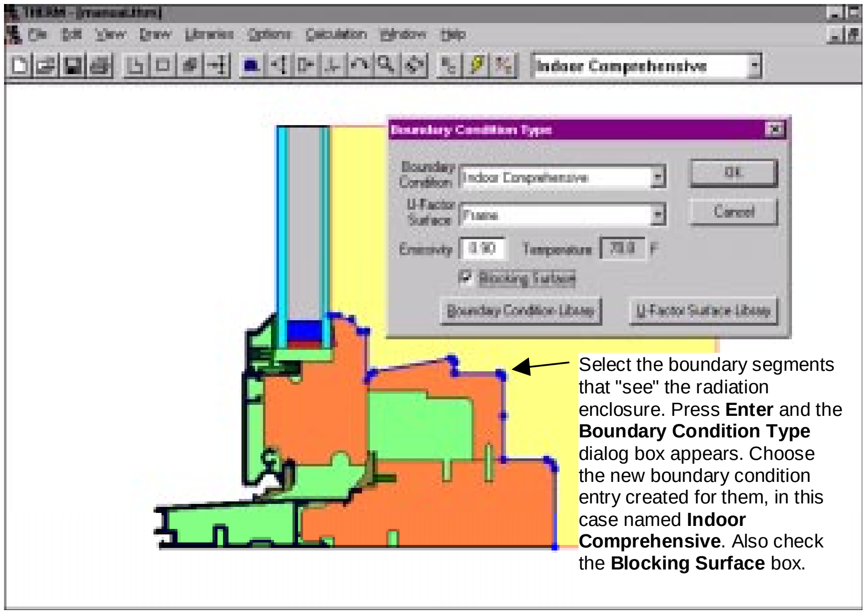


Figure 6-22. Set the boundary conditions for the surfaces that "see" the radiation enclosure.

7. CALCULATING RESULTS

7.1. Overview of THERM Calculation Methods

THERM is a two-dimensional (2D) finite-element heat-transfer analysis tool. Many excellent reference books describe the finite element method in detail[(8,9)]. THERM's steady-state conduction algorithm, CONRAD[(10)], is a derivative of the public-domain computer program TOPAZ2D[(6,7)]. THERM's radiation view-factor algorithm, VIEWER, is a derivative of the public-domain computer program FACET[(12)]. THERM contains an automatic mesh generator that uses the Finite Quadtree[(5)] algorithm. THERM checks solutions for convergence and automatically adapts the mesh as required using an error-estimation algorithm based on the work of Zienkiewicz and Zhu[(18,19)].

THERM's calculation routines evaluate conduction and radiation from first principles. Convective heat transfer is approximated through the use of film coefficients obtained from engineering references[(11,17)].

See Appendix C, "Theoretical Background" for more information about the calculation methods in THERM.

7.2. Calculations

When you have finished drawing a cross section, specifying its materials, and defining boundary conditions, you are ready to calculate the cross section's thermal performance.

7.2.1. Calculation Menu

The **Calculation** menu choices are shown in the figure below. If the results are not current, the **Display Options** and **Stop Current Calculation** menu choices will be inactive (grayed out) until you calculate current results.

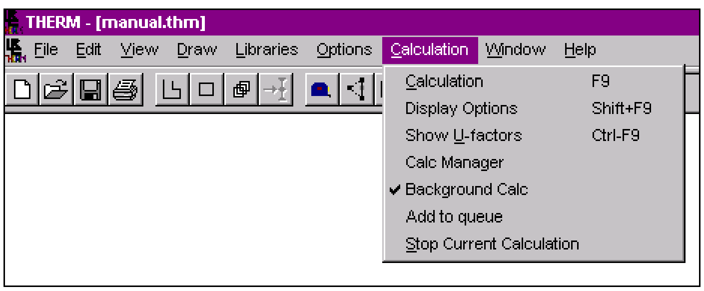


*Figure 7-1. The **Calculation** menu choices*

Calculation (F9) Use this menu choice or the **F9** keyboard short cut to start the calculation for a currently active THERM file. This is equivalent to pressing the **Calculation** toolbar button.

Display Options (Shift-F9) Use this choice or the **Shift-F9** keyboard short cut to access the **Results Display Options** dialog box. The default setting is to display isotherms when a calculation is finished. The results will display with the last choice selected. If the **Display Options** menu choice is inactive (gray), the results are not current, so you need to calculate them before you can access this choice.

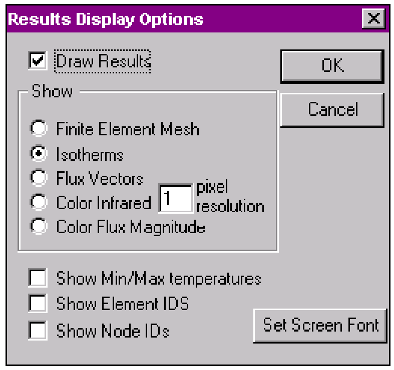


*Figure 7-2. The **Results Display Options** dialog box*

Show U-factors (Ctrl-F9) Use this choice or the **Ctrl-F9** keyboard short cut to display the **U-Factors** results dialog box. See Section 7.4.1, "U-factors" for more information about the **U-Factors** results display.

Calc Manager Use this choice to run a list of THERM files in batch mode. The program runs the files one at a time, starting from the file at the top of the list. Selecting the **Calc Manager** menu choice opens the **Calculation Manager** dialog box, shown in the figure below:

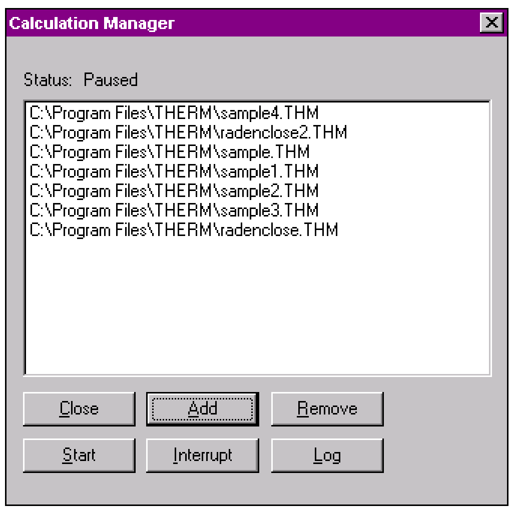


*Figure 7-3. Selecting the **Calculation/Calc Manager** menu choice opens the **Calculation Manager** dialog box which shows a list of THERM files that can be run.*

The **Calculation Manager** buttons are as follows:

- *Close:* The **Close** button closes the **Calculation Manager** dialog box. Even if this box is closed, the THERM files in the queue will still run.
- *Add*: The **Add** button is used to add another THERM file to the end of the list of files to run. Pressing **Add** opens a browse window which allows you to select files from anywhere on your computer or network. You can select multiple files using your mouse with either the **Shift** key (to select multiple contiguous files) or the **Ctrl** key (to select multiple noncontiguous files), in standard Microsoft Windows™ style.
- *Remove:* The **Remove** button deletes a file from the list. Highlight a file in the list by clicking on it once with your mouse; then, press **Remove**. The file will be deleted from the list.
- *Start or Pause:* The **Start** or **Pause** button toggles between starting and pausing files. If the button says **Start**, clicking on it causes the program to start running the files in

the list, beginning with the first file. If the button says **Pause**, clicking on the button causes the program to pause running all of the files in the list. You can click on the button when it says **Pause** (which changes the button to say **Start**) and then you can click the **Add** button to select additional files to be run. When you are ready for the files to begin running, click on the **Start** button.

- *Interrupt*: The **Interrupt** button causes the currently running THERM file to be stopped after the program finishes the current calculation step (such as Reading Geometry or Generating Input.)
- *Log*: The **Log** button displays a list of the files that have been run as well as any error messages generated during the calculation. This log can be cleared manually using the **Clear Log** button on the **Calculation Log** dialog box, and it is automatically cleared when you close the THERM program.

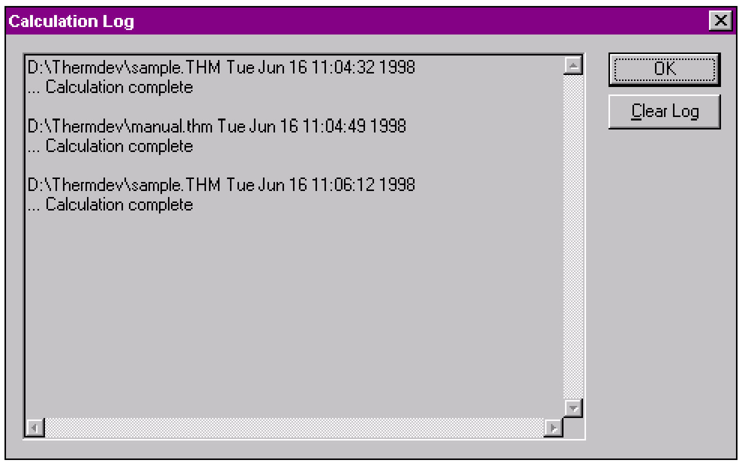


Figure 7-4. ***Calculation Log accessed from Calc/Calc Manager.***

Background Calc — Selecting the **Background Calc** menu choice causes the program to run the simulation in the background, allowing you to do other work in THERM while a model is being calculated. When **Background Calc** is on, the program displays the file name and the current calculation status (such as "Generating Input" or "Simulating") in the THERM status bar (see Chapter 4 for the location of the status bar). If **Background Calc** is selected, a check mark will display to the left of the menu choice, and the feature will remain on until it is unchecked. If this option is not turned on, an hourglass will be displayed while the program runs a THERM file, and you will not be able to do any other work until the calculation is finished.

Add to queue — Selecting the **Add to queue** menu choice causes the currently active THERM file to be added to the **Calc Manager** list.

Stop Current Calculation — Selecting the **Stop Current Calculation** menu choice causes the currently active and running THERM file to be cancelled. THERM will finish the current calculation step, such as "Reading Geometry" or "Generating Input", before stopping.

7.2.2. Calculating Results for a THERM file

To start calculating a THERM file, do the following:

1. Set the appropriate **Options/Therm File Options** and **Options/Preferences**, **Simulation** settings (see 7.2.3, "Calculation Options" for a description of these values).

2. Turn on **Background Calc** from the **Calculation** menu. When it is on, a check mark will appear to the left of the **Calculation/Background Calc** menu choice.

3. Press the **Calculation** toolbar button,  use the **Calculation/Calculation** menu item, or press **F9** to start the calculation.

4. The program will show the calculation steps being performed (because **Background Calc** is turned on). Depending on the complexity of the cross section and the fineness of the mesh as well as the CPU speed and amount of memory of your computer, calculation steps can take several seconds to a few minutes.

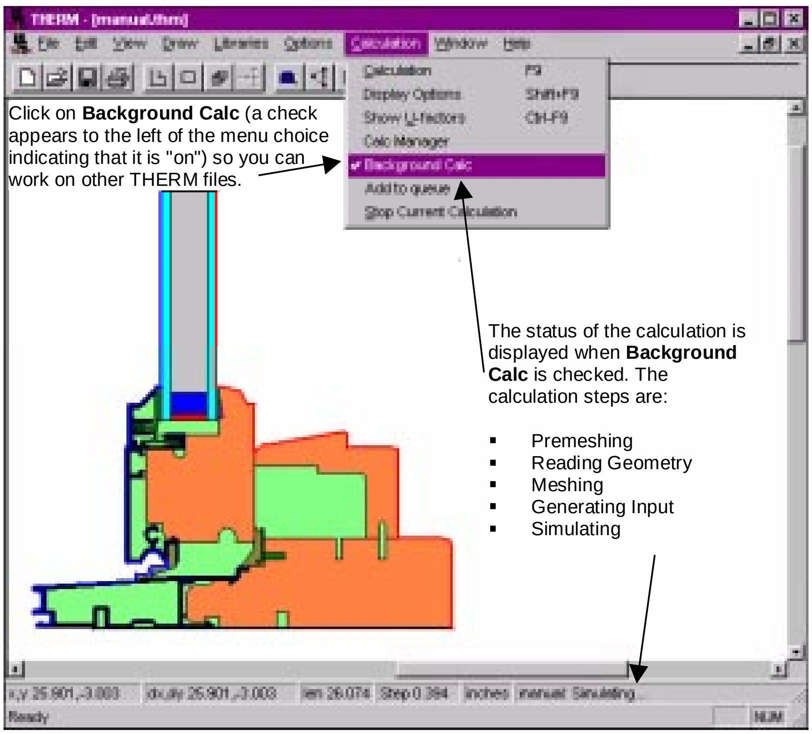


*Figure 7-5. THERM displays the steps in the calculation when the **Calculation/Background Calc** menu option is on.*

7.2.3. Calculation Options

There are several calculation options which may be changed. In most cases, the default values for these options will be appropriate.

One category of calculation options is accessed from the **Options/Therm File Options** menu, which opens a dialog box with the following settings:

Quad Tree Mesh Parameter: The mesh parameter determines the maximum element size in the mesh. The larger the mesh parameter, the smaller the maximum size. When modeling cross sections with very fine detail, you may need to increase this setting. Increasing the mesh parameter also increases the number of elements and, as a result, increases the time required to generate the mesh. Lower numbers produce coarser mesh, higher numbers produce a finer mesh. (*Note: The mesh parameter is based on geometry, and if a fine detail in the cross section is not being meshed to the level you think is necessary, and increasing the mesh parameter doesn't change this, you should check the **Run Error Estimator** check box and the mesh will be refined in this region if it is necessary.)* **Default:** 6.

Run Error Estimator

The **Run Error Estimator** check box governs whether THERM runs its error estimator. The purpose of the error estimator is to locate regions of potential error, so the mesh can be refined there to increase the accuracy of the program's calculation. The error estimator calculates a percent error energy norm. The percent error energy norm is a function of the heat flux integrated over the area of the element. If the percent error energy norm is high, the program refines the mesh so that the regions of high heat flux are integrated over smaller elements. This can be mathematically proven to result in a converged solution[18,19]. The default value for this percentage is 100%, which means THERM will not refine any areas of the mesh even when the error estimator runs. You can change the default value, but this is not necessary in all cases. The value of the percent error energy norm for your simulation will be displayed under **Calculation/Show U-factor**.

The following approach is suggested if you want to use the error estimator to have confidence that your solution is converged. First, simulate your cross section using the default setting for the error estimator. Check the percent error energy norm results, which appear under **Show U-factors** in the **U-Factor** dialog box, which is accessed by the **Calculation/Show U-factors** command under the **Calculation** menu. If the value is less than 10%, you can assume that your solution is converged. If the percent error energy norm is larger than 10%, check the values of the particular results (e.g., U-factor, local temperatures, etc.) that are of interest to you. Then rerun your problem by entering a value for the percent error energy norm that is smaller than the one currently reported in the U-factor dialog box. (Entering a target value for the percent error energy norm that is a factor of two smaller than the current reported value is reasonable way to proceed, or you can set the value between 5 and 10 %.) THERM will iterate until the percent error energy norm of every element in the cross section is less than the target value or until the specified maximum number of iterations is reached. Verify the change in percent error energy norm and recheck the results of interest (e.g. U-factors, etc.) for your cross section. If the results of interest have changed significantly, you may wish to rerun the calculation after decreasing the percent error energy norm until no significant change in results is observed. This strategy will allow you to achieve the desired level of accuracy required for your solution.

Note: The larger the mesh parameter and the smaller the percent error energy norm, the more elements will be included in your mesh and the longer your calculation time will be. A fine initial mesh will have a lower maximum percent error energy norm than a coarse mesh. A fine mesh may reach a converged solution with one run whereas a coarse mesh may take many iterations to reach the same level of convergence. You can attempt to achieve a percent error energy norm of less than 5%, but this may result in over-refinement of the mesh and accumulated round-off error, which will have an adverse effect on the accuracy of the solution.

Default: checked.

Maximum % Error Energy Norm

If the **Run Error Estimator** box is checked, the **Maximum % Error Energy Norm** value determines the percent error energy norm allowed. THERM will rerun the simulation until this value is reached. See **Run Error Estimator** above regarding when to change this value. **Default**: 100%

Maximum Iterations

This box specifies the maximum number of times the program will modify the mesh and resimulate if the **Run Error Estimator** box is checked. THERM will stop simulating and display a warning message when this value is exceeded even if the target value for the percent error energy norm has not been reached. The last set of results will be displayed. **Default**: 3.

Use CI Model for Window4 Glazing Systems — A check box indicates that you want to use the **Condensation Index Model** when WINDOW4 glazing systems are imported. **Default:** unchecked

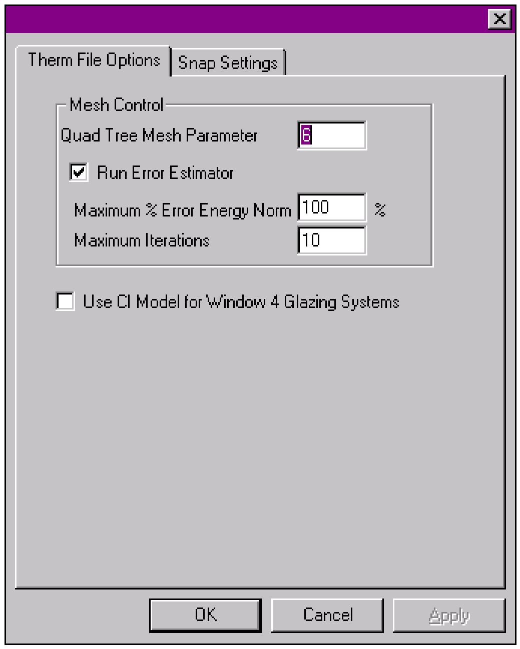


*Figure 7-6. **THERM File Options** affect how THERM calculates results.*

Another category of calculation options is accessed from the **Options/Preferences** menu, **Simulation** tab, which opens a dialog box with the following settings:

Convergence Tolerance — This feature is used when the detailed radiation model is chosen. This is an iterative model that calculates the nonlinear temperature relation that develops in combined conduction and radiation simulations. The convergence tolerance is the allowable maximum average temperature difference between two successive iterations. **Default:** 0.0001

Relaxation Parameter — The relaxation parameter is also related to convergence in problems containing the detailed radiation model. A relaxation parameter of less than one may help a problem to converge. **Default:** 1

View-Factor Smoothing — View-factor smoothing is the least square fit of view factors, which often provides betters results for view factors. **Default:** checked

Save Simulation Results in THM Files — This check box determines how the results are saved. You can save results with the THERM file, which creates a large file containing all of the mesh, temperature, and U-factor results for a cross section. If this box is not checked, the files are much smaller (1K) and only contains the U-factor results; you will be able to view the geometry of the cross section, the U-factors, and the report but not the mesh or temperature results. In this case, the results are current but not available and the **Calc/Display Options** menu choice will be inactive (grayed out). **Default:** checked

Save CONRAD Results file (.O) This option saves the CONRAD results files, that are named with the THERM file name with an extension of **.O**. This is an ASCII file that contains all the information about the model geometry, mesh, temperature and heat flux results. **Default:** unchecked

Save simulation Intermediate files This option is useful in determining why a simulation has failed. Intermediate files are saved in the SIM subdirectory of the THERM directory, and quickly use up disk space. This is primarily a research feature. **Default:** unchecked

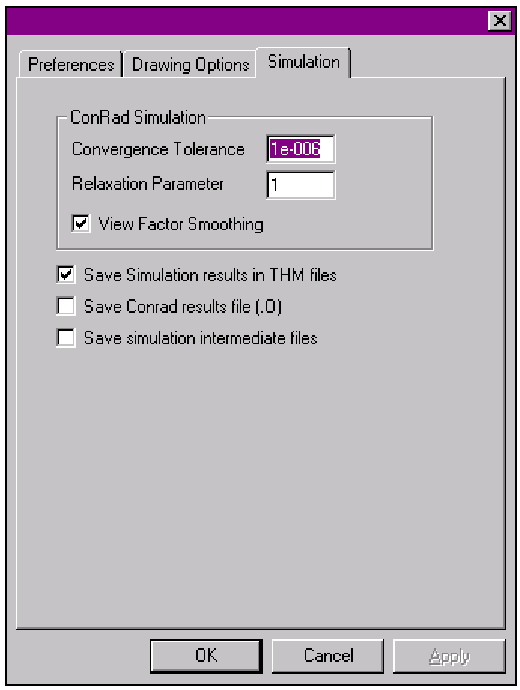


*Figure 7-7. The **Simulation** preferences accessed from the **Options/Preferences** menu*

7.3. Feedback and Troubleshooting During the Calculation Process

During the calculation process, the feedback bar will identify which stage of the automated calculation procedure THERM is working on. The stages of the calculation procedure are:

- Premeshing
- Reading Geometry
- Meshing
- Generating Input
- Simulating
- View Factors (optional)
- Simulating (optional)

- Error Estimator (optional)

This section briefly describes each stage. Also discussed are the possible errors which may occur at each stage and suggestions on what to do if they occur.

7.3.1. Premeshing

In this stage, the geometry is being prepared for meshing. THERM checks each polygon to make sure that there are no problems that the mesher cannot handle. If there is a problem with a polygon, you will get a message saying “Invalid Polygon found, Stopping Simulation,” followed by the polygon ID number. You can find this polygon by going to the **Edit/Special Select** menu choice and typing in the polygon ID number in the **ID=** field. THERM will then select the problem polygon.

The most common cause for invalid polygons is zero area extensions, shown in Figure 7-8. THERM may create zero area extensions in two ways.

(1) When very fine details are collapsed. This often happens with acute angles where one of the end points is less than 0.01 mm away from a line. If this is the case, another point will be added to the line when the boundary conditions are assigned and the two points will then be merged. The acute angle will be turned into a zero area extension and with the two points lying on top of one another.

(2) When the **Automatically adjust points within tolerance** option is chosen during the generation of boundary conditions. In this case, points within the checking tolerance (a distance of 0.1 mm) are merged automatically by the program and the possibility of creating an invalid polygon is very high. This is why it is recommended that you do not automatically adjust bad points without first inspecting them.

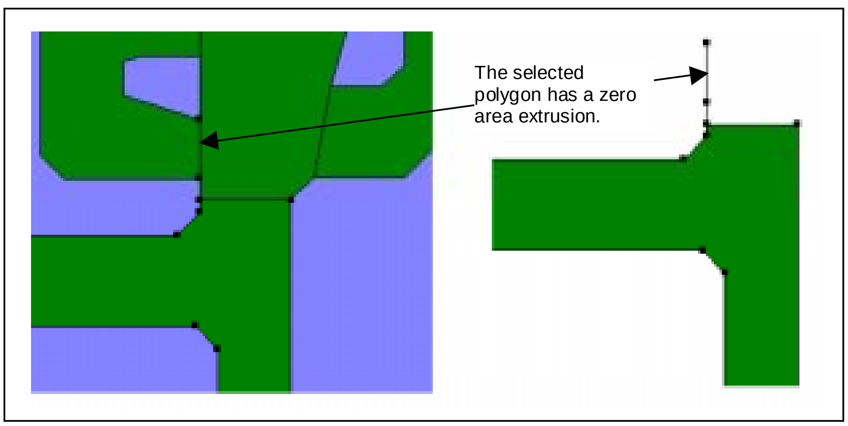


Figure 7-8. An example of a zero area extrusion which will result in an error message of "Invalid Polygon found, Stopping Simulation".

In order to correct these kinds of invalid polygons you need to delete the tip of the extension, any additional points that lie on it, and one of the two points that are lying on top of each other. Often it is easier to delete the polygon and recreate it using the void fill feature.

During the meshing stage, THERM also checks to see if U-factor surface tags have been assigned. The program was developed in large part for users interested in calculating U-factors so you will be warned if you have not selected any U-factor surface tags. (See Section 6.2.4, "Define U-factor Surface Tags" for more information). If none were selected intentionally, click on the **OK** button and the calculation will continue. This question is not asked if the models are simulated using the **Calc/Calculation Manager** menu option.

7.3.2. Reading Geometry

During this stage, the mesher is reading the input data and storing it in a database for the mesher to read. This should be a relatively short step, although occasionally a model will cause the program to hang during this step. If this step takes longer than five minutes you should close THERM using the Microsoft Windows™ Task Manager, restart the program and try to rerun the model. If you are running the program with the calculation in the background, and you forgot to save the model before calculating, you will be able to save the model before shutting down the program.

7.3.3. Meshing

In this stage, the mesh is being generated. This is the most time consuming portion of the simulation process.

Occasionally, when modeling building components with multiple cross sections such as greenhouse windows, or single cross sections with very fine detail, the mesh generator will have a problem creating a mesh. In this case you will get a message saying “Mesh Generation Error”.

If you get this error message, try simplifying the geometry near the identified point (the x and y coordinates are given in the error message) or try increasing the **Quad Tree Mesh Parameter** value in the **Options/THERM File Options** menu choice. (See Section 7.2.3, "Calculation Options" for more information on the **Quad Tree Mesh Parameter**.) The point indicated by this error will be circled in red for easy location. If the point is on the inside of a polygon, you will know which polygon is causing the problem. If the point is on an edge shared by one or more polygons, any one of these polygons could be causing the problem. If this is the case, the problem was probably caused by too many points too close together, leading to an over detailed mesh in these areas. The most common cause for this mistake is the over specification of curves (see section 5.4.1, "Suggestions for Drawing Cross Sections" for more information on modeling curves). It is likely that the problem with the geometry is near the circled area but the mesh generator could be failing at that point due to excessive detail somewhere else in the polygon. The polygon needs to be examined as a whole. Make all possible simplifications to the polygon that do not compromise the accuracy of the simulation. Usually very small modifications are all that is required.

If upon looking at the polygon there are no details you can change without compromising the model, try rerunning the model with the **Quad Tree Mesh Parameter** (in the **Options/THERM File Options** menu choice) set to a high value. This process may have to be repeated several times before the mesh can be generated. In rare cases, the mesher has problems meshing a model because of previous problems with another model. An easy way to determine if this is happening is to try to simulate a simple square and see if the mesher fails. If it fails, save your model, exit the program, restart the program and try rerunning the model. This often provides the necessary clearing of registers so the program can create a mesh.

7.3.4. Generating Input

This stage indicates that the mesher is creating the input file for the CONRAD module. This stage is fast and rarely produces an error.

7.3.5. Simulating

The finite element calculation takes place during this stage. This stage is fast and rarely produces errors. If your model does generate an error in this step please contact us.

7.3.6. View Factors (optional)

This stage indicates that the detailed radiation algorithm, VIEWER, is running.

7.3.7. Simulating (optional)

This stage will reappear if you are running the detailed radiation model. You may get an error message saying the problem is not converging. In this case, you can increase the convergence criterion found under the **Simulation** tab of the **Options/Preferences** menu choice. This criterion is a measure of the change in temperature between iterations. The maximum value for the convergence that will yield reasonable accuracy is 0.01. Do not modify this value unless you have to.

7.3.8. Preparing Input (optional)

You will see this stage if you are running a model that uses the detailed radiation model, such as a detailed radiation frame cavity, an external radiation enclosure, or the Condensation Index Model. This stage creates an input file for the VIEWER program.

7.3.9. Error Estimator (optional)

The error estimator is run during this stage. If the results from this analysis indicate that the mesh needs to be refined, the mesh will also be adapted during this stage. Then the steps of simulating and calculating view factors will be repeated until the solution meets the convergence criterion. The following warning message may appear: "The maximum number of meshing iterations reached before the error target was reached." If this happens, the last set of results will be displayed. If these results are not acceptable, you will have to increase the **Maximum Iterations** value in the **Options/THERM File Options** menu choice. You may also try performing the initial simulation at a higher value of the **Quad Tree Mesh Parameter**.

7.4. Viewing the results

When the program has finished the simulation (the calculation steps in the status bar or the hourglass have disappeared), you can view the results. There are several different types of results displays:

- U-Factors
- Finite-Element Mesh
- Isotherms
- Flux Vectors
- Color Infrared
- Color Flux Magnitude
- Report

Select the **Calculation/Display Options** menu choice to open the **Results Display Options** dialog box, shown in the figure below, which controls how the results are displayed.

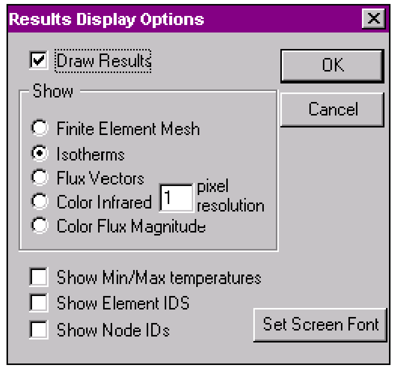


*Figure 7-9. The **Results Display Options** dialog box allows you to choose the results that are displayed.*

The following choices are available in the **Results Display Options** dialog box:

Draw Results — This box must be checked in order for the graphic results to be drawn. **Default:** checked.

Show — The **Show** section has the following choices, only one of which can be active at a time. The setting will determine what graphic results are drawn by the program when you click the **OK** button, as long as the **Draw Results** option is checked. These choices are discussed in detail later in Section 7.4.5, "Graphic Results."

- *Finite Element Mesh*
- *Isotherms*
- *Flux Vectors*
- *Color Infrared:* Drawing these results may take several minutes. You can press the **Esc** key if you want to cancel the drawing of these results.
  - *Pixel resolution:* this value controls how finely the color infrared is drawn. If you choose one pixel, THERM draws the image one pixel at a time, which can be slow but gives the smoothest image. If you choose a higher number, the program draws in blocks of pixels; the number you have chosen determines how many pixels make up the sides of the block. The image is drawn faster than using the one-pixel setting but does not appear as smooth.
- *Color Flux Magnitude:* Drawing these results may take several minutes. You can press the **Esc** key if you want to cancel the drawing of these results.

Show Min/Max temperatures — Check this box in order to display the minimum and maximum temperatures in the simulation. This is discussed in more detail in the *Min/Max Temperature* section.

Show Element Ids — Check this box to see the numerical ID values that have been assigned to each mesh element. This is primarily a research feature. **Default**: unchecked.

Show Node Ids — Check this box to see the numerical ID values that have been assigned to each mesh node. This is primarily a research feature. **Default**: unchecked.

Set Screen Font — Click this button to access a **Font** dialog box where you can select a font for the THERM graphic display from the list of fonts that are installed on your computer. This option can be useful if, for example, the labels of the isotherms do not display well on your monitor; changing the font may make them more readable.

7.4.1. U-factors

The U-factor results for the cross section are accessed by selecting the **Calculation/Show U-factors** menu or using the **Ctrl-F9** keyboard short cut.

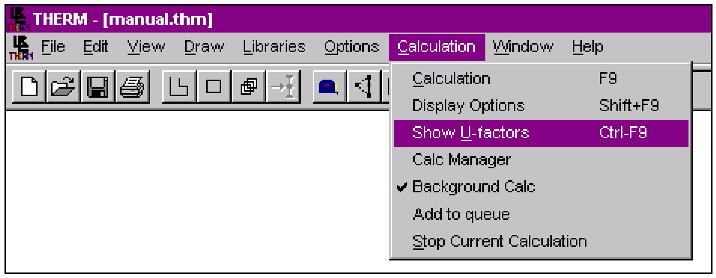


*Figure 7-10. Select the **Calculation/Show U-factors** menu to see the U-Factor results.*

The **U-factors** dialog box will appear, listing the results for the cross section. You can choose whether the U-factor is based on total surface, projected x dimension, or projected y dimension. The temperature difference, delta T, and the length used in the U-factor calculations are also displayed in this box. If the **Percent Error Energy Norm** value is **N/A**, this indicates that the simulation was run with the error estimator turned off. (See Section 7.2.3, "Calculation Options" for more information on the error estimator). If the error estimator was turned on for the simulation, the value of the **Percent Error Energy Norm** would appear in this box. Values between 5-10% are indications of a well converged solution. *(Note that "convergence" means that further refinement of the mesh will not increase the accuracy of THERM's solution, as discussed in Section 7.2.3, "Calculation Options".)* When more than two U-factors are defined in a model, the additional labels and results appear in a pull-down list.

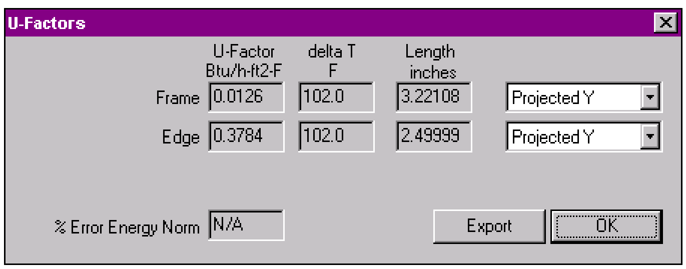


Figure 7-11. The U-factor results dialog box

Figure 7-12 illustrates the definition of Projected X and Projected Y.

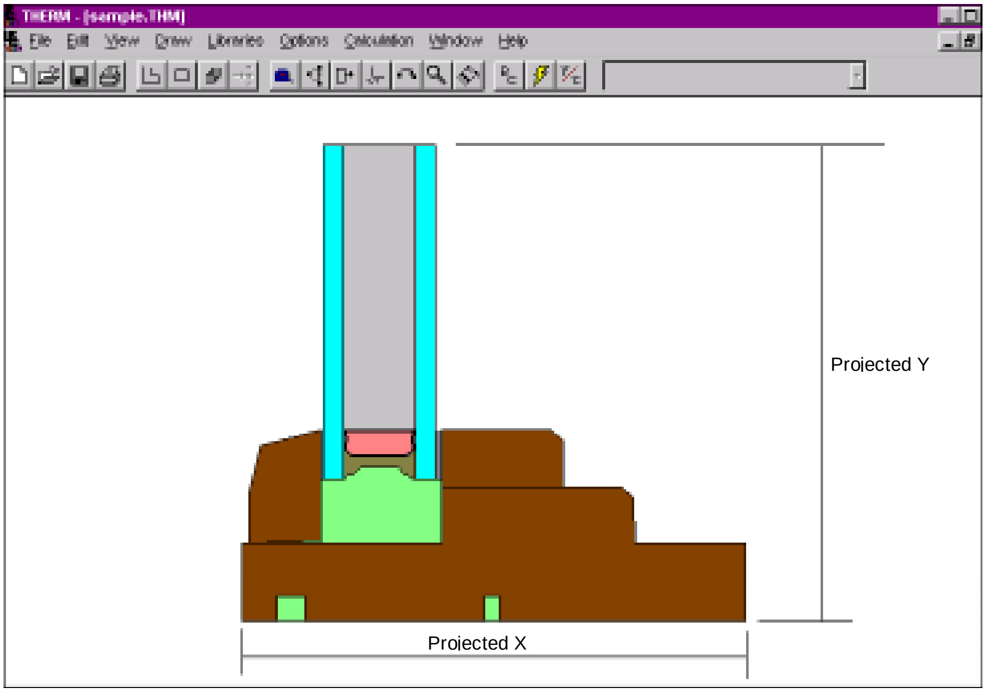


Figure 7-12. Projected X and Projected Y

To export the U-factor results to a text file, press the **Export** button, and a **Save** dialog box opens, allowing you to save the file using any file name and directory. A "txt" extension is automatically added to the file name. It is then possible to open the exported results file with any text editor (such as Notepad) or to import it into a spreadsheet.

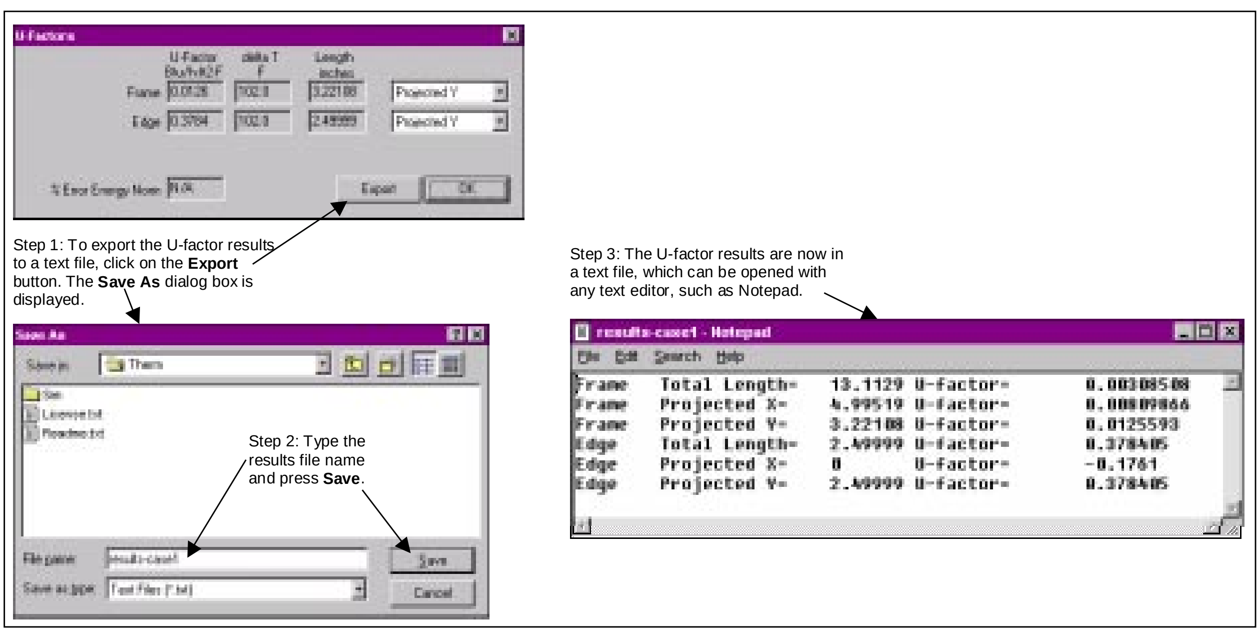


Figure 7-13. You can export the U-factor results to a text file using the ***Export*** *button on the* ***U-factors*** *dialog box.*

7.4.2. Min/Max Temperature

The maximum and minimum temperatures calculated for the cross section are displayed if the **Show Min/Max temperatures** box is checked in the **Calculation/Display Options** dialog box. When this choice is checked, the location of the temperatures is indicated by a blue "X" for the minimum and a red "X" for the maximum; a small dialog box appears, listing the temperature values and their x,y coordinates.

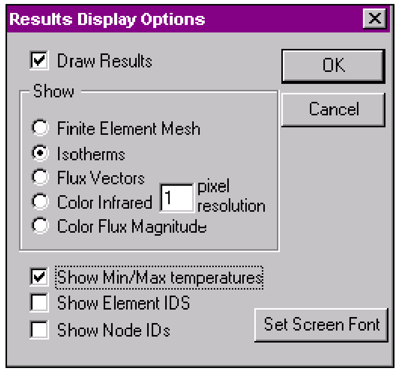


*Figure 7-14. Check the **Show Min/Max temperatures** box in **Results Display Options** dialog box.*

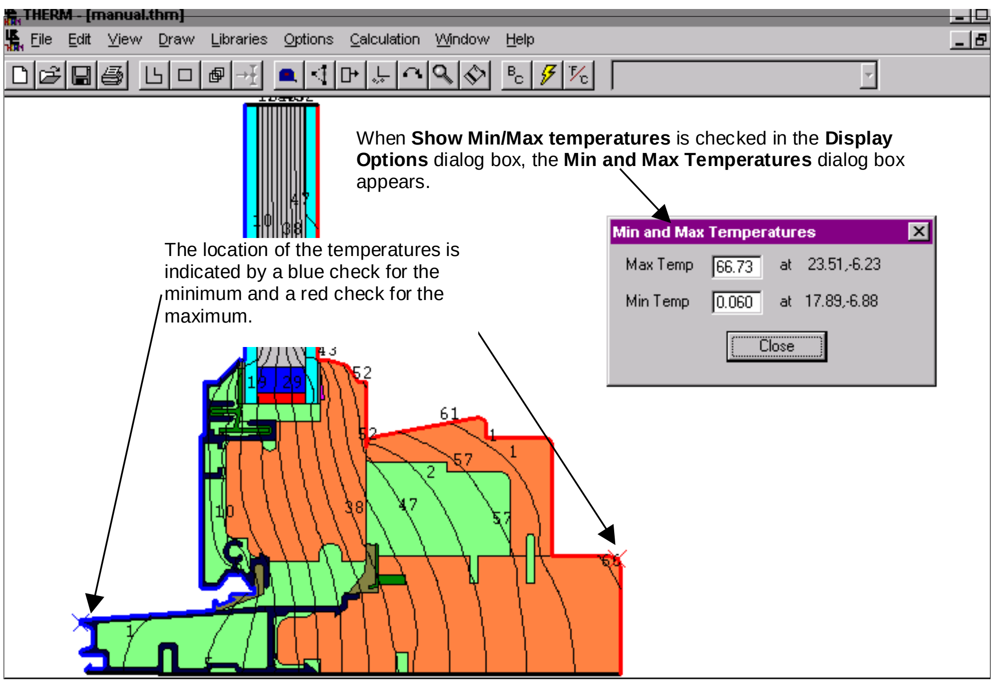


Figure 7-15. Display of minimum and maximum temperatures

7.4.3. Temperature at Cursor

When the calculations have been completed, you can view the temperature at any cursor location using the following technique:

- From the **View** menu, click the left mouse button on the **Temperature at Cursor** choice. When this option is active, a check mark will appear to the left of the choice. Clicking this choice will toggle it between active and inactive.

- A small **Temperature** dialog box will appear in the drawing area, and it will display the temperature value of the nearest mesh node point at any cursor location, with no interpolation. It may be helpful to use this feature with the Finite Element mesh displayed. The temperature is updated every time the cursor is moved. The value will be **N/A** for any cursor position outside the cross section.

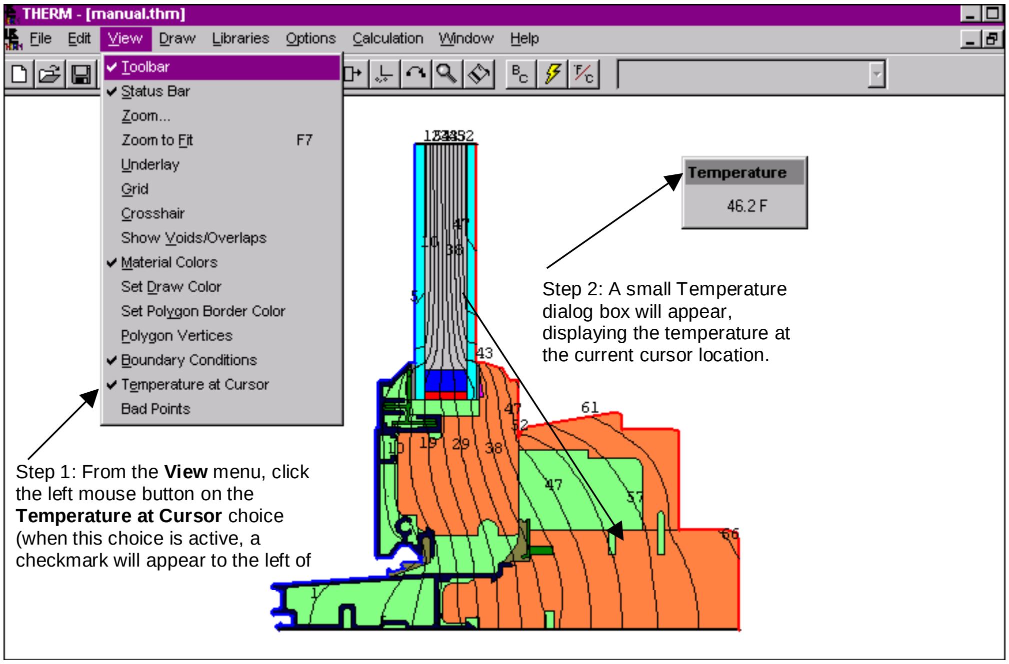


Figure 7-16. Display of the temperature at the cursor location

- If you want to know the temperature at a specific location in the cross section, you can add that point to the appropriate polygon (see Section 5.6.6, "Adding a Point"), which will force the mesher to put a node there. Then you can use the Temperature at Cursor feature to view the temperature at the point after you have calculated the results.

7.4.4. Average Temperature with Tape Measure

After the calculation has been completed, you can obtain the average temperature along the line between two points in the cross section using the **Tape Measure Average Temperature** feature of THERM.

The temperature of the sides of the equivalent rectangularized cavity are required inputs for the NFRC and CEN method for calculating effective conductivity in frame cavities. Standard default values are provided for these temperatures. You can use the average temperature feature after the model is run to check the applicability of these default temperatures to the specific cross section.

To turn on this feature, select the **Options/Preferences** menu and the **Drawing Options** tab; check the **Tape Measure Average Temperature** box.

If the **Tape Measure Average Temperature** is checked, the program will report the temperature with the **Tape Measure** toolbar button.

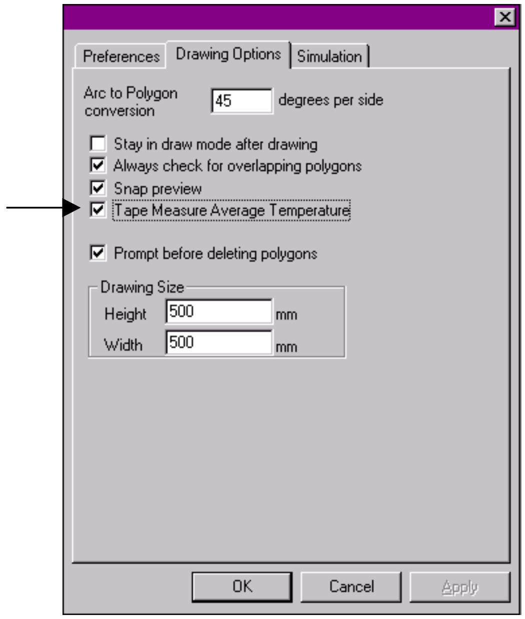


Figure 7-17. Turn on the ***Tape Measure Average Temperature*** *feature from the* ***Options/Preferences*** *menu in order to display the temperature at a specific point.*

With this feature turned on, click on the **Tape Measure** toolbar button, click the left mouse button on the starting point where you want to measure the average temperature, move the cursor to the ending point, click the left mouse button again, and a dialog box will display the average temperature between the two points. Note that the cross section results need to be calculated before the program can display these temperatures. Based on this temperature, using the simplified frame cavity model, you can edit the **Side 1** and **Side 2** temperature values in the **Properties for Selected Polygons** dialog box and then recalculate the results of the entire cross section.

Note: These temperatures are changed in each frame cavity separately. Access to the temperature input is obtained by double-clicking on the frame cavity or by using the **Libraries/Set Material** menu choice or pressing **F4** after the polygon is selected.

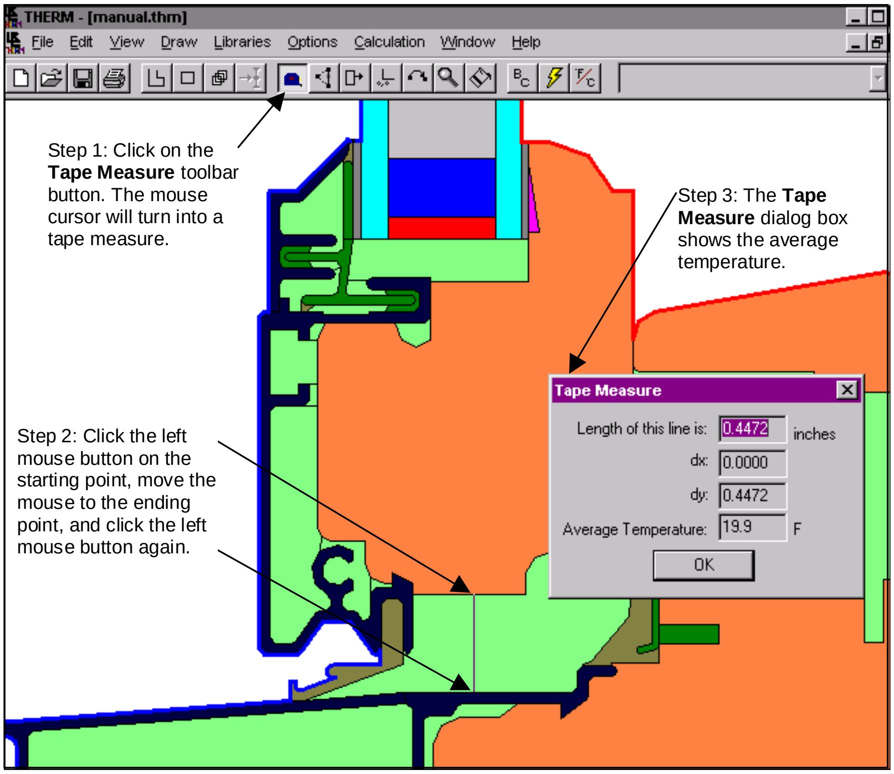


Figure 7-18. Use the Tape Measure to determine the average temperature across a particular surface.

7.4.5. Graphic Results

The graphic results can be displayed by selecting the **Calculation/Display Options** menu, which accesses the **Results Display Options** dialog box. The **Show** section offers different display options, only one of which can be active at a time. **Color Infrared** and **Color Flux Magnitude** results may take several minutes to draw. For a quick preview of the **Color Infrared** results, you can increase the value of the pixel resolution parameter.

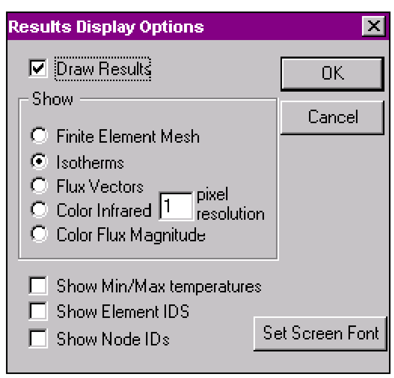


Figure 7-19. Graphic results display options

7.4.6. Finite-Element Mesh

THERM automatically generates a finite-element mesh at the beginning of its calculations. The cross section is broken into many discrete elements that are used to perform the finite-element calculation. Clicking on the **Finite Element Mesh** choice on the **Results Display Options** dialog box (you must also have the **Draw Results** box checked) causes the program to show the mesh used for the calculation. This reveals the areas in the cross section where the program has divided the building component being modeled into a very fine mesh (determined by the cross-section geometry and modified based on the percent error energy norm criterion; see Section 7.2.3, "Calculation Options" for a discussion of the calculation settings for mesh parameter and error estimator.)

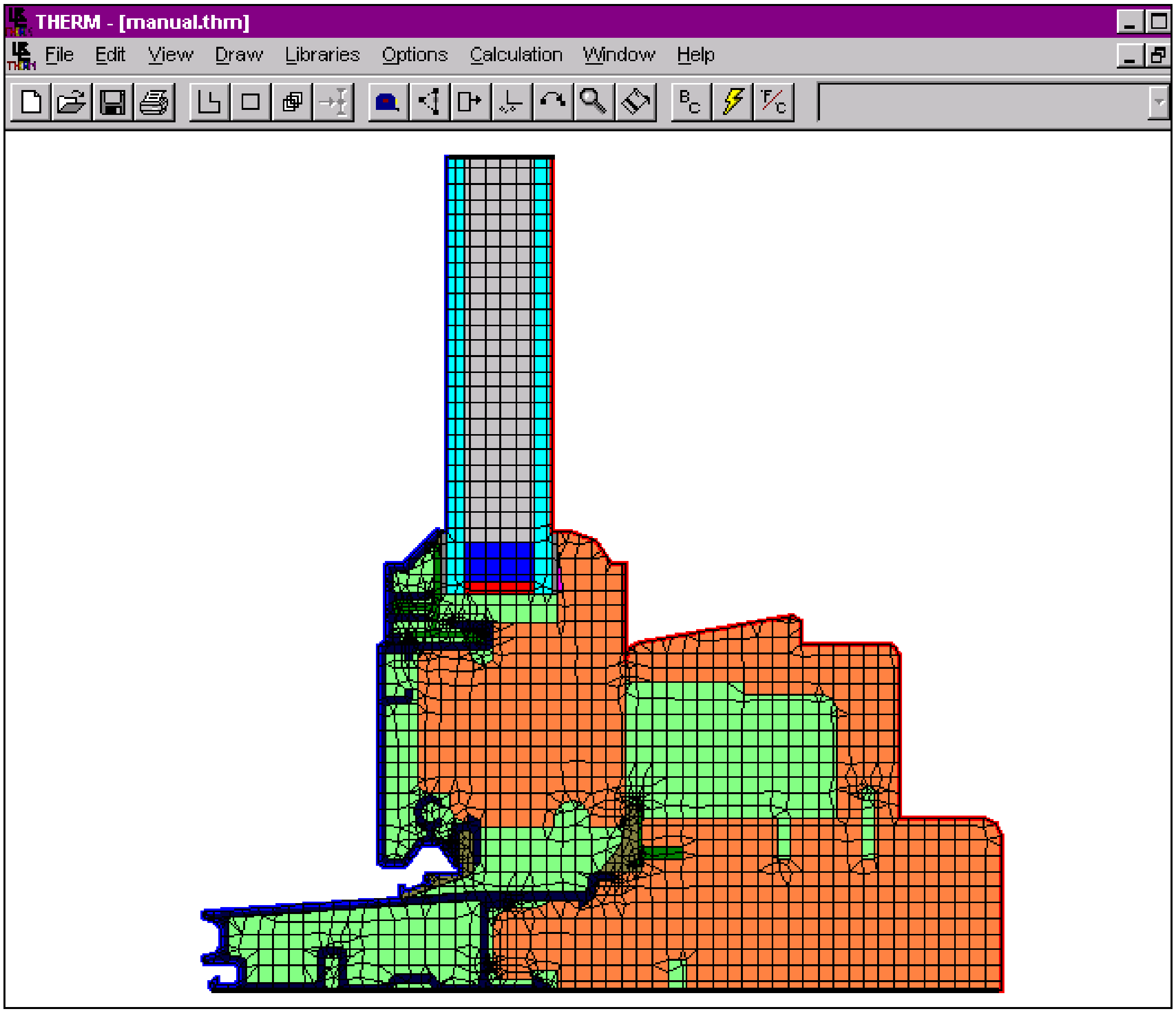


Figure 7-20. Graphic results: Finite-element mesh

7.4.7. Isotherms

When the heat-transfer analysis is complete, THERM can display the lines of isotherms through the cross section. Isotherms are useful for seeing where there are extreme temperature gradients (isotherms very close together) that may lead to thermal stress or structural problems. Isotherms are also useful for identifying hot or cold areas in the cross section, in order to predict thermal degradation or condensation.

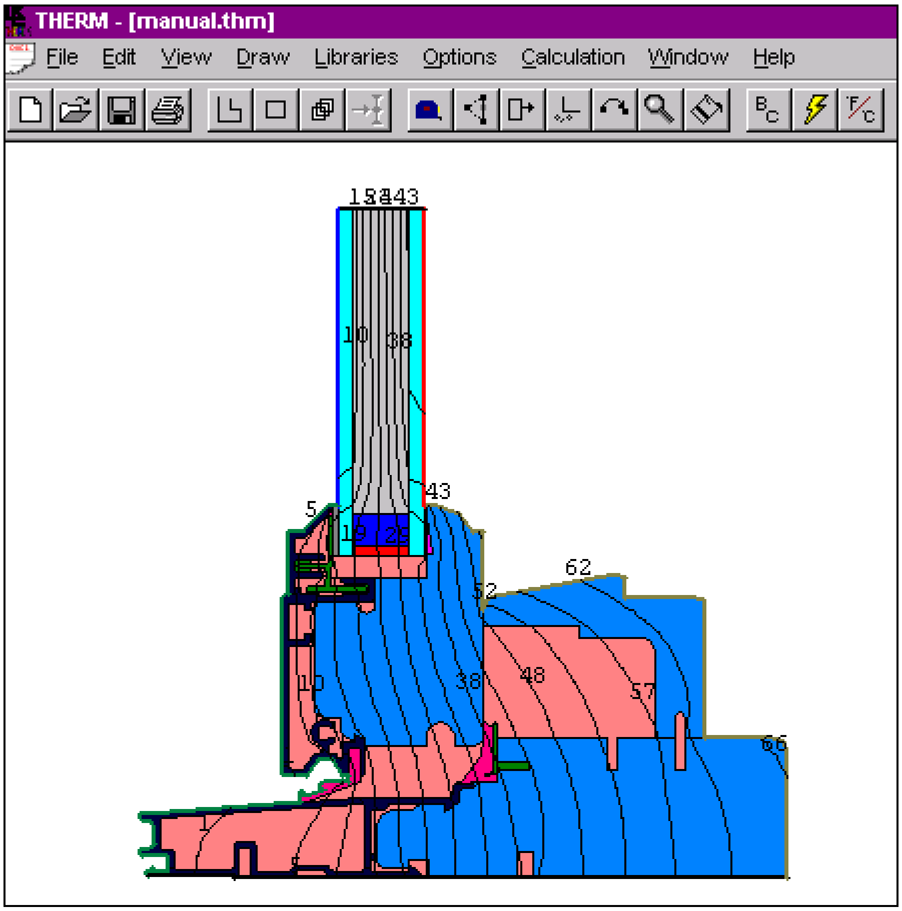


Figure 7-21. Graphic results: Isotherms

7.4.8. Flux Vectors

The flux vector results indicate the amount and direction of heat flow through the cross section. There is one flux vector for each mesh element (see the previous discussion in Section 7.4.6, "Finite Element Mesh" graphic results). The length of the vector corresponds to the amount of heat going through the element, which is a function of both the size of the element and the magnitude of the heat flux. These results can only be used for quantitative comparison with a uniform mesh. The direction of the arrow indicates the direction of heat flow. This representation of the data is most useful for determining the direction of heat flux.

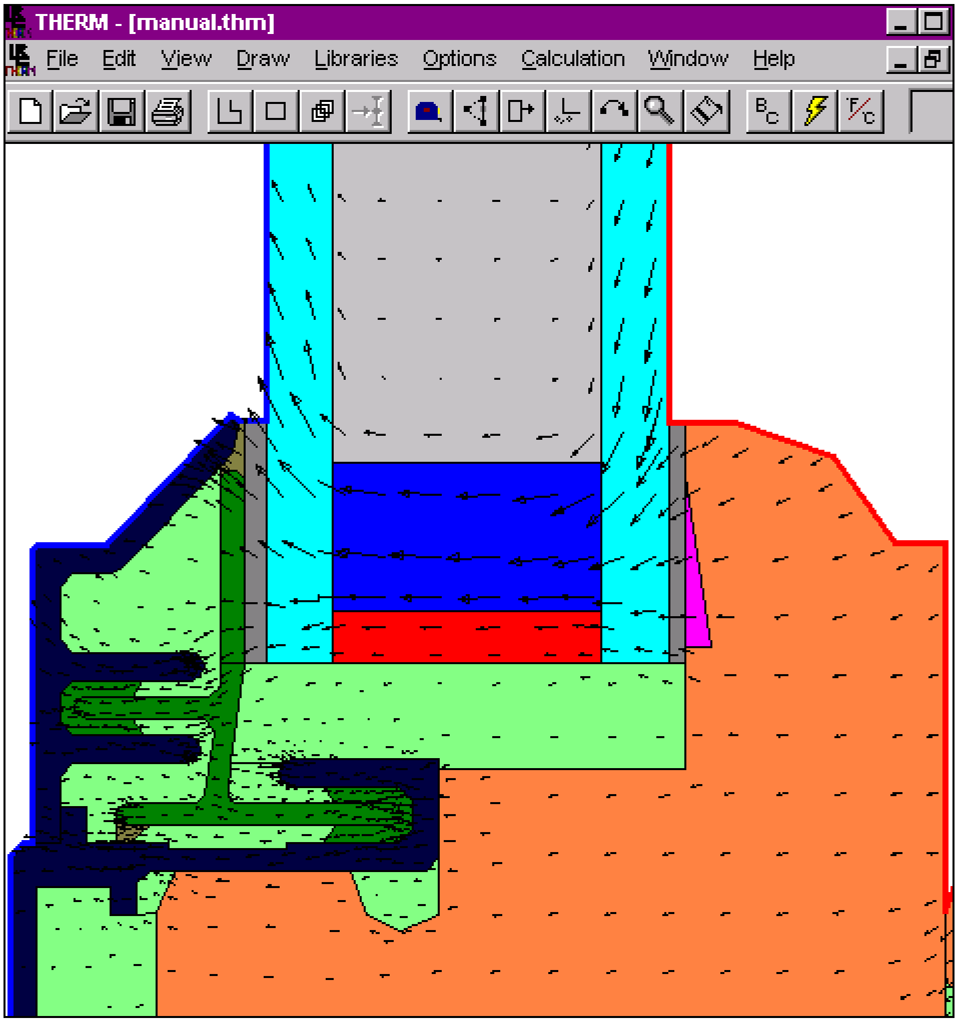


Figure 7-22. Graphic results: Flux vectors

7.4.9. Color Infrared

The color infrared results show temperature gradients in the cross section. Each temperature is represented by a different color; the cooler colors (purples and blues) are low temperatures, and warmer colors (yellows and reds) are higher temperatures.

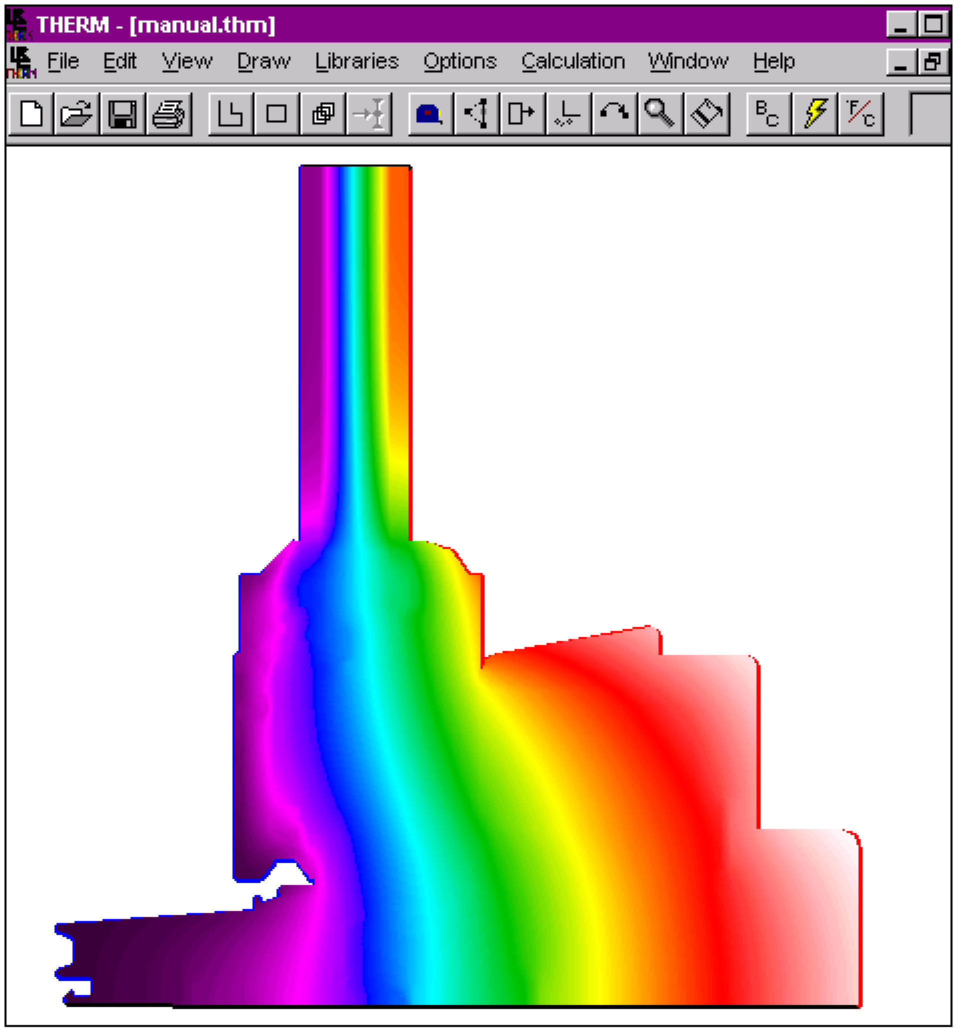


Figure 7-23. Graphic results: color infrared

7.4.10. Color Flux Magnitude

The color flux magnitude results represent the heat flux vectors, with the magnitude of the flux represented by color; the cooler colors (purples and blues) are low flux and warmer colors (yellows and reds) are higher flux. This display does not indicate the direction of the flux that is shown in the flux vector results.

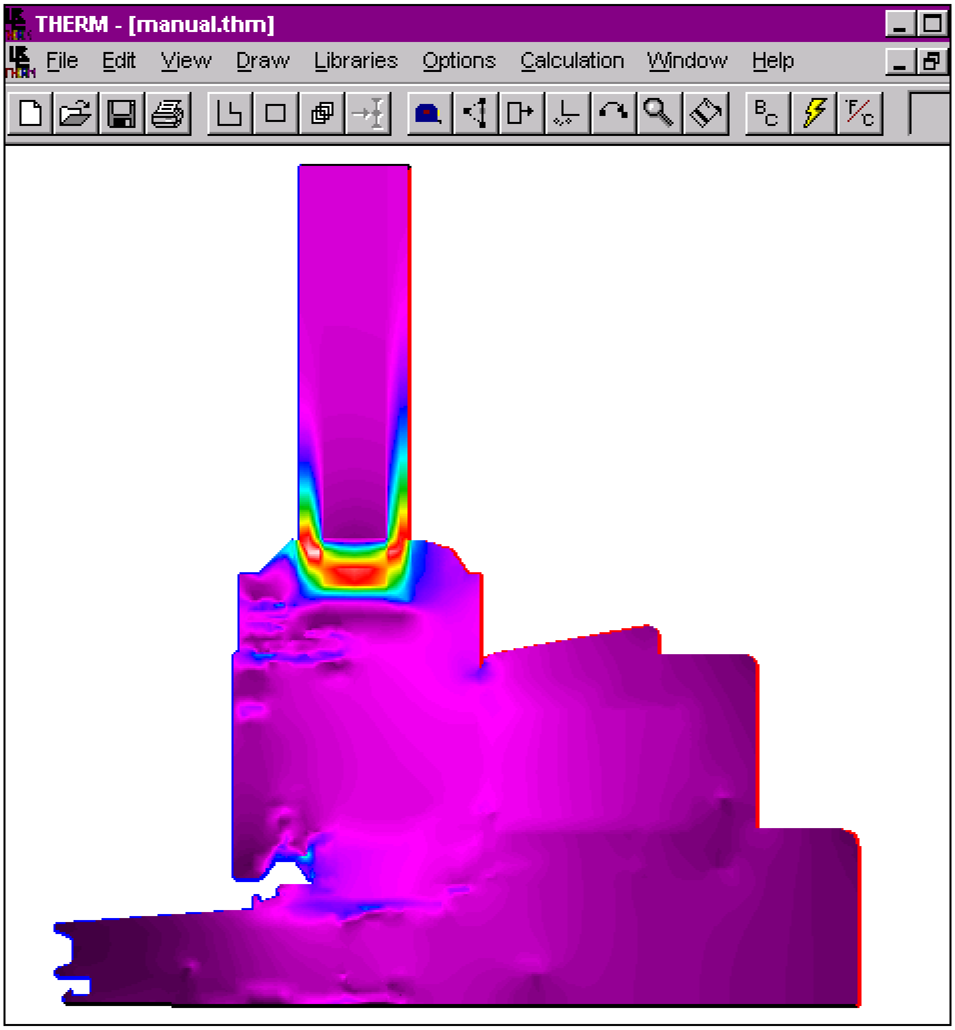


Figure 7-24. Graphic results: Color flux magnitude

7.4.11. Report

THERM generates a report each time a calculation is performed. This report contains a summary of the U-factor result as well as a description of the elements in the cross section. A sample report is shown below.

From the **File** menu, select the **Report** choice to display the report for the currently active THERM file, shown below. Click on the **Print** button to print the report; click on the **Close** button to close the report.

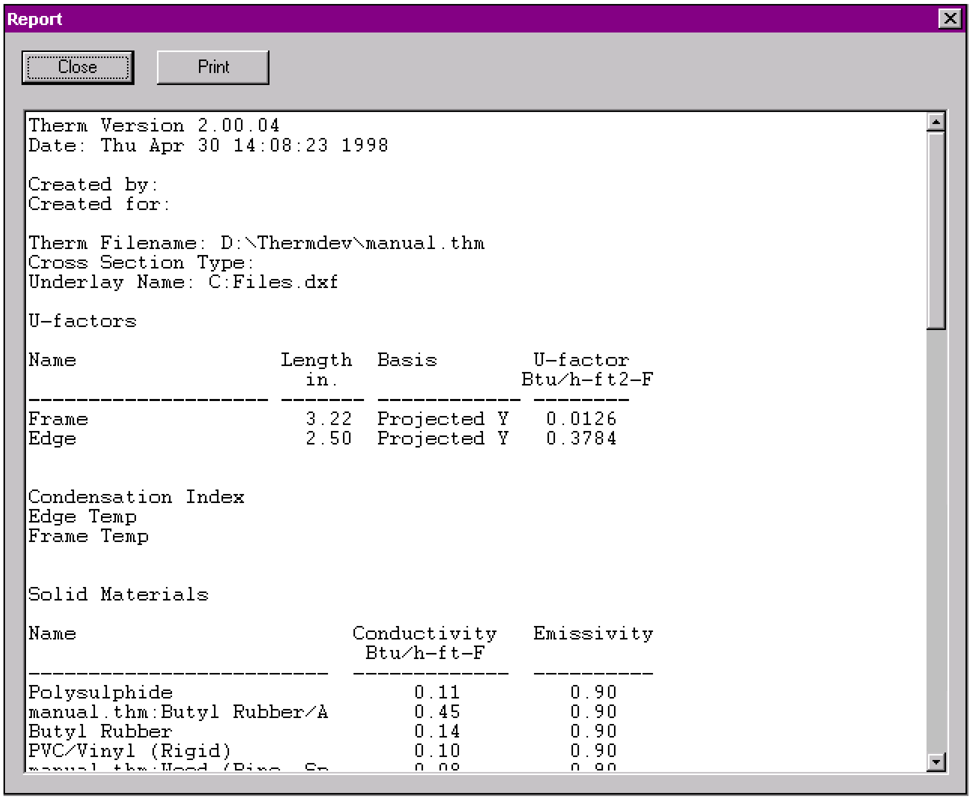


*Figure 7-25. You can display the THERM report by selecting the **File** menu and **Report** menu choice.*

The **THERM File Properties** dialog box, shown in Figure 7-26, is accessed using the **File** menu, **Properties** choice. You can input information that may be helpful in managing THERM files. In addition, some of the information in the report comes from this location. The following information can be entered into this dialog box:

Filename	The name of the THERM file. This is feedback from the program and cannot be edited.
Directory	The full path name where the file is saved. This is feedback from the program and cannot be edited.
Creation Date	The date the file was created. This is feedback from the program and cannot be edited.
Last Modified	The date the file was last changed. This is feedback from the program and cannot be edited.
Title	In this field you can give the model a name which is more descriptive than the THERM file name. For example, the serial number and product name could be included here.
Created by	In this field you can keep track of who created the model
Company	In this field you can indicate your company affiliation.
Client	In this field you can indicate who the work was for.

Cross Section Type The information in this field will be included in the file that is exported to WINDOW, indicating what type of window cross section you are modeling. The choices in this pull-down list are:

- Divider
- Head
- Jamb
- Meeting Rail
- Sill

Notes In this field you can include any other information about the model.

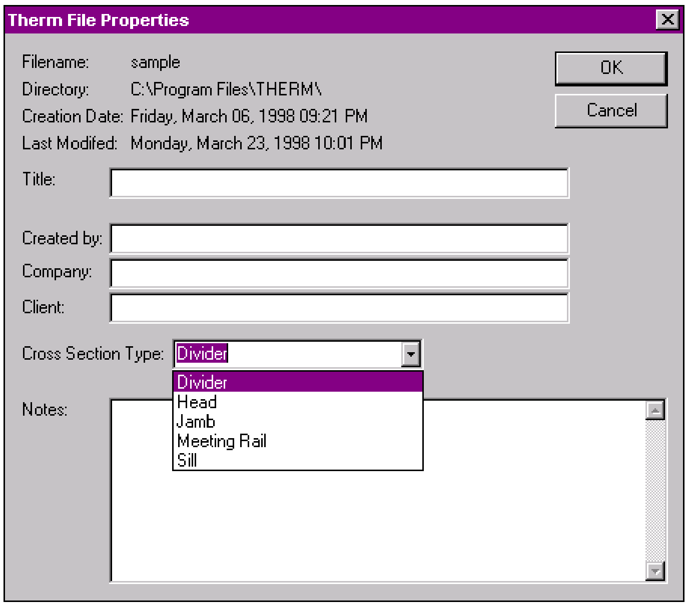


Figure 7-26. The File menu, Properties choice allows you to input information about the THERM file for the report.

Therm Version 2.00.04
Date: Thu Apr 30 14:08:23 1998

Created by:
Created for:

Therm Filename: D:\Thermdev\manual.thm
Cross Section Type:
Underlay Name: C:Files.dxf

U-factors

Name	Length in.	Basis	U-factor Btu/h-ft2-F
Frame	3.22	Projected Y	0.0126
Edge	2.50	Projected Y	0.3784

Condensation Index
Edge Temp
Frame Temp

Solid Materials

Name	Conductivity Btu/h-ft-F	Emissivity
Polysulphide	0.11	0.90
Butyl Rubber	0.14	0.90
PVC/Vinyl (Rigid)	0.10	0.90
Wood (Pine, Spruce, Fir)	0.08	0.90
Vinyl (flexible)	0.07	0.90
Fiberglass (PE Resin)	0.17	0.90
Silicone	0.21	0.90

Cavities

Name: Frame Cavity (NFRC Simplified)
Gas Fill: Air
Convection Model: NFRC
Radiation Model: Standard

Poly ID	Heat Flow Dir	Side 1 Temp F	Side 1 Emis	Side 2 Temp F	Side 2 Emis	Dimension Horz. in.	Dimension Vert. in.	Nu #	Keff Btu/h-ft-F
105	Horz	44.60	0.90	24.80	0.90	0.15	0.04	1.00	0.0179
10	Horz	44.60	0.90	24.80	0.90	0.85	0.19	1.00	0.0342
3	Horz	44.60	0.90	24.80	0.90	1.50	1.25	4.23	0.1263
4	Horz	44.60	0.90	24.80	0.90	0.09	0.50	1.00	0.0186
17	Horz	44.60	0.90	24.80	0.90	0.03	0.03	1.00	0.0155
21	Horz	44.60	0.90	24.80	0.90	0.32	0.10	1.00	0.0241
20	Horz	44.60	0.90	24.80	0.90	0.17	0.14	1.00	0.0218
100	Horz	44.60	0.90	24.80	0.90	1.79	0.58	2.42	0.0893
23	Horz	44.60	0.90	24.80	0.90	0.12	0.22	1.00	0.0203
30	Horz	44.60	0.90	24.80	0.90	0.29	0.20	1.00	0.0264
36	Horz	44.60	0.90	24.80	0.90	0.39	0.04	1.00	0.0174
28	Horz	44.60	0.90	24.80	0.90	0.20	0.07	1.00	0.0206
42	Horz	44.60	0.90	24.80	0.90	0.23	0.38	1.00	0.0257
43	Horz	44.60	0.90	24.80	0.90	0.06	0.10	1.00	0.0171
44	Horz	44.60	0.90	24.80	0.90	0.22	0.29	1.00	0.0249
61	Horz	44.60	0.90	24.80	0.90	0.33	0.06	1.00	0.0204
69	Horz	44.60	0.90	24.80	0.90	1.53	0.67	3.49	0.1043
19	Horz	44.60	0.90	24.80	0.90	0.15	0.25	1.00	0.0215
24	Horz	44.60	0.90	24.80	0.90	0.08	0.08	1.00	0.0176
40	Horz	44.60	0.90	24.80	0.90	0.16	0.13	1.00	0.0213
54	Horz	44.60	0.90	24.80	0.90	0.30	0.42	1.00	0.0287

Glazing Systems

Name	COG U-factor Btu/h-ft2-F	Overall Thickness in.	Cavity Height in.
Dbl,low-e, Ar	0.30	0.75	39.37

Standard Boundary Conditions

Name	Temperature F	Film Coefficient Btu/h-ft2-F
Exterior Surface	-0.00	5.112
Interior Wood/Vinyl	70.00	1.340
Dbl,low-e, Ar U-fact	70.00	1.353
Dbl,low-e, Ar U-fact	0.01	5.050

Calculation Specifications

Mesh Parameter: 7
Estimated Error: Not Calculated

Figure 7-27. An example of a printed THERM report

7.4.12. Export to WINDOW

THERM has a feature that allows you to export a THERM file to a WINDOW 4.1 file using the **File/Export** menu, which accesses a dialog box where you can specify the name of the new file (it is given a **t2w**, i.e., THERM to WINDOW extension) and the file format, which defaults to WINDOW 4.1. You can also specify a CSV or tab-delimited format by scrolling the **Format** pull-down list.

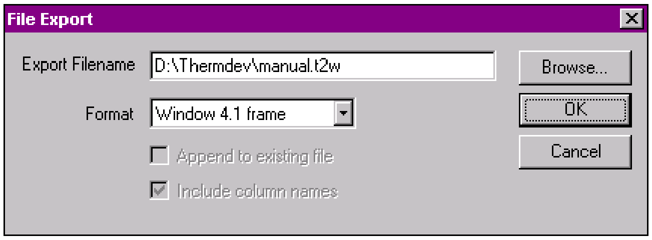


Figure 7-28. Exporting a THERM file to a WINDOW file format

You can choose to automatically save to a WINDOW 4.1 file every time you save a THERM file; use **Options/Preferences** and check the **Automatic WINDOW 4 Export on Save** box.

8. TUTORIAL

8.1. Tutorial Organization

The tutorial section of this manual is organized as follows:

8.2. Start THERM.

8.3. Draw the frame cross-section geometry using different techniques:

- 8.3.1. by using THERM's autoconvert DXF file feature,
- 8.3.2. by importing and tracing an underlay file,
- 8.3.3. by using a dimensioned drawing.

8.4. Import a glazing system from WINDOW 4.

8.5. Define the Boundary Conditions.

8.6. Run and View the Results.

8.2. Start THERM

- From the Windows 95™ or Windows NT™ **Start** button, **Programs** choice, click on THERM to activate the program.

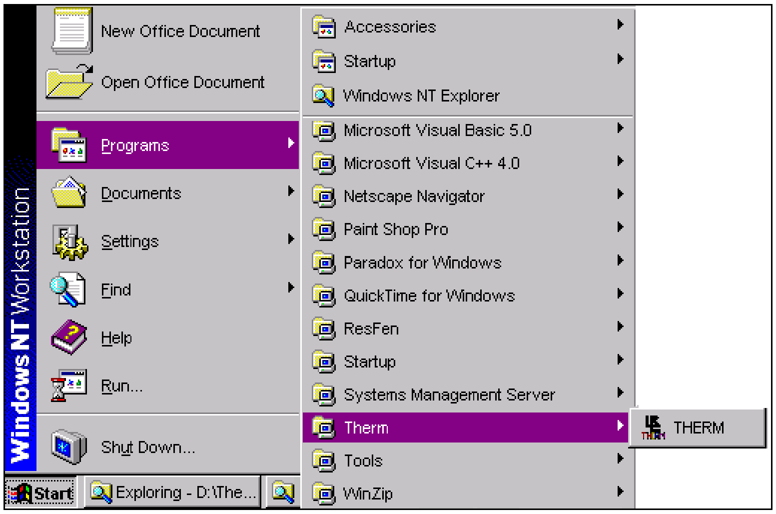


*Figure 8-1. Start THERM from the **Start** button on the Windows 95™ or Windows NT™ **Taskbar**.*

- THERM will open with a blank drawing area and no files loaded. By default, the program will assign a name of **Untitled-1** to the first drawing. When you save the cross section, you can change the name of the file.

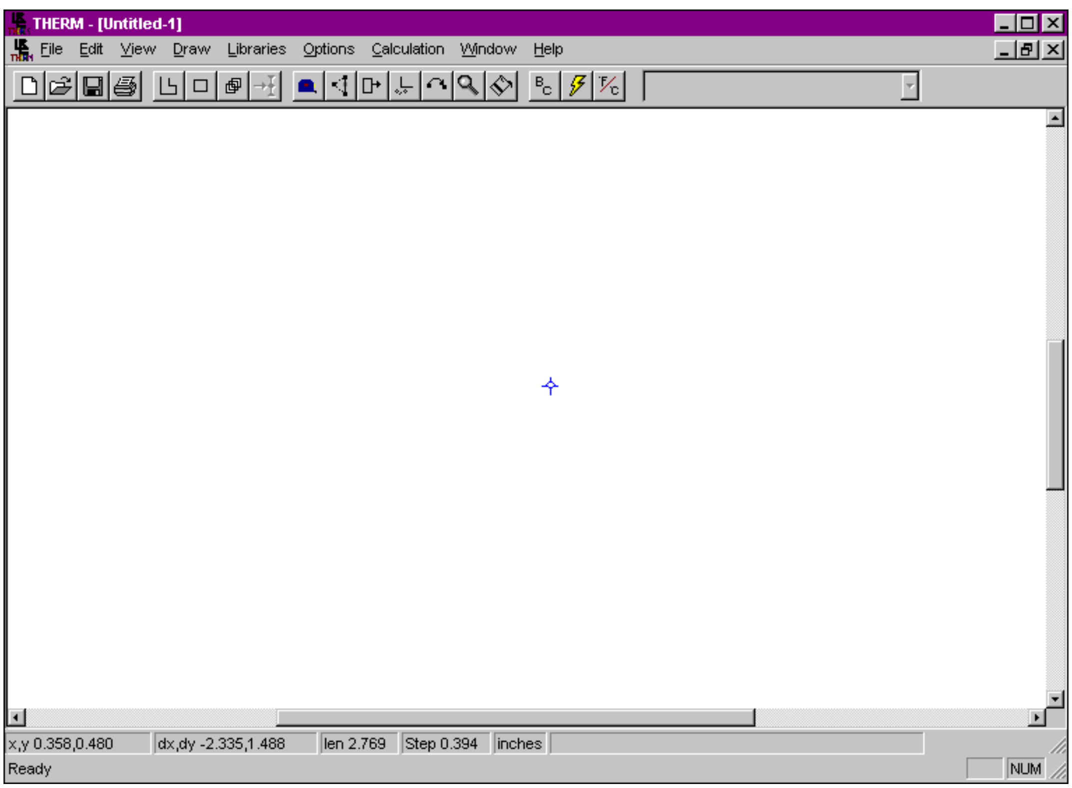


Figure 8-2. THERM starts with a blank drawing area and no files loaded.

8.3. Draw the Frame Cross-Section Geometry

You can use three methods to input a cross-section geometry. Section 8.3.1 shows how to input a cross section using the THERM underlay autoconvert feature, which automatically converts polygons from a DXF file. Section 8.3.2 shows how the same DXF underlay file can be used to trace the polygons. Section 8.3.3 shows how the same profile can be input from a dimensioned drawing.

8.3.1. Import Underlay with Autoconvert On

- From the **File** menu, select the **Underlay** menu choice.

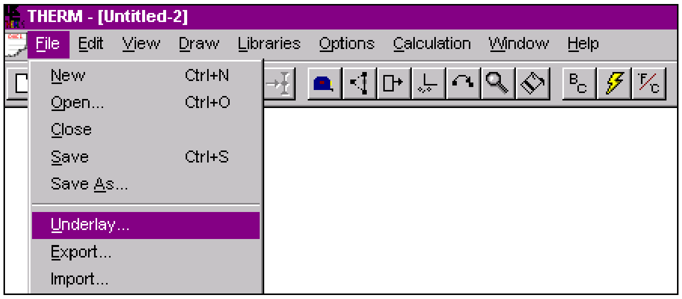


*Figure 8-3. To import an underlay, use the **File** menu, **Underlay** choice.*

- You will see the **Underlay** dialog box.

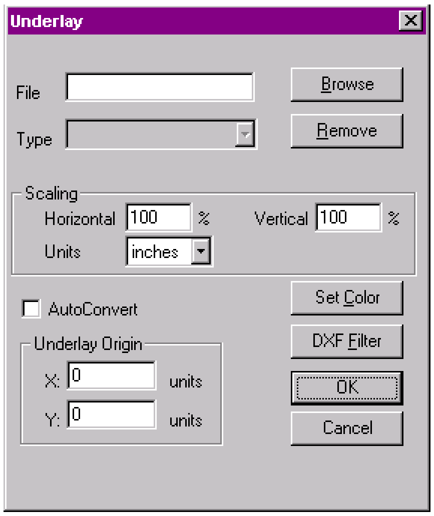


*Figure 8-4. The **Underlay** dialog box is used to import a file to use as an underlay for a THERM cross section.*

- Click on the **Browse** button.
- You will see a standard Windows™ **Open** dialog box, showing all files in your current working directory with either a BMP or DXF file extension; you can change working directories and drives if desired. You should see at least the following sample file that was installed with THERM:
  - SAMPLE.DXF
- Highlight SAMPLE.DXF by placing your cursor over it and clicking the left mouse button.

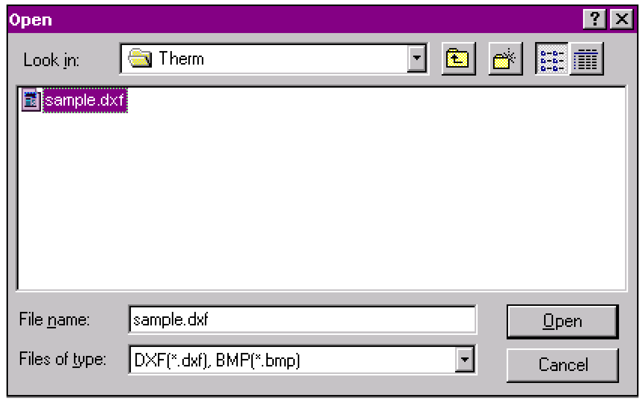


Figure 8-5. Select SAMPLE.DXF as the underlay file.

- Click on the **OK** button, and you will return to the **Underlay** dialog box.
- In the **Underlay** dialog box, click on the **Autoconvert** check box so it has a check mark, and click on the **OK** button.

Click on **AutoConvert**, so a check mark appears. This will cause THERM to automatically convert the polygons in the DXF file.

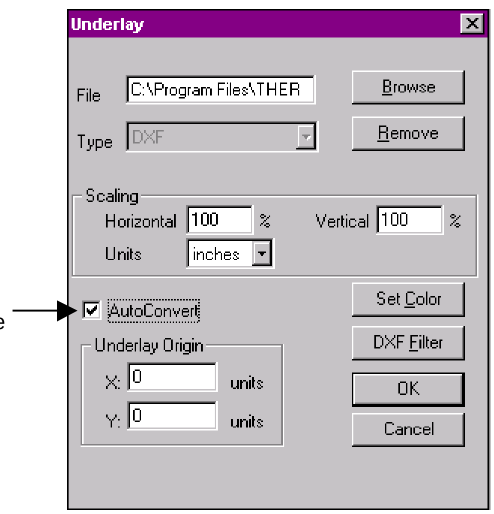


*Figure 8-6. Click the **Autoconvert** check box when importing the underlay file.*

- If the DXF file is complex and takes a few seconds to import, a small **DXF Import** dialog box may appear, showing a counter as THERM reads the DXF file.

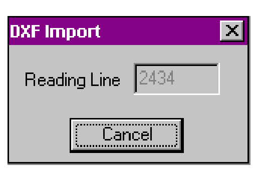


Figure 8-7. The DXF Import dialog box may display the underlay import status, depending on the complexity of the DXF file and the speed of your computer.

- When THERM has finished autoconverting the underlay file, the cross section is displayed with polygons defined for all the elements it could interpret. See Section 5.2, "Importing a DXF or Bitmap File as an Underlay" for details about defining the DXF file so that THERM can autoconvert the polygons properly.

- The DXF file will have been brought into the drawing area including all the white space in the original drawing. THERM attempts to fill the drawing area with the autoconverted cross section while still making the viewing scale as big as possible. In the figure below, you can see both the THERM origin (corresponding to the origin of the DXF file) and the cross section.

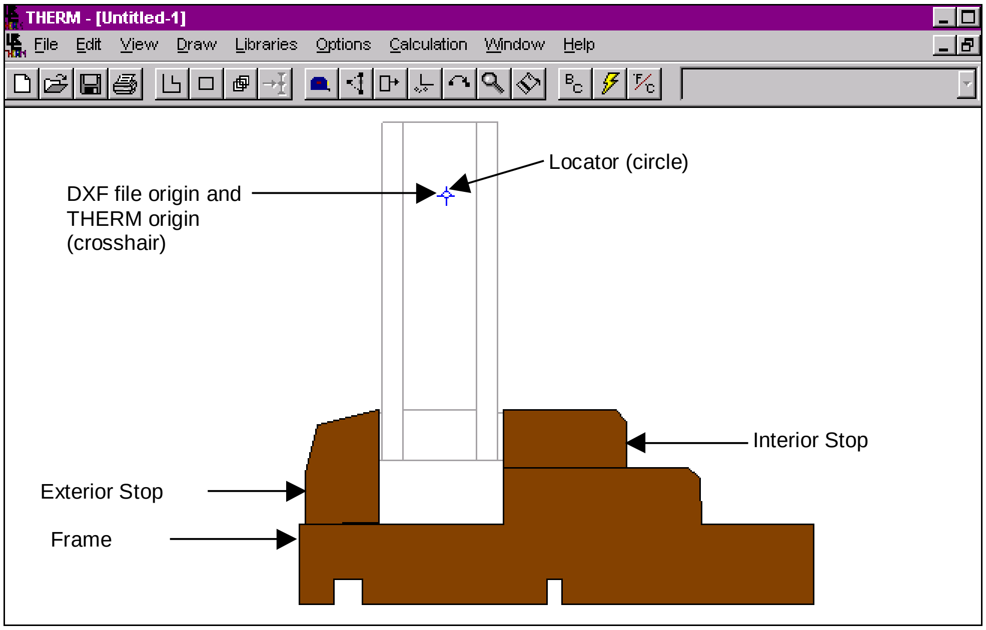


Figure 8-8. The components of the cross section

8.3.1.1. Save the File

- Save your work using the **Save File** toolbar button or the **File/Save** menu choice. Type a name that is not already being used, such as **Tutorial** (a file named Sample.thm is included in the installation, so pick a different name to avoid overwriting the sample file), and click on the **OK** button. THERM will automatically add the THM extension, so the file name will appear as TUTORIAL.THM.

8.3.1.2. Edit Cross Section

- If you want to zoom in, place the mouse cursor in the area of the underlay that you want to be in the center of the zoomed view and click the right mouse button. You only need to see the frame area of the cross section, not the glazing area. If you zoom in too far, hold the **Shift key** down and click the right mouse button to zoom back out. If you want to return to the original, centered view, hold down the **Ctrl** key and click the right mouse button.

- Because of the way this DXF file was defined, only the frame components of the underlay are converted but not the glazing system. The glazing system will be imported from WINDOW in a separate step later in the tutorial.

8.3.1.3. Assign Materials to Polygons

- Select the polygon representing the frame by placing the mouse cursor inside the polygon and clicking the left mouse button once. The vertices of the selected polygon will be displayed as squares. Select **Wood (Pine, Spruce, Fir)** from the **Materials** pull-down list on the right-hand side of the toolbar if the polygon is not already assigned that material. (See Section 5.10, "Assigning Materials After Drawing Polygons" for more information about selecting materials).

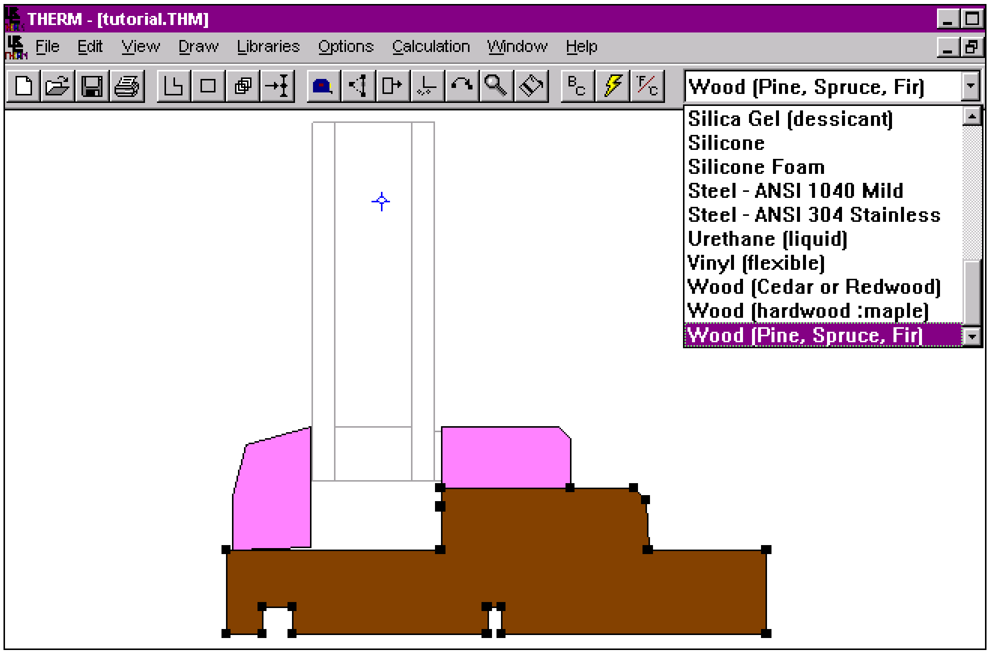


*Figure 8-9. Select the polygon representing the frame and assign it the **Wood (Pine, Spruce, Fir)** material.*

- Select the polygons representing the interior and exterior stop (you can select both of them by holding the **Shift** key down and clicking the left mouse button on each polygon), and select **Wood (Cedar or Redwood)** from the **Materials** list, as shown in Figure 8-10.

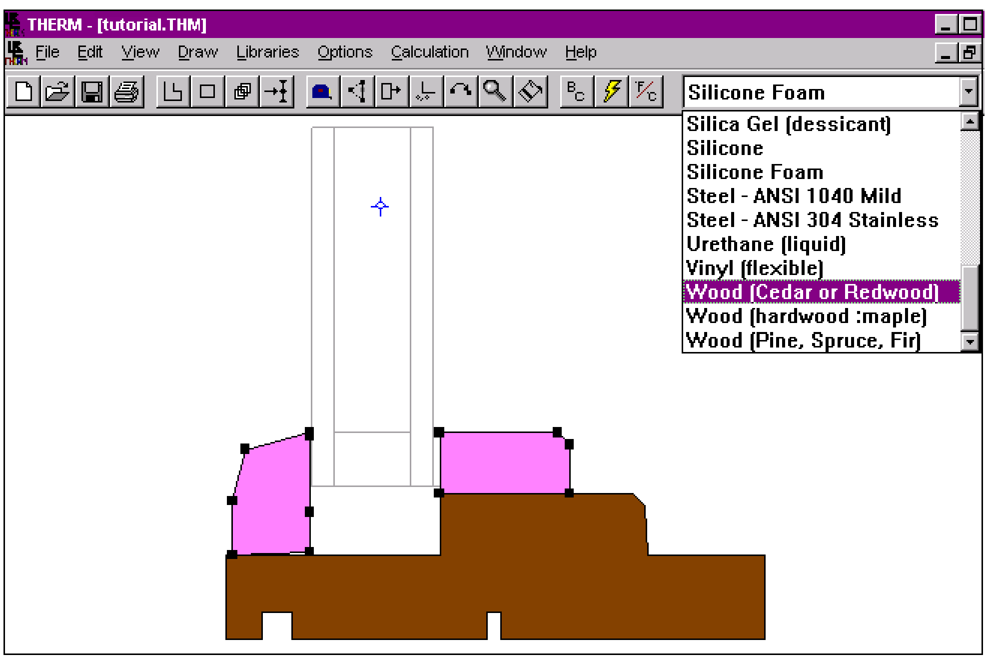


*Figure 8-10. Select multiple polygons by holding the **Shift** key down while clicking on multiple polygons.*

8.3.1.4. Save Your Work

- Save your work using the **File/Save** menu choice or the **Save** toolbar button . Remember to save your files often, because there is no autosave feature in THERM.

8.3.2. Tracing a DXF Underlay

If you don't want to use the DXF autoconvert feature, you can create the cross section polygons by tracing the underlay. This may be necessary for some DXF files that have not been created in a way that autoconvert can define the polygons properly.

- From the **File** menu, select the **Underlay** menu choice.

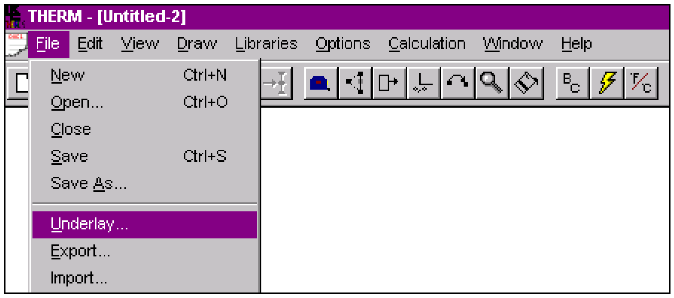


*Figure 8-11. To trace the DXF file, select the **Underlay** choice from the **File** menu.*

- You will see the **Underlay** dialog box.

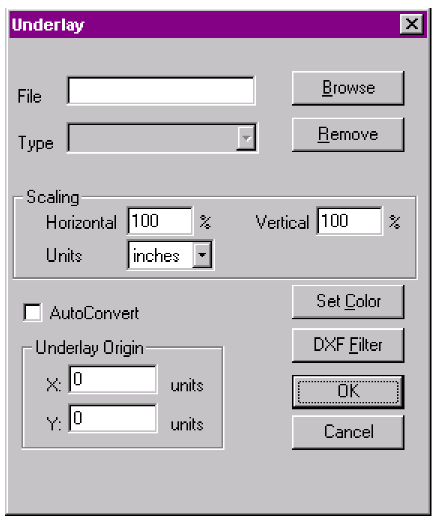


*Figure 8-12. The **Underlay** dialog box is used to select the DXF file to trace.*

- Click on the **Browse** button.
- You will see a standard Windows™ **Open** dialog box, showing all files in your current working directory with either a BMP or DXF file extension; you can change working directories and drives if desired. You should see at least the following sample file that was installed with THERM:
  - SAMPLE.DXF

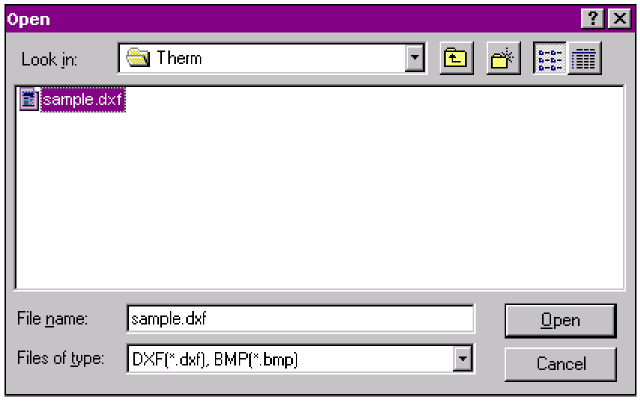


Figure 8-13. The Open dialog box shows all files with a DXF or BMP extension.

- Highlight SAMPLE.DXF by placing your mouse cursor over it and clicking the left mouse button.
- Click on the **OK** button, and you will return to the **Underlay** dialog box.
- Do not change any of the default values in the **Underlay** box; click on the **OK** button.
- You may see a small **DXF Import** dialog box, showing a counter as THERM reads the DXF file.

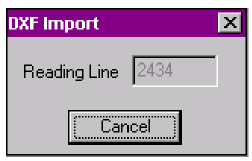


Figure 8-14. A DXF Import file may appear if the file is complex and takes a few seconds to import.

- When THERM has finished reading the DXF file, it will display the underlay cross section in gray, indicating that the lines are not true polygons but can be traced.

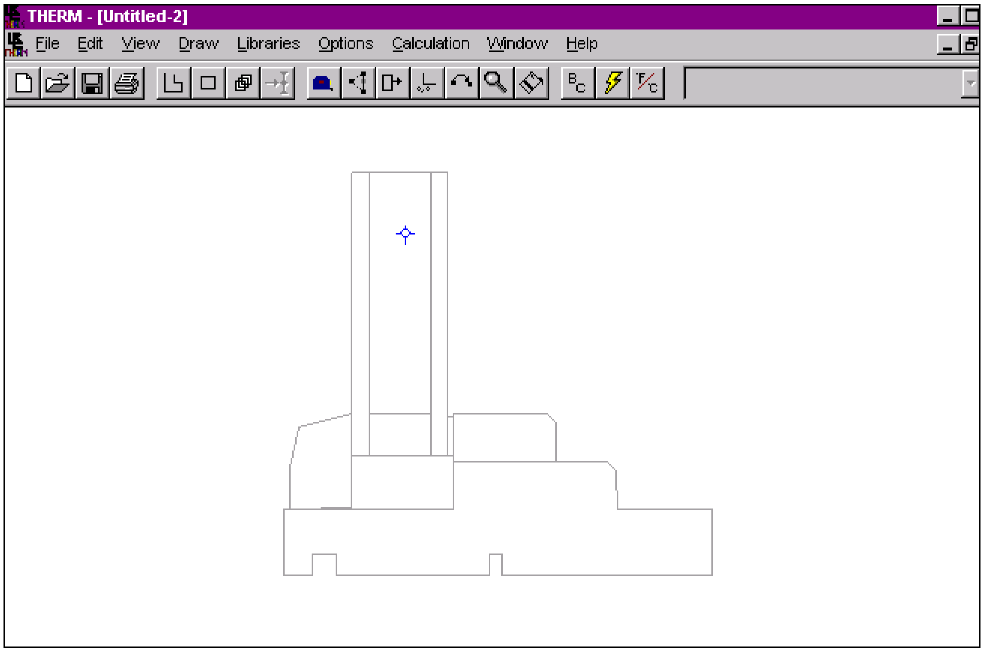
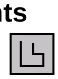


Figure 8-15. The DXF file has been imported as an underlay, which can now be traced to make the individual polygons.

- THERM attempts to fit the DXF cross section in the drawing area while still making the scale as big as possible. In Figure 8-15, you can see both the THERM origin (cross hairs corresponding to the origin of the DXF file) and the cross section.
- Before you start tracing the underlay, you may need to zoom in on the cross section in order for the cursor to snap to the correct underlay vertices. To zoom in, place your mouse cursor over the area of the cross section that you want centered in the new zoomed view and click the right mouse button. Each click of the right mouse button doubles the zoom percentage.
- You can use the horizontal and vertical scroll bars on the drawing area to move the underlay to the desired location in the drawing area. You only need to see the frame area of the cross section, not the glazing system.

8.3.2.1. Draw the Cross-Section Components

- Click on the **Draw Polygon** toolbar button  or use the **Draw/Polygon** menu choice to begin drawing.
- Click on the **Repeat Mode** toolbar button  or use the **Draw/Repeat Mode** menu choice, so you don't have to press the **Draw Polygon** button each time you finish one polygon and start to draw the next.

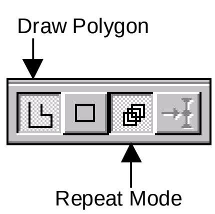


*Figure 8-16. Push the **Draw Polygon** and **Repeat Mode** toolbar buttons.*

- Start with the exterior stop on the outside of the glazing system. Because you are in **Draw Polygon** mode, the **Materials** pull-down list on the right side of the toolbar is active, and you can select the **Wood (Cedar or Redwood)** material for the exterior stop. (Alternatively, you could access the **Libraries** menu, select the **Material Library** or use the **Shift-F4** keyboard shortcut, and pick **Wood (Cedar or Redwood)** from the materials pull-down list). The material you select will be used by THERM when you start drawing the first polygon.

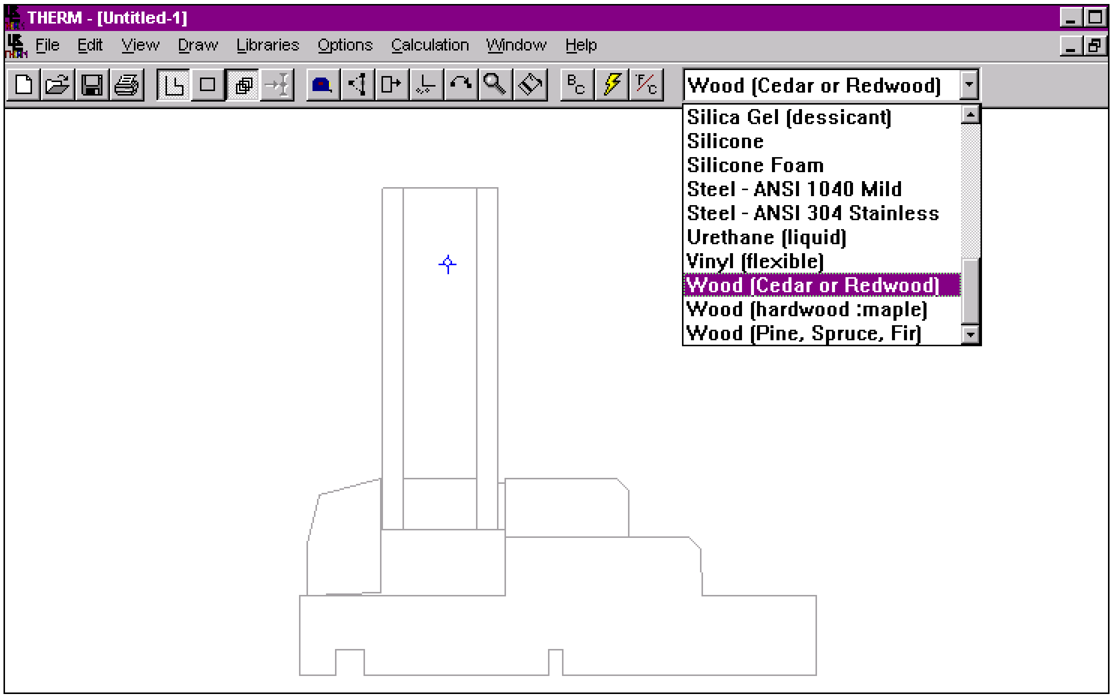


Figure 8-17. Select the appropriate material from the Materials pull-down list (on the right side of the toolbar) for the first polygon to be drawn.

- With the material selected, you are ready to start drawing the polygon for the exterior stop. Place the drawing cursor on the upper point of the stop nearest the glass and click the left mouse button. This sets the starting point of the polygon, as shown in Figure 8-18.
- Move the mouse to the left and down to the next point on the polygon, click the left mouse button, and a line will appear between the two points, shown in Figure 8-18. Because you are tracing a DXF file, THERM recognizes the vertices of the polygons;, therefore you don't have to be exactly on a point but only to be within "sticky" distance (defined as half of one side of the drawing cursor) of the point.

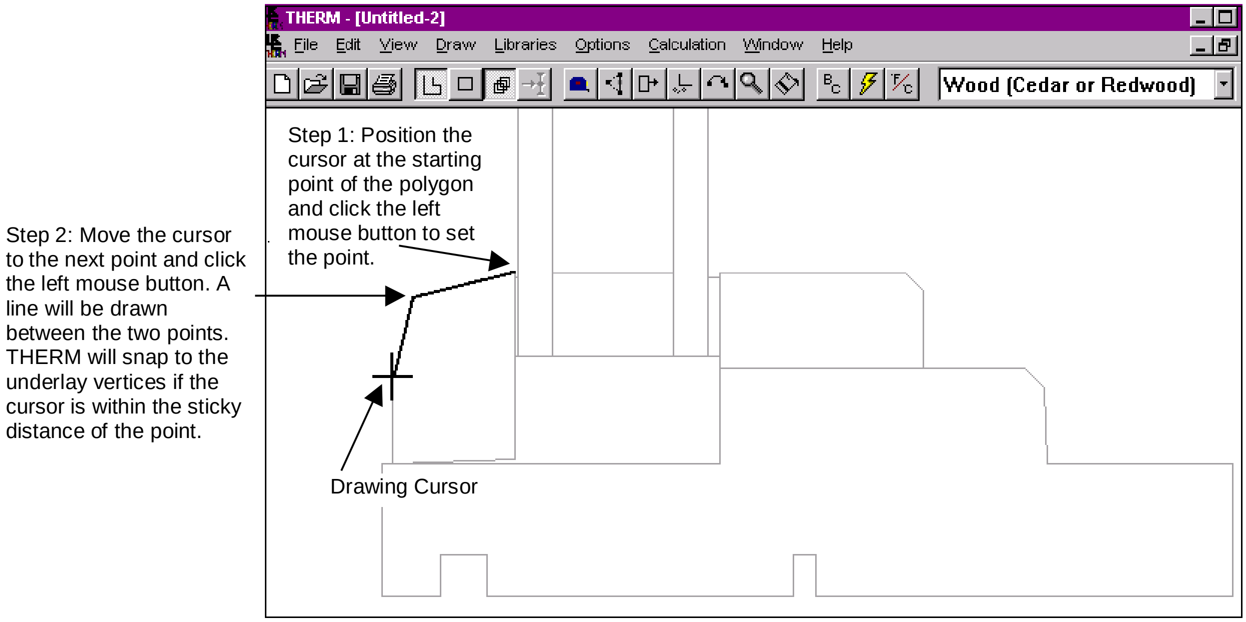


Figure 8-18. Start drawing the first polygon.

- Move the mouse to the next point in a counterclockwise direction (moving in a counterclockwise direction is not required, but is recommended) and click the left mouse button when you have reached the next point. Continue around the component until you only have one more line to draw. To draw the last line and close the polygon, you can either press **C** (to close) or move the mouse to the first point and click the left mouse button. Either of these actions will cause THERM to complete the polygon, as shown in Figure 8-19, and fill it in with the color corresponding to the material definition (or to a shade of gray on monochrome monitors).

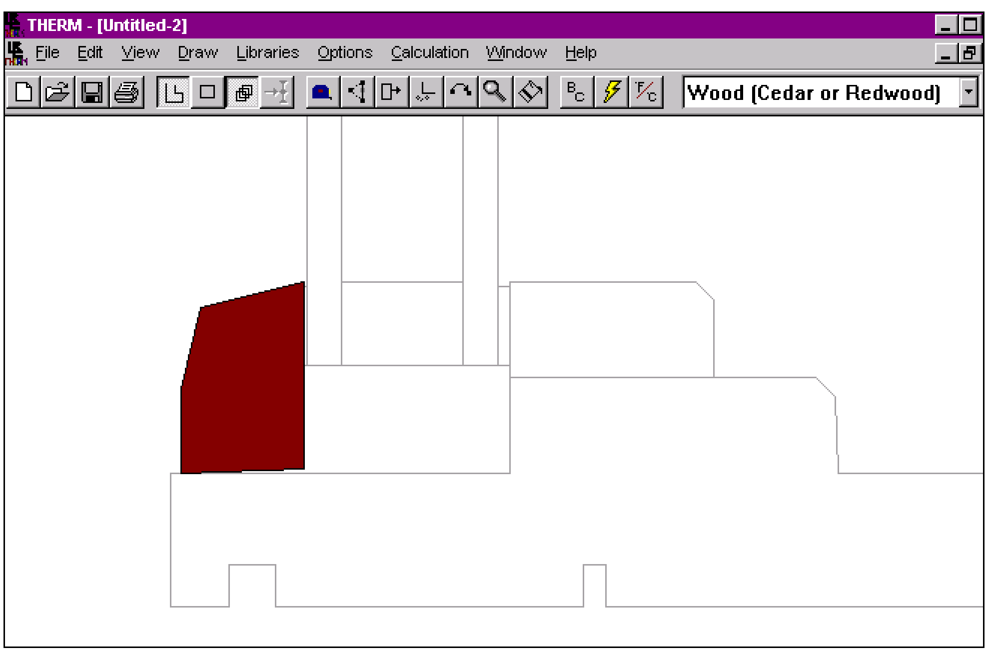


Figure 8-19. The first polygon, representing the exterior stop, has been drawn.

- By default, the **Snap to Underlay** setting is enabled, so the drawing cursor will snap to vertices on the underlay. If you find the cursor snapping to points you don't want, zoom in on the drawing by clicking the right mouse button. **Snap to Underlay** will snap within the distance of half of one side of the cursor, so you don't have to be exactly on a point for the snap to occur.
- Next draw the frame in which the glazing sits using the same method above; after that, draw the inside stop, as shown in Figure 8-20.
- If you find that a component you are tracing is too large (and therefore is out of the drawing area) or too small (the cursor is snapping to the wrong points), you can use the Zoom feature to zoom in and out even when you are in **Draw Polygon** mode. For example, if you are drawing a line that ends outside the THERM drawing area, hold the **Shift** key down and click the right mouse button. This will cause the drawing view to be zoomed out so that you can probably see the entire component you are tracing. Then you can continue to draw the line. When you have finished that line, you may want to zoom back in to get more detail by clicking the right mouse button.

8.3.2.2. Assign Different Materials to Polygons

- The frame components of the cross section can each be assigned a different material. To assign a different material to a polygon, highlight it by placing the mouse cursor inside the polygon and clicking the left mouse button once. The vertices of the selected polygon will be displayed as squares. Select, for example, **Wood (Pine, Spruce, Fir)** from the **Materials** pull-down list on the right-hand side of the toolbar.

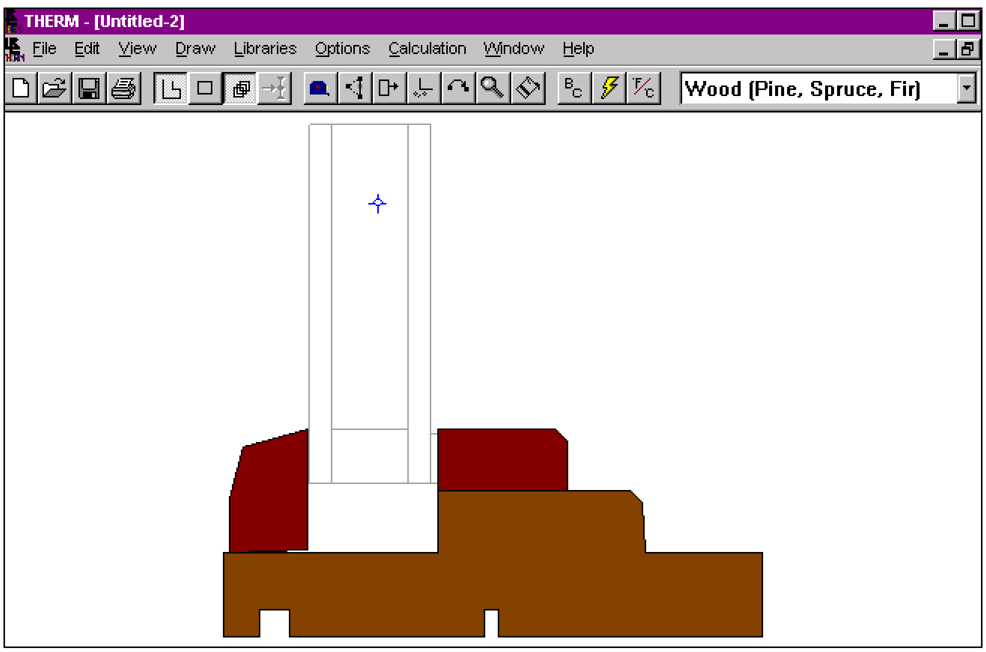


Figure 8-20. The wood components have been defined.

- Now you have three polygons representing the three main components of the wood frame cross section.
- Press both the **Polygon** and the **Repeat Mode** toolbar buttons to turn off the drawing mode, or use the **Draw** menu and uncheck (by clicking on them with your left mouse button) **Polygon** and **Repeat Mode** (they are "on" when there is a check mark to the left; this selection toggles on or off each time you click on the menu choice).

8.3.2.3. Save the File

- Save your work using the **Save File** toolbar button or the **File/Save** menu choice. Type a name that is not already being used, such as **Tutorial2** (a file named Sample.thm is included in the installation, so pick a different name to avoid overwriting that file), and click on the **OK** button. THERM will automatically add the THM extension, in this case creating the file name TUTORIAL2.THM.

8.3.3. Using a Dimensioned Drawing

This section explains how to use THERM to draw from a dimensioned drawing. You will learn how to:

- Start a drawing from scratch,
- Use the keyboard and mouse to navigate and draw,
- Use keyboard short cuts.

8.3.3.1. The Dimensioned Drawing

- You will be drawing the frame section of the following dimensioned drawing, using a combination of the keyboard and the mouse.
- *Note: be sure you are in °F mode (IP) for this exercise. If you are in °C (SI) units, set the units to °F using the "switch units" button on the menu bar or select it from the Options menu.*

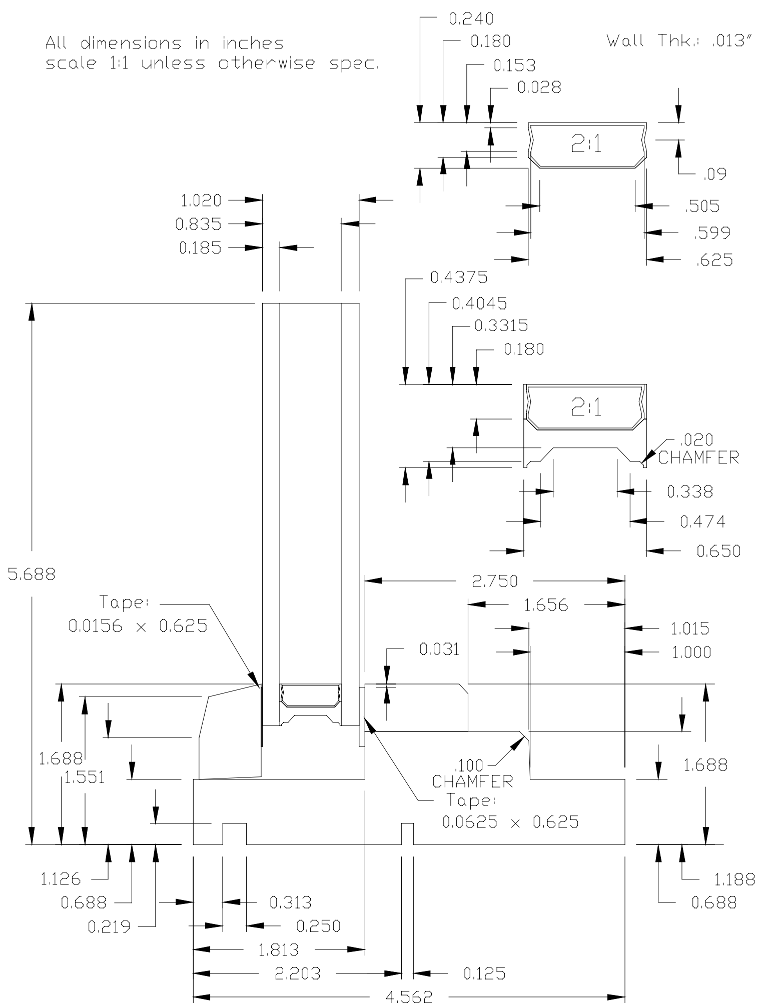


Figure 8-21. Dimensioned drawing of cross section

8.3.3.2. Keyboard Short cuts

The following keys will be used in this exercise:

- **F2**: Turns Draw Polygon on and off
- **Shift+F7:** Sets the origin
- **Home:** Centers the cursor over the origin cross hair
- **Space bar:** Snaps the drawing cursor to the nearest point, to ensure that the cursor is *exactly* on the point. This can be used when the sticky distance, which also causes the cursor to snap exactly on a point, doesn't apply.
- **Esc:** Cancels drawing mode (and other modes)
- **Any Number:** Enters the step size for cursor / line movement

8.3.3.3. Positioning the Drawing

- With your mouse, you can use the horizontal and vertical scroll bars on the drawing area to position the origin (represented by cross hairs) in the lower right corner of the drawing, so you can see the entire drawing without scrolling. (This position is based on the assumption that you want to start the drawing on the origin; if not, this step is not necessary.)

8.3.3.4. Setting the Material

- Press the **F2** key to enable the **Draw Polygon** mode. This causes the **Materials** pull-down list on the right side of the toolbar to become active; you can use your mouse to select the **Wood (Pine, Spruce, Fir)** material for the frame. (Alternatively, you can access the **Material Library** by pressing **Shift-F4**, or pressing **Alt-L** to access the **Libraries** menu, then typing **M** to access the **Material Library**. Use your mouse to view the choices in the pull-down materials list, or type the first few letters of the material and use the up and down arrow keys to scroll through the list. Press **Enter** or click your left mouse button on the **Close** button to close the dialog box.)
- This defines the material for all polygons you draw until you select another material.

8.3.3.5. Drawing the Dimensioned Part

- The polygons representing the frame and the interior and exterior stops will be drawn using the following procedure:
  - Type in the line dimension,
  - Move the dimension length using the arrow keys,
  - Press **Enter** to draw the line.
- Press the **Home** key on your keyboard, which causes the drawing cursor to be positioned at the drawing origin. If you are not in **Draw Polygon** mode, press **Alt-D**, **P**. Press **Enter** to establish the current cursor position as the starting point for drawing the polygon.
- With the dimensioned drawing example in front of you, begin your drawing from the lower right corner and move counterclockwise (the program does not require that you draw in a counterclockwise direction, but this is recommended). The extension at the corner is 0.688” high, so type **0.688** from the number row or numeric keypad (be sure **Num Lock** is “on”) of your computer keyboard.
- *Note: Do not press **Enter** (or the **OK** button on the **Step Size** dialog box) to accept the step-size;- as soon as you type the number, the step size is set, and you can immediately use the arrow keys to move the distance you have selected.*

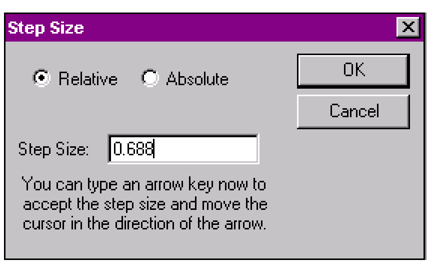


Figure 8-22. The Step Size dialog box appears any time you type a numeric value. You can click on the OK button to set the step size, or just use the arrow keys while the dialog box is open to move the step size distance.

- Press the **up arrow key** once. A line will be drawn between your starting point and the new cursor location.

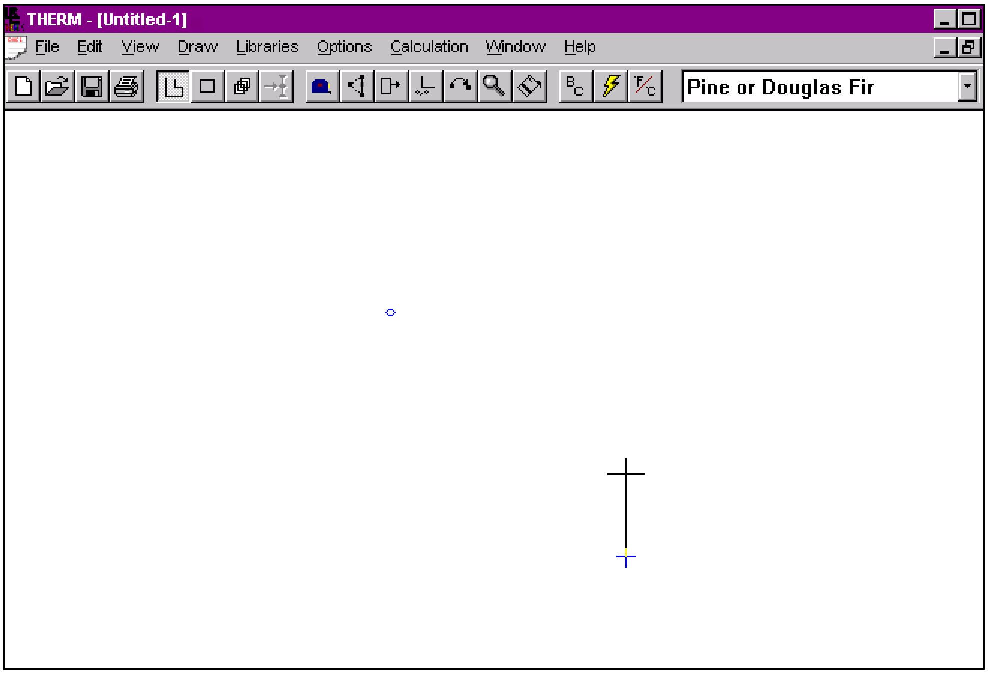


Figure 8-23. To draw a line using the keyboard: from the origin, type in the step size, move the ***up arrow key****, and press* ***Enter****.*

- Press **Enter** to draw the line and anchor the end point.
- The next line length the extension is 1.000”. Type **1**, and you will notice that, on the status bar at the bottom of the screen, the **x,y** gives your position relative to the origin. The **dx,dy** shows change in x and y, telling you where you are relative to the last point anchored; **len** tells you the actual length of the line. If you move the arrow keys, **dx**, **dy** and **len** will reflect the values for the current dimension being drawn.
- Press the **left arrow key** once, then press **Enter** to draw the line.
- The height of the next step is 0.500” (1.188 - 0.688). Type **0.5**, press the **up arrow key** once, then press **Enter** to draw the line
- For this example, we will initially simplify the drawing process by squaring off the beveled edge, marked as .100 CHAMFER in Figure 8-21. This change will have very little effect on thermal performance. If you would like to include the bevel in your model follow the steps in Section 8.3.3.8.
- The next horizontal length is 1.750” (2.75 - 1.00). Type **1.75**, press the **left arrow key** once, then press **Enter** to draw the line.
- If the interior stop had been made of the same material, we could have included it as part of this polygon. It is up to you to decide how to combine cross section elements. If there is any chance that elements may be used with other drawings or may be moved, it is best to keep them separate. If two polygons are made of different materials, they must be drawn separately.
- The next vertical length is 0.500”. Type **0.5**, press the **down arrow key** once, then press **Enter** to draw the line.

- The next horizontal length is 1.813 (seen at the bottom of the drawing). Type **1.813**, press the **left arrow key** once, then **Enter** to draw the line.
- We're now about half done with the example. Your drawing should look like this:

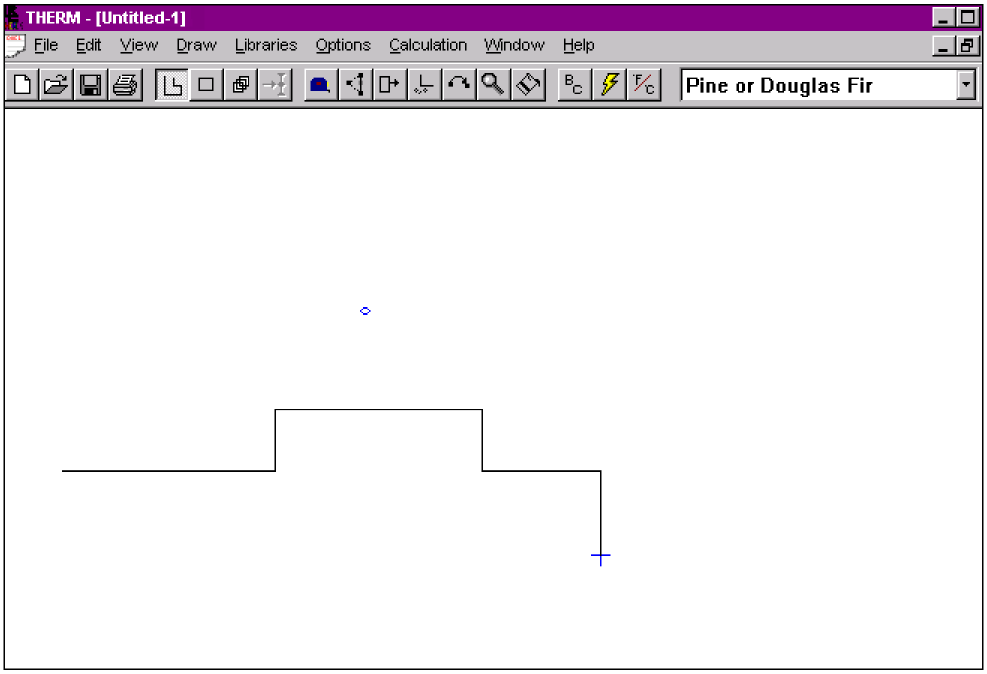


Figure 8-24. Using the keyboard, you can type the polygon dimensions and use the arrow keys to draw the lines.

8.3.3.6. Completing the Polygon

- The next vertical length is 0.688". Type **0.688**, press the **down arrow key** once, then press **Enter** to draw the line.
- The next horizontal length is 0.313". Type **0.313**, press the **right arrow key** once, then press **Enter** to draw the line.
- The next vertical length is 0.219". Type **0.219**, press the **up arrow key** once, then press **Enter** to draw the line.
- The next horizontal length is 0.250". Type **0.250**, press the **right arrow key** once, then press **Enter** to draw the line.
- The next vertical length is 0.219". Type **0.219**, press the **down arrow key** once, then press **Enter** to draw the line.
- The next horizontal length is 1.64" (2.203 – 0.313 – 0.250). Type **1.64**, press the **right arrow key** once, then press **Enter** to draw the line. Another technique, which lets the program do the arithmetic for you, would be to type **2.203**, press the **right arrow key** once, type **0.313**, press the **left arrow key** once, type **0.250**, press the **left arrow key** once, then press **Enter** to draw the line and anchor the end point.
- The next vertical length is 0.219". Type **0.219**, press the **up arrow key** once, then press **Enter** to draw the line.
- The next horizontal length is 0.125". Type **0.125**, press the **right arrow key** once, then press **Enter** to draw the line.

  - The next vertical length is 0.219”. Type **0.219**, press the **down arrow key** once, then press **Enter** to draw the line.

  - Press **C** to close the polygon, and the program will automatically draw a line between the current cursor location and the starting point of the polygon.

- When you are finished and the polygon is filled in, press the **Esc** key to cancel the polygon mode.

- Your finished polygon should look like the picture in Figure 8-25.

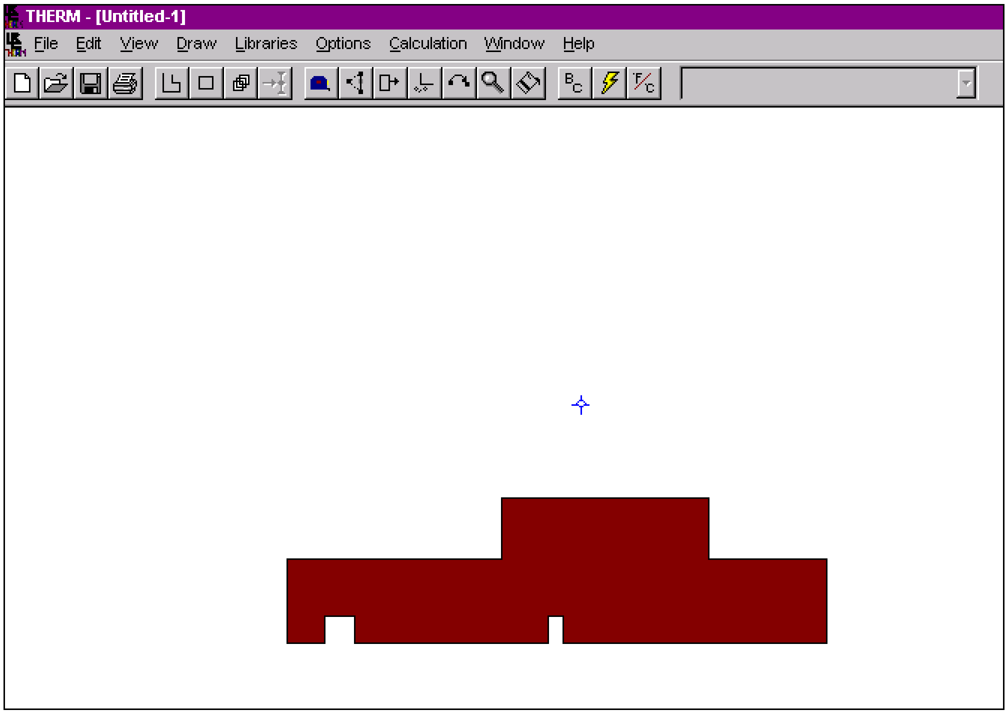


Figure 8-25. The completed polygon that represents the frame

8.3.3.7. Correcting Mistakes While Drawing

If you make mistakes while drawing a polygon, there are several ways to make corrections without deleting the entire polygon. Here are a few techniques:

- *Wrong Dimension:* If you accidentally enter the wrong dimension and move the cursor using the arrow keys, simply use the arrow keys to backtrack to the last point drawn (len = 0.000). Alternatively, you can use the mouse to position the cursor over the last good point and press the **Space bar**, which causes the cursor to snap exactly to that point. Then type in the correct dimension and start over (you can also use the technique below to move back).

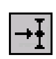
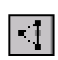
- *Accidental Mouse Movement:* Movement of the mouse while drawing a polygon may cause the line to shift, changing the dimensions. If you accidentally move the cursor, use the mouse to position the cursor over the last good point and press the **Space bar**, which causes the cursor to snap exactly to that point. Be sure len = 0.000 before you begin drawing again.

- *Wrong Point:* If you’ve entered the wrong dimension (as in the first technique above) but pressed the **Enter** key to anchor the point, you may delete the point by pressing the **Esc** key once. Use the mouse to reposition the cursor as in the technique above. Pressing the **Esc** key twice produces the following dialog box shown in Figure 8-26 for deleting additional points or the entire polygon.

Figure 8-26. This dialog box allows you to delete the last point in a polygon. You can use your mouse or the arrow keys to move to each button in the box.

8.3.3.8. Drawing Additional Polygons

Interior Stop:

- Place the mouse cursor over the corner of the frame where the lower left corner of the interior stop will be located. Press **F3** or **Alt-D,R** to activate the **Draw Rectangle** toolbar button. Place the cursor within the sticky distance (half of one side of the cursor cross hair) and press **Enter** to set that location as the starting point for the rectangle. Now you are ready to draw the rectangle representing the interior stop.
- Type the horizontal dimension of the rectangle, **1.094**, and press the **right arrow key** once to draw the first side of the rectangle. (*Do not press **Enter** yet !)*
- Type the vertical dimension of the rectangle, **0.50**, press the **up arrow key** once, then press **Enter**, and the program will draw the rectangle, assigning it the last used **Material**, which should be **Wood (Pine, Fir, Spruce)**.
- You can add the bevel on the stop using the mouse and the keyboard as follows:
  - Highlight the interior stop polygon by placing your mouse cursor inside the polygon and clicking the left mouse button once. The vertices of the highlighted polygon will be indicated by squares.
  - You may want to zoom in on the drawing by clicking the right mouse button once or twice in order to see more detail.
  - Type **0.100** and press **Enter** (or click the **OK** button) to set the step size to 0.100".
  - Click on the **Insert Points** toolbar button  and place the mouse cursor over the upper right corner. Press **Space bar** to locate the cursor exactly on that corner point, press the **left arrow key** once to move the cursor 0.100" to the left, and press **Enter** to insert the new point.
  - Click on the **Edit Points** toolbar button  and place the mouse cursor over the same upper right corner point as in the previous step, so we can move the point. Press the **Space bar** to locate the cursor exactly on the point to be moved. A double-pointed arrow should appear, which indicates that you can move the point. If a four-pointed arrow appears, this indicates that you are on a line rather than a point, which means that the cursor was outside the sticky distance of the point. (If you press the **down arrow** under this condition, the entire line will move. You will need to zoom in so that you can position the cursor on the point you wish to select).
  - Press the **down arrow key** once, which will move the point down 0.100".
  - Press **Enter** to set the point in the new location. The lines attached to the point will be redrawn at the new angle.

Exterior Stop:

- We will draw the polygon for the exterior stop in a similar fashion. The drawing in Figure 8-27 contains the dimensions needed to draw this polygon. We will first draw a rectangle, and then insert and move points to make the beveled sides of the polygon.

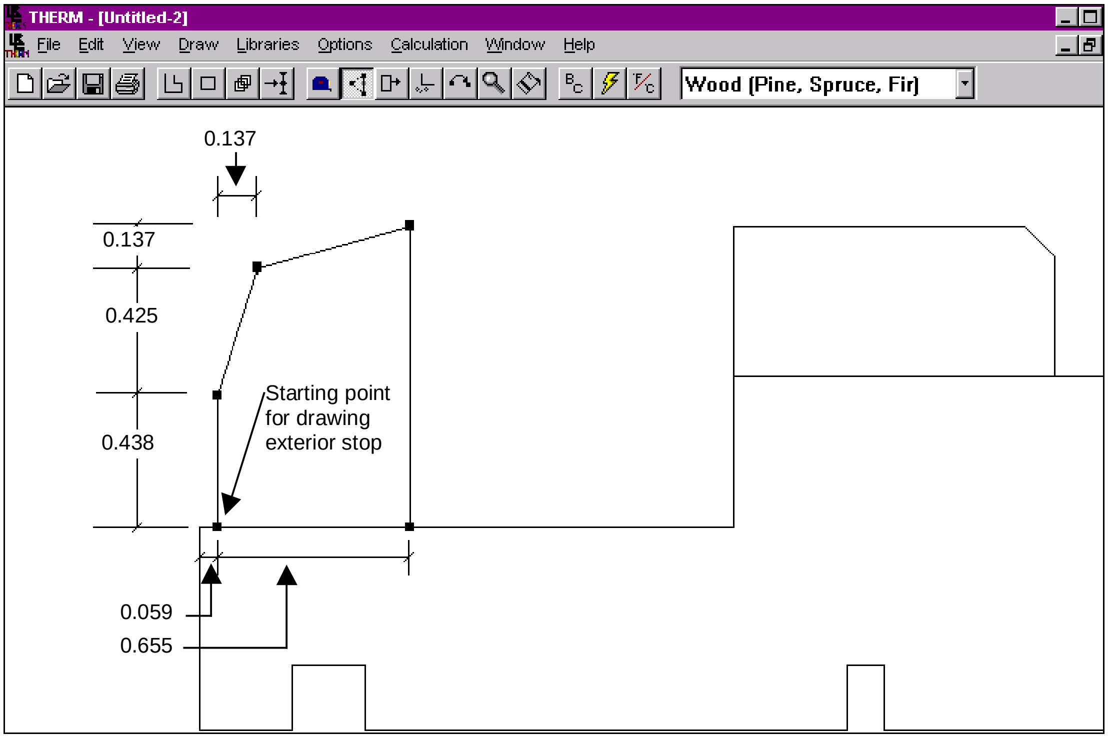
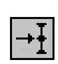


Figure 8-27. Dimensions for the exterior stop.

- We will use the lower left corner of the stop as the starting point for the rectangle. To locate the drawing cursor at this corner point, first place the cursor on the left corner of the frame polygon closest to the point. Press the **Space bar** to locate the cursor exactly on the point. The lower left corner of the exterior stop is 0.059" to the right of this point, so type **0.059** and press the **right arrow key** once. The location of the cursor will be the starting point of the stop.
- Press **F3** to activate the **Draw Rectangle** mode. Press **Enter** to set the current cursor position as the starting point for the rectangle.
- The exterior stop is 0.655" wide, so type **0.655**, and press the **right arrow** once. This establishes the width dimension of the rectangle. *Do not press Enter yet !*
- The exterior stop is 1.00" high, so type **1**, press the **up arrow** once, and press **Enter**. You now have a rectangle 0.655" by 1.00".
- Next, we are going to insert a point on the left side of the stop where the bevel starts. Highlight the exterior stop by placing the drawing cursor inside the rectangle and clicking the left mouse button once. The vertices of the highlighted rectangle will be displayed as squares.
- Click your mouse on the **Insert Points** toolbar button , and place your mouse cursor over the lower left point (the starting point) of the exterior stop. Press the **Space bar** to position the cursor exactly on the point. The point on the left side of the stop where the bevel starts is

0.438" above the starting point, so type **0.438** and press the **up arrow key** once. Press **Enter**, and a point will be inserted at your cursor position.

- Click your mouse on the **Edit Points** toolbar button , and place the mouse cursor over the upper left point on the exterior stop. When you are over the point, the cursor will turn into a double-pointed arrow. Press the **Space bar** to place the cursor exactly on the point.
- The bevel is 0.137" off the horizontal and vertical lines, so type **0.137**, and press the **right arrow key** once, then the **down arrow key** once. Press **Enter**, and the point will be redrawn in the proper position, with the attached lines following and forming the beveled edge.
- You have now finished drawing all the frame and stop elements and can proceed to the next part of the tutorial: inserting the glazing system.

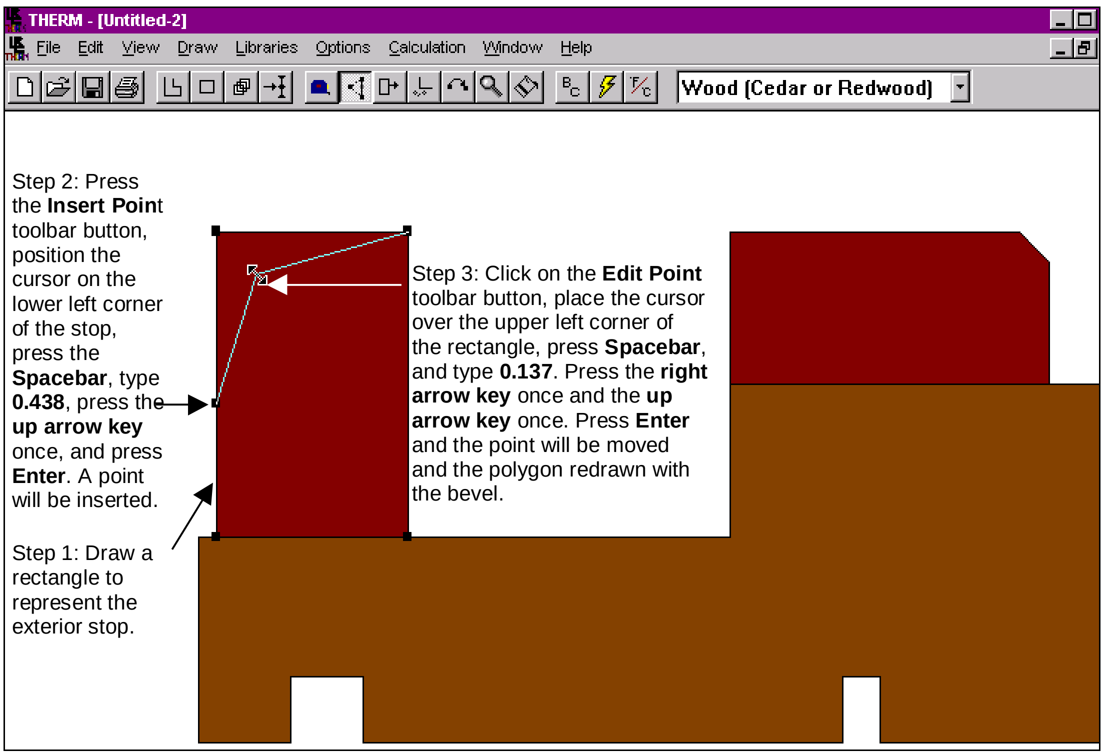


Figure 8-28. Adding the beveled edge to the exterior stop

8.3.3.9. Save the File

- Save your work using the **Save File** toolbar button , or the **File/Save** menu choice. Type a name that is not already being used, such as **Tutorial3** (a file named Sample.thm is included in the installation, so pick a different name to avoid overwriting that file), and click on the **OK** button. THERM will automatically add the THM extension, resulting in this case in creating a file name of TUTORIAL3.THM.

8.4. Import a Glazing System from WINDOW 4

Now that you have drawn the frame and stop components of the cross section, you are ready to import a glazing system from the WINDOW program. A sample WINDOW glazing system library is provided with the THERM installation.

- From the **Draw** menu, click your left mouse button on the **Locator** choice.

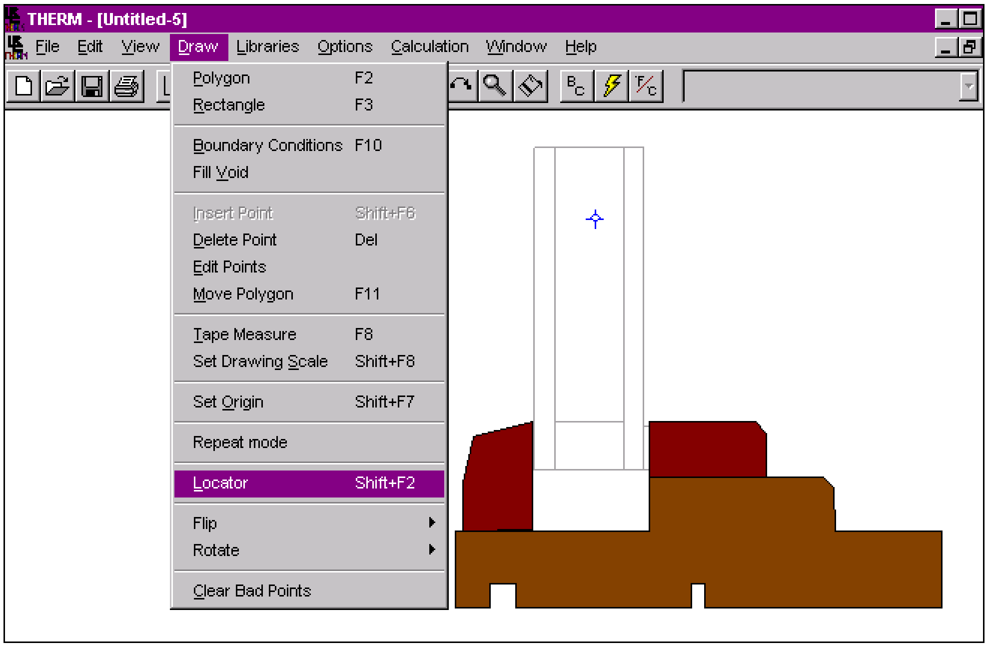


Figure 8-29. Select the Locator choice from the Draw menu.

- Move the mouse cursor to the lower left corner where the glazing system is to be inserted, and click on the left mouse button. A small circle will appear indicating the position of the **Locator,** as shown in Figure 8-30. (If you are not using an underlay, first draw the glazing tape to the right of the exterior stop, to serve as a guide for placing the **Locator**.)

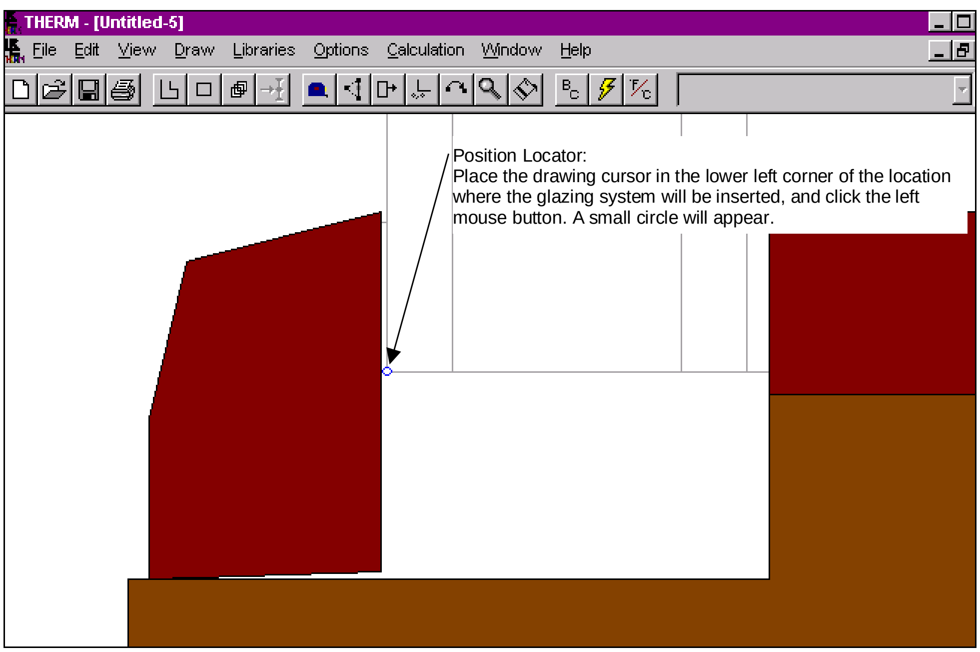


Figure 8-30. Position the locator where you want the lower left corner of the glazing system to be placed.

- From the **Libraries** menu, select the **Glazing Systems** choice.

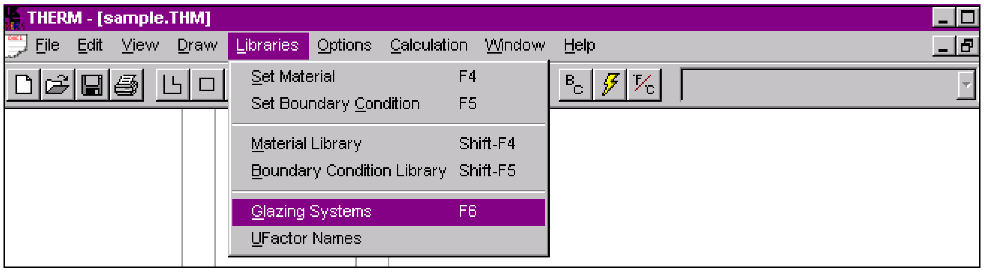


Figure 8-31. Select the Glazing Systems choice from the Libraries menu.

- The **Glazing Systems** dialog box will appear with a pull-down list showing all the choices from the WINDOW "GLZSYS.W4" file that was installed with THERM. You can use the **Browse** button to locate other GLZSYS.W4 files if desired. However, for this example, you should use the library that is provided with THERM.

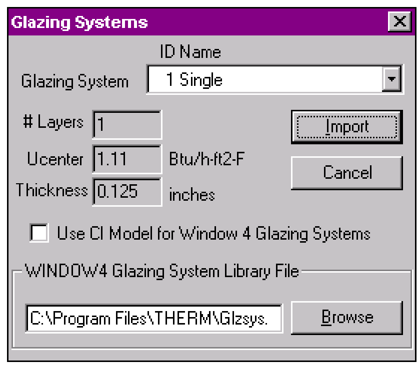


Figure 8-32. The Glazing Systems dialog box displays the information about each glazing system choice.

- Put the mouse cursor on the pull-down list arrow, click the left mouse button, and the list of glazing systems will be displayed. Scroll through the list (using your mouse or arrow keys) and pick the "THERM SAMPLE" choice.

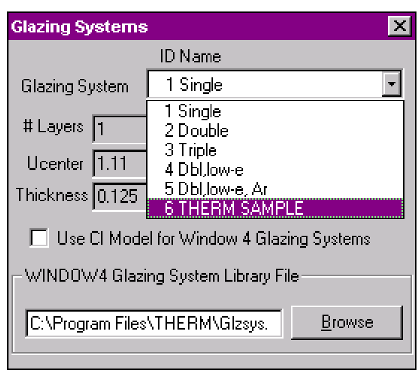


Figure 8-33. Select the THERM SAMPLE glazing system to be inserted in the tutorial example.

- Press the **Import** button, and the **Insert Glazing System** dialog box will appear. Leave all the default values as they are, and press the **OK** button. *(The **Draw Spacer** box is left unchecked because we will insert a custom spacer later in the tutorial).*

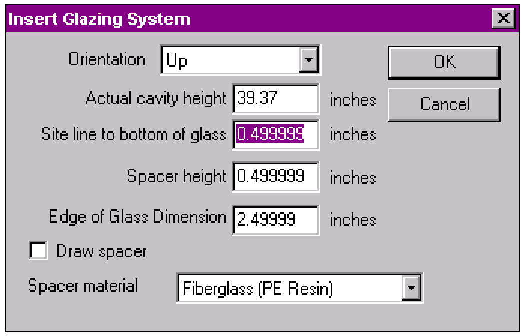


*Figure 8-34. The **Insert Glazing System** dialog box appears when you are importing a glazing system from WINDOW.*

- The glazing system will be inserted into the frame with the lower left corner positioned where the **Locator** was placed.

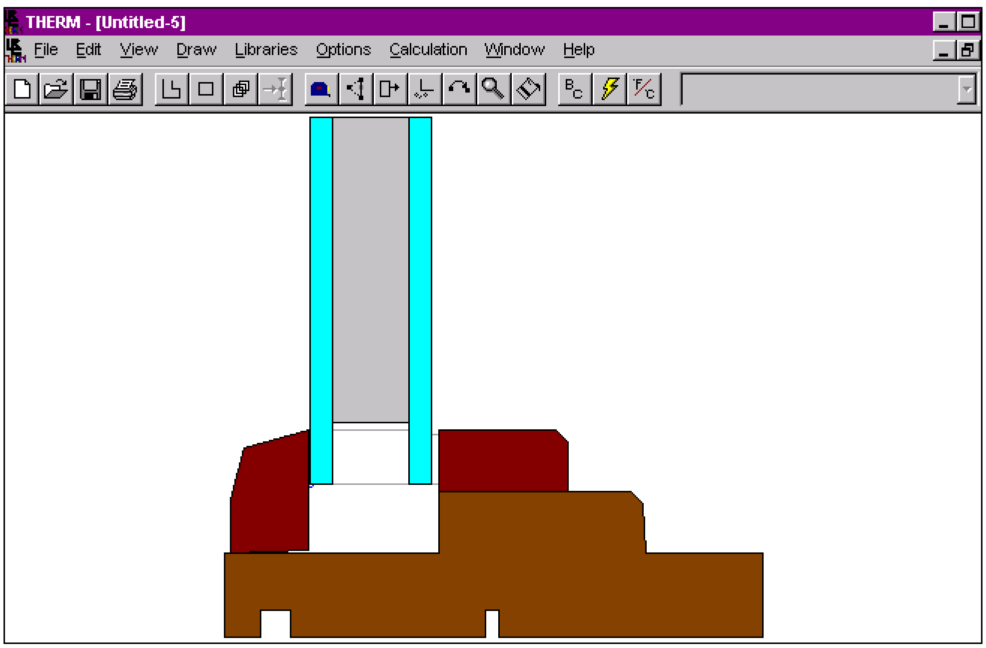
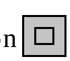


Figure 8-35. The glazing system from WINDOW has been inserted into the frame components.

- Now that the glazing system has been inserted, glazing tape can be drawn between the glazing system and the interior and exterior stops (if you haven't already completed this step), as follows:
  - Click the right mouse button as many times as you need in order to enlarge the drawing so that you can snap to the points described below. You may have to enlarge it quite a bit.
  - Press the **Draw Rectangle** toolbar button .
  - Select **Polysiobutylene** from the materials pull-down list on the right-hand side of the toolbar.
  - Position the drawing cursor in the upper left corner of the space to the left of the glazing system, and click the left mouse button. Keeping the left mouse button down, move the drawing cursor down to the lower right corner of the gap; release the mouse button. A rectangle will be drawn that exactly fits in the space.

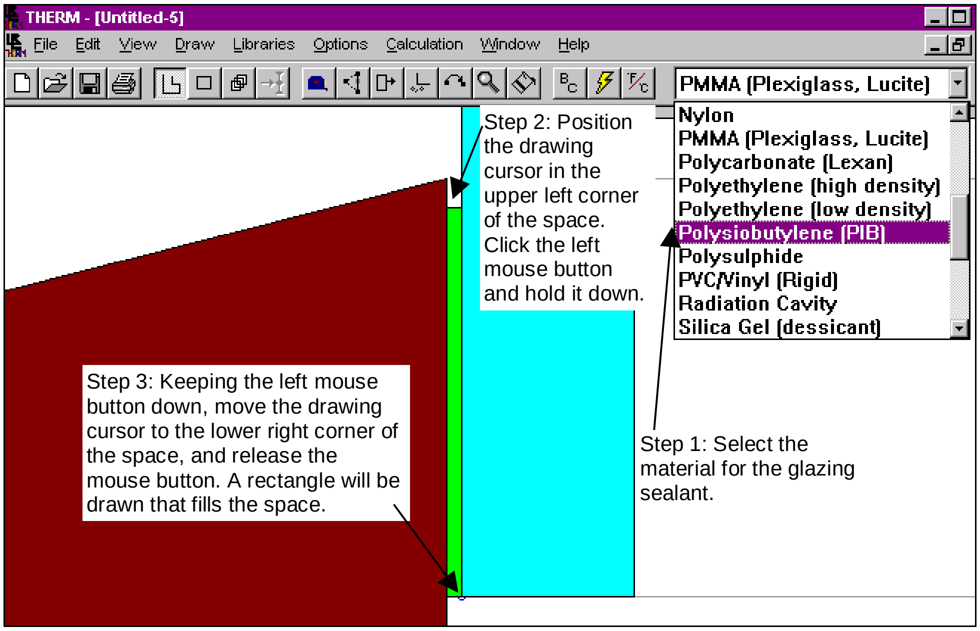


Figure 8-36. Draw a rectangle that fills the space between the exterior stop and the glazing system.

- In the same manner, draw a rectangle for the glazing tape between the interior stop and the glazing system.

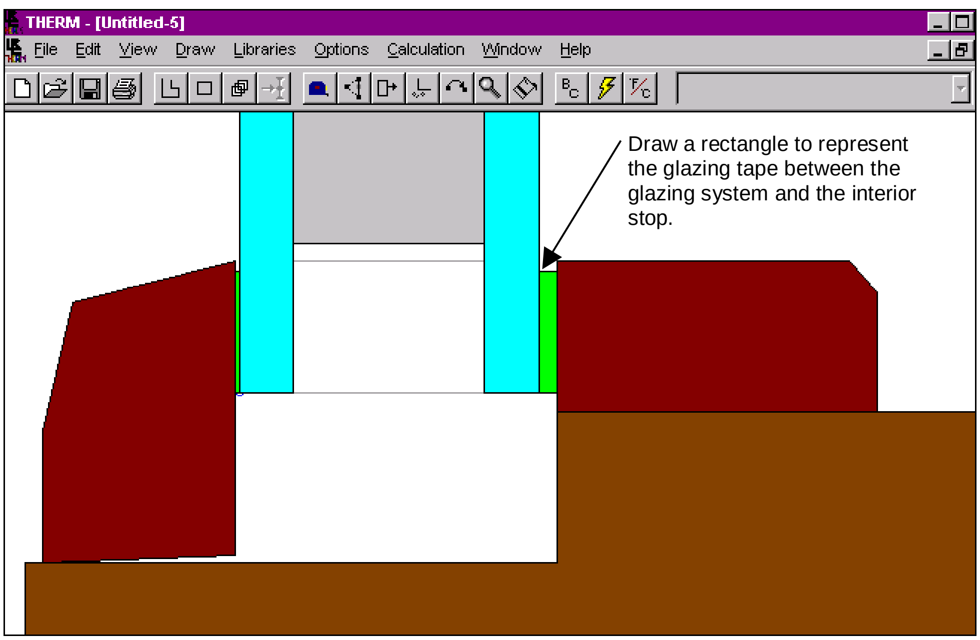
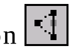


Figure 8-37. Draw a rectangle that fills the space between the interior stop and the glazing system.

- The lower right corner of the exterior stop does not lie exactly on the frame, and because it is necessary to either fill or delete all voids, we have to decide what to do with this area. One solution is to move that corner down to the frame, as follows:
  - Highlight the exterior stop by placing the mouse cursor inside the polygon and clicking the left mouse button once. The vertices of the selected polygon will be displayed as squares.

- Click the left mouse button on the **Edit Point** toolbar button .
- You may want to zoom in (click the right mouse button) one or two times to enlarge the cross section. Position the mouse cursor over the lower right-hand corner of the exterior stop. The cursor will be displayed as a double-pointed arrow, which means you can move the point.
- Hold down the Shift key to restrain the cursor movement to the vertical direction. Then, holding the left mouse button down, move the mouse (and the point will follow) down to the frame. Release the mouse button.

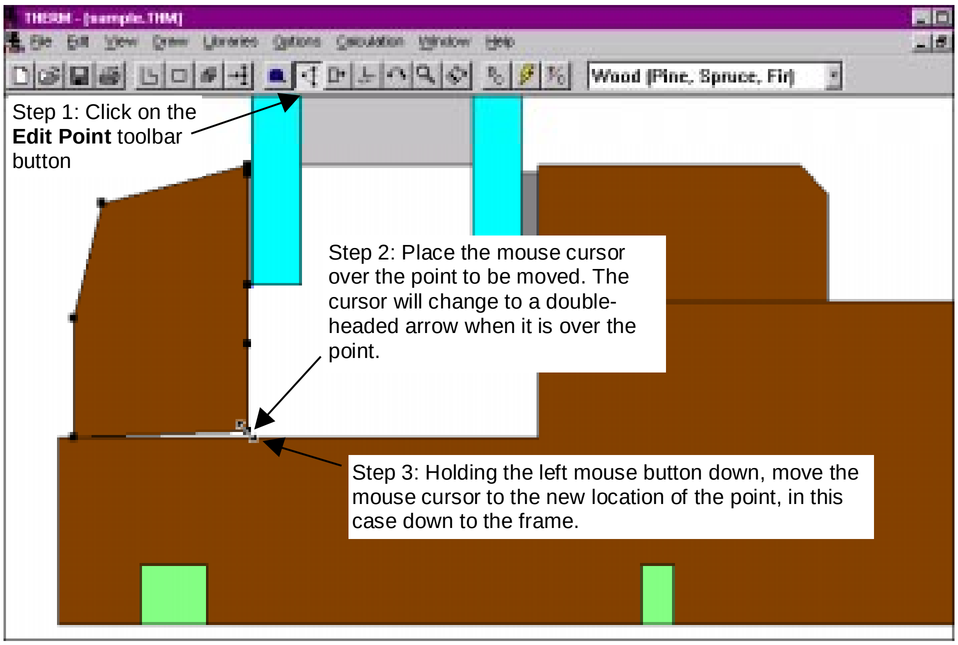


*Figure 8-38. With the **Edit Point** toolbar button activated, place the mouse cursor over the point to be moved, hold the left mouse button down, and move the mouse to the new point location.*

- The point and attached lines are moved, and the polygon is redrawn, as shown in Figure 8-39.

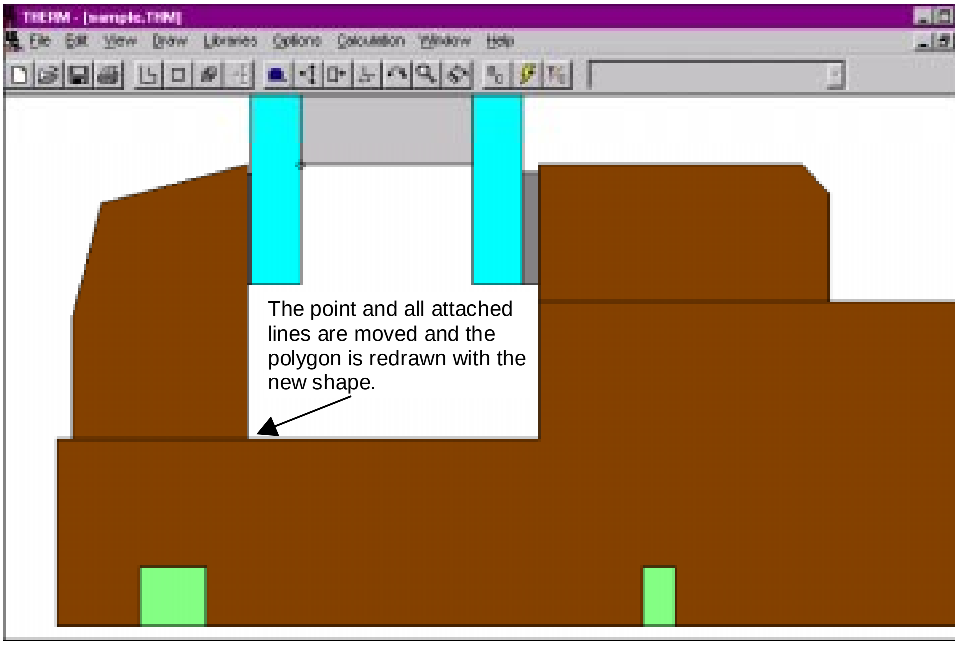


Figure 8-39. THERM will move the point and the lines attached to that point and redraw the cross section with the new geometry.

- Another option for this space would be to define it as a cavity. In that case, you should make this sliver a separate cavity polygon from the main cavity area. The effective conductivity of the cavity is based on the overall dimensions of a cavity and if you include the sliver you effectively double the size of the main cavity. The first solution, to bring the exterior stop corner down to the frame and model that area as wood, is a better thermal approximation than modeling the whole area as one cavity.
    - You can copy and paste into the cross section a spacer from another THERM file, as follows:
        - Use the **File** menu, **Open** option to open SPACER.THM, another sample file that was installed with the THERM program. This file contains a cross section of a custom spacer that we can copy and paste into our tutorial cross section.

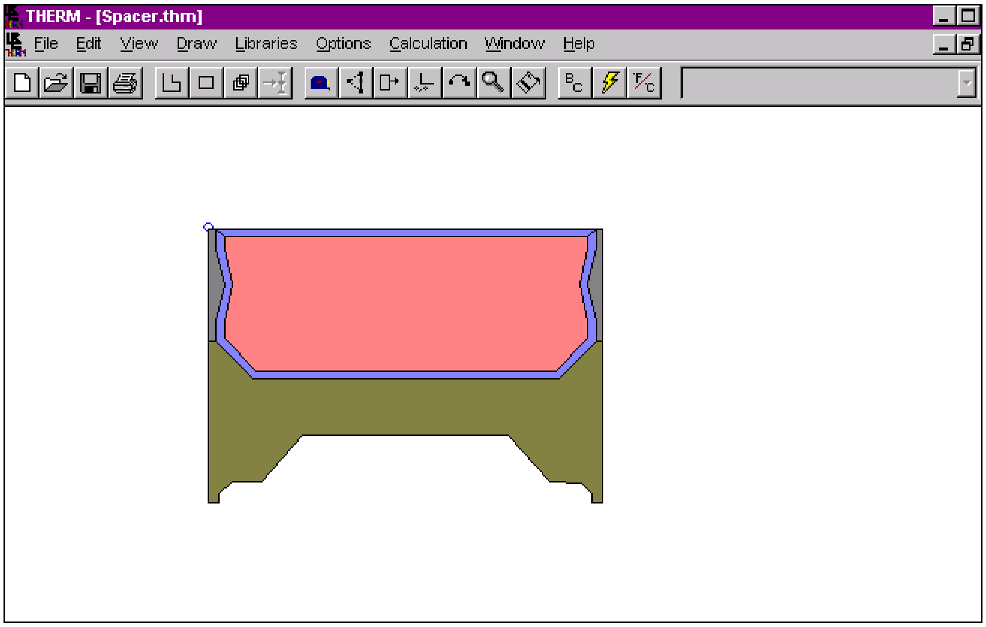


*Figure 8-40. Use the **Draw/Locator** menu choice to position the **Locator** in the upper left corner of the spacer.*

- Use the **Draw/Locator** menu to place the locator circle in the upper left corner of the spacer drawing (if it is not already there), as shown in Figure 8-41.

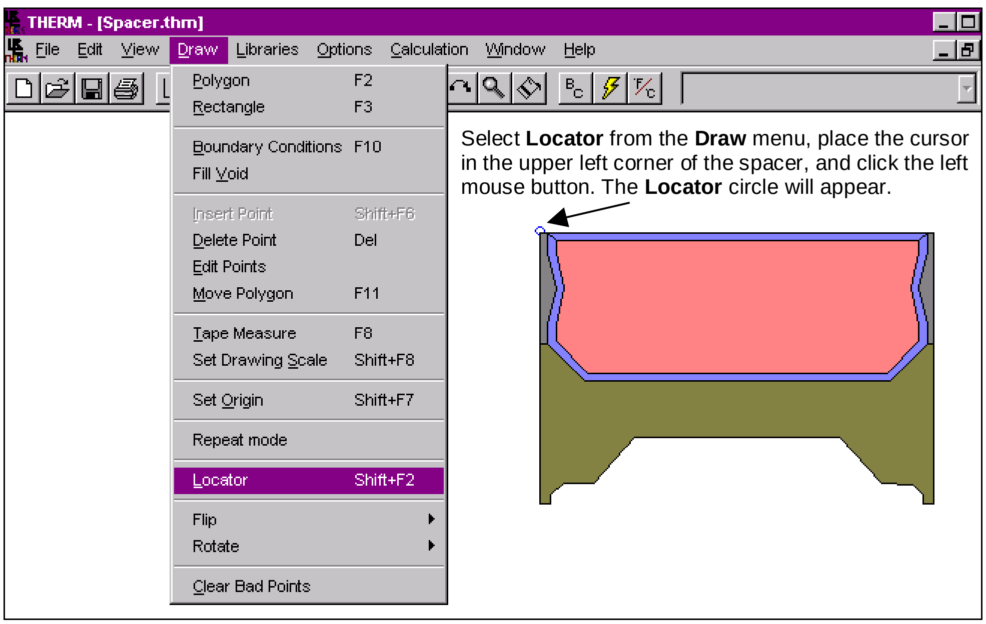


*Figure 8-41. Position the **Locator** in the upper left corner of the spacer.*

- Use the **Edit/Select All** menu to select all the polygons in the spacer file. Or you can click on a point above and to the left of the spacer, and holding the left mouse key down, move the cursor to a point below and to the right of the spacer. A dashed box will be displayed showing the area you have moved your cursor, and all the polygons inside that area will be automatically selected.

- Use the **Edit/Copy** menu, or **Ctrl-C**, to copy the selected polygons to the Windows™ clipboard.

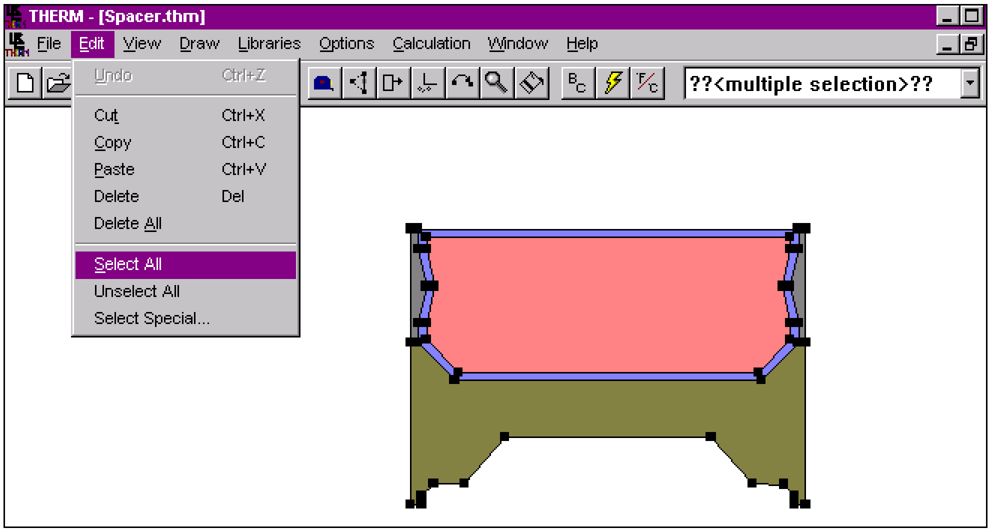


*Figure 8-42. Use the **Edit/Select All** menu choice to select all the polygons in the spacer; then, use **Edit/Copy** (or **Ctrl-C**) to copy these polygons into memory.*

- Use the **Window** menu to select the tutorial cross section file.
- Use the **Draw/Locator** menu to place the locator circle in the upper left corner of the recess under the glazing system cavity where you want the upper left corner of the spacer to be located.
- Use the **Draw/Paste menu**, or **Ctrl-V**, to paste the spacer into the SAMPLE cross section. The spacer will be placed in the cross section by matching up the positions of the two Locators.

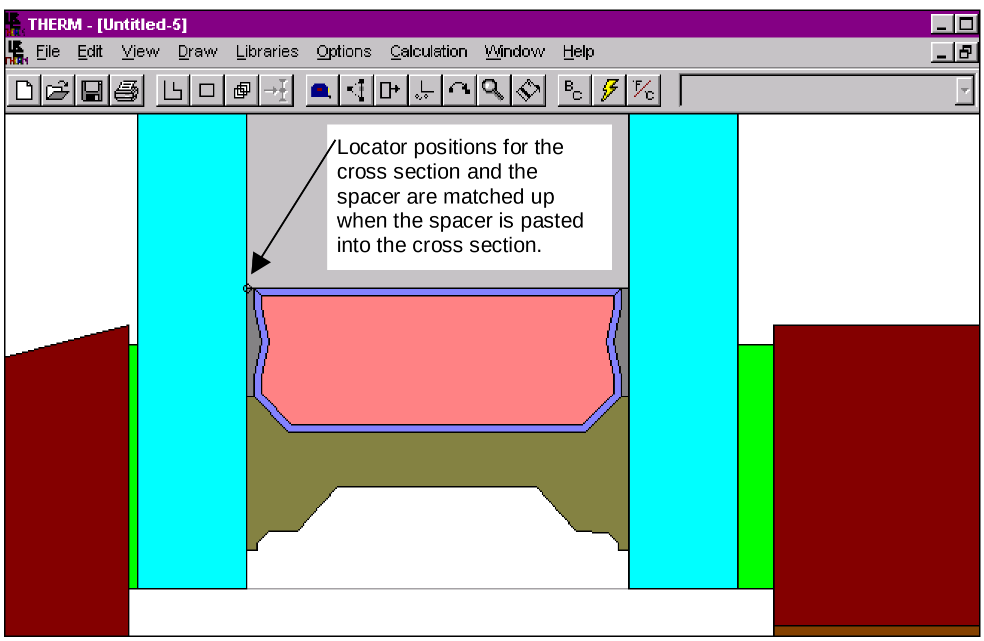


*Figure 8-43. Use the **Edit/Paste** menu choice to paste the spacer into the cross section.*

- Select the **Libraries** menu, **Material Library** choice (or press **Shift-F4**), and select the **Frame Cavity NFRC Simplified** material from the **Materials** list.

- Select the **Fill** tool and click in the area between the glass and the stop. This area will automatically be filled by the program with a polygon of the material **Frame Cavity NFRC Simplified**.

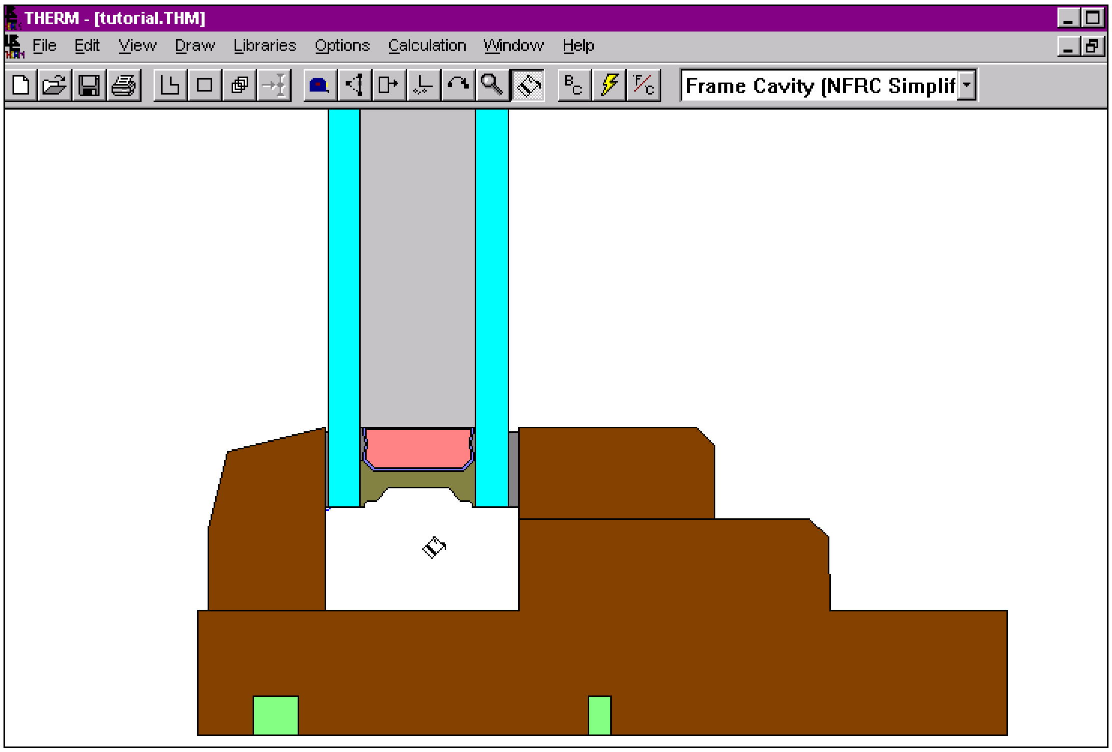


*Figure 8-44. Use the **Fill** tool to fill in the cavities in the cross section.*

- When you have finished these polygons, **Zoom** the drawing back out, either holding down the **Shift** key while clicking the right mouse button repeatedly until the drawing is the size you want it to be, or **Ctrl right** mouse click to zoom to fit the drawing in the screen.

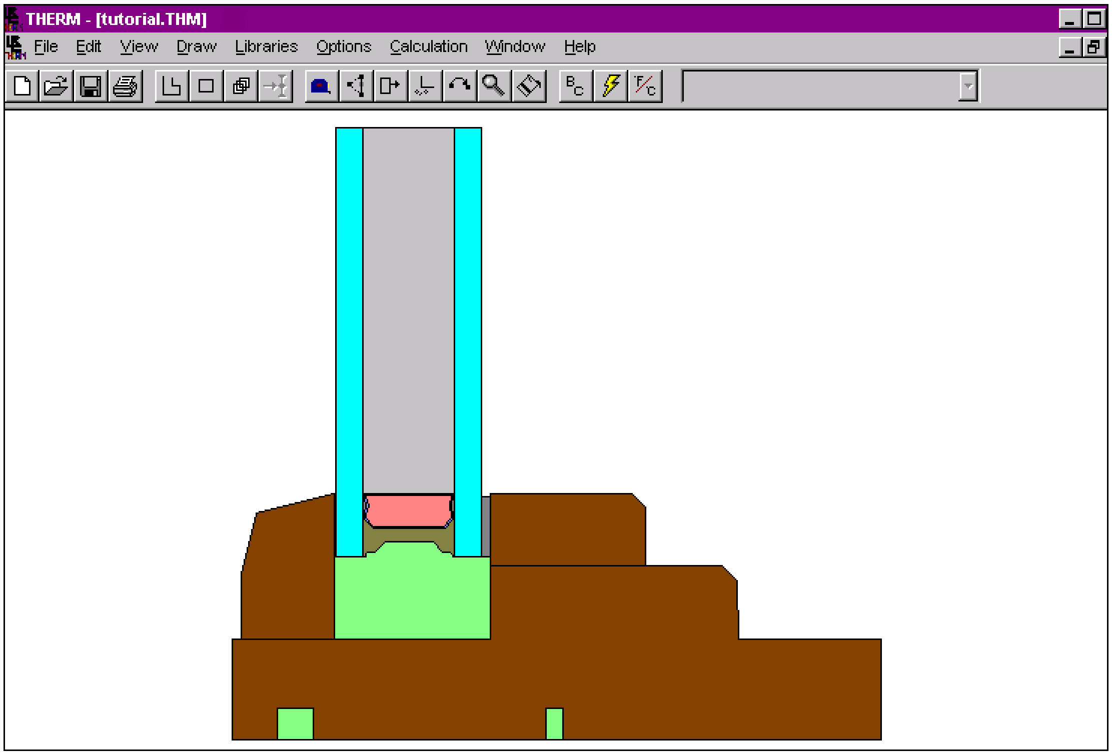
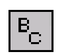


Figure 8-45. The finished cross section.

8.5. Define Boundary Conditions

- Next, you need to define the boundary conditions for the cross section.
- Press the **Boundary Conditions** toolbar button , or use the **Draw/Boundary Conditions** menu choice.
- If the program detects any flaws in the drawing while assigning the boundary conditions, a message will appear, and you may have to fix the problems before the program can continue defining the boundary conditions. See Section 6.3, "Error Detection in THERM" for detailed discussions of this situation.
- THERM will automatically draw an exterior boundary around all the cross-section components, indicated by a heavy line. The boundary condition is actually made up of many boundary segments, which can each be given different boundary condition definitions.

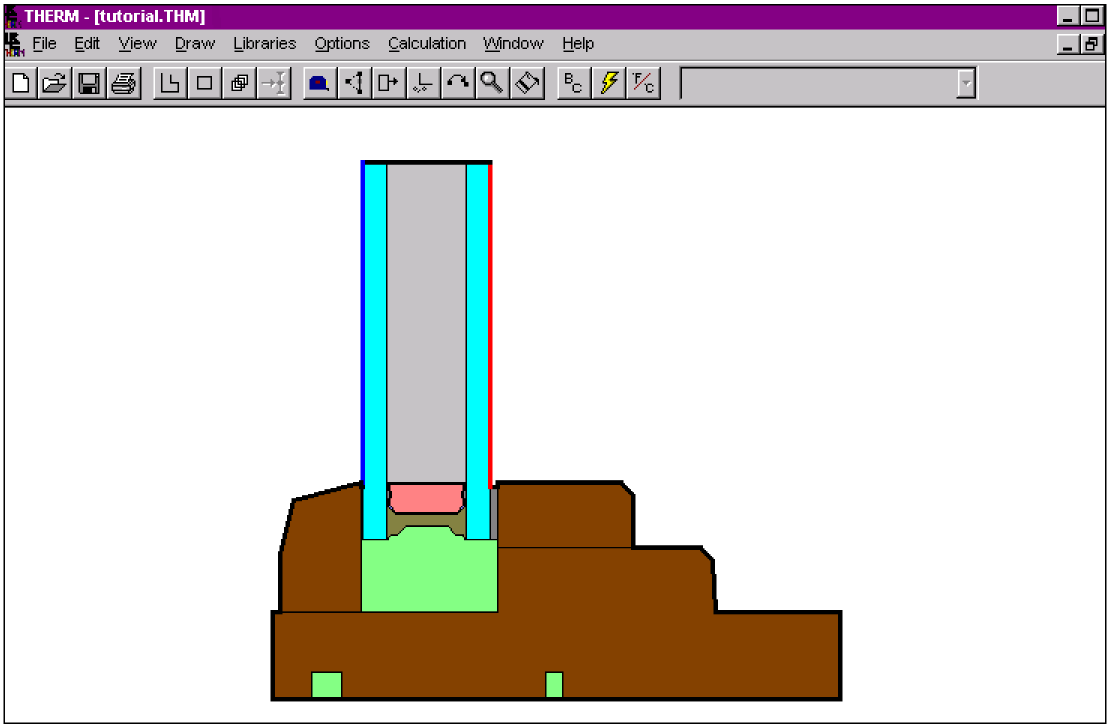


Figure 8-46. When you press the Boundary Conditions toolbar button, THERM draws an exterior boundary around all the cross-section components.

- The default boundary condition for all the boundary elements except the glazing system is adiabatic. The glazing system is automatically given **THERM SAMPLE U-factor Inside** and **THERM SAMPLE U-factor Outside** boundary conditions (the program automatically adds the glazing system name, in this case **THERM SAMPLE**, as a prefix to the boundary condition name) if the glazing system was inserted from a WINDOW glazing system.

- Select the lower right boundary segment on the interior frame (the right side) by moving the mouse cursor to it and pressing the left mouse button. Round points on each end of the segment indicate that it has been selected.

- Hold the **Shift** key down, and click on the segment on the interior stop. The program automatically selects all boundary segments between the first and last selected segments, *moving in a counterclockwise direction*, as shown in Figure 8-47.

- Press **Enter** and the **Boundary Condition Type** dialog box will appear. Select **Interior Wood/Vinyl Frame** for the boundary condition and **Frame** for the **U-Factor Surface** as shown in Figure 8-47. Press the **OK** button.

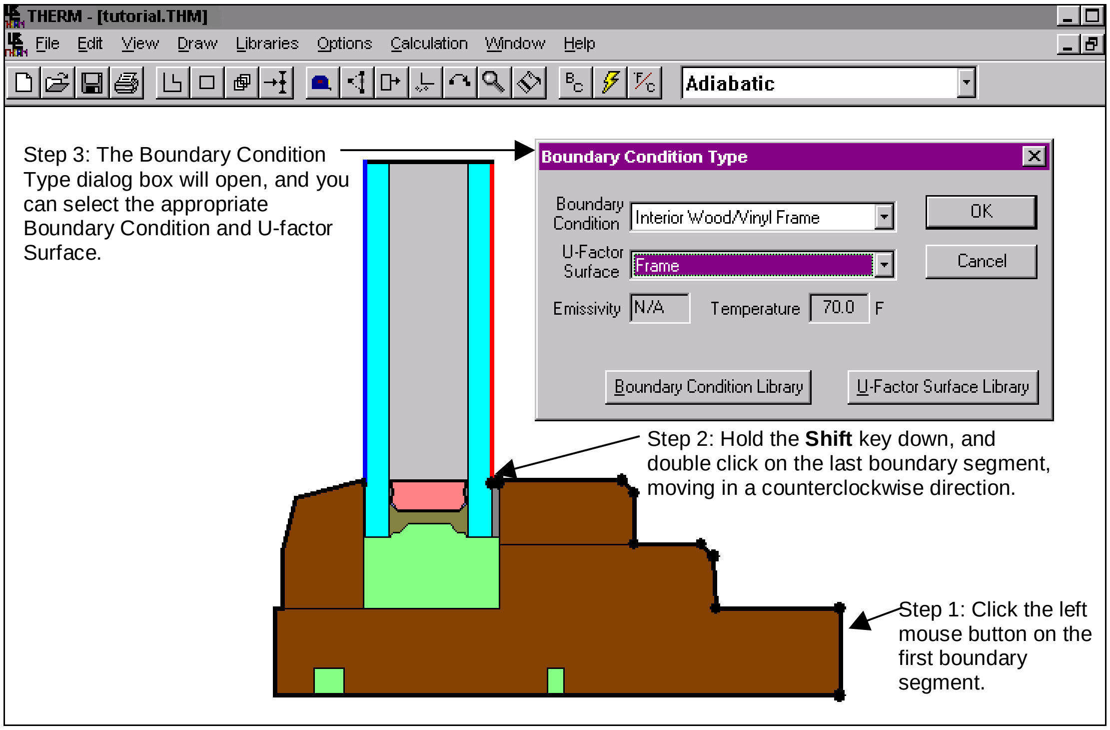


Figure 8-47. Select multiple boundary segments for the interior side of the cross section, and define their boundary conditions.

- Use the same technique to define the boundary conditions for the exterior (left side) of the cross section, as shown in Figure 8-48. Select the exterior boundary segments for the exterior stop and the frame, and assign them the **Boundary Condition** of **Exterior Surface**.

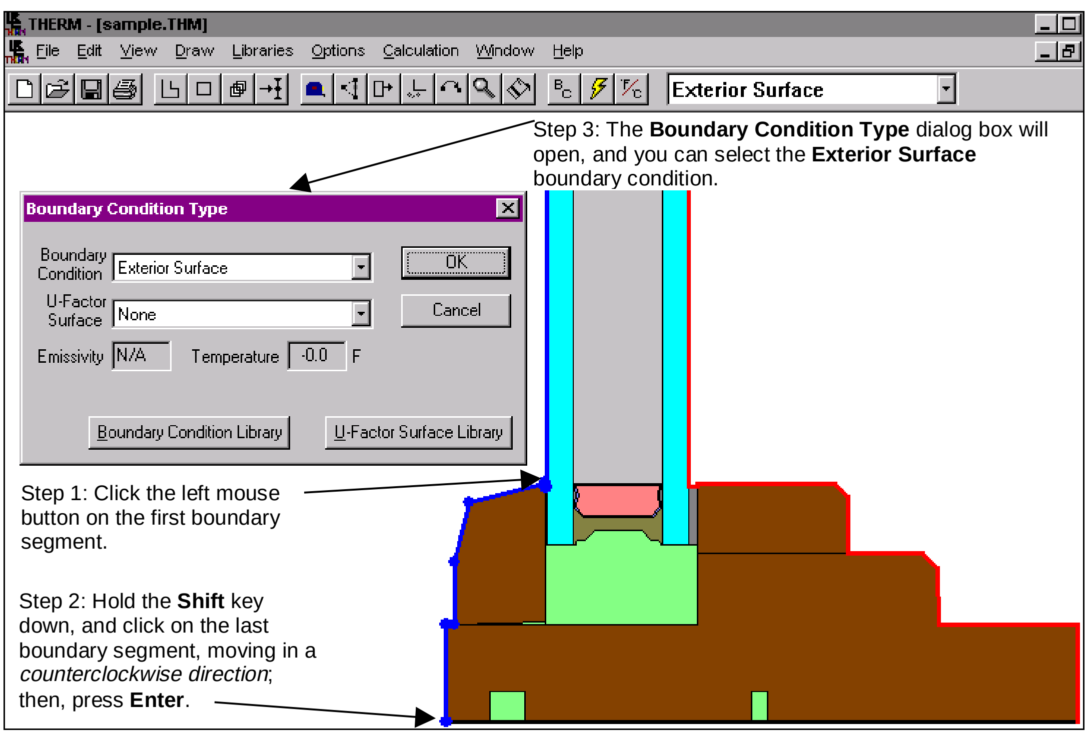


Figure 8-48. Select multiple boundary segments for the exterior side of the cross section, and define their boundary conditions.

- To view the boundary conditions automatically defined for the imported WINDOW glazing system, place the mouse cursor on the inside boundary segment of the glazing system, and double-click the left mouse cursor. The **Boundary Condition Type** dialog box will appear. **THERM SAMPLE: Inside U-factor** will already be selected for the **Boundary Condition**, and **Edge** for the **U-Factor Surface**, as shown in Figure 8-49. Press the **OK** button.

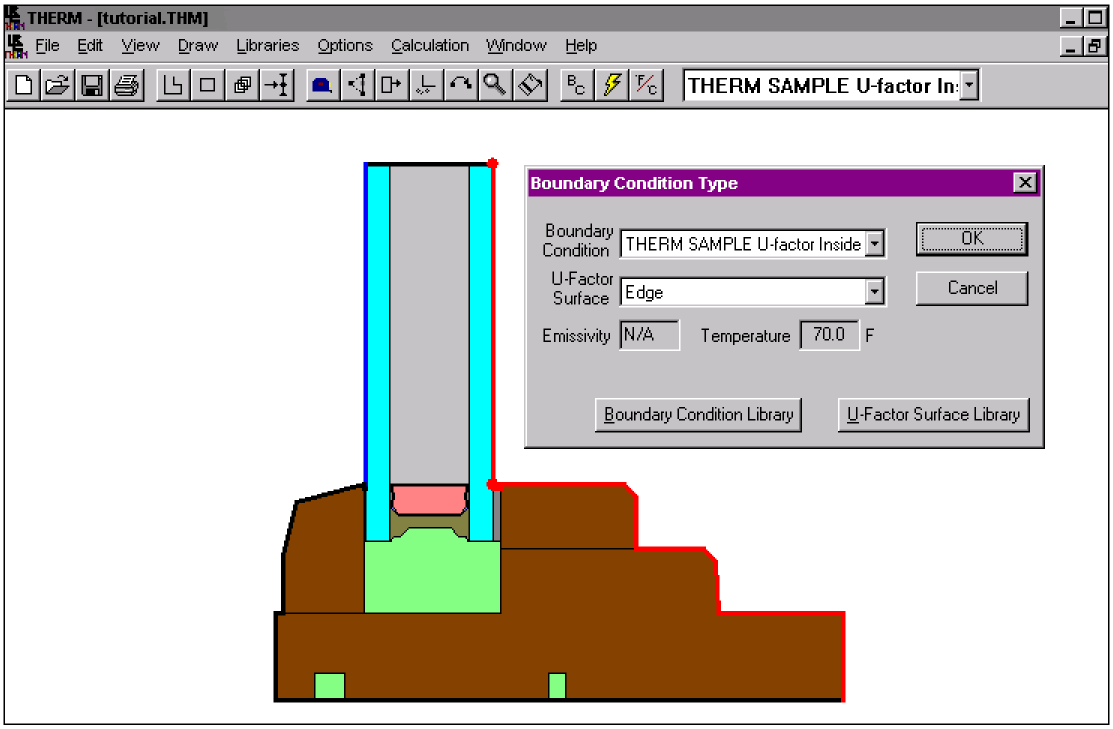


Figure 8-49. You can view the boundary condition definitions for the glazing system by double-clicking the left mouse button on the glazing system boundary.

- Save the file using the **Save File** toolbar button , or the **File/Save** menu choice.

8.6. Run and View the Results

- Press the **Calculation** toolbar button , or select the **Calculation/Calculation** menu choice.
- The status bar at the bottom of the THERM screen will display the steps of the calculation, as shown in Figure 8-50.

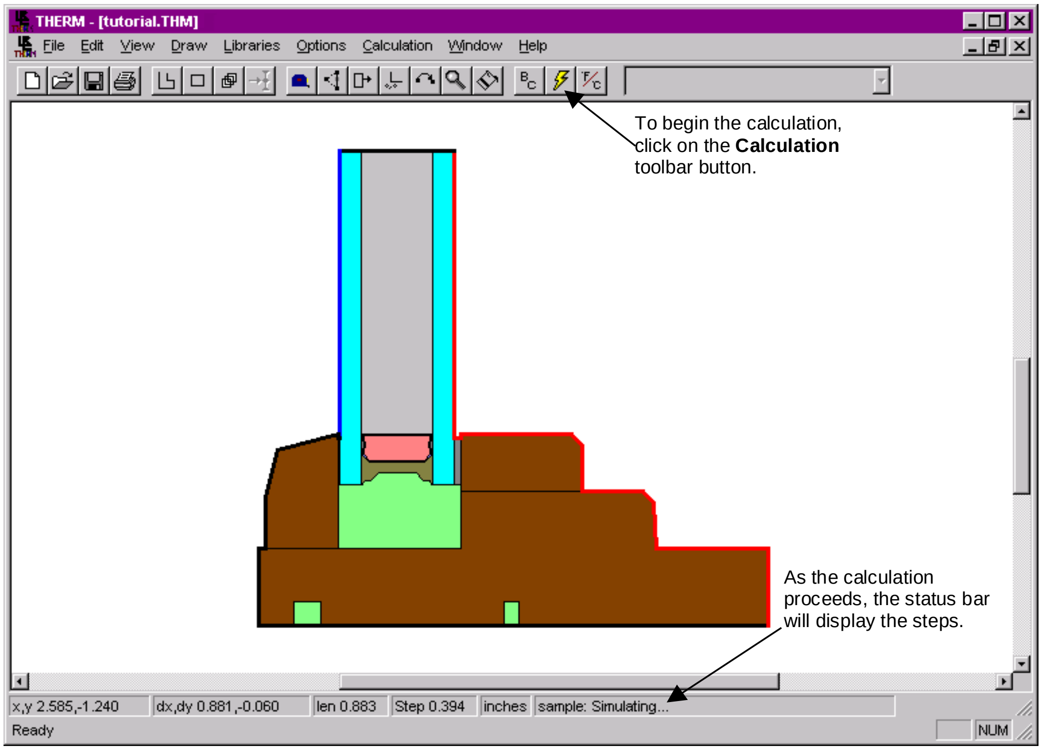


Figure 8-50. Click on the Calculation toolbar button; as the calculation proceeds, the steps will be displayed on the status bar.

- When the simulation is finished, the default isotherm results will be drawn on the cross section, as shown in Figure 8-51.

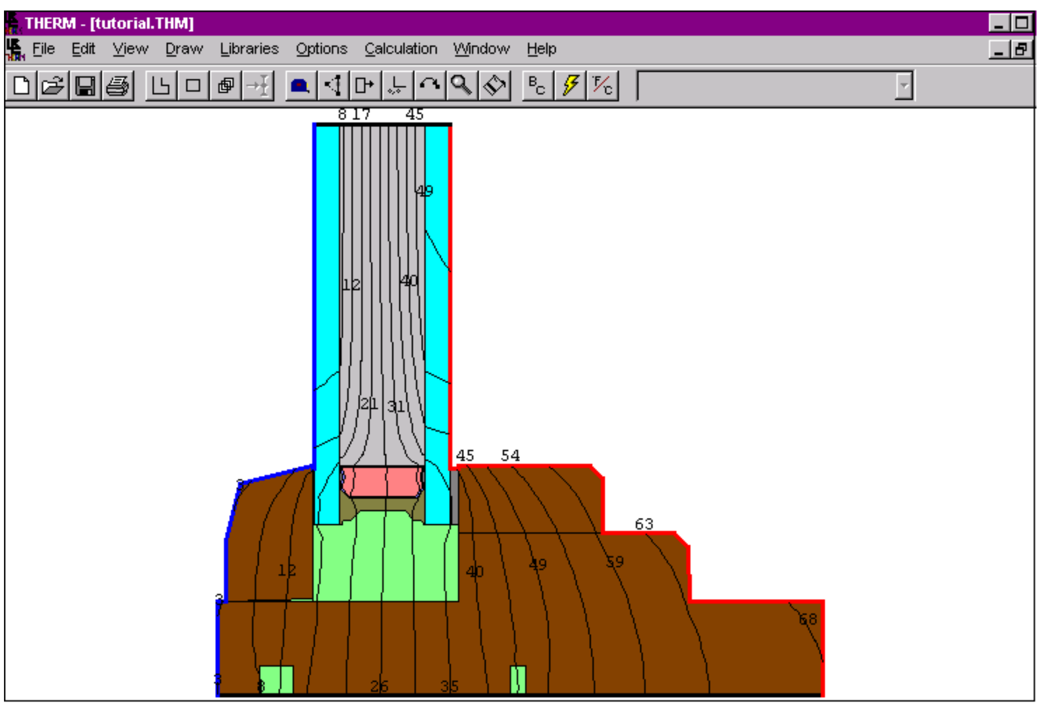


Figure 8-51. When the calculation has finished, the program will display the results as isotherms.

- You can view the U-factor results using the **Calculation/Show U-Factors** menu choice.

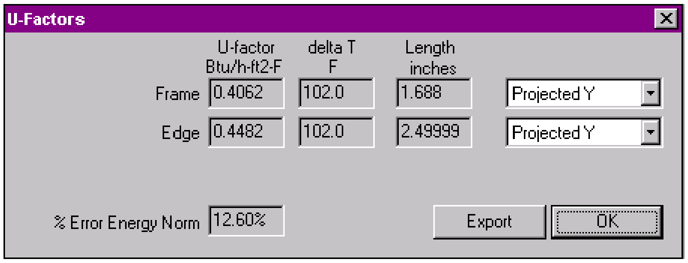


*Figure 8-52. U-Factor results, accessed from **Calculation/Show U-Factors** menu choice.*

- You can view other results in other formats using the **Calculation/Display Options** menu choice, which opens the **Results Display Options** dialog box. For more detailed information about the types of results available from THERM, see Chapter 7, "Results."

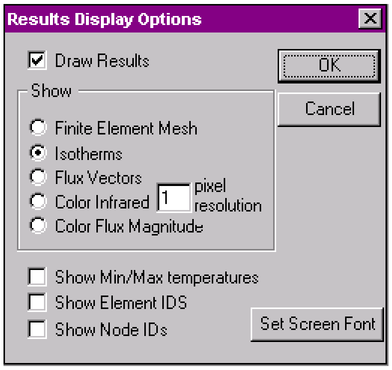


*Figure 8-53. Other results can be displayed by changing the values in the **Results Display Options** dialog box, access from the **Calculation** menu, **Display Options** choice.*

9. REFERENCES

1. Arasteh, D. K. Finlayson, E. U., and Huizenga, C. 1994. "WINDOW 4.1: A PC Program for Analyzing Window Thermal Performance in Accordance with Standard NFRC Procedures." LBL Report 35298. Berkeley, CA.

2. Finlayson, E. U., Arasteh, D. K., Huizenga, C., Rubin, and M. D., Reilly, M. S., 1993. "WINDOW 4.0: Documentation of Calculation Procedures." LBL Report 33943. Berkeley CA.

3. Enermodal Engineering Ltd. 1991. "FRAME™ -- A Computer Program to Evaluate the Thermal Performance of Window Frame Systems, Version 3.0." Waterloo, Ontario, Canada.

4. Finlayson, E. U. , Arasteh, D. K., Rubin, M. D., Sadlier, J, Sullivan, R., Huizenga, C., Curcija, D., and Beall, M. 1995. "Advancements in Thermal and Optical Simulations of Fenestration Systems: The Development of WINDOW 5." *Proceedings of the Thermal Envelopes Conference VI*, Clearwater, Florida.

5. Baehmann, P. L. Wittchen, S.L., Shephard, M.S., Grice, K.R. and Yerry, M.A., 1987. "Robust, Geometrically Based, Automatic Two-Dimensional Mesh Generation." *International Journal for Numerical Methods in Engineering*. 24, 1043 - 1078..

6. Shapiro, A. B. 1986. "TOPAZ2D - A Two-Dimensional Finite Element Code for Heat Transfer Analysis, Electrostatic, and Magnetostatic Problems" Lawrence Livermore National Laboratory, Rept. UCID-20824.

7. Shapiro, A. B. 1990. "TOPAZ2D Heat Transfer Code Users Manual and Thermal Property Data Base", Lawrence Livermore National Laboratory, Rept. UCRL-ID-104558, May 1990.

8. Zienkiewicz, O. C.; and Taylor, R. L. 1989. *The Finite Element Method*. 4th ed. Vol. 1. McGraw Hill, Maidenhead, UK.

9. Pepper, P. W. and Heinrich, J. C. 1992. *The Finite Element Method Basic Concepts and Applications*. Hemisphere Publishing Corporation, Washington.

10. Curcija, D.; Power, J. P.; and Goss, W. P. 1995. "CONRAD: A Finite Element Method Based Computer Program Module for Analyzing 2-D Conductive and Radiative Heat Transfer in Fenestration Systems". Draft Report, University of Massachusetts at Amherst.

11. Rohsenow, W. M., Harnett, J. P., and Ganic, E. N., 1985. *Handbook of Heat Transfer Fundamentals*, 2nd Edition, McGraw Hill.

12. Shapiro, A. B., 1983. FACET – A Radiation View Factor Computer Code for Axisymetric, 2D Planar, and 3D Geometries with Shadowing. Lawrence Livermore National Laboratory, Rept. UCID-19887.

13. Sullivan, H. F., Wright, J. L., and Fraser, R. A., 1996, Overview of a Project to Determine the Surface Temperatures of Insulated Glazing Units: Thermographic Measurements and Two-Dimensional Simulation. PP 516-522, *ASHRAE Transactions*, Vol 102, Part 2.

14. Zhao, Y., Curcija, D., and Goss, W. P., 1996, Condensation Resistance Validation Project - Detailed Computer Simulations Using Finite - Element Methods. PP 508-515, *ASHRAE Transactions*, Vol 102, Part 2.

15. Curcija; D., and Goss, W. P. The 'Variable-h' Model For Improved Prediction of Surface Temperatures in Fenestration Systems, DRAFT June 1998 University of Massachusetts Report.

16. ASHRAE Standard 142P, Standard Method for Determining and Expressing the Heat Transfer and Total Optical Properties of Fenestration Products, Public Review Draft, October 1996, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, GA.

17. *1997 ASHRAE Handbook of Fundamentals*, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, GA.

18. Zienkiewicz, O.C. and Zhu, J.Z. 1992a. "The Superconvergent Patch Recovery and A Posteriori Error Estimates. Part 1: The Recovery Technique." *International Journal for Numerical Methods in Engineering.* Vol 33. pp. 1331-1364.

19. Zienkiewicz, O.C. and Zhu, J.Z., 1992b. "The superconvergent patch recovery and a posteriori error estimates. Part 2. The error estimates and adaptivity." *International Journal for Numerical Methods in Engineering.* Vol 33. pp. 1365-1382.

10. ACKNOWLEDGEMENTS

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technology, State and Community Programs, Office of Building Systems of the U.S. Department of Energy under Contract No. DE-AC03-76SF00098. The program authors wish to thank the window manufacturers who supplied data on their products for use in the program; Ken Nittler, Jeff Baker, and Ross DePaola of WestLab for their review of pre-release versions; Mark Shephard, Mark Beall, and Scientific Computation Research Center (SCOREC) at Rensselaer Polytechnic Institute for their contribution to the mesh generation and error estimation algorithms in the program; Art Shapiro of Lawrence Livermore National Laboratory for his help in modifying TOPAZ2D for use with THERM; Brent Griffith and Christian Kohler for debugging and testing the program; Nan Wishner for her time and effort editing this document; and Pat Ross for all her many daily contributions including the final production of the program and manual.

Appendix A: Glossary of Terms

Adiabatic

A boundary condition that assumes a perfectly insulated or a symmetric condition.

Adjacent polygon

A polygon that shares a common side with another polygon.

ASHRAE

The American Society for Heating, Refrigerating and Air-Conditioning Engineers, Inc.

ASHRAE summer conditions

Environmental conditions, used to determine Solar Heat Gain boundary conditions, with the following characteristics:

- Exterior Temperature: 31.7°C
- Room Temperature: 23.9°C
- Wind Speed on Exterior Surface: 3.4 m / sec, windward
- Direct Solar Radiation: 783.0 W / m2

ASHRAE winter conditions

Environmental conditions, used to determine U-factor boundary conditions, with the following characteristics:

- Exterior Temperature: –17.8°C,
- Room Temperature: 21.1°C,
- Wind Speed on Exterior Surface: 6.7 m / sec, windward
- Direct Solar Radiation: 0.0 W / m2.

Aspect ratio

The dimension perpendicular to the heat flow (vertical for horizontal heat flow) divided by the dimension parallel to the heat flow (horizontal for horizontal heat flow). This value is used to determine the effective conductivity in frame cavities.

Assembly drawings

Drawings that show many separate parts put together. These drawings often show deformable pieces that overlap. When creating a THERM model from an assembly drawing the user must turn these overlapping pieces into polygons with no voids or overlaps.

Autoconvert

An option when importing DXF files into THERM that allows closed polylines and curves to be turned into polygons.

AutoLISP programs

Programs created to automate commands in AutoCAD ™ that can be used to improve the applicability of a DXF file for the THERM autoconvert feature.

Bad Points

Two points that are closer than 0.1 mm are considered "bad points", because they should actually be the same point. These bad points are marked with red circles when the boundary conditions are drawn. If they are defects in the drawing they should be fixed, but if they are areas of fine detail they can be ignored. The bad points can be cleared using the **Draw/Clear Bad Points** menu choice. View the red circles is controlled using the **View/Bad Points** menu choice.

Bitmap file

An file format that THERM can use as an underlay to be traced. Bitmap files do not contain information about lines and vertices, so THERM cannot snap to vertices as it can with a DXF underlay. Bitmap files sometimes distort the dimensions of the cross section and should be used with care.

Black Body Radiation

Electro-magnetic radiation governed by the Stefan Boltzman law:

$$q = \sigma T^4$$

The sun is a source of black body radiation. The emissivity of a surface is a measure of how a surface compares to a black body, which is defined with an emissivity of 1.0.

Blocking surfaces

When calculating radiation heat transfer between the model and a radiation enclosure, THERM automatically calculates the view factor between all surfaces. The view factor calculation takes into account which surfaces block the view between other surfaces. By default, all surfaces of a radiation enclosure are blocking surfaces, meaning that they can *potentially* block the view between two other surfaces. Optionally, the blocking surface option can be turned off for any surface. Turning off the blocking surface option will speed the view factor calculation process, but should only be done for surfaces that will never block the view between two other surfaces.

Boundary conditions

The properties and/or relationships that define the edges of the model are called boundary conditions. For the steady state energy equation solved in THERM, the boundary conditions either specify the heat flux and/or the temperature. Environmental conditions, film coefficients, surface emissivities, radiation view factors, and thermal conductivities are used to determine the boundary conditions of the model. The default boundary condition in THERM is an adiabatic or perfectly insulating boundary.

Boundary segment

A boundary segment is the line segment between any two points on the boundary (or edge) of the model. Boundary segments must be assigned boundary conditions and may be tagged to be included in the U-factor calculations.

CAD

Computer Aided Design. A category of computer drafting programs that can create a DXF file that can be used as an underlay in THERM.

Cavities

There are two types of cavities mentioned in the THERM manual. A glazing cavity is the gas filled gap between glazing layers in a multilayer glazing system. A frame cavity is an air filled cavity that occurs in window frames, especially in extruded vinyl and aluminum frames.

Cavity height

The total height of the glazing system cavity, including two edge regions and a center of glass region. This value is a necessary input for the Condensation Index calculation.

Cavity Model

The cavity model refers to one of several options for modeling frame cavities in THERM. The various cavity models have different methods of approximating non-rectangular cavity geometry, effective conductivity and radiative heat transfer.

CEN

The standard body of the European Community analogous to ASTM.

CEN radiation model

The simplified radiation model used with the CEN frame cavity models. This model approximates a frame cavity as a rectangular cavity with isothermal walls. The radiative conductance (hr) uses a view factor (F) for a rectangular section and a linearized Stephan Boltzman law:

$$h_r = E \cdot F \cdot 4\sigma \cdot T_{avg}^3$$

$$E = (\varepsilon_1^{-1} + \varepsilon_2^{-1} - 1)^{-1}$$

$$F = \frac{1}{2}\left(1 + \sqrt{1 + (L/H)^2} - L/H\right)$$

Where T is Temperature, ε is emissivity, L is the cavity dimension in the direction parallel to the heat flux and H is the cavity dimension in the direction perpendicular to the heat flux.

Center-of-glass

The region in a glazing system where the frame and spacer effects are negligible and one-dimensional heat transfer occurs. The thermal properties for the center-of-glass region are determined in the WINDOW program.

Condensation Index

Refers to the calculation procedure that uses local film coefficients and the radiation enclosure model to obtain local temperatures that are more accurate than those obtained using the effective conductivity method for glazing systems.

Conduction

Heat transfer resulting from a temperature difference between adjacent objects governed by the thermal conductivity.

Conductivity

A measure of the rate at which heat flows through a material. Materials that are good conductors of electricity (i.e. metals) are usually good conductors of thermal energy as well and have a high thermal conductivity. An insulating material is a poor conductor of thermal energy and has a low thermal conductivity. Units are Btu / hr-ft-F, Btu-in / hr-ft^2-F or W / m-K.

Constant heat flux magnitude

Heat flux is vector quantity, meaning that the heat flux at any point in the model has both a magnitude and a direction. The Constant Heat Flux Magnitude result only indicates the magnitude of the heat flux, which is useful in identifying thermal bridges and evaluating thermal breaks.

Convection

Heat transfer resulting from gas movement caused either by a temperature gradient or by an incident wind.

Convection correlations

Convection heat transfer is treated approximately in THERM using correlations developed from experimental and computational research.

These correlations are used to determine the effective conductivity in frame cavities. They provide results that are a function of the thermal properties, geometry, and temperature difference in the cavity.

Cross section geometry

A two dimensional view cut through a three dimensional object, represented in THERM by polygons. This geometry is derived from a DXF file or a dimensioned drawing.

Cursor sticky distance

The area of influence of the cursor relative to a vertex. If the mouse is clicked when the drawing cursor is within this distance from a vertex, the cursor will "stick" to the vertex. This helps prevent voids and overlaps from being unintentionally created in the drawing. The sticky distance is 3mm at 100% scale, which is about half the length of one side of the drawing cross hair. This distance is fixed, so as you zoom in, it becomes smaller with respect to the detail of the drawing. If you are having trouble with the program snapping to the wrong vertices, try zooming in on the area of interest which effectively reduces the sticky distance.

Curve fit model

The simplified radiation model associated with the NFRC and User Defined frame cavity models that uses the length (L), height (H) and average absolute temperature (Tavg) to determine the approximate radiative conductance (hr)[16]:

$$h_r = \exp(1.53 - 0.194 * L/H) * (T_{avg}/273)^3$$

Deformable pieces

Components of a cross section, such as glazing stops, glazing tape and weather-stripping, which will change size and shape (i.e., deform) when they are assembled into a real product. They are often drawn in their undeformed state in assembly drawings, resulting in overlapping regions. Because there can be no overlapping regions or voids in a THERM cross section, these objects should be modeled as they will actually occur in the real product rather than how they appear in the assembly drawing.

Detailed radiation model

The detailed radiation model uses an element to element view factor method (see Appendix C for more information) to determine the radiant exchange between every element in the enclosure it is applied to. The detailed radiation model accounts for the temperature and emittance of these elements.

Dialog box

A THERM program window that displays information and allows user input. For example, if the user selects the **File/Open** menu choice, a small program window, or dialog box, opens, allowing the user to choose the file to be opened.

Dimensioned drawing

A representation that includes all the dimensions needed to represent an object as a cross section in THERM.

DXF file

An export file format from a CAD program that can be used as an underlay in THERM. THERM can preferentially snap to the vertices in a DXF file and can convert the closed polylines into polygons.

DXFOUT command

The command used in a CAD program to create a DXF file.

Edge effects

The two dimensional heat transfer that occurs in the glazing system due to the thermal bridging effects of the spacer and frame.

Effective conductivity

A method that converts the convection and radiation effects of a volume of gas into a conductance that is combined with the thermal conductivity to model the gas volume as if it were a solid.

Emissivity

The ratio of the actual emission of radiation by an object to the emission of a black body at the same temperature.

Entities

An object such as a line, circle, arc or polyline treated as a single element in a CAD program in order to simplify creation, manipulation and modification of those objects.

Environmental conditions

The conditions that define the environment surrounding the model from which the boundary conditions are derived (see ASHRAE summer conditions and ASHRAE winter conditions for examples).

External Radiation Enclosure

A THERM material type used to model a space through which radiation heat exchange takes place between the model and other external surfaces. Each of these external surfaces can have its own temperature and emissivity.

Finite element mesh

A collection of non-overlapping three and four sided elements upon which the finite element analysis is performed. The mesh is generated automatically by THERM based on the geometry of the model. Subsequent adaptation of the mesh based on the heat flux in the model can be achieved using the error estimator option.

Finite-element analysis

The numerical method used in THERM to solve the two dimensional energy equation. The equation is set up on each element of the finite element mesh and then a solution for each element that minimizes the global error of the entire mesh is obtained by solving a matrix of related variables.

Floating point tolerance

The minimum distance between points in THERM, which is set at 0.01 mm. Points that are closer together than this distance are merged together.

Flux

See heat flux.

Frame cavities

See Cavities.

FRAME™ F30 file

A file created by the FRAME 3.0 program developed by Enermodal Engineering Ltd.

Glazing cavity

See Cavities.

Glazing layers

Transparent layers, such as glass and acrylic, combined with spacers and fill gases, to create glazing systems.

Glazing system

In THERM, a glazing system imported from the WINDOW4 computer program. It is a series of polygons that represent the thermal properties, heat transfer characteristics, and boundary conditions of a glazing unit. A sample WINDOW4 glazing system library is provided with THERM. Additional glazing systems can be created using the WINDOW4 program.

Heat flux

Heat flux is the vector quantity determined from the thermal conductivity and the local temperature gradient surrounding a node on the finite element mesh.

Heat transfer analysis

Heat transfer analysis is a term applied to the solution of the energy equation in its many forms. There are three main modes of heat transport: conduction, convection and radiation. All three of these modes are modeled in THERM.

Heat-flux vectors

Heat flux is a vector quantity that has a magnitude and a direction, resulting from the finite element analysis. These vectors are determined for every node in the finite element mesh. They are a function of the thermal conductivity and the local temperature gradient. The heat flux vector plot shows one heat flux vector for each element. This vector is determined by integrating the heat flux over the entire element. Its magnitude is a function of the heat flux and also the size of the element. The heat flux vector plot gives an qualitative representation of the direction of heat flux through the cross section.

Horizontal heat flow

The default assumption in frame cavity models is that heat flow is occurring largely between the two vertical sides of the cavity (or its equivalent rectangle) and thus the flow is horizontal. This heat flow is driven by a temperature difference between these two sides. Other possible models are vertical flow up and vertical flow down.

Isotherms

Lines of constant temperature.

Keff

See Effective Conductivity.

Local film coefficients

Local convective film coefficients vary as a function of position. They provide improved accuracy in the calculation of local temperatures and are used in the Condensation Index Model.

Local temperatures

Temperatures at a specific location in contrast to temperatures averaged over an area.

Material properties

The thermal conductivity and emissivity characteristics of a material. For example, the material properties for a frame cavity are the geometry, the emissivity, the thermal properties of the gas, and the temperature difference across the cavity.

Mesher

The algorithm based on the Finite Quadtree method that automatically creates the finite element mesh.

NFRC

The National Fenestration Rating Council, whose mission is to provide a fair, accurate, and credible rating system for windows and other fenestration products.

Non-contiguous polygons

Polygons that do not share a common edge.

Non-planar surfaces

A surface that has dimensions in more than one plane, i.e., the horizontal and the vertical plane. For example, a greenhouse window has non-planar surfaces.

Nusselt number

The Nusselt number is the ratio of convective to conductive heat transfer. A Nusselt number of 1.0 is an indication of still air.

Overlapping regions

A situation caused when part of one polygon lies on top of another polygon. THERM cannot perform the calculation if this situation exists, and the program tries to prevent it during the drawing process. Sometimes the check for overlapping regions is overly cautious and a warning comes even when an overlap doesn't exist. If this happen you can turn off the "always check for overlapping polygons" option. In general it is best to keep this option turned on when drawing. After the model is completed and the boundary conditions are drawn the program will identify any overlapping regions, which must be corrected before the model can be simulated.

Polygon

A closed shape made up of a minimum of three sides. In THERM, a polygon cannot intersect itself; therefore shapes such as "figure eights" and donuts are not acceptable polygons.

Polylines

A term used in CAD programs to describe a series of line segments connected into a continuous line. A closed polyline in a DXF file can be autoconverted into a polygon by THERM.

Projected length

The dimension obtained by projecting the frame profile on either a horizontal (Projected X) or vertical plane (Projected Y). This value is used as a basis for the U-factor, which is the heat flow per unit temperature difference per unit projected length.

Radiation

The electromagnetic radiation modeled in THERM is the long wavelength thermal radiation that is emitted by opaque bodies at temperatures greater than absolute zero.

Radiation blocking surface

See blocking surface.

Radiation heat transfer

The net energy transfer between two bodies that are radiating to each other. The amount of radiation heat transfer between two bodies depends on the surface emissivities, the radiation view factors and the absolute temperatures of the bodies.

Radiation Model

There are three radiation heat transfer models used in THERM. The detailed radiation model, the curve-fit radiation model and the CEN radiation model. See glossary listings for information on each of these models.

Radiation view-factor

The radiation view-factor is a measure of how much one surface sees of another. A view-factor of 1 means that one surface sees only another surface and nothing else. A view-factor of 0 means that one surface does not see another. Partial viewing will result in a view factor that falls between these extremes. The radiation view factor is calculated by THERM using the cross string method.

Self viewing

The ability of a non-planar product to radiatively see itself. If there is a significant temperature gradient along the surface the radiation heat transfer caused by self-viewing can have a marked effect on the overall heat transfer in the cross section. This is especially true in projecting fenestration products such as greenhouse windows.

Sight line

The point on the room side of the glazing system corresponding to the highest point on the window frame (either exterior or room side). This point is used to determine whether boundary segments on a glazing system are tagged as edge or frame for the U-factor calculation. The height of the edge region is measured from the sight line.

Simulation

The simulation process involves everything necessary for solving the energy equation , including mesh generation, finite element analysis, error estimation, and mesh refinement.

Snap-in pieces

Parts of an assembly that are "snapped" into a main extrusion to create separate cross sections. For example, a window frame head and sill might have a single main extrusion and may differ only in a few snap-in pieces. In THERM the main extrusion could be modeled once, and then cut and paste could be used to model both the head and the sill.

Solar gain

The thermal energy that enters a room through transmission, absorption, and subsequent radiation of solar energy.

Solar Heat Gain Inside Film

The boundary condition that is imported with a glazing system from WINDOW4 corresponding to ASHRAE summer conditions for application to the room side boundary of the glazing system.

Solar Heat Gain Outside Film

The boundary condition that is imported with a glazing system from WINDOW4 corresponding to ASHRAE summer conditions for application to the exterior boundary of the glazing system.

Solid material

An opaque material that is fully defined for analysis purposes by a thermal conductivity and emissivity.

Spacer height

The distance from the top of the spacer to the bottom of the glazing. The height determines the height of the glazing cavity in a glazing system that is imported from WINDOW4. A rectangle the height of the spacer height and a width of the glazing cavity will be left empty when the glazing system is imported. THERM can automatically fill that rectangle with a solid material to approximate a spacer if the **Draw spacer** check box is checked when importing the glazing system.

Step size function

Used in conjunction with the arrow keys to move the cursor a specified amount. Typing in a numeric value defines the step size. Holding down the control key causes the step size to be temporarily decreased by a factor of 10.

Sticky distance

See cursor sticky distance.

Temperature difference

The difference in temperature between the warm and cold sides of a cross section. The temperature difference is the driving force behind conduction, natural convection, and radiation heat transfer.

Thermal bridges

Materials of high thermal conductivity that connect a cold area with a warm area, facilitating heat transfer between these areas.

Thermal conductivity

See Conductivity.

Two-dimensional (2D) conduction heat transfer analysis

An analysis assumption that energy is transported in the horizontal (x) and vertical (y) directions but that nothing is driving the heat flow in the transverse (z) direction.

U-factor

The U-factor is a measure of the heat transfer characteristics of a cross section under specific environmental conditions. In THERM, the U-factor calculation integrates the heat flux over the tagged boundary segment or segments, divides that flux by the projected length of the segment and the defined temperature difference, and returns a U-factor.

U-factor Inside Film

The boundary condition that is imported with a glazing system from WINDOW4 corresponding to ASHRAE winter conditions for application to the room side boundary of the glazing system.

U-factor Outside Film

The boundary condition that is imported with a glazing system from WINDOW4 corresponding to ASHRAE winter conditions for application to the exterior side boundary of the glazing system.

U-factor tags

Labels given to boundary segments over which the heat flow will be integrated in order to obtain U-factors. These boundary segments need not be contiguous nor do they need to be defined with the same boundary condition.

Underlay

A DXF file or a Bitmap file that can be traced over to create the cross-section geometry in THERM.

Vertex (Vertices)

A vertex is a point where two line segments come together in a DXF file or a point on a polygon in a THERM cross section.

Void

An undefined space created between two polygons with unmatched edges. After the model is completed and the boundary conditions are drawn the program will identify any voids. These must be corrected before the model can be simulated.

WINDOW4

A software program developed by the Lawrence Berkeley National Laboratory to calculate center of glass and total window thermal performance indices.

WINDOW4 glass library

The database of glazing layers used to create glazing systems in the WINDOW4 program. This library contains information on the thickness, optical properties, surface coatings and thermal conductivity of the glazing layers.

Appendix B: Error Messages

The following is a partial list of THERM warning and error messages, their probable causes, and possible solutions. Updated information on error messages can be found at http://windows.lbl.gov/software/software.html.

Array overflow in Mesher. Try reducing the Quad Tree Mesh Parameter in Therm File Options.

Cause: A very detailed mesh can exceed the limits of the automatic mesh generator.

Solution: Reduce the Quad Tree Mesh Parameter value under **Options/Therm File Options**. If you are getting this error message because you increased the mesh parameter in response to another message telling you to "simplify the geometry or increase the mesh parameter", you should reduce the Quad Tree Mesh Parameter and then simplify the geometry. Often very minor simplifications in the geometry in areas of fine detail will result in the mesher being able to generate a mesh.

Calc Manager is Paused. The file won't be run until you start it. Do you want to start it?

Cause: The Calc Manager has been set to pause the file on which you are now trying to do a calculation.

Solution: Unpause it in the Calc Manager, or click on the **Yes** button to this message. If you do not want the file to be calculated, click on either the **No** or **Cancel** buttons.

Calculation not successful

Cause: This error message can have many possible causes.

Solution: Read the calculation log by going to the **Calc Manager** option of the Calc menu.

Can't delete a point from a 3-sided polygon.

Cause: Deleting a point from a 3-sided polygon will create an invalid polygon.

Solution: This error is often caused by the cursor sticking to a different point than you intended. Try zooming in on the area of interest.

Can't find glass library (glass.dat) in the glazing system directory. All glass surface emittances will default to 0.84.

Cause: Emittance data is read from the WINDOW4 library file "glass.dat". If THERM does not find this file, it will default all glass emittances to 0.84.

Solution: Copy the file glass.dat to the same folder as the glazing system library file glzsys.w4. You only need to do this if you are using the condensation index or the radiation enclosure model.

Can't insert a point on top of another point. Try zooming in.

Cause: You are trying to insert a point that is too close to an adjoining point (within 0.01 mm).

Solution: Use the Zoom feature (click the right mouse button) one or more times to enlarge the area enough that you can insert the new point at a distance greater than 0.01 mm from the adjacent point.

Condensation Index Model currently does not support user defined gas types. Glazing Cavity has a cavity with gas type

Cause: If you import a glazing system from WINDOW that has a user-defined gas (i.e. something other than air, argon or krypton), you will not be able to use the Condensation Index Model with this version of THERM.

Error: Autofill created an invalid polygon.

Cause: In rare cases, the autofill feature can't resolve the geometry and will not be able to fill the cavity.

Solution: Either draw the polygon by hand or try resolving any fine detail in the surrounding polygons. This often occurs when there are small gaps in a cavity. Occasionally it occurs when there is detail less than 0.01 mm apart.

Error: There is a radiation enclosure that does not have any boundary conditions on the model with enclosure radiation enabled. To change this, the boundary conditions must be changed to "comprehensive" and the enclosure radiation model must be checked. Enclosure Polygon ID=

Cause: Every boundary condition that faces a radiation enclosure must have the enclosure radiation model enabled.

Solution: Edit the boundary condition, make sure the **Comprehensive** model is selected and check the **Radiation Enclosure Model** check box. The boundary conditions that come in with an imported glazing system from WINDOW are not compatible with the radiation enclosure model.

Invalid polygon found, stopping simulation. Polygon ID=

Cause: An invalid polygon has been found by THERM, preventing the calculation from continuing. The error message will list the polygon ID # of the offending polygon.

Solution: Using the **Edit/Select Special** menu choice, you can locate the offending polygon and either correct the problem, or delete the polygon and redraw it. Invalid polygons are usually created as a result of the point merging algorithms in THERM. Automatic adjustment of bad points can result in invalid polygons. The most common invalid polygon has a zero area extension. These extension have two points on top of one another. In order to fix this kind of invalid polygon you must delete one of these points as well as the end of the extension.

Library names must be unique.

Cause: When defining a new library entry, this error message indicates that you have used a name that is already in use.

Solution: Use a different name for the new entry that isn't already being used.

Maximum number of cavities (9) is exceeded

Cause: A cross-section geometry cannot have more than 9 glazing cavities.

Solution: Reduce the number of glazing cavities in the cross section, possibly by combining adjacent cavities.

Mesh generation error. Try simplifying the geometry near point (or try increasing Quad Tree Mesh Parameter in THERM File Options.

Cause: The finite element mesh generator has encountered problems generating the mesh in the geometry because the cross section contains too much detail.

Solution: Simplify the geometry by reducing the number of points in polygons or combining similar polygons. In addition, you can try increasing the value of the Quad Tree Mesh Parameter in the Options/Therm File Options menu. The maximum value allowed is 11.

Model geometry and Boundary Conditions need to be properly defined before a calculation can be performed.

Cause: If boundary conditions have not been defined for a geometry, you will get this error message when trying to do a calculation. This can occur if you do not regenerate the boundary condition after fixing voids and overlaps.

Solution: Generate the boundary conditions before doing a calculation.

Overlapping regions are not allowed!

Cause: THERM cannot simulate a cross section that has polygons which overlap adjacent polygons. As you are drawing, if the program detects that you are about to generate a polygon which overlaps another polygon, it will display this error message.

Solution: Do not try to draw a polygon which overlaps another polygon. It may be necessary to zoom in on the area you are drawing in order to avoid potential overlaps. Sometimes the algorithm for checking for overlaps thinks you are creating an overlap when you really are not. If you are getting this message you can turn off the checking under **Options/Preferences/Drawing**

Options/Always check for overlapping polygons. In general it is good practice to keep this feature turned on.

Please enter a positive number for the step size, or choose absolute coordinate mode.

Cause: You can only enter positive numbers in the **Step Size** dialog box, if you are entering **Relative** values.

Solution: Enter a positive number and use the arrow keys in the direction opposite to the one you want to go in, or change the **Step Size** setting to **Absolute** in order to enter a negative number.

Points of a polygon cannot overlap. Try Zooming in for detailed work.

Cause: You are working at too low of a zoom level and the point you are trying to insert is snapping back to an existing point. This would create two points on top of each other and THERM does not allow this.

Solution: Zoom in, effectively reducing the sticky distance. This can cause a problem if you are working with WINDOWS 95 which has limited zooming abilities. If this is the case try drawing everything around this polygon and create the polygon using the fill void feature.

Polygon already exists at selected point

Cause: You are trying to use the fill void feature and the cavity is already filled.

Solution: This can happen when you are filling cavities with the material colors turned off so turn on the colors and you will be able to tell if the cavity is filled. It can also happen if you are zoomed out and the cursor in not located within the cavity that you want to fill, in which case try zooming in.

Solution failed to converge. Try changing relaxation parameter or decrease convergence tolerance.

Cause: The comprehensive radiation model is an iterative solution and the convergence criterion is not being met.

Solution: Make the convergence criterion **OPTIONS/Preferences/ Simulation** larger. The maximum allowable value in 0.01.

The Glazing system can't be inserted because it overlaps with existing materials. Try repositioning the Glazing System Locator or adjacent materials.

Cause: THERM will not allow polygons to overlap one another. This error message is issued when the glazing system you are trying to insert will overlap with other polygons in the cross section.

Solution: One possible solution is to move the Locator, which determines where the lower left corner of the glazing system will be inserted. You may need to Zoom in to the drawing in order to see accurately where the Locator is currently positioned. If changing the Locator position does not solve the problem, it may be necessary to delete some polygons adjacent to the glazing system (such as those representing the glazing tape), insert the glazing system, and then redraw the polygons once the glazing system is in place. (Glazing tape is specified in its undeformed state, but it will deform when a real glazing system is inserted into a window. Drawing the polygons representing the glazing tape in their compressed state is a more realistic model.) Note that THERM allows you to specify the width of the glazing system to four decimal points in IP units but WINDOW only specifies it to three, so measuring the glazing cavities in SI units (reported to three decimal places in THERM) may help the glazing system fit. Since the match has to be exact this difference can cause the glazing system not to fit. You can try pasting the glazing system outside of the cross section and then cutting and pasting it into location. This may cause overlapping regions but they can be corrected.

There are materials that lie outside of the Boundary Conditions

Cause: This is caused by noncontiguous polygons.

Solution: THERM can only analyze one continuous cross section at a time. You will have to delete any other polygons. The major cause for this error is very small cracks in the model that are open to the environment.

There are no Ufactor tags defined. Simulate anyway?

Cause: This is a warning based on the assumption that most people are going to be using THERM to determine U-factors

Solution: If you want to obtain U-factor calculations from THERM, you must define the U-factor Surface Tags when defining the cross-section boundary conditions. If you do not, you can ignore this message.

This file is currently being simulated. The changes you have made may effect the results, so the calculation is being stopped.

Cause: If you make a change to a file that is currently being calculated, the program will issue this error message.

Solution: Cancel the calculation (using the Calc Manager), save the changes to the file, and redo that calculation.

Trouble loading library file.

Cause: When loading one of the libraries (material.lib, bc.lib, or ufactor.lib) the program either couldn’t find the file or couldn't read it.

Solution: Make sure all the libraries are in the THERM working directory. If they are there, they may be corrupted. You can reinstall the default libraries from the installation diskettes.

Trouble reading Bitmap file.

Cause: If THERM can not read a bitmap file that you are bringing in as an underlay, this message will be displayed, due to a file format incompatibility.

Solution: Try generating the bitmap file with another drawing program.

Trouble reading DXF file.

Cause: If THERM can not read a DXF file that you are bringing in as an underlay, this message will be displayed, due to a file format incompatibility.

Solution: Try generating the DXF file with another drawing program, or in a different manner.

Trouble with Auto BC Generation near point:

Cause: Very fine detail that the program cannot resolve.

Solution: Make minor adjustments to the geometry to avoid details less than 0.01 mm.

Trouble with Auto Fill near point:

Cause: Very fine detail that the program cannot resolve.

Solution: Make minor adjustments to the geometry to avoid details less than 0.01 mm.

Trouble with geometry at segment

Cause: Very fine detail that the program cannot resolve.

Solution: Make minor adjustments to the geometry to avoid details less than 0.01 mm.

U-factors are not valid for exporting to Window 4. Check the U-factor tags and U-factors.

Cause: U-factors other than Frame and Edge are included in the model

Solution: Only Frame and Edge U-factors can be imported into WINDOW.

Unable to fix some trouble with detailed geometry. The problem areas are circled in red.

Cause: Automatic adjustment of bad points failed.

Solution: Make sure to save the file before you perform the automatic adjustment of points because this command cannot be undone. If you have saved the file, when you see this error message redo the boundary conditions but don't adjust any bad points. If you have not saved the file you will have to fix the geometry that THERM has created.

Warning: The maximum number of meshing iterations reached before the error target was reached.

Cause: Too low of a percent error energy norm target value for the number of iterations specified.

Solution: Check the results of interest to make sure that the level of the percent error energy norm you have specified is required. If it is, increase the number of iterations. You can also reduce the number of iterations required by starting with a finer mesh (higher mesh control parameter).

Warning: Can't find matching glass types for all glazing systems in the glass.dat library. Emittances for these glass types will be defaulted.

Cause: Emittance data is read from the WINDOW4 library file "glass.dat". If THERM can not find all of the glazings in a glazing system in this library, it will default all glass emittances to 0.84.

Solution: Add the glazing layer to the glass library within the WINDOW program. You only need to do this if you are using the condensation index or the radiation enclosure model.

Warning: There are Materials and/or Boundary Conditions used by this file that were not stored in the file and are not in the Libraries. These will display with ""??"" after the name when they are selected and you need to redefine them before simulating.

Cause: This sometimes occurs when you are importing files into THERM that were created in previous versions.

Solution: Unfortunately you will have to redefine the material properties and boundary conditions.

You can't close a file while calculations are being performed. Do you want to stop the calculations and close the file?

Cause: You have tried to close a file that is currently being simulated.

Solution: Wait until the calculation has finished to close the file, or allow the program to stop the calculation and close the file.

You need to calculate results first.

Cause: You have tried to view the U-factors for a cross section without simulation results.

Solution: Calculate the results before trying to view the U-factors. The **Calculation/Display Results** is grayed out if the results have not been calculated for a model. If a file has been saved without the results (the **Save Simulation Results in THM Files** box is *not* checked in the **Options/Preferences** menu choice, **Simulation** tab), the **Calculation/Display**

Options menu will be grayed out, but the **Calculation/Show U-factors** will be available.

You need to have at least two non-adiabatic boundary conditions to do a simulation. Check your boundary condition definitions.

Cause: The cross section does not have two non-adiabatic boundary conditions.

Solution: The default boundary condition for all boundary segments, except glazing systems imported from WINDOW4, is Adiabatic. You need to assign realistic boundary conditions to the boundary segments before the program can proceed with the calculation.

You need to select a Polygon before doing this

Cause: Trying to add a point to an unselected polygon

Solution: Select the polygon before trying to add a point. A selected polygon has squares on the vertices.

Appendix C: Theoretical Background

This manual describes how to specify the geometry, material properties, and boundary conditions required by THERM. Once those have been defined by the user, THERM automatically, meshes the cross section, performs the heat transfer analysis, runs an error estimation, refines the mesh if necessary and finally returns the converged solution. This appendix describes the technical algorithms behind each of these functions in detail.

AUTOMATIC MESH GENERATION

The first step in a finite-element analysis is the definition of a mesh. This mesh is made up of a finite number of non-overlapping subregions, known as elements, that cover the whole region under analysis. The manual creation of a well-conditioned finite-element mesh requires sophisticated knowledge of the solution method. This labor-intensive component of the finite-element analysis is accomplished for the user in THERM by the automatic meshing algorithm, Finite Quadtree.[5] The Finite Quadtree mesh generator is based on a spatial decomposition procedure that represents the domain of an object as a set of non-overlapping squares, referred to as quadrants, that are stored in a hierarchic tree. The object to be meshed is placed in a square universe that entirely encloses it. The square represents the “parent” quadrant; it is subdivided into four, “daughter”, quadrants. The remainder of the tree is then defined in a recursive manner by subdividing the boundary and interior quadrants until all quadrants are at the level dictated by the geometry of the object. The subdivisions continue until each quadrant contains only one material. The mesh control parameter determines the minimum number of subdivisions preformed by Quadtree. The higher the mesh control parameter the smaller the maximum size of the finite element in the model. After the quadtree has been defined, all terminal quadrants are checked to ensure that only one tree level difference exists between them and their neighbors so that there is not a large size difference between adjacent finite elements in the final mesh. The final step in the algorithm is to convert the quadtree into a combination of well shaped quadrilaterals and triangles. For analysis purposes all elements are treated as quadrilaterals. A triangle is defined as a quadrilateral element in which two of the nodes are coincident. Once the mesh is created it is sent to the finite element solver.

FINITE ELEMENT SOLUTION

The finite element solver in THERM, Conrad, is derived from the public-domain computer programs TOPAZ2D and FACET[6,7,12]. The governing equation for two-dimensional heat conduction, under the assumption of constant physical properties, is derived from the general energy equation and is given by the partial differential equation shown in Eq. 1.

$$k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+q_g=0 \tag{1}$$

where,

q_g = internal heat generation

subject to the following set of boundary conditions:

$q_f = 0$ adiabatic boundary condition

$T = f(x, y)$ Temperature boundary condition

$q_f = \overline{q}$ known heat flux, $\overline{q}$, boundary condition

$q_c = h(T - T_\infty)$ convection / linearized radiation boundary condition,

$q_r = \varepsilon_i \sigma T_i^{\,4} - \alpha_i H_i$ radiation boundary condition.

For radiation boundary conditions, THERM assumes that radiating surface segments are gray and isothermal, so absorptance equals emittance, or $\alpha = \varepsilon$. **H** is irradiation on the surface. Equation 2 shows the value of **H** for a surface **i**.

$$H_i = \frac{1}{1-\varepsilon_i}\left(B_i - \varepsilon_i \sigma T_i^{\,4}\right) \tag{2}$$

where,

B_i = radiosity of the surface "i"

B_j = radiosity of the surface "j" that views surface "i"

$$B_i = \varepsilon_i \sigma T_i^{\,4} + (1-\varepsilon_i)\sum_{j=1}^{n} F_{ij} B_j \,. \tag{3}$$

Equation 3 represents a system of **n** linear algebraic equations which is solved for $\mathbf{B_i}$, and substituted in Eq. 2.

View factors are calculated using the "cross-string" rule which is illustrated in Figure C-1 and given by Eq. 4.

$$F_{ij} = \frac{r_{12} + r_{21} - (r_{11} + r_{22})}{2L_i} \tag{4}$$

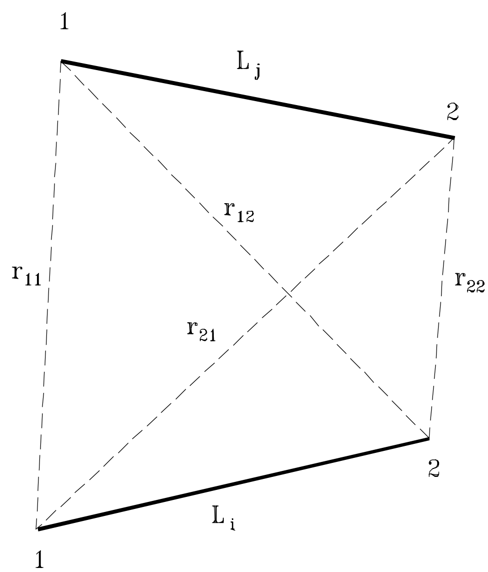


Figure C-1. Cross-String Rule

When partial, or third-surface shadowing exists, the two radiating surfaces are subdivided into **n** finite subsurfaces and contributions to the summation in Eq. 5 of those subsurfaces in which ray $\mathbf{r}_{kl}$ intersects a shadowing surface are excluded, as shown in Figure C-2.

$$F_{ij} = \sum_{k=1}^{n} \sum_{l=1}^{n} F_{kl} \tag{5}$$

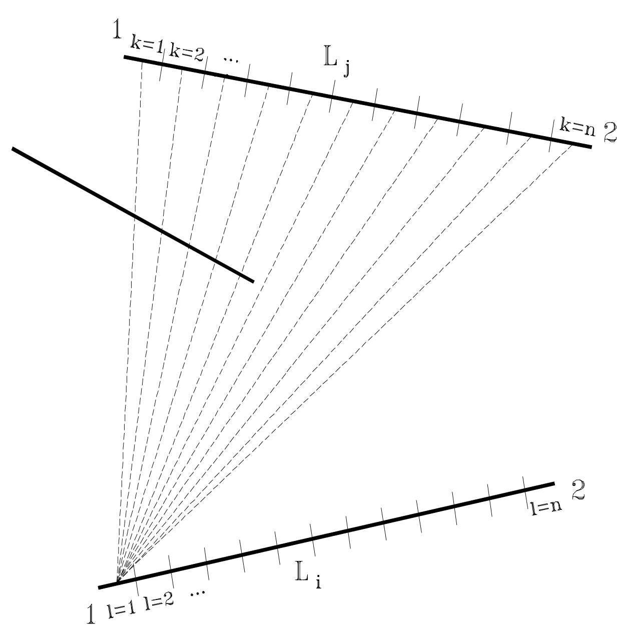


Figure C-2. Third-Surface Shadowing

The magnitude of the heat flux vector normal to the boundary, $\mathbf{q} = \mathbf{q_f} + \mathbf{q_c} + \mathbf{q}_r$ is given by Fourier's law, shown in Eq. 6.

$$q_f + q_c + q_r = -k\left(\frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y\right) \tag{6}$$

THERM uses finite-element analysis based on the method of weighted residuals. The method of weighted residuals seeks to solve an equation of the form shown in Equation 7.

$$\iint_\Omega W\Re dxdy=0 \tag{7}$$

The residual function, $\Re$, results when an approximation for the actual temperature field is substituted in Eq. 1. The residual function is zero for the exact temperature field. The weighted residual method does not seek to force the residual function to zero everywhere in the domain; instead, the residual function is multiplied by a weighting factor, W, forcing the integral of the weighted expression to zero and minimizing the error of the approximate solution. This is accomplished by first integrating Eq. 6 by parts and substituting in the boundary conditions as shown in Eq. 8.

$$\begin{aligned} k\iint_\Omega\left(\frac{\partial W}{\partial x}\frac{\partial T}{\partial x} + \frac{\partial W}{\partial y}\frac{\partial T}{\partial y} - Wq_g\right)dx\,dy + \int_{\Gamma_h} Wh_c(T - T_\infty)\,d\Gamma_h - \int_{\Gamma_q} W\overline{q}d\Gamma_q \\ - \int_{\Gamma_r} W\left[\varepsilon_i\sigma\left(4\overline{T}_i^{\,3}T_i - 3\overline{T}_i^{\,4}\right) - \alpha_i H_i\right]d\Gamma_r = 0 \end{aligned} \tag{8}$$

Because the radiation boundary conditions make the problem non-linear, it is necessary to use an iterative technique to obtain the final solution. Term $\mathbf{T}_i^4$ is linearized by using the first two terms of its Taylor series expansion about $\overline{\mathbf{T}}_i$, the temperature from the previous iteration (the initial temperature is a guess for the first iteration).

Equation 8 is the "weak" form of the energy equation because it contains only first derivatives. The Galerkin form of the weighted residual method, used in THERM, relies on algebraic shape functions, $\mathbf{N}_i$, for the weighting function, $\mathbf{W} = \mathbf{N}_i$, and to approximate the temperature field, $\mathbf{T} = \mathbf{N}_i\mathbf{T}_i$. Equation 4 is set up for each element in a mesh. The integration over each element is approximated by a numerical integration using second-order Gaussian quadrature where the integration points are as shown in Figure C-3. The local systems of equations are combined into a global matrix, and a solution is obtained for the unknown nodal temperatures and heat fluxes.

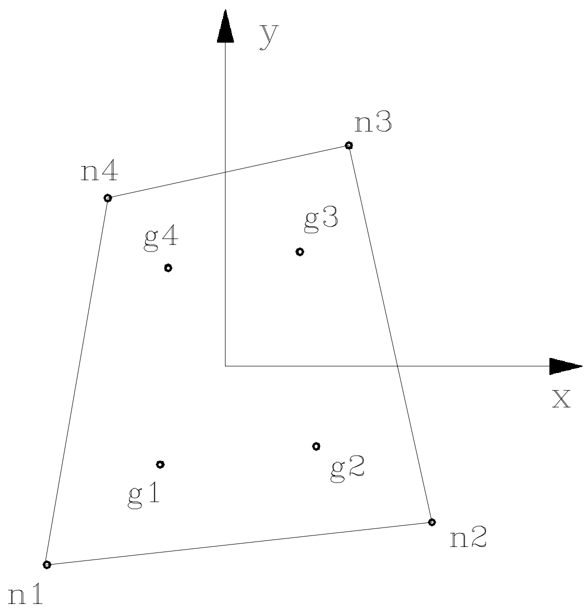


Figure C-3. Integration Points for Second-Order Gaussian Quadrature

Once the solution has been obtained on the initial mesh, it is sent to the error estimator, to check for convergence. The error estimator follows a published method [18,19]. Although a detailed description of this method is beyond the scope of this appendix we include a conceptual overview. Recall that the finite-element method solves the "weak" form of the energy equation, Eq. 4. We know that some points in the element have a higher order of convergence than others have, these are called superconvergent points. For quadrilateral elements used in CONRAD, the superconvergent points coincide with the Gaussian integration points, (g1-g4 in Figure C-3). The first step in error estimation is to use this knowledge of the finite-element solution to make a good guess at a better solution at all the nodal points. To accomplish this, a least squares method is used to fit a smooth function to the values of the gradient at the supercovergent points in a patch of elements around the node. The patch of elements around a node is defined as all the elements sharing that node. A better solution for the vector field of heat fluxes, $\mathbf{q_r}$, at the nodal points is recovered from this smooth function. The shape functions from the original problem are then used to interpolate the nodal solutions in order to obtain values for the recovered solutions throughout each element. Using the recovered fluxes, $\mathbf{q_r}$, and the original fluxes, $\mathbf{q}$, we calculate the contribution to the global error for each element as:

$$\|\mathbf{e}\| = \left(\int_\Omega (\mathbf{q_r} - \mathbf{q})^T (\mathbf{q_r} - \mathbf{q}) d\Omega \right)^{1/2} \tag{9}$$

If the global error is above a specified value, then the error estimator signals the mesh generator, and the mesh is refined in areas where the potential for error is high. The refined mesh is sent back to CONRAD, and a new solution is obtained. This process continues until the global error is less than the predetermined convergence value. The percent error energy norm is the maximum value of the error energy norm divided by the energy norm of the sum of the recovered fluxes and the error multiplied by 100:

$$\%\,\text{Error Energy Norm} = \frac{\|\mathbf{e}\|}{\|\mathbf{q_r}\|} \times 100 \tag{10}$$

NOMENCLATURE

$\mathbf{e}$	error estimate
h	Surface heat transfer coefficient
k	Thermal conductivity
n_x, n_y	Vector components of the outward facing normal to the boundary
q_g	Internal energy source
$\overline{q}$	Known heat flux on the boundary
$\mathbf{q}$	Nodal flux vector used in error estimation
T	Temperature
x,y	Spatial coordinates
N	Shape functions
$\Re$	Residual function
W	Weighting function
Γ	Boundary surface
Ω	Element or problem domain.
Subscripts	
∞	Ambient Conditions
Superscripts	
T	Transpose

Index

G

H

I

L

M

N

O

P

Q

R

S

T

U

V

Z